An $n$-sided polygonal edge-based smoothed finite element method ($n$ES-FEM) for solid mechanics

T. Nguyen-Thoi$^{1,3,*}$, G. R. Liu$^{1,2}$ and H. Nguyen-Xuan$^{2,3}$

$^1$Center for Advanced Computations in Engineering Science (ACES), Department of Mechanical Engineering, National University of Singapore, 9 Engineering Drive 1, Singapore 117576, Singapore
$^2$Singapore-MIT Alliance (SMA), E4-04-10, 4 Engineering Drive 3, Singapore 117576, Singapore
$^3$Department of Mechanics, Faculty of Mathematics and Computer Science, University of Science, Vietnam National University—HCM, 227 Nguyen Van Cu, District 5, Hochiminh City, Vietnam

SUMMARY

An edge-based smoothed finite element method (ES-FEM) using triangular elements was recently proposed to improve the accuracy and convergence rate of the existing standard finite element method (FEM) for the elastic solid mechanics problems. In this paper the ES-FEM is further extended to a more general case, $n$-sided polygonal edge-based smoothed finite element method ($n$ES-FEM), in which the problem domain can be discretized by a set of polygons, each with an arbitrary number of sides. The simple averaging point interpolation method is suggested to construct $n$ES-FEM shape functions. In addition, a novel domain-based selective scheme of a combined $n$ES/NS-FEM model is also proposed to avoid volumetric locking. Several numerical examples are investigated and the results of the $n$ES-FEM are found to agree well with exact solutions and are much better than those of others existing methods. Copyright © 2010 John Wiley & Sons, Ltd.

Received 7 July 2009; Revised 23 November 2009; Accepted 23 November 2009

KEY WORDS: numerical methods; finite element method (FEM); smoothed finite element methods (S-FEM); edge-based smoothed finite element method (ES-FEM); node-based smoothed finite element method (NS-FEM); polygonal element

1. INTRODUCTION

The strain smoothing technique has been proposed by Chen et al. [1] to stabilize the solutions in the context of the meshfree method [2, 3] and then applied in the natural element method [4]. Liu [5] has generalized the gradient (strain) smoothing technique and applied it in the meshfree context to formulate the node-based smoothed point interpolation method (NS-PIM or LC-PIM) [6, 7] and the node-based smoothed radial point interpolation method (NS-RPIM or LC-RPIM) [8]. Applying the same idea to the finite element method (FEM), a cell-based smoothed finite element method (SFEM or CS-FEM) [9–12] and a node-based smoothed finite element method (NS-FEM) [13, 14] have also been formulated.

In the CS-FEM, the domain discretization is still based on quadrilateral elements as in FEM; however, the stiffness matrices are calculated based over smoothing cells (SC) located inside the quadrilateral elements as shown in Figure 1. When the number of SC of the elements equals 1, the CS-FEM solution has the same properties with those of FEM using reduced integration.

$^*$Correspondence to: T. Nguyen-Thoi, Center for Advanced Computations in Engineering Science (ACES), Department of Mechanical Engineering, National University of Singapore, 9 Engineering Drive 1, Singapore 117576, Singapore.
$^1$E-mail: g0500347@nus.edu.sg

Copyright © 2010 John Wiley & Sons, Ltd.
Figure 1. Division of quadrilateral element into the smoothing cells (SCs) in CS-FEM by connecting the mid-segment-points of opposite segments of smoothing cells: (a) 1 SC; (b) 2 SCs; (c) 4 SCs; and (d) 8 SCs.

When SC approaches infinity, the CS-FEM solution approaches to the solution of the standard displacement compatible FEM model [10]. In practical calculation, using four smoothing cells for each quadrilateral element in CS-FEM is easy to implement, work well in general and hence advised for all problems. The numerical solution of CS-FEM (SC = 4) is always stable, accurate, much better than FEM, and often very close to the exact solutions. The CS-FEM has been developed for general n-sided polygonal elements (nCS-FEM) [15] dynamic analyses [16], incompressible materials using selective integration [17, 18], plate and shell analyses [19–23], and further extended for the extended finite element method (XFEM) to solve fracture mechanics problems in 2D continuum and plates [24].

In the NS-FEM, the strain smoothing domains and the integration of the weak form are performed over the smoothing domains associated with nodes. The NS-FEM works well for triangular elements, and can be applied easily to general n-sided polygonal elements [13, 14] for 2D problems and tetrahedral elements for 3D problems. For n-sided polygonal elements, the smoothing domain \( \Omega^{(k)} \) associated with the node \( k \) is created by connecting sequentially the mid-edge-point to the central points of the surrounding n-sided polygonal elements of the node \( k \) as shown in Figure 2. When only linear triangular or tetrahedral elements are used, the NS-FEM produces the same results as the method proposed by Dohrmann et al. [25] or to the LC-PIM [6] using linear interpolation. The NS-FEM [13, 14] has been found immune naturally from volumetric locking and possesses the upper bound property in strain energy as presented in [26]. Hence, by combining NS-FEM and FEM with a scale factor \( \alpha \in [0, 1] \), a new method named as the alpha finite element method (\( \alpha \)FEM) [27] is proposed to obtain nearly exact solutions in strain energy using triangular and tetrahedral elements. The NS-FEM has been developed for adaptive analysis [28], primal-dual shakedown [29], 2D and 3D visco-elastoplastic analyses [30]. One disadvantage of NS-FEM is its larger bandwidth of stiffness matrix compared with that of FEM, because the number of nodes related to the smoothing domains associated with nodes is larger than that related to the elements. The computational cost of NS-FEM therefore is larger than that of FEM for the same meshes used.

Recently, an edge-based smoothed finite element method (ES-FEM) [31] was also been formulated for static, free and forced vibration analyses in 2D problems. In the ES-FEM, the problem domain is also discretized using triangular elements as in FEM; however, the stiffness matrices are calculated based on smoothing domains associated with the edges of the triangles. For triangular elements, the smoothing domain \( \Omega^{(k)} \) associated with the edge \( k \) is created by connecting two endpoints of the edge to the centroids of the adjacent elements as shown in Figure 3. The numerical results demonstrated that the ES-FEM [31] possesses the following excellent...
properties: (1) ES-FEM is often found super-convergent and much more accurate than FEM using triangular elements (FEM-T3) and even more accurate than FEM using quadrilateral elements (FEM-Q4) with the same sets of nodes; (2) there are no spurious non-zeros energy modes and hence ES-FEM is both spatial and temporal stable and works well for vibration analysis; (3) no additional degree of freedom is used; (4) a novel domain-based selective scheme is proposed leading to a combined ES/NS-FEM model that is immune from volumetric locking and hence works very well for nearly incompressible materials. Note that similar to NS-FEM, the bandwidth of stiffness matrix in ES-FEM is larger than that in FEM-T3; hence, the computational cost of ES-FEM is larger than that of FEM-T3. However, when the efficiency of computation (computation time for the same accuracy) in terms of both energy and displacement error norms is considered, ES-FEM is more efficient [31]. ES-FEM has been developed for 2D piezoelectric [32], 2D visco-elastoplastic [33], plate [34] and primal–dual shakedown analyses [35]. In addition, the idea of the ES-FEM is quite straightforward to extend for the 3D problems using tetrahedral elements to give a so-called the face-based smoothed finite element method (FS-FEM) [36, 37].

In this paper, we aim to extend the ES-FEM to even more general case, i.e. $n$-sided polygonal edge-based smoothed finite element method (or $n$ES-FEM) in which the problem domain can be discretized by a set of polygons, each with an arbitrary number of sides. The simple averaging point interpolation method is suggested to construct $n$ES-FEM shape functions. In addition, a novel domain-based selective scheme of a combined $n$ES/NS-FEM model is also proposed to avoid volumetric locking. Several numerical examples are investigated and the results found agree well with exact solutions and are much better than those of other existing methods. Note that,
more information about the interpolants as well as the new integration techniques for the \( n \)-sided polygonal elements can be found in Refs \[38–43\].

The paper is outlined as follows. In Section 2, the idea of the \( n \)ES-FEM is introduced. In Section 3, the construction of \( n \)ES-FEM shape function is presented and the domain-based selective scheme \( n \)ES/NS-FEM for nearly incompressible materials is described briefly in Section 4. Section 5 presents numerical implementations of the \( n \)ES-FEM. Some numerical examples are presented and examined in Section 6 and some concluding remarks are made in the Section 7.

2. BRIEFING OF \( n \)ES-FEM

2.1. Briefing on the FEM \[44–46\]

The discrete equations of the FEM are generated from the Galerkin weakform and the integration is performed on the basis of element as follows:

\[
\int_{\Omega} (\nabla_s \delta u)^T D (\nabla_s u) \, d\Omega - \int_{\Omega} \delta u^T b \, d\Omega - \int_{\Gamma_t} \delta u^T \bar{t} \, d\Gamma = 0
\]

(1)

where \( b \) is the vector of external body forces, \( D \) is a symmetric positive-definite (SPD) matrix of material constants, \( \bar{t} \) is the prescribed traction vector on the natural boundary \( \Gamma_t \), \( u \) is trial functions, \( \delta u \) is test functions and \( \nabla_s u \) is the symmetric gradient of the displacement field.

The FEM uses the following trial and test functions:

\[
u^h(x) = \sum_{I=1}^{N_n} N_I(x) d_I, \quad \delta u^h(x) = \sum_{I=1}^{N_n} N_I(x) \delta d_I
\]

(2)

where \( N_n \) is the total number of nodes of the problem domain, \( d_I \) is the nodal displacement vector and \( N_I(x) \) is the shape function matrix of \( I \)th node.

By substituting the approximations, \( u^h \) and \( \delta u^h \), into the weakform and invoking the arbitrariness of virtual nodal displacements, Equation (1) yields the standard discretized system of algebraic equation:

\[
K_{FEM} d = f
\]

(3)

where \( K_{FEM} \) is the stiffness matrix, \( f \) is the element force vector that is assembled with entries of

\[
K_{ij}^{FEM} = \int_{\Omega} B_I^T D B_J \, d\Omega
\]

(4)

\[
f_I = \int_{\Omega} N_I^T(x) b \, d\Omega + \int_{\Gamma_t} N_I^T(x) \bar{t} \, d\Gamma
\]

(5)

with the strain–displacement matrix defined as

\[
B_I(x) = \nabla_s N_I(x)
\]

(6)

2.2. An \( n \)ES-FEM

In the \( n \)ES-FEM, the domain discretization is still based on polygonal elements of arbitrary number of sides, but the integration using strain smoothing technique \[1\] is performed based on the edges of the elements. In such an edge-based integration process, the problem domain \( \Omega \) is divided into smoothing domains associated with edges such that \( \Omega = \bigcup_{k=1}^{N_{ed}} \Omega^{(k)} \) and \( \Omega^{(i)} \cap \Omega^{(j)} = \emptyset \), \( i \neq j \), in which \( N_{ed} \) is the total number of edges of the problem domain. For \( n \)-sided polygonal elements, the smoothing domain \( \Omega^{(k)} \) associated with the edge \( k \) is created by connecting two endpoints of the edge to the two central points of the two adjacent elements as shown in Figure 4.
Applying the edge-based smoothing operation, the compatible strains $\varepsilon = \nabla \mathbf{u}$ used in Equation (1) is used to create a smoothed strain on the smoothing domain $\Omega^{(k)}$ associated with edge $k$:

$$\bar{\varepsilon}_k = \int_{\Omega^{(k)}} \varepsilon(x) \Phi_k(x) \, d\Omega = \int_{\Omega^{(k)}} \nabla \mathbf{u}(x) \Phi_k(x) \, d\Omega$$

(7)

where $\Phi_k(x)$ is a given smoothing function that satisfies at least unity property

$$\int_{\Omega^{(k)}} \Phi_k(x) \, d\Omega = 1$$

(8)

Using the following constant smoothing function:

$$\Phi_k(x) = \begin{cases} 1/A^{(k)}, & x \in \Omega^{(k)} \\ 0, & x \notin \Omega^{(k)} \end{cases}$$

(9)

where $A^{(k)}$ is the area of the smoothing domain $\Omega^{(k)}$, and applying a divergence theorem, one can obtain the smoothed strain

$$\bar{\varepsilon}_k = \frac{1}{A^{(k)}} \int_{\Gamma^{(k)}} \mathbf{n}^{(k)}(x) \mathbf{u}(x) \, d\Gamma$$

(10)

where $\Gamma^{(k)}$ is the boundary of the domain $\Omega^{(k)}$ as shown in Figure 4, and $\mathbf{n}^{(k)}(x)$ is the outward normal vector matrix on the boundary $\Gamma^{(k)}$ and has the following form for 2D problems:

$$\mathbf{n}^{(k)}(x) = \begin{bmatrix} n_x^{(k)} & 0 \\ 0 & n_y^{(k)} \\ n_y^{(k)} & n_x^{(k)} \end{bmatrix}$$

(11)

In the nES-FEM, the trial function $\mathbf{u}^h(x)$ is the same as in Equation (2) of the FEM and therefore the force vector $\mathbf{f}$ in the nES-FEM is calculated in the same way as in the FEM.
Substituting Equation (2) into Equation (10), the smoothed strain on the domain $\Omega^{(k)}$ associated with edge $k$ can be written in the following matrix form of nodal displacements:

$$\bar{\varepsilon}_k = \sum_{I \in N_n^{(k)}} \bar{B}_I(x_k) d_I$$

(12)

where $N_n^{(k)}$ is the total number of nodes of elements attached with the common edge $k$ and $\bar{B}_I(x_k)$ is termed as the smoothed strain–displacement matrix on the domain $\Omega^{(k)}$,

$$\bar{B}_I(x_k) = \begin{bmatrix} \bar{b}_{Ix}(x_k) & 0 \\ 0 & \bar{b}_{Iy}(x_k) \end{bmatrix}$$

(13)

and it is calculated numerically using

$$\bar{b}_{ih}(x_k) = \frac{1}{A^{(k)}} \int_{\Gamma^{(k)}} n^{(k)}_h(x) N_I(x) d\Gamma, \quad (h = x, y)$$

(14)

When a linear compatible displacement field along the boundary $\Gamma^{(k)}$ is used, one Gaussian point is sufficient for line integration along each segment of boundary $\Gamma_i^{(k)}$ of $\Omega^{(k)}$, the above equation can be further simplified to its algebraic form

$$\bar{b}_{ih}(x_k) = \frac{1}{A^{(k)}} \sum_{i=1}^{M} n^{(k)}_{ih} N_I(x^{GP}_i) J_i^{(k)}, \quad (h = x, y)$$

(15)

where $M$ is the total number of the boundary segments of $\Gamma_i^{(k)}$, $x^{GP}_i$ is the midpoint (Gaussian point) of the boundary segment of $\Gamma_i^{(k)}$, whose length and outward unit normal are denoted as $l_i^{(k)}$ and $n_{ih}^{(k)}$, respectively.

Equation (15) implies that only shape function values at some particular points along segments of boundary $\Gamma_i^{(k)}$ are needed and no explicit analytical form is required. This gives tremendous freedom in shape function construction. It is therefore that a simple averaging method is suggested to construct nES-FEM shape functions in Section 3.

The stiffness matrix $\bar{K}$ of the system is then assembled by a similar process as in the FEM

$$\bar{K}_{ij} = \sum_{k=1}^{N_{ed}} \bar{K}_{ij}^{(k)}$$

(16)

where $\bar{K}_{ij}^{(k)}$ is the stiffness matrix associated with edge $k$ and is calculated by

$$\bar{K}_{ij}^{(k)} = \int_{\Omega^{(k)}} \bar{B}_I^T \bar{D} \bar{B}_J \ d\Omega = \bar{B}_I^T \bar{D} \bar{B}_J A^{(k)}$$

(17)

3. CONSTRUCTION OF nES-FEM SHAPE FUNCTION

In the nES-FEM, by using the smoothed strain of the cell $\Omega^{(k)}$ associated with the edge $k$, the domain integration becomes line integration along the boundary $\Gamma^{(k)}$ of the cell. Only the shape functions themselves at some particular points along segments of boundary are used and no explicit analytical form is required. When a linear compatible displacement field along the boundary $\Gamma^{(k)}$ is used, one Gauss point at midpoint on each edge is sufficient for accurate boundary integration. The values of the shape functions at these Gauss points, e.g. $#1$ on the segment 1-A shown in Figure 5, are evaluated averagely using the values of shape functions of two related nodes on the segment: points #1 and #A.

In order to construct a linear compatible displacement field along the boundary $\Gamma^{(k)}$ of the cell, the following conditions of the shape functions for the discrete points of an $n$-sided polygonal
element need to be satisfied: (i) delta function: $N_i(x_j) = \delta_{ij}$; (ii) partition of unity: $\sum_{i=1}^{n} N_i(x) = 1$; (iii) linear consistency: $\sum_{i=1}^{n} N_i(x)x_i = x$; (iv) linear shape functions along lines connecting the central point and field points, e.g. line A-6 or line B-6; (v) values of the shape functions for the central points of $n$-sided polygonal elements, e.g. points #A or #B, are evaluated as

$$
\begin{bmatrix}
\frac{1}{n} & \frac{1}{n} & \ldots & \frac{1}{n}
\end{bmatrix} \quad \text{(size: } 1 \times n) 
$$

with the coordinates of the central points calculated simply using

$$
x_c = \frac{1}{n} \sum_{i=1}^{n} x_i, \quad y_c = \frac{1}{n} \sum_{i=1}^{n} y_i
$$

where the number of nodes $n$ of the $n$-sided polygonal element may be different from one element to the other and $x_i, y_i$ are the coordinates of nodes of the $n$-sided polygonal element.

The condition (iii) is essential to reproduce the linear polynomial fields such as in the standard patch test. With conditions (iv) and (v), a linear compatible displacement field along the boundary $\Gamma^{(k)}$ of the cell $\Omega^{(k)}$ associated with the edge $k$ is constructed.

Figure 5 and Table I demonstrate explicitly the shape function values at different points of the smoothing domain associated with the edge 1–6. The number of support nodes for the smoothing domain is 9 (from #1 to #9). We have 4 segments $\Gamma_i^{(k)}$ on $\Gamma^{(k)} (1A, 6B, B1)$. Each segment needs only one Gauss point, and therefore, there are a total of four Gauss points ($g_1, g_2, g_3, g_4$) used for the entire smoothing domain $\Omega^{(k)}$, and the shape function values at these four Gauss points can be tabulated in Table I by simple inspection.

The simple averaging shape functions presented above create a continuous displacement field on the whole domain of the problem. The strain field of these non-mapped shape functions includes the piecewise constant functions that are not continuous on the edges of elements and on the edges of smoothing cells. With such a construction, the simple averaging shape functions of the nES-FEM are similar to those in the nCS-FEM [15] and the nNS-FEM [13] using the $n$-sided polygonal elements.

It should be mentioned that the purpose of introducing the central points is to facilitate the evaluation of the shape function at the Gauss points and to ensure the linear compatible property along the edges of the cell connecting the central point and field points. No extra degrees of freedom are associated with these points. In other words, these points carry no additional field variables. This means that the nodal unknowns in the nES-FEM are the same as those in the FEM of the same mesh.

It is easy to see that the linear shape function of triangular elements of the standard FEM satisfy naturally the five above conditions. Hence, the nES-FEM can be applied easily to traditional triangular elements without any modification.
Different from the standard isoparametric finite elements, the nES-FEM obtains shape functions without using coordinate transformation or mapping. Hence, the field domain can be discretized in a much more flexible way.

4. A DOMAIN-BASED SELECTIVE SCHEME: A COMBINED nES/NS-FEM

Volumetric locking appears when the Poisson’s ratio approaches 0.5. The application of selective formulations in the conventional FEM [47] has been found effectively to overcome such a locking and hence the similar idea is employed in this paper. However, different from the FEM using selective integration [47], our selective scheme will use two different types of smoothing domains selectively for two different material ‘parts’ (µ-part and λ-part). Since the node-based smoothing domains used in the NS-FEM were found effective in overcoming volumetric locking [13] and the λ-part is known as the culprit of the volume locking, we use node-based domains for the λ-part and edge-based domains for the µ-part. The details are given below.

The material property matrix D for isotropic materials can be decomposed as

\[ D = D_1 + D_2 \] (20)

where \( D_1 \) relates to the shearing modulus \( \mu = E/(2(1+\nu)) \) and hence is termed as µ-part of D, and \( D_2 \) relates to Lame’s parameter \( \lambda = 2\mu\nu/(1-2\nu) \) and hence is termed as λ-part of D. For plane strain cases, we have

\[
D = \begin{bmatrix}
\lambda + 2\mu & \lambda & 0 \\
\lambda & \lambda + 2\mu & 0 \\
0 & 0 & \mu
\end{bmatrix} = \mu \begin{bmatrix}
2 & 0 & 0 \\
0 & 2 & 0 \\
0 & 0 & 1
\end{bmatrix} + \lambda \begin{bmatrix}
1 & 1 & 0 \\
1 & 1 & 0 \\
0 & 0 & 0
\end{bmatrix} = D_1 + D_2
\] (21)

and for axis-symmetric problems, we have

\[
\begin{align*}
D &= \mu \begin{bmatrix}
2 & 0 & 0 \\
0 & 2 & 0 \\
0 & 0 & 0
\end{bmatrix} + \lambda \begin{bmatrix}
1 & 1 & 0 \\
1 & 1 & 0 \\
0 & 0 & 0
\end{bmatrix} = D_1 + D_2
\end{align*}
\] (22)

Copyright © 2010 John Wiley & Sons, Ltd.

DOI: 10.1002/cnm
In our domain-based selective scheme proposed, we use the \( n \text{NS-FEM} \) to calculate the stiffness matrix related to \( \lambda \)-part and the \( n \text{ES-FEM} \) to calculate the one related to the \( \mu \)-part. The stiffness matrix of the combined \( n \text{ES/NS-FEM} \) model becomes

\[
\bar{K} = \sum_{i=1}^{N_{\text{ed}}} (\bar{B}_i^1)^T D_1 \bar{B}_i^1 A_i^1 + \sum_{j=1}^{N_n} (\bar{B}_j^2)^T D_2 \bar{B}_j^2 A_j^2
\]

where \( \bar{B}_i^1 \) and \( A_i^1 \) are the smoothed strain–displacement matrix and area of the smoothing domain \( \Omega_{\text{ed}}^i \) associated with edge \( i \), correspondingly; \( \bar{B}_j^2 \) and \( A_j^2 \) are the smoothed strain–displacement matrix and area of the smoothing domain \( \Omega_{n}^{(j)} \) associated with node \( j \), correspondingly; \( N_n \) is the total number of nodes of the problem domain.

5. NUMERICAL IMPLEMENTATION

5.1. Domain discretization with polygonal elements of the Voronoi diagram

The discretization of a problem domain into \( n \)-sided convex polygonal elements can be performed using the Voronoi diagram, and the procedure can be described as follows [48]. The problem domain and its boundaries are first discretized by a set of properly scattered points \( P := \{p_1, p_2, \ldots, p_n\} \). Based on the given points, the domain is further decomposed into the same number of Voronoi cells \( C := \{C_1, C_2, \ldots, C_n\} \) according to the nearest-neighbor rule defined by

\[
C_i = \{x \in \mathbb{R}^2 : d(x, x_i) < d(x, x_j), \forall j \neq i\} \quad \forall i
\]

In numerical implementation, as illustrated in Figure 6(a), the obtained Voronoi diagram has not cell vertices along the boundaries, then the artificial nodes along the boundary outside the domain need to be added to enclose the cells at the boundaries. With this, the boundary cells are then bounded, but the vertices of these cells do not fall within the problem domain as shown in Figure 6(b). Next, these vertices will be ‘shifted’ inwards onto the boundaries as shown in Figure 6(c). The final shape of these Voronoi cells is generally irregular but they are all convex polygons. The initial point \( p_i \) is regarded as the representative point of the \( i \)-th element. Once we get the information of these Voronoi diagrams, a set of polygonal elements is then formed for our numerical analysis.

![Figure 6. (a) Voronoi diagram without adding the artificial nodes along the boundary outside the domain; (b) Voronoi diagram with the artificial nodes added along the boundary outside the domain; and (c) final Voronoi diagram.](www.goldenbase.vn)
The following points need to be noted: (i) The original discrete points \( P \) only serve as numerical devices for domain decomposition and do not function in the following numerical analysis; (ii) If we prefer more regular elements, such as rectangular elements, hexagon elements, we need to arrange a special point pattern \( P \) before the generation of Voronoi diagrams; (iii) For demonstration purpose, we arrange the initial points in an arbitrary form in the following numerical examples regardless of the issue of computational cost. As a result, the number of elements sides is generally changing from element to element; (iv) The \( n \)ES-FEM does not require the elements to be convex; (v) Triangular elements used in the FEM can be used directly in the \( n \)ES-FEM without any alterations.

5.2. Procedure of the \( n \)ES-FEM

The numerical procedure for the \( n \)ES-FEM is briefly as follows:

1. Divide the domain into a set of elements and obtain information on nodes’ coordinates and element connectivity;
2. Determine the area of cells \( \Omega^{(k)} \) associated with edge \( k \) and find neighboring elements of each edge;
3. Loop over all the edges:
   (a) Determine the nodal connecting information of cell \( \Omega^{(k)} \) associated with edge \( k \);
   (b) Determine the outward unit normal of each boundary side for cell \( \Omega^{(k)} \);
   (c) Compute the matrix \( \bar{B}_I(x_k) \) using Equation (13);
   (d) Evaluate the stiffness matrix using Equation (17) and force vector of the current cell;
   (e) Assemble the contribution of the current smoothing domain to form the system stiffness matrix using Equations (16) and force vector;
4. Implement essential boundary conditions;
5. Solve the system of equations to obtain the nodal displacements;
6. Evaluate strains and stresses at nodes of interest.

In the \( n \)ES-FEM, the value of strains (or stresses) at the node \( i \) will be average value of strains (or stresses) of the smoothing domains \( \Omega^{(k)} \) associated with edges \( k \) around the node \( i \), and is calculated numerically by

\[
\bar{\varepsilon}_i = \frac{1}{A^{(i)}} \sum_{k=1}^{N_{ed}^{(i)}} \bar{\varepsilon}_k A^{(k)}
\]  

(25)

where \( N_{ed}^{(i)} \) is the total number of edges connecting directly to node \( i \), \( \bar{\varepsilon}_k \) and \( A^{(k)} \) are the strain and the area of the smoothing domain \( \Omega^{(k)} \) associated with edge \( k \) around the node \( i \), respectively, and \( A^{(i)} = \sum_{k=1}^{N_{ed}^{(i)}} A^{(k)} \).

5.3. Rank test for the stiffness matrix

Property 1: The \( n \)ES-FEM possesses only ‘legal’ zero-energy modes that represents the rigid motions, and there exists no spurious zero-energy mode. This is ensured by the following reasons:

(i) The numerical integration used to evaluate Equation (17) in the \( n \)ES-FEM satisfies the necessary condition given in Section 6.1.3 in Reference [2]. This is true for all possible \( n \)ES-FEM models, as detailed in Table II.
(ii) The edge strain smoothing operation ensures linearly independent columns (or rows) in the stiffness matrix due to the independence of these smoothing domains.
(iii) The shape functions used in the \( n \)ES-FEM are of partition of unity.
Table II. Existence of spurious zero-energy modes in an element.

<table>
<thead>
<tr>
<th>Type of element</th>
<th>nES-FEM</th>
<th>FEM with reduced integration</th>
</tr>
</thead>
<tbody>
<tr>
<td>Triangle</td>
<td>$n_Q = 3, N_Q = 3 \times n_Q = 9, n_t = 3, N_u = 2 \times n_t = 6$</td>
<td>$n_Q = 1, N_Q = 3 \times n_Q = 3, n_t = 3, N_u = 2 \times n_t = 6$</td>
</tr>
<tr>
<td></td>
<td>$N_Q &gt; N_u - N_R$</td>
<td>$N_Q = N_u - N_R$</td>
</tr>
<tr>
<td></td>
<td>$\Rightarrow$ spurious zero energy modes not possible</td>
<td>$\Rightarrow$ spurious zero energy modes possible</td>
</tr>
<tr>
<td>Quadrilateral</td>
<td>$n_Q = 4, N_Q = 3 \times n_Q = 12, n_t = 4, N_u = 2 \times n_t = 8$</td>
<td>$n_Q = 1, N_Q = 3 \times n_Q = 3, n_t = 4, N_u = 2 \times n_t = 8$</td>
</tr>
<tr>
<td></td>
<td>$N_Q &gt; N_u - N_R$</td>
<td>$N_Q &lt; N_u - N_R$</td>
</tr>
<tr>
<td></td>
<td>$\Rightarrow$ spurious zero energy modes not possible</td>
<td>$\Rightarrow$ spurious zero energy modes possible</td>
</tr>
<tr>
<td>$n$-sided polygon</td>
<td>$n_Q = n, N_Q = 3 \times n_Q = 3n, n_t = n, N_u = 2 \times n_t = 2n$</td>
<td>Not applicable</td>
</tr>
<tr>
<td></td>
<td>$N_Q &gt; N_u - N_R$</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\Rightarrow$ spurious zero energy modes not possible</td>
<td></td>
</tr>
</tbody>
</table>

Note: $N_R$: number of DOFs of rigid motion; $n_t$: number of nodes; $n_Q$: number of quadrature points/cells; $N_u$: number of total DOFs; $N_Q$: number of independent equations.

Therefore, no deformed zero-energy mode will appear in the nES-FEM. In other words, any deformation (except the rigid motions) will result in strain energy in an nES-FEM model.

5.4. Standard patch test

Satisfaction of the standard patch test requires that the displacements of all the interior nodes follow ‘exactly’ (to machine precision) the same linear function of the imposed displacement. A domain discretization of a square patch using 36 $n$-sided polygonal elements is shown in Figure 7. The
following error norm in displacements is used to examine the computed results

\[ e_d = \frac{\sum_{i=1}^{2N} |u_i - u_i^h|}{\sum_{i=1}^{2N} |u_i|} \times 100\% \]  

(26)

where \( u_i \) is the exact solution and \( u_i^h \) is the numerical solution.

The parameters are taken as \( E = 100, \nu = 0.3 \) and the linear displacement field is given by

\[ u = x \]
\[ v = y \]  

(27)

It is found that the nES-FEM can pass the standard patch test within machine precision with the error norm in displacements: \( e_d = 2.56e^{-12} \) (%).

6. NUMERICAL EXAMPLES

In this section, some example will be analyzed to demonstrate the effectiveness, accuracy and convergence properties of the present method. The results of the nES-FEM will be compared with those of the n-sided polygonal element-based smoothed finite element method (nCS-FEM) [15] and the n-sided polygonal node-based smoothed finite element method (nNS-FEM) [13]. In order to study the convergence rate of the present method, two norms are used here. The displacement error norm is given in Equation (26) and the energy error norm is calculated by

\[ e_e = \left( \frac{1}{2} \sum_{k=1}^{N_s} (\varepsilon - \bar{\varepsilon}^{(k)})^T D (\varepsilon - \bar{\varepsilon}^{(k)}) A^{(k)} \right)^{1/2} \]  

(28)

where \( \bar{\varepsilon}^{(k)} \) and \( A^{(k)} \) are the smoothing strain and the area of the smoothing domain \( \Omega^{(k)} \), respectively; \( \varepsilon \) is the analytical strain calculated at the nodes (for nNS-FEM), or at edge-mid-points (for nES-FEM), or at the central points of smoothing domains \( \Omega^{(k)} \) (for nCS-FEM); \( N_s \) is the total number of smoothing domains.
number of smoothing domains. Note that the convergence rates of the error norms are calculated based on the average length of sides of the elements by

\[ h = \sqrt{\frac{A_\Omega}{N_e}} \quad (29) \]

where \( A_\Omega \) is the area of the whole problem domain.

6.1. Cantilever loaded at the end

A cantilever with length \( L \) and height \( D \) is studied as a benchmark here, which is subjected to a parabolic traction at the free end as shown in Figure 8. The cantilever is assumed to have a unit thickness so that plane stress condition is valid. The analytical solution is available and can be found in a textbook by Timoshenko and Goodier \[49\].

\[
\begin{align*}
ux &= \frac{Py}{6EI} \left[ (6L - 3x)x + (2 + \nu) \left( y^2 - \frac{D^2}{4} \right) \right] \\
uy &= -\frac{P}{6EI} \left[ 3vy^2(L - x) + (4 + 5\nu) \frac{D^2x}{4} + (3L - x)x^2 \right]
\end{align*}
\]  

(30)

where the moment of inertia \( I \) for a beam with rectangular cross section and unit thickness is given by \( I = D^3/12 \).

The stresses corresponding to the displacements equation (30) are

\[
\begin{align*}
\sigma_{xx}(x, y) &= \frac{P(L - x)y}{I}, & \sigma_{yy}(x, y) &= 0, & \tau_{xy}(x, y) &= -\frac{P}{2I} \left( \frac{D^2}{4} - y^2 \right)
\end{align*}
\]  

(31)

The related parameters are taken as \( E = 3.0 \times 10^7 \) kPa, \( \nu = 0.3 \), \( D = 12 \) m, \( L = 48 \) m and \( P = 1000 \) N. In the computations, the nodes on the left boundary are constrained using the exact displacements obtained from Equation (30) and the loading on the right boundary uses the distributed parabolic shear stresses in Equation (31). The domain discretization using \( n \)-sided polygonal elements is shown in Figure 9.
Figure 10 shows that all the computed stresses using nES-FEM agree well with the analytical solutions. The convergence of the strain energy is shown in Figure 11. It is seen that the nCS-FEM model behaves overly stiff and hence gives a lower bound, and nNS-FEM behaves overly soft and gives an upper bound. The nES-FEM has a very close-to-exact stiffness and hence is most accurate. The convergences of displacement and energy norms are demonstrated in Figures 12 and 13, respectively. It is observed that nES-FEM not only has smaller errors than those of nNS-FEM and nCS-FEM, but also has higher convergence rates in both norms.

Figure 15 shows the influence of the point distribution in the nES-FEM by comparing the evolution of strain energy of two sets of meshes: one set with regular meshes as shown in Figure 14.

Figure 10. Normal stress $\sigma_{xx}$ and shear stress $\sigma_{xy}$ along the section of $x=0$ using nES-FEM for the cantilever subjected to a parabolic traction at the free end.

Figure 11. Convergence of the strain energy solution of nES-FEM in comparison with other methods for the cantilever subjected to a parabolic traction at the free end.
and one set with irregular meshes as shown in Figure 9. It is seen that the regular meshes give more accurate results than the irregular meshes.

Figure 16 compares the computation time of three methods using the same direct full-matrix solver. It is found that with the same sets of nodes, the computation time of the $n$ES-FEM is only shorter than that of the $n$NS-FEM. However, when the efficiency of computation (CPU time for the same accuracy) in terms of both displacement and energy norms is considered as shown in Figures 17 and 18, the $n$ES-FEM model stands out clearly as a winner.

Table III compares the evolution of the conditioning number with mesh density between $n$CS-FEM and $n$ES-FEM. It is seen that the conditioning numbers of $n$ES-FEM are little less than those of $n$CS-FEM.

6.2. Semi-infinite plane

The semi-infinite plane shown in Figure 19 is studied subjected to a uniform pressure within a finite range ($-a \leq x \leq a$). The plane strain condition is considered. The analytical stresses are given by [49]

$$
\sigma_{11} = \frac{p}{2\pi} [2(\theta_1 - \theta_2) - \sin 2\theta_1 + \sin 2\theta_2] \\
\sigma_{22} = \frac{p}{2\pi} [2(\theta_1 - \theta_2) + \sin 2\theta_1 - \sin 2\theta_2] \\
\tau_{12} = \frac{p}{2\pi} [\cos 2\theta_1 - \cos 2\theta_2]
$$

The directions of $\theta_1$ and $\theta_2$ are indicated in Figure 19. The corresponding displacements can be expressed as

$$
u_1 = \frac{p(1-v^2)}{\pi E} \left[ \frac{1-2v}{1-v} \left( (x+a)\theta_1 - (x-a)\theta_2 \right) + 2y \ln \frac{r_1}{r_2} \right]
$$

$$
u_2 = \frac{p(1-v^2)}{\pi E} \left[ \frac{1-2v}{1-v} \left( y(\theta_1 - \theta_2) + 2H \arctan \frac{1}{c} \right) + 2(x-a) \ln r_2 \\
- 2(x+a) \ln r_1 + 4a \ln a + 2a \ln (1+c^2) \right]
$$
Figure 13. Error in displacement norm of $n$ES-FEM in comparison with other methods for the cantilever subjected to a parabolic traction at the free end.

Figure 14. A domain discretization using regular meshes of the cantilever subjected to a parabolic traction at the free end using $n$-sided polygonal elements.

Figure 15. Influence of the point distribution in the strain energy solution of the $n$ES-FEM for the cantilever subjected to a parabolic traction at the free end.
Figure 16. Comparison of the computation time of different methods for solving the cantilever subjected to a parabolic traction at the free end. For the same distribution of nodes, the nCS-FEM is the fastest to deliver the results.

Figure 17. Comparison of the efficiency (computation time for the solutions of same accuracy measured in displacement norm) for solving the cantilever subjected to a parabolic traction at the free end. The nES-FEM stands out as a winner.

where $H=ca$ is the distance from the origin to point $O'$, the vertical displacement is assumed to be zero and $c$ is a coefficient.

Owing to the symmetry about the $y$-axis, the problem is modeled with a $5a \times 5a$ square with $a=0.2\text{m}$, $c=100$ and $p=1\text{MPa}$. The left side is subjected the symmetric boundary condition of displacement and the bottom side is constrained using the exact displacements given by Equation (32). The right side is subjected to tractions computed from Equation (33). Figure 20 gives the discretization of the domain using $n$-sided polygonal elements.

From Figures 21 and 22, it is observed that all the computed displacements and stresses using nES-FEM agree well with the analytical solutions. It is also noticed that the present stresses are

Figure 18. Comparison of the efficiency of computation time in terms of energy norm of the cantilever subjected to a parabolic traction at the free end. The nES-FEM stands out clearly as a winner.

Table III. Comparison of the evolution of the conditioning number with mesh density between nCS-FEM and nES-FEM.

<table>
<thead>
<tr>
<th>Mesh density</th>
<th>16 x 4</th>
<th>24 x 6</th>
<th>32 x 8</th>
<th>40 x 10</th>
<th>48 x 12</th>
</tr>
</thead>
<tbody>
<tr>
<td>nCS-FEM</td>
<td>6.58e+11</td>
<td>8.25e+13</td>
<td>1.22e+14</td>
<td>1.99e+13</td>
<td>2.25e+14</td>
</tr>
<tr>
<td>nES-FEM</td>
<td>6.46e+11</td>
<td>8.18e+13</td>
<td>1.21e+14</td>
<td>1.96e+13</td>
<td>2.20e+14</td>
</tr>
</tbody>
</table>

Figure 19. Semi-infinite plane subjected to a uniform pressure.

very smooth though no post-processing is performed for them. The convergence of the strain energy is shown in Figure 23. Again, it is seen that the nCS-FEM gives a lower bound, nNS-FEM gives an upper bound and nES-FEM is most accurate. The convergence rates in displacement and stresses are displayed in Figure 24 and Figure 25, respectively. It is observed again that both displacement
and energy norms of the nES-FEM not only have smaller errors than those of nNS-FEM and nCS-FEM, but also have the faster convergence rate.

Figure 26 shows the error in displacement for nearly incompressible material when Poisson’s ratio is changed from 0.4 to 0.4999999. The results show that the proposed selective scheme in Section 4 can overcome the volumetric locking for nearly incompressible materials and gives better results than those of the nNS-FEM.

6.3. A cylindrical pipe subjected to an inner pressure

Figure 27 shows a thick cylindrical pipe, with internal radius \(a = 0.1\) m, external radius \(b = 0.2\) m, subjected to an internal pressure \(p = 600\) kN/m\(^2\). Because of the axi-symmetric characteristic of the problem, we only calculate for one quarter of cylinder as shown in Figure 27, and Figure 28 gives the discretization of the domain using \(n\)-sided polygonal elements. Plane strain condition is considered and Young’s modulus \(E = 21\ 000\) kN/m\(^2\), Poisson’s ratio \(\nu = 0.3\). Symmetric conditions are imposed on the left and bottom edges, and outer boundary is traction free. The exact solution for the stress components is [49]

\[
\sigma_r(r) = \frac{a^2 p}{b^2 - a^2} \left(1 - \frac{b^2}{r^2}\right), \quad \sigma_\phi(r) = \frac{a^2 p}{b^2 - a^2} \left(1 + \frac{b^2}{r^2}\right), \quad \sigma_{r\phi} = 0
\]  

whereas the radial and tangential exact displacements are given by

\[
u_r(r) = \frac{(1 + \nu)a^2 p}{E(b^2 - a^2)} \left\{(1 - 2\nu)r + \frac{b^2}{r}\right\}, \quad u_\phi = 0
\]

where \((r, \phi)\) are the polar coordinates, and \(\phi\) is measured counter-clockwise from the positive \(x\)-axis.

From Figure 29, it is observed again that all the computed displacements and stresses agree well with the analytical solutions. The present stresses are also very smooth. The convergence
of the strain energy is shown in Figure 30. It is again seen that the $n$CS-FEM gives a lower bound, $n$NS-FEM gives an upper bound and $n$ES-FEM is most accurate. The convergence rates in displacement and stresses are displayed in Figures 31 and 32, respectively. Once again, it is observed that both displacement and energy norms of the $n$ES-FEM not only have smaller errors than those of $n$NS-FEM and $n$SFEM, but also have the faster convergence rate.

Figure 34 shows the influence of the point distribution in the $n$ES-FEM by comparing the evolution of strain energy of two sets of meshes: one set with regular meshes as shown in Figure 33 and one set with irregular meshes as shown in Figure 28. It is seen that the regular meshes give more accurate results than the irregular meshes.

Copyright © 2010 John Wiley & Sons, Ltd.


DOI: 10.1002/cnm
7. CONCLUSIONS

ES-FEM [31] was recently proposed to improve the accuracy and convergence rate of the standard FEM. In this work, the ES-FEM is extended to more general case, i.e. n-sided polygonal edge-based smoothed finite element method (or nES-FEM) in which the problem domain can be discretized by a set of polygons, each with an arbitrary number of sides. The simple averaging method is suggested
Figure 25. Error in displacement norm of nES-FEM in comparison with other methods for the semi-infinite plane problem.

Figure 26. Error in displacement norm for different Poisson’s ratios of the selective nES/NS-FEM in comparison with other methods for the semi-infinite plane problem.

Figure 27. A thick cylindrical pipe subjected to an inner pressure and its quarter model.
to construct $n$ES-FEM shape functions. The novel domain-based selective scheme ES/NS-FEM proposed in the ES-FEM is extended to the $n$ES-FEM to give a combined $n$ES/NS-FEM model that is immune from volumetric locking. Several numerical examples are investigated and the results of the $n$ES-FEM are found to agree well with exact solutions and are much better than those of others existing methods in displacement and energy norms, both in the accuracy and the convergence rate. In addition, in comparison of the stiffness of models, the $n$ES-FEM is softer than the $n$CS-FEM (which gives a lower bound) and stiffer than the $n$NS-FEM (which gives an upper bound). The $n$ES-FEM is hence between the $n$CS-FEM and $n$NS-FEM, and is closer than the exact solution compared with the $n$CS-FEM and $n$NS-FEM. Depending on the specific problems,

Figure 28. Discretization of the domain using $n$-sided polygonal elements of the thick cylindrical pipe subjected to an inner pressure.

Figure 29. Computed and exact results of nodes along the $x$-axis ($y=0$) of the thick cylindrical pipe subjected to an inner pressure: (a) radial displacement $u_r$; (b) radial stress $\sigma_r$ and tangential stress $\sigma_\theta$. 
Figure 30. Convergence of the strain energy solution of nES-FEM in comparison with other methods for the thick cylindrical pipe subjected to an inner pressure.

Figure 31. Error in energy norm of nES-FEM in comparison with other methods for the thick cylindrical pipe subjected to an inner pressure.

Figure 32. Error in displacement norm of nES-FEM in comparison with other methods for the thick cylindrical pipe subjected to an inner pressure.
Figure 33. A domain discretization using regular meshes of the thick cylindrical pipe subjected to an inner pressure.

Figure 34. Influence of the point distribution in the strain energy solution of the nES-FEM for the thick cylindrical pipe subjected to an inner pressure.

Figure 35. Error in displacement norm for different Poisson’s ratios of the selective nES/NS-FEM in comparison with other methods for the thick cylindrical pipe subjected to an inner pressure.
the strain energy of nES-FEM can be overestimated or underestimated compared with the exact solution.

REFERENCES


