An s-version finite element method without generation of coupling stiffness matrix by using iterative technique

Yosuke YUMOTO*, Yasunori YUSA** and Hiroshi OKADA***

* Department of Mechanical Engineering, Graduate School of Science and Technology, Tokyo University of Science
** Department of Mechanical Engineering, Faculty of Science and Technology, Tokyo University of Science
2641 Yamazaki, Noda, Chiba 278-8510, Japan
E-mail: yyusa@rs.tus.ac.jp

Received 29 December 2015

Abstract

In s-version finite element method (s-FEM) proposed by Fish (1992), a local mesh that represents the local feature such as a hole or a crack is superposed on a global mesh that represents the shape of the whole analysis model. The interaction between global and local meshes is represented by coupling stiffness matrices. Since the global and local meshes can be generated independently, mesh generation efforts are reduced remarkably. However, s-FEM has a common issue. The generation of coupling stiffness matrices takes a considerable amount of program development efforts, which include constructing accurate cross-element integration methodology and programming it for various element types. For such an issue, we propose an iterative s-FEM that does not require the generation of coupling stiffness matrices at all. The coupling term is now evaluated by global and local stresses that are computed on the respective mesh and then transferred to the other by interpolation techniques. The global and local stresses are treated as initial stress in the finite element computations. The global and local analyses are performed alternately under assumed initial stress, and converged solution is achieved by iteration with a monitored residual being sufficiently small. In proposed iterative s-FEM, an issue about linear independence of global and local elements, which is known to occur in the original s-FEM, does not occur. In numerical experiments, converged solution was successfully obtained within several hundred iteration counts. Accurate stress distribution for a stress concentration problem and an accurate stress intensity factor for a linear elastic fracture mechanics problem were computed by proposed iterative s-FEM. In addition, several stress interpolation techniques were compared in the numerical experiments. Nearest neighbor interpolation for the global stress and local least squares interpolation for the local stress showed good convergence and accurate solution.

Key words: S-version finite element method, Coupling stiffness matrix, Iterative method, Initial stress, Local least squares interpolation, Nearest neighbor interpolation, Stress concentration, Stress intensity factor

1. Introduction

In s-version finite element method (s-FEM) proposed by Fish (1992), a local mesh that represents the local feature such as a hole or a crack is superposed on a global mesh that represents the shape of the whole analysis model. The local feature does not have to be modeled by the global mesh at all (Nakasumi et al., 2001). The interaction between global and local meshes is represented by coupling stiffness matrices. Since the global and local meshes can be generated independently, mesh generation efforts, which are drawing a dominant bottleneck in recent computer-aided engineering (Arai et al., 2015), are reduced remarkably. S-FEM has been applied to basic problems (Fish, 1992; Fish and Markolefas, 1993), multiscale structural problems (Suzuki et al., 1999), linear elastic fracture problems (Okada et al., 2005; Maitireyimu et al., 2009; Kamaya et al., 2010; Wada et al., 2014), elastic–plastic fracture problems (Okada et al., 2007), and composite material problems (Okada et al., 2004; Tanaka et al., 2006; Kikuchi et al., 2014). However, s-FEM has a common issue. The generation of coupling stiffness matrices takes a considerable amount of program development efforts (Okada et al., 2004; Tanaka et al., 2006; Kikuchi et al., 2014), which include constructing accurate cross-element integration methodology and programming it for various element types. This is because finite element integration computation is troublesome.
when a local element is partly superposed on a part of a global element. Information about how the elements overlap each other needs to be obtained and numerical integration schemes must be constructed accordingly. Thus, processes depend on the types of finite elements adopted in the analysis. That is why the generation of coupling stiffness matrices when the elements partly overlap each other is very troublesome.

Here, several methods that utilize such global and local meshes with iterative technique have been proposed. Suzuki et al. (1999; 2002) modified the s-FEM using iterative technique. A block linear equation system generated by s-FEM is solved by an iterative method. In this linear equation system solution procedure, the global and local analyses are performed independently and iteratively. Iterative substructure method (Murakawa et al., 2005; Nishikawa et al., 2007) employs superposed global and local meshes. The two meshes are analyzed alternately under assumed enforced displacement and external force boundary condition on the global–local interface. Converged solution satisfies both geometrical continuity and force equilibrium on the global–local interface. Partitioned coupling method (Yusa et al., 2012; Yusa and Yoshimura, 2013; 2014) adopts non-overlapped global and local meshes. The global and local analyses are performed similarly to the iterative substructure method. SGBEM–FEM (symmetric Galerkin boundary element method–finite element method) alternating method (Nikishkov et al., 2001; Han and Atluri, 2002) uses boundary elements for the local mesh with a crack and finite elements for the global mesh. Assumed external force boundary condition on the global–local interface and on the crack surface is updated by iteration. Such global–local alternating procedures are called block Gauss–Seidel method (Suzuki et al., 1999; 2002; Yusa et al., 2012; Yusa and Yoshimura, 2013).

In this paper, an s-FEM with iterative technique is proposed. Proposed iterative s-FEM does not require the generation of coupling stiffness matrices at all. Thus, proposed method is different from the iterative s-FEM proposed by Suzuki et al. (1999; 2002) that requires coupling stiffness matrices. Suzuki et al. (1999; 2002) solved, using an iterative method, a monolithic linear equation system that contains generated coupling stiffness matrices. In present approach, information about how the elements in global and local meshes overlap each other is not necessary to express their mechanical interactions. Instead, we use the information about the locations of integration points. We can avoid the generation of coupling stiffness matrices, greatly simplifying the numerical processes and reducing the program development efforts. Hence, with our iterative s-FEM, one can easily perform the s-FEM simulations utilizing existing finite element programs without the generation of coupling stiffness matrices. In the next section, formulation of the original s-FEM is explained first, followed by that of proposed iterative s-FEM. Then, numerical experiments of a patch test problem, a stress concentration problem, and a linear elastic fracture mechanics problem are performed to demonstrate the performance of proposed method.

2. Method
2.1. Original s-version finite element method

In s-version finite element method (s-FEM) proposed by Fish (1992), a local mesh is superposed on a global mesh as shown in Fig. 1. The local mesh is usually finer than the global mesh in order to accurately represent stress concentration near the local feature such as a hole or a crack. A local domain, $\Omega^L$, a global domain, $\Omega^G$, and a global–local interface, $\Gamma^{GL}$, are defined. $\Omega^G$ denotes the whole problem domain. It is noted that $\Omega^G$ includes $\Omega^L$ and that the global mesh ranges on $\Omega^G$ as well as $\Omega^L$. Displacement and traction are prescribed over the boundaries, $\Gamma_u$ and $\Gamma_t$, respectively. In the s-FEM, a continuous displacement field, $u$, is assumed to be

$$
\begin{align*}
u &= \begin{cases} u^G & \text{in } \Omega^G \setminus \Omega^L, \\ u^G + u^L & \text{in } \Omega^L, \end{cases} \\
\end{align*}
$$

(1)

where $u^G$ and $u^L$ are displacement fields on global and local meshes, respectively. To satisfy geometrical continuity,

$$
\begin{align*}
u^L &= 0 \\
\end{align*}
$$

(2)

on $\Gamma^{GL}$. In addition, the variation of the displacement field can also be written in the same manner, as:

$$
\begin{align*}
\delta u &= \begin{cases} \delta u^G & \text{in } \Omega^G \setminus \Omega^L, \\
\delta u^G + \delta u^L & \text{in } \Omega^L, \end{cases} \\
\end{align*}
$$

(3)

and thus

$$
\begin{align*}
\delta u^L &= 0 \\
\end{align*}
$$

(4)

on $\Gamma^{GL}$. Here, the principle of virtual work is introduced, as:

$$
\int_{\Omega^G} \delta u^T D e d\Omega = \int_{\Gamma_u} \delta u^T t d\Gamma + \int_{\Gamma_t} \delta u^T b d\Omega, \\
$$

(5)
where $\varepsilon = \frac{1}{2} \left\{ \frac{\partial \mathbf{u}}{\partial x} + \left( \frac{\partial \mathbf{u}}{\partial x} \right)^T \right\}$ is strain, $\delta \varepsilon = \frac{1}{2} \left\{ \frac{\partial \delta \mathbf{u}}{\partial x} + \left( \frac{\partial \delta \mathbf{u}}{\partial x} \right)^T \right\}$ is the variation of the strain, $\mathbf{D}$ is a D matrix, $\mathbf{t}$ is prescribed traction on $\Gamma_t$, and $\mathbf{b}$ is external body force. In $\Omega^G \setminus \Omega^L$, inserting $\mathbf{u}$, $\delta \mathbf{u}$, $\varepsilon$, and $\delta \varepsilon$ into Eq. (5), one can derive

$$
\int_{\Omega^G} \delta \varepsilon^G \mathbf{D} \varepsilon^G \mathrm{d} \Omega = \int_{\Gamma_t} \delta \mathbf{u}^G \mathbf{t} \mathrm{d} \Gamma + \int_{\Omega^G} \delta \mathbf{u}^G \mathbf{b} \mathrm{d} \Omega.
$$

In $\Omega^L$, one can similarly derive

$$
\int_{\Omega^L} \delta \varepsilon^l \mathbf{D} \varepsilon^l \mathrm{d} \Omega = \int_{\Gamma_t} \delta \mathbf{u}^l \mathbf{t} \mathrm{d} \Gamma + \int_{\Omega^L} \delta \mathbf{u}^l \mathbf{b} \mathrm{d} \Omega.
$$

These equations are reconstructed as

$$
\int_{\Omega^G} \delta \varepsilon^G \mathbf{D} \varepsilon^G \mathrm{d} \Omega = \int_{\Gamma_t} \delta \mathbf{u}^G \mathbf{t} \mathrm{d} \Gamma + \int_{\Omega^G} \delta \mathbf{u}^G \mathbf{b} \mathrm{d} \Omega
$$

and

$$
\int_{\Omega^L} \delta \varepsilon^l \mathbf{D} \varepsilon^l \mathrm{d} \Omega = \int_{\Gamma_t} \delta \mathbf{u}^l \mathbf{t} \mathrm{d} \Gamma + \int_{\Omega^L} \delta \mathbf{u}^l \mathbf{b} \mathrm{d} \Omega.
$$

It is noted that $\varepsilon$ and $\delta \varepsilon$ also follow the superposition assumption as Eqs. (1) and (3). $\delta \mathbf{u}$, $\varepsilon$, and $\delta \varepsilon$ are discretized to be $\mathbf{N} \delta \mathbf{u}$, $\mathbf{B} \varepsilon$, and $\mathbf{B} \delta \varepsilon$, respectively, by using a shape function matrix, $\mathbf{N}$, and a B matrix, $\mathbf{B}$. An overbar $(\bar{\cdot})$ denotes that it is a discretized variable. Finally, a linear equation system is obtained, as:

$$
\begin{bmatrix}
\mathbf{K}^G & \mathbf{K}^{GL} \\
\mathbf{K}^{LG} & \mathbf{K}^L
\end{bmatrix}
\begin{bmatrix}
\mathbf{u}^G \\
\mathbf{u}^L
\end{bmatrix}
= \begin{bmatrix}
\mathbf{f}^G \\
\mathbf{f}^L
\end{bmatrix},
$$

where

$$
\mathbf{K}^G = \int_{\Omega^G} \mathbf{B}^G \mathbf{D} \mathbf{B}^G \mathrm{d} \Omega,
$$

$$
\mathbf{K}^L = \int_{\Omega^L} \mathbf{B}^l \mathbf{D} \mathbf{B}^l \mathrm{d} \Omega,
$$

$$
\mathbf{K}^{GL} = \int_{\Omega^G} \mathbf{B}^G \mathbf{D} \mathbf{B}^l \mathrm{d} \Omega,
$$

$$
\mathbf{K}^{LG} = \int_{\Omega^L} \mathbf{B}^l \mathbf{D} \mathbf{B}^G \mathrm{d} \Omega = \mathbf{K}^{LG},
$$

$$
\mathbf{f}^G = \int_{\Gamma_t} \mathbf{N}^G \mathbf{t} \mathrm{d} \Gamma + \int_{\Omega^G} \mathbf{N}^G \mathbf{b} \mathrm{d} \Omega,
$$

and

$$
\mathbf{f}^L = \int_{\Gamma_t} \mathbf{N}^L \mathbf{t} \mathrm{d} \Gamma + \int_{\Omega^L} \mathbf{N}^L \mathbf{b} \mathrm{d} \Omega.
$$

The stiffness matrices, $\mathbf{K}^G$ and $\mathbf{K}^L$, and the external force vectors, $\mathbf{f}^G$ and $\mathbf{f}^L$, can be generated in the same manner as conventional finite element method. However, the generation of the coupling stiffness matrices, $\mathbf{K}^{GL}$ and $\mathbf{K}^{LG}$, takes a considerable amount of program development efforts (Okada et al., 2004; Tanaka et al., 2006; Kikuchi et al., 2014).

### 2.2. Iterative s-version finite element method without generation of coupling stiffness matrices

In proposed iterative s-FEM, the generation of the coupling stiffness matrices, $\mathbf{K}^{GL}$ and $\mathbf{K}^{LG}$, is eliminated as described in the following. From Eq. (8), one can write

$$
\int_{\Omega^G} \delta \varepsilon^G \mathbf{D} \varepsilon^G \mathrm{d} \Omega = -\int_{\Omega^G} \delta \varepsilon^G \sigma^G \mathrm{d} \Omega + \int_{\Gamma_t} \delta \mathbf{u}^G \mathbf{t} \mathrm{d} \Gamma + \int_{\Omega^G} \delta \mathbf{u}^G \mathbf{b} \mathrm{d} \Omega,
$$

where $\sigma^G = \mathbf{D} \varepsilon^G$. It is noted that the stress field, $\sigma$, also follows the superposition assumption, as:

$$
\sigma = \begin{cases}
\sigma^G & \text{in } \Omega^G \setminus \Omega^L, \\
\sigma^G + \sigma^L & \text{in } \Omega^L.
\end{cases}
$$

In this paper, we call $\sigma^G$ and $\sigma^L$ to be global and local stresses. After that, Eq. (17) is discretized similarly to the previous subsection, and a linear equation system at the $(k+1)$-th iteration step is obtained and it can be written to be:

$$
\mathbf{K}^G \delta \mathbf{u}^{\text{(G)}(k+1)} = -\int_{\Omega^G} \mathbf{B}^G \delta \sigma^G \mathrm{d} \Omega + \mathbf{f}^G.
$$
The coupling stiffness matrix, \( K^{GL} \), is eliminated. From Eq. (19), one can design a recurrence formula. Global discretized stress at the \((k+1)\)-th iteration step, \( \sigma^{G(k+1)} \), can be calculated from local discretized stress at the \(k\)-th iteration step, \( \sigma^{L(k)} \), by:

\[
\sigma^{G(k+1)} = D \hat{e}^{G(k+1)} = DB^G \hat{u}^{G(k+1)} = DB^G K^{-1} \left( - \int_{\Omega} B^G T \sigma^{L(k)} d\Omega + f^G \right).
\] (20)

Similarly from Eq. (9), one can derive an expression, as:

\[
\int_\Omega \delta e^T D e d\Omega = - \int_\Omega \delta e^T \sigma^G d\Omega + \int_\Omega \delta u^T t d\gamma + \int_\Omega \delta u^T b d\Omega.
\] (21)

Thus, a linear equation system at the \((k+1)\)-th iteration step is obtained, as:

\[
K^L \hat{u}^{L(k+1)} = - \int_{\Omega^L} B^L T \sigma^{G(k+1)} d\Omega + f^L.
\] (22)

The coupling stiffness matrix, \( K^{LG} \), is eliminated. Local discretized stress at the \((k+1)\)-th iteration step, \( \sigma^{L(k+1)} \), can be calculated from global discretized stress at the \((k+1)\)-th iteration step, \( \sigma^{G(k+1)} \), by:

\[
\sigma^{L(k+1)} = D \hat{e}^{L(k+1)} = DB^L \hat{u}^{L(k+1)} = DB^L K^{L-1} \left( - \int_{\Omega^L} B^L T \sigma^{G(k+1)} d\Omega + f^L \right).
\] (23)

From Eqs. (20) and (23), local discretized stress at the \((k+1)\)-th iteration step, \( \sigma^{L(k+1)} \), can be calculated from that at the \(k\)-th iteration step, \( \sigma^{L(k)} \), by:

\[
\sigma^{L(k+1)} = DB^L K^{L-1} \left( - \int_{\Omega^L} B^L T DB^G K^{-1} \left( - \int_{\Omega} B^G T \sigma^{L(k)} d\Omega + f^G \right) d\Omega + f^L \right).
\] (24)

The computational procedure is summarized in Fig. 2, and it is repeated until converged solution is achieved. The computational procedure at an iteration step includes a global analysis, global stress transferring, a local analysis, local stress transferring, and convergence checking, in order. The global and local analyses can be performed by existing finite element programs without the generation of coupling stiffness matrices. \( LL/LLD \) factorization or preparation of a preconditioning matrix has to be performed only once for each stiffness matrix, because the stiffness matrices, \( K^G \) and \( K^L \), remain constant throughout the iteration. The local stress, \( \sigma^L \), and the global stress, \( \sigma^G \), are treated as initial stress in the finite element computations, as shown in Eqs. (19) and (22). The local stress, \( \sigma^L \), is evaluated at local integration points from nodal displacement of the local mesh, \( \hat{u}^L \), and then transferred to global integration points by interpolation. The global stress, \( \sigma^G \), is evaluated similarly on the global mesh and transferred to the local mesh. It should be noted that since proposed technique has a structure as block Gauss–Seidel method (Suzuki et al., 1999; 2002; Yusa et al., 2012; Yusa and Yoshimura, 2013), its convergence is guaranteed.

Convergence at the \((k+1)\)-th iteration step is checked based on force equilibrium by

\[
\begin{bmatrix}
  f^G - K^G \hat{u}^{G(k+1)} - \int_{\Omega^G} B^G T \sigma^{L(k+1)} d\Omega \\
  f^L - K^L \hat{u}^{L(k+1)} - \int_{\Omega^L} B^L T \sigma^{G(k)} d\Omega
\end{bmatrix} = \begin{bmatrix}
  f^G - \int_{\Omega^G} B^G T \left( \sigma^{G(k+1)} + \bar{\sigma}^{L(k+1)} \right) d\Omega \\
  f^L - \int_{\Omega^L} B^L T \left( \sigma^{L(k+1)} + \bar{\sigma}^{G(k)} \right) d\Omega
\end{bmatrix} \leq \varepsilon,
\] (25)

where \( \varepsilon \) is a tolerance parameter. The numerator of the left-hand side is algebraically equal to the norm of the left-hand side minus the right-hand side of Eq. (10). This type of convergence check is generally used in linear system solution. In Eq. (25), internal forces at each iteration step with respect to the constrained degrees of freedom are included in the external force vector in order to account for the contribution of enforced displacement boundary condition.

Various interpolation techniques can be applied to the transfers of \( \sigma^G \) and \( \sigma^L \). We tried shape functions-based interpolation and nearest neighbor interpolation for \( \sigma^G \), and local least squares interpolation with three types of sampling regions and nearest neighbor interpolation for \( \sigma^L \).

The shape functions-based interpolation procedure for \( \sigma^G \) is shown in Fig. 3 (a). First, \( \sigma^G \) at global nodes is calculated by averaging \( \sigma^G \) at near global integration points. Then, the nodal global stress is interpolated by using shape functions and \( \sigma^G \) at local integration points is calculated. In the nearest neighbor interpolation for \( \sigma^G \), \( \sigma^G \) at the nearest global integration point is transferred to the local integration point as depicted in Fig. 3 (b). Global integration points that are located outside the local mesh are ignored by the local integration point even if one of them is the nearest. It is noted that the interpolated \( \sigma^G \) becomes continuous between every adjacent elements in the shape functions-based interpolation, whereas that it does discontinuous in the nearest neighbor interpolation. Discretized stress should be discontinuous.
between every adjacent elements because it is discretized by using derivatives of shape functions. Continuous stress
distribution may degrade numerical accuracy.

The local least squares interpolation procedure for $\sigma^L$ is shown in Fig. 4. In a sampling region, each component of
local stress, $\sigma^L_{ij}$, is assumed to be distributed linearly, as:

$$
\sigma^L_{ij} = f(x, y) = a_0 + a_1 x + a_2 y.  \tag{26}
$$

The coefficients, $a_0$, $a_1$, and $a_2$, are determined by the least squares method. When the sampling region is outside
the local domain or inside the hole, namely, when there are no local integration points in the sampling region, $\sigma^L$ at
the global integration point becomes zero. If the number of local integration points in the sampling region is not sufficient,
namely, if the coefficient matrix of the local least squares goes singular, the local least squares approach is switched to the
averaging approach, which assumes $f(x, y) = a_0$. Three types of the sample regions are tried in this study. In the approach
of (a) a shifted global element, its sampling region is set to be a shifted global element whose central point is the global
integration point. In this approach, $\sigma^L$ becomes continuous between every adjacent elements. In the approaches of (b) the
whole of a global element and (c) a part of a global element, their sampling regions fit the global element and a part of
the global element, respectively. We consider, in the case of a four-node quadrangular finite element with four integration
points, that an integration point represents quarter area of the finite element. This strategy to design the sampling region
can be extended to any type of finite elements including the three-dimensional cases. In these approaches, $\sigma^L$ becomes
discontinuous between every adjacent elements. The approach of (b) the whole of a global element has an advantage that
it can easily be applied to various integration schemes such as selectively reduced integration and negative integration
weights.

In above interpolation techniques for $\sigma^G$ and $\sigma^L$, four pairs, which are described in Table 1, are tried in the section
of numerical experiments. Scheme 1 is the combination of the shape function-based interpolation and one of the local least
squares interpolations. Schemes 2 and 3 utilize the nearest neighbor interpolation and the local least squares interpolation.
Scheme 4 uses the nearest neighbor interpolation only. Scheme 1 gives continuous stress distribution, whereas schemes
2–4 do not. It should be noted that local least squares interpolation would not be applicable to $\sigma^G$ because a local element
is usually smaller than a global element and the sampling region becomes too small to perform the local least squares
interpolation.

3. Numerical experiments

3.1. Patch test

Generally in s-FEM, it is known that the monolithic stiffness matrix of Eq. (10) goes singular when shape functions
of global and local elements are not linearly independent (Ooya et al., 2009). When a direct solver is used for linear
equation system solution, zero division will occur in LL/LDL factorization. When an iterative solver is used, its iteration
cannot converge in most cases. In proposed iterative s-FEM, this issue does not occur because global and local analyses
are performed independently with their rigid body motion mode perfectly constrained. In this subsection, a patch test
problem is analyzed to demonstrate this advantage.
Fig. 3  Procedures of (a) shape functions-based interpolation and (b) nearest neighbor interpolation to transfer global stress, $\bar{\sigma}^G$, from global integration points to a local integration point.

Fig. 4  Three types of sampling regions of local least squares interpolation to transfer local stress, $\bar{\sigma}^L$, from local integration points to a global integration point.

Table 1  Interpolation techniques for $\bar{\sigma}^L$ and $\bar{\sigma}^G$, which will be compared in the section of numerical experiments.

<table>
<thead>
<tr>
<th>Identifier</th>
<th>$\bar{\sigma}^L$ transferred from global to local integration points</th>
<th>$\bar{\sigma}^G$ transferred from local to global integration points</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scheme 1</td>
<td>Shape functions-based</td>
<td>Local least squares (a shifted global element)</td>
</tr>
<tr>
<td>Scheme 2</td>
<td>Nearest neighbor</td>
<td>Local least squares (the whole of a global element)</td>
</tr>
<tr>
<td>Scheme 3</td>
<td>Nearest neighbor</td>
<td>Local least squares (a part of a global element)</td>
</tr>
<tr>
<td>Scheme 4</td>
<td>Nearest neighbor</td>
<td>Nearest neighbor</td>
</tr>
</tbody>
</table>

Dimension parameters and applied boundary conditions are shown in Fig. 5. To satisfy Eq. (2), local nodes on $\Gamma^{GL}$ are entirely constrained. Young’s modulus and Poisson’s ratio are set to be 210 GPa and 0.3, respectively. A plane stress state is assumed and the thickness of the plate is set to be 1 mm. Tensile load is set to be 1 MPa. Global and local meshes with linear quadrangular finite elements are depicted in Fig. 6. The numbers of elements and nodes of the global mesh are 16 and 25, respectively. Those of the local mesh are 64 and 81, respectively. The scheme 3 of the stress interpolation techniques is used.

Converged solution was obtained with only one iteration count. Since the global stress, $\bar{\sigma}^G$, was analyzed to be uniform after the first global analysis, the right-hand side of Eq. (22) became zero and $\bar{u}^L$ was also analyzed to be zero. The relative residual of the left-hand side of Eq. (25) after one global analysis and one local analysis was $1.57 \times 10^{-10}$. This very small nonzero residual is due to the preconditioned conjugate gradient solver that is used to solve linear equation systems. This tolerance of the residual of the preconditioned conjugate gradient solver was set to be $10^{-10}$ in this numerical experiment.

3.2. Flat plate with a circular hole

In this subsection, a stress concentration problem is analyzed using proposed iterative s-FEM. Four pairs of stress interpolation techniques, which are described in Table 1, are compared. Convergence performance is measured and compared. Computed stress is compared to that computed by conventional FEM. Representation of free boundary condition on the local mesh is also discussed. It should be noted that both traction, $t$, and body force, $b$, of the local domain are
Fig. 5 Dimension parameters and applied boundary conditions of a patch test problem.

Fig. 6 Global and local meshes with linear quadrangular finite elements of a patch test problem for s-FEM.

set to be zero in this numerical experiment. However, one can use nonzero traction or body force in the methodology described above.

A flat plate with a circular hole subject to uniform tension is analyzed. Dimension parameters and applied boundary conditions are shown in Fig. 7. To satisfy Eq. (2), local nodes on $\Gamma^{GL}$ are entirely constrained. It is noted that the surface of the circular hole is not constrained. Young’s modulus and Poisson’s ratio are set to be 210 GPa and 0.3, respectively. A plane stress state is assumed and the thickness of the plate is set to be 1 mm. Tensile load at the top and bottom edges of the plate is set to be 1 MPa. Global, local, and monolithic meshes with linear quadrangular finite elements are depicted in Fig. 8. The hole does not have to be modeled by the global mesh (Nakasumi et al., 2001). The monolithic mesh is used to obtain reference solution. The numbers of elements and nodes of the global mesh are 256 and 289, respectively. Those of the local mesh are 800 and 880, respectively. Those of the monolithic mesh for conventional FEM are 2,000 and 2,120, respectively. The tolerance parameter, $\varepsilon$, of Eq. (25) is set to be $10^{-6}$.

The relative residuals are measured and their variations are plotted in Fig. 9. The horizontal axis denotes the iteration count, $k$, and the vertical one does the relative residual of the left-hand side of Eq. (25). Converged solution was successfully obtained with a few tens of iteration counts in scheme 1 of stress interpolation, and with about one thousand iteration counts in schemes 2 and 3 of stress interpolation. Scheme 4 could not achieve the convergence.

Distributions of stresses $\sigma_y$, $\sigma_z$, and $\tau_{xy}$ that are computed by using scheme 3 of stress interpolation is visualized in Figs. 10, 11, and 12, respectively. Although the global stress, $\sigma^G$, in the local domain shows strange distribution, the sum of the global and local stresses, $\sigma^G + \sigma^L$, shows good stress concentration for the three directions. Stress concentration in the vicinity of the circular hole is shown in Fig. 13. The horizontal axis denotes the $x$ coordinate from the center of the circular hole, and the vertical one does the stress $\sigma_y$. Stress concentration was accurately computed by proposed iterative s-FEM with schemes 2 and 3 of stress interpolation. However scheme 1 could not give accurate stress concentration even though it gave converged solution. This is probably because scheme 1 could not capture the hole edge accurately. Although scheme 1 interpolates discretized stress continuously, it should change discontinuously to zero on the hole edge. Stresses $\sigma_r$, $\sigma_\theta$, and $\sigma_{\theta r}$ along the edge of the circular hole, which are transformed from those of $\sigma_x$, $\sigma_y$, and $\tau_{xy}$, are shown in Fig. 14. That computed by schemes 1–3 of stress interpolation is compared with that of reference solution. Free boundary condition ($\sigma_r = \sigma_\theta = 0$) is well represented by proposed iterative s-FEM with schemes 1–3. Although some oscillation is seen at the angles around 25° and 65° due to the effect of global element interfaces, scheme 2 and 3 provided accurate distribution of $\sigma_\theta$ along the edge of the circular hole. However scheme 1 of stress interpolation could not evaluate the stress $\sigma_\theta$ accurately.

3.3. Flat plate with a crack

In this subsection, a linear elastic fracture mechanics problem is analyzed using proposed iterative s-FEM. This problem produces stress singularity near a crack. It is demonstrated that such a severe stress concentration problem can be analyzed accurately by proposed iterative s-FEM. A computed stress intensity factor is compared to that computed by conventional FEM and the reference formula.

A flat plate with a mode-I crack subject to uniform tension is analyzed. Dimension parameters and applied boundary conditions are shown in Fig. 15. Young’s modulus and Poisson’s ratio are set to be 210 GPa and 0.3, respectively. A plane stress state is assumed and the thickness of the plate is set to be 1 mm. Tensile load at the top and bottom edges of the plate is set to be 1 MPa. Global, local, and monolithic meshes with linear quadrangular finite elements are depicted in Fig. 8. The hole does not have to be modeled by the global mesh (Nakasumi et al., 2001). The monolithic mesh is used to obtain reference solution. The numbers of elements and nodes of the global mesh are 256 and 289, respectively. Those of the local mesh are 800 and 880, respectively. Those of the monolithic mesh for conventional FEM are 2,000 and 2,120, respectively. The tolerance parameter, $\varepsilon$, of Eq. (25) is set to be $10^{-6}$.

The relative residuals are measured and their variations are plotted in Fig. 9. The horizontal axis denotes the iteration count, $k$, and the vertical one does the relative residual of the left-hand side of Eq. (25). Converged solution was successfully obtained with a few tens of iteration counts in scheme 1 of stress interpolation, and with about one thousand iteration counts in schemes 2 and 3 of stress interpolation. Scheme 4 could not achieve the convergence.

Distributions of stresses $\sigma_y$, $\sigma_z$, and $\tau_{xy}$ that are computed by using scheme 3 of stress interpolation is visualized in Figs. 10, 11, and 12, respectively. Although the global stress, $\sigma^G$, in the local domain shows strange distribution, the sum of the global and local stresses, $\sigma^G + \sigma^L$, shows good stress concentration for the three directions. Stress concentration in the vicinity of the circular hole is shown in Fig. 13. The horizontal axis denotes the $x$ coordinate from the center of the circular hole, and the vertical one does the stress $\sigma_y$. Stress concentration was accurately computed by proposed iterative s-FEM with schemes 2 and 3 of stress interpolation. However scheme 1 could not give accurate stress concentration even though it gave converged solution. This is probably because scheme 1 could not capture the hole edge accurately. Although scheme 1 interpolates discretized stress continuously, it should change discontinuously to zero on the hole edge. Stresses $\sigma_r$, $\sigma_\theta$, and $\tau_{\theta r}$ along the edge of the circular hole, which are transformed from those of $\sigma_x$, $\sigma_y$, and $\tau_{xy}$, are shown in Fig. 14. That computed by schemes 1–3 of stress interpolation is compared with that of reference solution. Free boundary condition ($\sigma_r = \sigma_\theta = 0$) is well represented by proposed iterative s-FEM with schemes 1–3. Although some oscillation is seen at the angles around 25° and 65° due to the effect of global element interfaces, scheme 2 and 3 provided accurate distribution of $\sigma_\theta$ along the edge of the circular hole. However scheme 1 of stress interpolation could not evaluate the stress $\sigma_\theta$ accurately.

3.3. Flat plate with a crack

In this subsection, a linear elastic fracture mechanics problem is analyzed using proposed iterative s-FEM. This problem produces stress singularity near a crack. It is demonstrated that such a severe stress concentration problem can be analyzed accurately by proposed iterative s-FEM. A computed stress intensity factor is compared to that computed by conventional FEM and the reference formula.

A flat plate with a mode-I crack subject to uniform tension is analyzed. Dimension parameters and applied boundary conditions are shown in Fig. 15. Young’s modulus and Poisson’s ratio are set to be 210 GPa and 0.3, respectively. A plane stress state is assumed and the thickness of the plate is set to be 1 mm. Tensile load at the top and bottom edges of the
Figure 7: Dimension parameters and applied boundary conditions of a problem of circular hole.

Figure 8: Global, local, and monolithic meshes with linear quadrangular finite elements of the problem of circular hole.

Figure 9: A convergence history of proposed iterative s-FEM of a circular hole analysis.

Figure 10: Distribution of stress $\sigma_y$ of the problem of circular hole computed by proposed iterative s-FEM with scheme 3 of stress interpolation.

Figure 11: Distribution of stress $\sigma_x$ of the problem of circular hole computed by proposed iterative s-FEM with scheme 3 of stress interpolation.

Figure 12: Distribution of stress $\tau_{xy}$ of the problem of circular hole computed by proposed iterative s-FEM with scheme 3 of stress interpolation.

The plate is set to be 1 MPa. Global, local, and monolithic meshes with linear quadrangular finite elements are depicted in Fig. 16. Due to the symmetry, a quarter of the cracked plate was analyzed by proposed iterative s-FEM and conventional FEM. The numbers of elements and nodes of the global mesh are 64 and 81, respectively. Those of the local mesh are 2,650 and 2,746, respectively. Those of the monolithic mesh for conventional FEM are 4,900 and 5,061, respectively. The minimum element edge length near the crack is 0.03125 mm. The tolerance parameter, $\varepsilon$, of Eq. (25) is set to be $10^{-6}$. The scheme 3 of the stress interpolation techniques is used.

The relative residual is measured and its variation is plotted in Fig. 17. The horizontal axis denotes the iteration step, $k$, and the vertical one does the relative residual of the left-hand side of Eq. (25). Converged solution was successfully obtained with 320 iteration counts.

Distribution of stress $\sigma_y$ is visualized in Fig. 18. That computed by conventional FEM for reference solution is also visualized in Fig. 19. Computed stress seems to be discontinuous on the global–local interface, $\Gamma^{GL}$, due to the difference of the resolution of the global and local meshes. $\Gamma^{GL}$ is located across the global elements. Stress concentration can be seen. Free boundary condition ($\sigma_y = 0$) on the crack surface is well represented. Stress concentration in the vicinity of the crack tip is shown in Fig. 20. The horizontal axis denotes the $x$ coordinate from the crack tip, and the vertical one does the stress $\sigma_y$. Severe stress concentration with singularity was accurately computed by proposed iterative s-FEM with respect to comparing with conventional FEM. The slope in Fig. 20, which is fitted in the range of $10^{-1}$ to $10^{0}$ mm of the horizontal axis, is $-0.487$. It indicates that the $r^{-\frac{1}{2}}$ stress singularity is approximated by conventional FEM as well as
fig. 13 Stress concentration in the vicinity of the circular hole.

fig. 14 Distributions of Stresses \( \sigma_{rr}, \sigma_{\theta\theta}, \) and \( \tau_{r\theta} \) along the edge of the circular hole.

proposed iterative s-FEM.

Displacement on the top edge where the prescribed traction is applied is compared in Fig. 21 to show the accuracy of proposed iterative s-FEM with respect to the global behavior that is influenced by the local feature. It is indicated that proposed iterative s-FEM gave the far-field displacement accurately.

Stress intensity factors calculated by virtual crack closure-integral method for s-FEM (Okada et al., 2007) are presented in Table 2. The reference stress intensity factor (Murakami et al., 1987), \( K_1 \), is calculated by

\[
K_1 = \sigma_0 \sqrt{\pi a} F(\alpha, \beta),
\]

(27)

where \( \sigma_0 \) is applied tensile load, \( a \) is half crack width, and \( \pi \) is the circular constant. \( F(\alpha, \beta) \) is a boundary correction factor and is 1.123 from the table (Murakami et al., 1987) when \( \alpha = \frac{2a}{W} = 0.3 \) and \( \beta = \frac{2H}{W} = 1 \). \( W \) is plate width and \( H \) is half plate height. Accuracy of the reference stress intensity factor is less than 1%. The stress intensity factors computed by proposed iterative s-FEM, conventional FEM, and the reference formula are in good agreement.

4. Conclusion

In this paper, an iterative s-version finite element method (s-FEM) without the generation of coupling stiffness matrices is proposed. The generation of coupling stiffness matrices, which takes a considerable amount of program development...
efforts in the original s-FEM, is eliminated. The coupling term is now evaluated by global and local stresses that are computed on the respective mesh and then transferred to the other by interpolation techniques. This global and local stresses are treated as initial stress in the finite element computations. The global and local analyses are performed alternately, and converged solution is achieved by iteration with a monitored residual being sufficiently small. One can easily perform the s-FEM simulations utilizing existing finite element programs. The following features of proposed iterative s-FEM were demonstrated by the numerical experiments.

1. An issue about linear independence of global and local elements, which is known to occur in the original s-FEM, does not occur in proposed iterative s-FEM. This is because global and local analyses are performed independently with their rigid body motion mode perfectly constrained.

2. Converged solution was successfully obtained within several hundred iteration counts.

3. A stress concentration problem was accurately analyzed with respect to stress distribution as well as the representation of free boundary condition. In a linear elastic fracture mechanics problem, the stress intensity factor was computed accurately. Its error was 1.50%.

In addition, several stress interpolation techniques were compared in the numerical experiments. It was investigated that nearest neighbor interpolation for global stress and local least squares interpolation for local stress are appropriate.
Table 2  Stress intensity factors computed by proposed iterative s-FEM, conventional FEM, and the reference formula (Murakami et al., 1987).

<table>
<thead>
<tr>
<th></th>
<th>Proposed iterative s-FEM</th>
<th>Conventional FEM</th>
<th>Reference solution</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.349 94 MPa m^-1</td>
<td>0.344 76 MPa m^-1</td>
<td>0.344 76 MPa m^-1</td>
</tr>
<tr>
<td>(1.50% error)</td>
<td>(1.02 x 10^-4% error)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

to obtain accurate converged solution. An appropriate sampling region of the local least squares is the whole of a global element or a part of a global element. The former sampling region is applicable to various integration schemes such as selectively reduced integration and negative integration weights, whereas the latter sampling region would give slightly fewer iteration counts.

Proposed iterative s-FEM requires computational time because a significant number of iteration counts is required to obtain a converged solution. The authors are now investigating methodologies to reduce the iteration counts of the iterative s-FEM analysis. Various iterative solvers for a block linear equation system such as block SOR (successive over-relaxation) method, CG (conjugate gradient) method, CG method with block Jacobi preconditioner, and CG method with static condensation have been studied so far. Their concept is probably applicable to proposed iterative s-FEM even though there are no coupling stiffness matrices.

References


Okada, H., Endoh, S., and Kikuchi, M., Application of s-version finite element method to two-dimensional fracture