Analysis of a many-hole problem using coupling-matrix-free iterative s-version FEM with multiple local meshes

Yasunori YUSA*, Joe OKAMOTO*, Daiji TOYAMA** and Hiroshi OKADA*

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

Received: 24 May 2018; Revised: 11 July 2018; Accepted: 1 August 2018

Abstract
The coupling-matrix-free iterative s-version finite element method is extended to multiple local meshes in order to model multiple local features such as holes, inclusions, and cracks. The formulation with multiple local meshes is presented, along with the stress transfer method between the local meshes. The present method does not require the generation of coupling stiffness matrices with a very sophisticated numerical integration method between the global and local meshes or between the local meshes. Instead, stress transfers between pairs of overlapping meshes are performed. Several numerical examples, such as a patch test, a two-hole problem, and a structure with many holes, are presented. The examples demonstrate that the present method is capable of representing a uniform stress distribution as well as capturing the stress concentration of multiple holes that are located in the vicinity of each other. Moreover, numerical integration methods of the global mesh and local meshes are found to have significant influences on the convergence of the iteration. In these numerical examples, the straightforward Gaussian quadrature could not achieve convergence, whereas the element subdivision technique could. Thus, sufficient element subdivisions in the global mesh and the local meshes are necessary in order to produce a converged solution.

Keywords: S-version finite element method, Multiple local meshes, Coupling stiffness matrix, Iterative method, Numerical integration, Stress concentration problem

1. Introduction

The s-version finite element method (s-FEM) (Fish, 1992) was proposed to tractably analyze structures with local features such as holes, inclusions (Okada et al., 2004; Tanaka et al., 2006), and cracks (Kikuchi et al., 2008; Kamaya et al., 2010). In this method, single or multiple local meshes that represent local features are superposed on the global mesh for the whole structure. When the number, shape, or size of local features changes due to design demands, only local meshes should be recreated, whereas the global mesh can remain the same. This approach can considerably reduce tasks in the preparation of the finite element model. In the methodology of s-FEM, the global and local meshes are analyzed monolithically with a global stiffness matrix, which includes coupling effects between the global and local meshes. In early studies of s-FEM, a single local mesh was assumed, so that only one local feature could be modeled. However, multiple local features often appear in realistic scenarios. For example, gas turbine blades that suffer from severe thermal conditions have many cooling holes. Moreover, composite materials generally have a very large number of inclusions. Special care should be devoted to the treatment of multiple local features that are modeled by multiple local meshes. Okada et al. (2004) and Tanaka et al. (2006) introduced the formulation of multiple local meshes in s-FEM and analyzed particulate composite materials. This formulation considers coupling effects between pairs of local meshes, in which volume integration of partly superposed finite elements of the local meshes is conducted. This volume integration requires a much more complicated algorithm than that of the global and local meshes, because global–local volume integration can assume a finite element of the global mesh to be much larger than that of the local mesh. Kikuchi et al. (2008) and Kamaya et al. (2010) analyzed the interaction of two fatigue cracks. Two local meshes were used to model two adjacent cracks.
Recently, s-FEM has been applied to sophisticated modeling of damage and failure problems (Chen et al., 2014; Xu et al., 2018). Moreover, the concept of s-FEM has been used in multi-scale analysis in conjunction with another method such as the extended finite element method (Loehnert and Belytschko, 2007) and the phase field method (Gerasimov et al., 2018).

Other methods have been proposed to tractably deal with multiple local features. Kaneko et al. (2012) analyzed crack propagation problems with coalescence using the conventional finite element method (FEM) with an automatic mesh generation technique based on Delaunay triangulation. Theoretically, a sophisticated automatic mesh generation technique must enable us to model every local feature in single mesh. In the extended finite element method (XFEM) (Moës et al., 1999), enrichment functions are introduced to represent crack face discontinuities or crack tip singularities. They enrich the finite element interpolations using the property of the partition of unity. When multiple cracks are located in an element, a special volume integration technique should be used (Budyn et al., 2004). The accuracy of XFEM is affected by the size of finite elements in the vicinity of the crack. In order to overcome this problem, Lee et al. (2004) and Nakasumi et al. (2008) combined XFEM with s-FEM. Lee et al. (2004) placed local meshes of s-FEM at the crack tips. These local meshes move as the cracks propagate. Nakasumi et al. (2008) placed a local mesh that contains the cracks and does not move. In the elastic–plastic finite element alternating method (EPFEAM) (Pyo et al., 1995a, 1995b), analytical solutions of cracks in an infinite elastic media and an uncracked structure model are analyzed alternately, resulting in a converged solution. Pyo et al. (1995a, 1995b) analyzed multi-site damage problems of aircraft panels using EPFEAM. In EPFEAM, any crack in a structure can easily be simulated, as long as the crack has an analytical solution.

Recently, the coupling-matrix-free iterative s-FEM (Yumoto et al., 2016a, 2016b; Yusa et al., 2018) was proposed as an extension of the original s-FEM. The principal advantage of this method as compared to the original s-FEM is the elimination of coupling stiffness matrices, which requires a very sophisticated volume integration algorithm, as mentioned above. Instead of the coupling stiffness matrices, stresses computed in one mesh are transferred to another mesh. Then, the solution is obtained by an iterative algorithm. Such a method can be viewed as an iterative solver such as the Gauss–Seidel method (Yusa et al., 2018). However, in our previous studies (Yumoto et al., 2016a, 2016b; Yusa et al., 2018), the stress transfers were designed for a single overlapping local mesh. This method is applicable to multiple local meshes when local meshes do not overlap each other (Yusa et al., 2018).

In the present study, the coupling-matrix-free iterative s-FEM is extended to multiple local meshes. The present method does not require the generation of coupling stiffness matrices. A very sophisticated numerical integration method between the global and local meshes as well as between the local meshes is no longer needed. Instead, stress transfers between pairs of meshes are performed. In the present paper, the formulation of the coupling-matrix-free iterative s-FEM with multiple local meshes is explained, followed by the stress transfer method between the local meshes. These are computational techniques presented in the present paper. Then, several numerical examples, such as a patch test, a simple two-hole problem, and a structure with many holes, are presented. It is shown that many holes can be introduced at arbitrary positions in a structure. Moreover, the element subdivision technique (Yumoto et al., 2016b) of the global mesh and the local meshes is found to play a significant role in achieving convergence. In this technique, a finite element is divided into multiple element subdivisions, by which numerical integrals based on the Gaussian quadrature are performed. In our previous study (Yumoto et al., 2016b), the element subdivision technique was used only in the global mesh to improve the accuracy of the analysis, whereas it is used in both the global and local meshes for the convergence in the present study.

2. Coupling-matrix-free iterative s-version FEM with multiple local meshes

In this section, the formulation of the coupling-matrix-free iterative s-FEM with multiple local meshes is explained first. Part of the formulation follows studies involving the original s-FEM with multiple local meshes (Okada et al., 2004; Tanaka et al., 2006; Kikuchi et al., 2008; Kamaya et al., 2010). Then, interpolation methods in stress transfers between meshes are presented. In particular, the present method with multiple local meshes requires stress transfers between pairs of overlapping local meshes. Finally, the element subdivision technique (Yumoto et al., 2016b) is described. This technique is the key to achieving convergence of coupling-matrix-free iterative s-FEM in multiple local mesh problems.

2.1. Formulation for multiple local meshes

In s-FEM with multiple local meshes (Okada et al., 2004; Tanaka et al., 2006; Kikuchi et al., 2008; Kamaya et al., 2010), $N$ local domains, $\Omega_i$ ($i = 1, 2, \ldots, N$), are considered in a global domain, $\Omega$, as shown in Fig. 1. The global domain, $\Omega$, and the local domains, $\Omega_i$ ($i = 1, 2, \ldots, N$), are discretized by finite element models, which are referred
The displacements, strains, and their variations are expressed as

\[ \tilde{\mathbf{u}} = \tilde{\mathbf{u}}^G + \sum_{i=1}^{N} \tilde{\mathbf{u}}^L \]  

in \( \mathcal{D}^G \),

where

\[ \tilde{\mathbf{u}}^L = \mathbf{0} \]  
in \( \mathcal{D}^G \setminus \mathcal{D}^L \)  
(i = 1, 2, \ldots, N),

and \( \tilde{\mathbf{u}}^G \) and \( \tilde{\mathbf{u}}^L \) are associated with the global and local finite element models, respectively. In order for the displacements, \( \tilde{\mathbf{u}} \), to be continuous in \( \mathcal{D}^G \), we set

\[ \tilde{\mathbf{u}}^L = \mathbf{0} \]  
on \( \Gamma^{GL} \)  
(i = 1, 2, \ldots, N).

Moreover, the variations of displacements, \( \delta \tilde{\mathbf{u}} \), follow the superposition expressed by Eq. (1), i.e.,

\[ \delta \tilde{\mathbf{u}} = \delta \tilde{\mathbf{u}}^G + \sum_{i=1}^{N} \delta \tilde{\mathbf{u}}^L \]  
in \( \mathcal{D}^G \).

Here, \( \delta \tilde{\mathbf{u}}^G \) and \( \delta \tilde{\mathbf{u}}^L \) are the variations of \( \tilde{\mathbf{u}}^G \) and \( \tilde{\mathbf{u}}^L \), respectively. From the compatibility equation in linear elasticity, the strains, \( \tilde{\varepsilon} \), and their variations, \( \delta \tilde{\varepsilon} \), are similarly \( \tilde{\varepsilon} = \tilde{\varepsilon}^G + \sum_{i=1}^{N} \tilde{\varepsilon}^L \) and \( \delta \tilde{\varepsilon} = \delta \tilde{\varepsilon}^G + \sum_{i=1}^{N} \delta \tilde{\varepsilon}^L \), respectively. Here, \( \tilde{\varepsilon}^G \) and \( \tilde{\varepsilon}^L \) are the strains in terms of \( \tilde{\mathbf{u}}^G \) and \( \tilde{\mathbf{u}}^L \), respectively. Moreover, \( \delta \tilde{\varepsilon}^G \) and \( \delta \tilde{\varepsilon}^L \) are the strains expressed by \( \delta \tilde{\mathbf{u}}^G \) and \( \delta \tilde{\mathbf{u}}^L \), respectively.

Here, the principle of virtual work is introduced as

\[ \int_{\mathcal{D}^G} \delta \tilde{\varepsilon}^T \mathbf{D} \delta \mathbf{u} \, d\Omega = \int_{\Gamma_f} \delta \tilde{\mathbf{u}}^T \mathbf{t} \, d\Gamma + \sum_{j=1}^{N} \int_{\Gamma_f^j} \delta \tilde{\mathbf{u}}^T \mathbf{b} \, d\Gamma, \]

where \( \mathbf{D} \) is the elasticity matrix, and \( \mathbf{t} \) and \( \mathbf{b} \) are the prescribed traction and body forces, respectively. Then, from the expressions shown in Eqs. (1), (4), and (5), we have

\[ \int_{\mathcal{D}^G} \delta \tilde{\varepsilon}^G T \mathbf{D} \delta \mathbf{u}^G \, d\Omega + \sum_{j=1}^{N} \int_{\mathcal{D}^{Lj}} \delta \tilde{\varepsilon}^G T \mathbf{D} \delta \mathbf{u}^L \, d\Omega = \int_{\Gamma_f} \delta \tilde{\mathbf{u}}^G T \mathbf{t} \, d\Gamma + \sum_{j=1}^{N} \int_{\Gamma_f^j \setminus \Gamma_f^j} \delta \tilde{\mathbf{u}}^G T \mathbf{b} \, d\Gamma, \]

\[ \int_{\mathcal{D}^G} \delta \tilde{\varepsilon}^L T \mathbf{D} \delta \mathbf{u}^L \, d\Omega + \sum_{j=1}^{N} \int_{\mathcal{D}^{Lj} \setminus \Gamma_f^j \setminus \Gamma_f^j} \delta \tilde{\varepsilon}^L T \mathbf{D} \delta \mathbf{u}^L \, d\Omega = \int_{\Gamma_f} \delta \tilde{\mathbf{u}}^L T \mathbf{t} \, d\Gamma + \sum_{j=1}^{N} \int_{\Gamma_f^j \setminus \Gamma_f^j} \delta \tilde{\mathbf{u}}^L T \mathbf{b} \, d\Gamma \quad (i = 1, 2, \ldots, N). \]

Equations (6) and (7) are then discretized using the shape functions. The shape functions for the global and local finite element models are expressed by matrices, \( \mathbf{N}^G \) and \( \mathbf{N}^L \). Their respective strain–displacement matrices are \( \mathbf{B}^G \) and \( \mathbf{B}^L \). The displacements, strains, and their variations are expressed as \( \tilde{\mathbf{u}}^G = \mathbf{N}^G \mathbf{u}^G, \tilde{\mathbf{u}}^L = \mathbf{N}^L \mathbf{u}^L, \delta \tilde{\mathbf{u}}^G = \mathbf{N}^G \delta \mathbf{u}^L, \delta \tilde{\mathbf{u}}^L = \mathbf{N}^L \delta \mathbf{u}^L, \)

Fig. 1 Boundary value problem with local domains.
\[ \ddots = B^G u^G, \ddots = B^L u^L, \ddots = B^G \ddot{u}^G, \text{ and } \ddots = B^L \ddot{u}^L. \] Here, \( u^G \) and \( u^L \) are global and local nodal displacements, respectively, and \( \ddot{u}^G \) and \( \ddot{u}^L \) are their variations. Then, we have the following linear system of equations:

\[
\begin{bmatrix}
K^{GG} & K_{GL} & \cdots & K_{GLN} \\
K_{LG} & K^{LL} & \cdots & K_{LLN} \\
\vdots & \vdots & \ddots & \vdots \\
K_{LG} & K_{LL} & \cdots & K^{LN}
\end{bmatrix}
\begin{bmatrix}
u^G \\
u^L \\
\vdots \\
u^L
\end{bmatrix}
= \begin{bmatrix}
f^G \\
f^L \\
\vdots \\
f^L
\end{bmatrix}
\tag{8}
\]

In this equation, \( K^{GG} = \int_{\Omega^G} B^G D B^G d\Omega, K^{LL} = \int_{\Omega^L} B^L D B^L d\Omega, K_{LG} = \int_{\Omega^L} B^L D B^G d\Omega, K_{GL} = \int_{\Omega^G} B^G D B^L d\Omega, f^G = \int_{\Omega^G} B^G \sigma d\Omega, f^L = \int_{\Omega^L} B^L \sigma d\Omega, \) and \( f^L = \int_{\Omega^L} B^L \sigma d\Omega + \int_{\Omega^L} N^L \ddot{u}^L d\Omega. \) Note that \( K_{LG} \) vanishes when \( \Omega^G \) and \( \Omega^L \) do not overlap each other, i.e., \( \Omega^G \cap \Omega^L = \emptyset. \)

In the coupling-matrix-free iterative s-FEM (Yumoto et al., 2016a, 2016b; Yusa et al., 2018), we rewrite Eq. (8) for the Gauss–Seidel method as

\[
u^{G(i+1)} = K^{GG}^{-1} \left( f^G - \sum_{j=1}^{N} K_{GL} u_{j}^{(i)} \right)
= K^{GG}^{-1} \left( f^G - \sum_{j=1}^{N} \int_{\Omega^G} B^G \sigma_{j} d\Omega \right)
\tag{9}
\]

\[
u^{L(i+1)} = K^{L(j-1)} \left( f^L - K_{jL} u^{G(i+1)} - \sum_{j'=1}^{i-1} K^{Lj} u^{G(i+1)} - \sum_{j'=1}^{N} K_{jL} u_{j}^{(i)} \right)
= K^{L(j-1)} \left( f^L - \int_{\Omega^L} B^L \sigma d\Omega - \int_{\Omega^L} \sum_{j'=1}^{i-1} \int_{\Omega^L} B^L \sigma d\Omega - \int_{\Omega^L} \sum_{j'=1}^{N} \int_{\Omega^L} B^L \sigma d\Omega \right) \tag{10}
\]

where

\[
\sigma^{G(i)} = B^G u^{G(i)},
\sigma^{L(i)} = B^L u^{L(i)} \tag{11, 12}
\]

Here, \( k \) is the iteration step. These equations can be regarded as an extension of our previous studies on the coupling-matrix-free iterative s-FEM for a single local mesh (Yumoto et al., 2016a, 2016b; Yusa et al., 2018). In the present method, the computations of Eqs. (9) and (10) are carried out in order, in the context of the Gauss–Seidel method. If \( \Omega^G \) and \( \Omega^{L+1} \) do not overlap, they can be computed in parallel. Although other linear iterative solvers such as Krylov subspace methods can be applied to Eq. (8), the coupling-matrix-free iterative s-FEM is based on the Gauss–Seidel method due to its simplicity. In Eqs. (9) and (10), the stiffness matrices, \( K^{GG} \) and \( K^{L(j-1)} \), are generated explicitly in the same manner as the conventional FEM and are used as a coefficient matrix of a linear system of equations. In contrast, the coupling stiffness matrices, \( K_{LG}, K_{GL}, \text{ and } K_{LG(i)} (i \neq j) \), are not generated. Instead of generating the coupling stiffness matrices, we use the concept of stress transfer, in which the stresses of the global mesh, \( \sigma^G \), and the stresses of the local mesh, \( \sigma^L \), computed by Eqs. (11) and (12) in their meshes are transferred to other meshes. Hence, the computations of the right-hand sides of Eqs. (9) and (10) are carried out. The interpolation methods of the stress transfers are described in the subsequent subsection.

After conducting the solution procedures of Eqs. (9) and (10), the convergence is checked by

\[
\left| \begin{bmatrix}
f^G \\
f^L \\
\vdots \\
f^L
\end{bmatrix}
- \begin{bmatrix}
f^{G(i+1)} \\
f^{L(i+1)} \\
\vdots \\
f^{L(i+1)}
\end{bmatrix}
\right|
= \left| \begin{bmatrix}
\int_{\Omega^G} B^G \sigma d\Omega - \sum_{j=1}^{N} \int_{\Omega^G} B^G \sigma_{j} d\Omega \\
\int_{\Omega^L} B^L \sigma d\Omega - \sum_{j=1}^{N} \int_{\Omega^L} B^L \sigma_{j} d\Omega \\
\vdots \\
\int_{\Omega^L} B^L \sigma d\Omega - \sum_{j=1}^{N} \int_{\Omega^L} B^L \sigma_{j} d\Omega
\end{bmatrix}
\right| \leq \tau \tag{13}
\]
where $\tau$ is a tolerance. The left-hand side is referred to as a relative residual norm. This convergence criterion is an extension of our previous studies (Yumoto et al., 2016a, 2016b; Yusa et al., 2018). Note that reaction forces at constrained degrees of freedom are included in the denominator of Eq. (13) in order to consider the contribution of the prescribed displacements.

2.2. Stress transfers between local meshes

In order to compute Eqs. (9) and (10), the stresses $\sigma^G$ and $\sigma^{L_j}$ defined in the global and local meshes should be transferred between the global and local meshes as well as between the local meshes. In our previous study (Yumoto et al., 2016a), a combination of the local least squares and nearest neighbor interpolations was found to give a good convergence in the iterative computations and to result in accurate stress solutions. The local least squares interpolation is applied to the stress transfers from the local meshes to the global mesh. The nearest neighbor interpolation is used to transfer the stresses from the global mesh to the local meshes. In the local least squares interpolation (Fig. 2 (a)), the stresses of the local meshes, $\sigma^{L_j}$, are transferred from the integration points of the local mesh that are located inside the element of the global mesh to the integration point of the global mesh. A linear basis function is used in the local least squares interpolation. In the nearest neighbor interpolation (Fig. 2 (b)), the stresses of the global mesh, $\sigma^G$, are transferred from the nearest integration point of the global mesh to those of the local mesh. The nearest neighbor interpolation is able to retain stress discontinuities at the element interfaces because of the shape functions of finite elements. Hence, we also adopt the nearest neighbor interpolation for the stress transfers between the local meshes. The stresses of the $j$-th local mesh, $\sigma^{L_j}$, are transferred from the $j$-th local mesh to the $i$-th local mesh by the nearest neighbor interpolation, as shown in Fig. 2 (c).

2.3. Element subdivision technique

In our previous study (Yumoto et al., 2016b), the element subdivision technique was used in the global mesh to improve the accuracy. This technique is also adopted in the present study in order to improve the accuracy as well as to achieve convergence of the iterative procedure. For the latter goal, the technique is used in both the global mesh and the local meshes.

Generally, strains and stresses of FEM have discontinuities at the element interfaces due to shape functions. Such discontinuous functions should be integrated numerically in s-FEM because the finite elements of the global mesh and the local meshes overlap each other. In addition, the finite elements of the local meshes overlap each other in the present study. However, the straightforward Gaussian quadrature cannot accurately integrate discontinuous functions. Thus, a finite element is divided into subdivisions, by which numerical integrals based on the Gaussian quadrature are performed. This technique is shown schematically in Fig. 3. For the purpose of explanation, the two-dimensional quadrangular finite element is shown, although this technique can be applied to other finite elements. In this figure, $4\times 4$ element subdivisions
are placed in \((\xi, \eta)\) coordinates. Each element subdivision has 2×2 integration points of the Gaussian quadrature.

3. Numerical results and discussions

3.1. Patch test

In general, a patch test examines the capability of an analysis method if the uniform strain/stress state could be reproduced. In the context of s-FEM, local meshes without any local features, such as holes, are superposed on the global mesh subject to uniform stress. Our previous study with a single local mesh (Yumoto et al., 2016a) indicated two major features of the coupling-matrix-free iterative s-FEM for a patch test. The first feature is that the number of iteration steps is always one. This is because the global–local coupling term, \(\int_{\Omega} B^L \sigma^G \, d\Omega\), becomes zero due to the uniform distribution of \(\sigma^G\). In the present method with multiple local meshes, the local–local coupling terms, \(\int_{\Omega} B^L \sigma^L \, d\Omega\), should be zero, which is confirmed in this patch test. The second feature is the problem of linear dependency of the displacement functions of the superimposed finite element models (Ooya et al., 2009). The coefficient matrix of Eq. (8) becomes singular due to the linearly dependent shape functions of the global and local meshes. In the coupling-matrix-free iterative s-FEM, the coefficient matrices of Eqs. (9) and (10) are generated in the same manner as the conventional FEM and are thus regular.

A model for the patch test used in the present study is a flat plate, the dimensions and boundary conditions of which are described in Fig. 4. Uniform tension is prescribed on the top and bottom edges of the plate, and rigid body modes are constrained. Moreover, the displacements at the boundaries of the local models, \(T^{GL_1}\) and \(T^{GL_2}\), are constrained as expressed in Eq. (3). The meshes are shown in Fig. 5, where a global mesh and two local meshes are visualized simultaneously. The two local meshes that overlap each other are superposed on the global mesh. Linear hexahedral finite elements in conjunction with the 2×2×2-point Gaussian quadrature were used. The numbers of elements and nodes of the global mesh are 2,408 and 3,267, respectively, and the numbers of elements and nodes of each local mesh are 1,024 and 1,445, respectively. Young’s modulus and Poisson’s ratio were set to be 210 GPa and 0.3, respectively.

In the analysis, a sufficiently small relative residual norm, \(7.87 \times 10^{-14}\), was achieved at the first iteration step. The maximum and minimum values of components of \(\sigma^{G}\) are [1.43×10^{-12}, 1.00×10^1, 5.16×10^{-12}, 4.87×10^{-12}, 2.09×10^{-12}, 3.61×10^{-12}]^T (MPa) and [−1.44×10^{-11}, 1.00×10^1, −4.18×10^{-12}, −7.35×10^{-12}, −3.71×10^{-12}, −5.17×10^{-12}]^T (MPa), respectively. These are x, y, z, xy, yz, and zx components of stresses. All the components of \(\sigma^{G}\) except for the y component are almost zero, whereas the y component is uniform. Those of \(\sigma^{L_1}\) are [2.07×10^{-13}, 1.48×10^{-13}, 1.46×10^{-13}, 6.36×10^{-14}, 1.01×10^{-13}, 9.98×10^{-14}]^T (MPa) and [−1.60×10^{-13}, −1.69×10^{-13}, −1.21×10^{-13}, −7.35×10^{-14}, −9.93×10^{-14}, −1.02×10^{-13}]^T (MPa), respectively. Those of \(\sigma^{L_2}\) are [3.57×10^{-13}, 1.96×10^{-13}, 1.58×10^{-13}, 1.67×10^{-13}, 7.33×10^{-14}, 1.34×10^{-13}]^T (MPa) and [−3.35×10^{-13}, −1.53×10^{-13}, −1.34×10^{-13}, −1.25×10^{-13}, −9.49×10^{-14}, −1.18×10^{-13}]^T (MPa), respectively. All the components of \(\sigma^{L_1}\) and \(\sigma^{L_2}\) are almost uniformly zero. The present method with multiple local meshes was shown to pass the patch test.

3.2. Two-hole problem

In order to investigate the basic performance of the coupling-matrix-free iterative s-FEM with multiple local meshes, a two-hole problem was analyzed. In this problem, two circular holes are located in a flat plate subject to uniform tension.
In order to model the two holes, two local meshes that overlap each other are used. As described below, the element subdivision technique is the key to achieving convergence.

The dimensions and boundary conditions of the flat plate with two circular holes are described in Fig. 6. Uniform tensile loads are prescribed on the top and bottom edges of the plate, and rigid body modes are constrained. Stress concentration should occur in the vicinities of the two holes. The global mesh and two local meshes of the problem are visualized simultaneously in Fig. 7, and an enlarged view is shown in Fig. 8. A donut-shaped mesh visualized in Fig. 9 is used for the two local meshes. The diameter of the mesh is 100 mm, which is twice that of the circular hole. The displacements at the boundaries of the local meshes, $\Gamma_{GL1}$ and $\Gamma_{GL2}$, are constrained as shown in Eq. (3). The numbers of elements and nodes of the global mesh are 2,048 and 3,267, respectively, and the numbers of elements and nodes of each local mesh are 8,192 and 10,880, respectively. The element subdivision technique was used for the global mesh and/or local meshes. Each element subdivision has $2 \times 2 \times 2$ integration points of the Gaussian quadrature. Moreover, as a reference solution, conventional finite element analysis was performed using a monolithic mesh visualized in Figs. 10 and 11. The numbers of elements and nodes are 9,760 and 12,795, respectively. Young’s modulus and Poisson’s ratio were set to be 210 GPa and 0.3, respectively. The tolerance of the convergence criterion was set to be $10^{-3}$.

The convergence histories of the iteration of the coupling-matrix-free iterative s-FEM are plotted in Figs. 12, 13, 14, and 15 for $1 \times 1 \times 1$ subdivision, $2 \times 2 \times 2$ subdivisions, $3 \times 3 \times 3$ subdivisions, and $4 \times 4 \times 4$ subdivisions, respectively, for the global mesh. The horizontal axes represent the iteration step, whereas the vertical axes represent the relative residual norm, which is shown in the left-hand side of Eq. (13). The numbers of iteration steps are summarized in Table 1.

Generally, the straightforward Gaussian quadrature ($1 \times 1 \times 1$ element subdivision) could not achieve a converged solution. When the analysis did not converge, the relative residual norm decreases first and then increases. Similar convergence histories can be found in our previous study (Yumoto et al., 2016a), in which the nearest neighbor interpolation was used for both the global–local and local–global stress transfers. In the present study, the nearest neighbor interpolation is also
used for the stress transfers between the local meshes. In order to obtain a converged solution, the integration points of the local meshes should be increased, so that nearest neighbor interpolation becomes accurate. In this problem, at least 2×2×2 subdivisions for both the global and local meshes are necessary for convergence, with the exception of 3×3×3 subdivisions for the global mesh in conjunction with 1×1×1 subdivision for the local mesh. The number of iteration steps does not change greatly for 2×2×2 subdivisions, 3×3×3 subdivisions, and 4×4×4 subdivisions and remains in the range of 100 to 122.

The distribution of stress in the $y$ direction normalized by the remote tensile stress is visualized in Figs. 16 and 17. The result for 2×2×2 element subdivisions for both the global mesh and the local meshes is shown. Finite elements of the global mesh inside the holes are not shown in the visualizations. The stress distribution appears to be sufficiently smooth, although our previous study (Yumoto et al., 2016b) recommended 4×4 subdivisions for two-dimensional quadrangular finite elements to obtain a smooth stress distribution. In three-dimensional analysis, 4×4×4 subdivisions require a huge amount of memory, namely, 16.3 GB for this simple problem in the developed program. Thus, 2×2×2 subdivisions are again used in the next subsection. Moreover, a reference solution computed by the conventional FEM is visualized in

Table 1 Number of iteration steps in the two-hole problem.

|                  | Local meshes: 1×1×1 subdivision | Local meshes: 2×2×2 subdivisions | Local meshes: 3×3×3 subdivisions | Local meshes: 4×4×4 subdivisions |
|------------------|---------------------------------|----------------------------------|----------------------------------|----------------------------------|
| Global mesh:     | Not converged                    | Not converged                    | Not converged                    | Not converged                    |
| 1×1×1 subdivision|                                 |                                  |                                  |                                  |
| Global mesh:     | Not converged                    | 110                              | 102                              | 100                              |
| 2×2×2 subdivisions|                                |                                  |                                  |                                  |
| Global mesh:     | Not converged                    | 105                              | 108                              | 108                              |
| 3×3×3 subdivisions|                                |                                  |                                  |                                  |
| Global mesh:     | Not converged                    | 122                              | 111                              | 112                              |
| 4×4×4 subdivisions|                                |                                  |                                  |                                  |
Fig. 16. Normalized stress in the $y$ direction of the two-hole problem computed by the coupling-matrix-free iterative s-FEM.

Fig. 17. Enlarged view of Fig. 16.

Fig. 18. Enlarged view of normalized stress in the $y$ direction of the two-hole problem computed by FEM (reference solution).

Fig. 19. Stress in the $y$ direction normalized by the remote tensile stress in the vicinity of the holes of the two-hole problem.

Fig. 20. Stress in the $y$ direction normalized by the remote tensile stress in the overlapping region of the two local meshes of the two-hole problem.

Fig. 18. The stress in the $y$ direction on the central line in the plate height ($y$) direction as well as in the plate thickness ($z$) direction is plotted in Fig. 19. The horizontal axis represents the $x$ coordinate from the center of the plate, whereas the vertical axis represents the stress in the $y$ direction normalized by the remote tensile stress. The present method is shown to be able to express the stress concentration even when multiple local meshes overlap each other. Yumoto et al. (2016b), in which the element subdivision technique was originally proposed for the coupling-matrix-free iterative s-FEM with non-overlapping local meshes, pointed out that stress solutions tend to be accurate when the element size of the local mesh is much smaller than that of the global mesh. In the present analysis, however, there is an inherent difficulty in that the element sizes of the local meshes are approximately the same because the same local mesh is superimposed on the global mesh repeatedly. Hence, the superposition of the local meshes may become a cause of loss of accuracy. The stress distribution in Fig. 19 shows an oscillation in the overlapping location of two local meshes, which is enlarged in Fig. 20. This figure indicates that $4 \times 4 \times 4$ element subdivisions produce a smoother stress distribution than $2 \times 2 \times 2$ element subdivisions. In addition, even with $4 \times 4 \times 4$ subdivisions, the stress values are approximately 10% smaller than those of the reference solution, which is the result of an ordinary finite element analysis. That is probably caused by the capability of linear hexahedral finite elements. It was indicated in our previous study with a single local mesh (Yumoto et al., 2016b) that very fine local mesh is required to evaluate the stress concentration accurately. Even when the ratio of the local to global mesh size was less than 3%, the stresses were approximately 10% smaller than those of the conventional FEM. Although further investigations are necessary, the loss of accuracy due to the superposition of local meshes is considered to be a cause of the convergence failure.

For reference, computational time was measured and compared. A computer having the Intel Core i5-4590 CPU with the DDR3-1600 memory was used. The program was coded by the C99 language and compiled by the GNU Compiler.
300
10 MPa
30×⌀10
local
models

Fig. 21 Dimensions and boundary conditions of a turbine blade problem.

Fig. 22 Global and local meshes of the turbine blade problem.

Fig. 23 Enlarged view of Fig. 22.

Fig. 24 Local mesh of the turbine blade problem.

Collection (gcc) with the –O2 option. The computational time of the conventional FEM was 25 s, in which 23 s was used for LDL factorization. In the present study, a skyline-based LDL factorization solver was used for solving linear systems of equations. The computational time of the coupling-matrix-free iterative s-FEM with $2 \times 2 \times 2$ subdivisions was 543 s. Most time was devoted to the iterative procedures.

3.3. Turbine blade structure with many holes

In order to investigate the capability of the present method with many local meshes for many local features, a section of a gas turbine blade structure with 30 holes was analyzed. Note that a gas turbine blade structure with multiple holes was previously analyzed by the coupling-matrix-free iterative s-FEM with a single local mesh (Yusa et al., 2018), which cannot deal with overlaps of local meshes. In practical situations, holes may be very close to each other, so that overlapping of local meshes is unavoidable.

The dimensions and boundary conditions of the turbine blade structure are described in Fig. 21. A uniform tensile load of 10 MPa is prescribed on the top edge, whereas the bottom edge is constrained. A total of 30 circular holes of 10 mm in diameter are situated as shown in the figure. The global and local meshes are visualized simultaneously in Fig. 22. A total of 30 local meshes are used to model the 30 holes. The enlarged view of Fig. 22 is shown in Fig. 23. One of the local meshes is visualized in Fig. 24. The diameter of the donut-shaped local mesh is 24 mm. The displacements at...
the boundaries of the local meshes, $F^{GL}$, are constrained as shown in Eq. (3). The numbers of elements and nodes of the global mesh are 130,560 and 163,108, respectively, and the numbers of elements and nodes of each local mesh are 16,384 and 19,584, respectively. The element subdivision technique with 1×1×1 subdivision and 2×2×2 subdivisions was used, because more than 3×3×3 subdivisions required a very large amount of memory. Each element subdivision has 2×2×2 integration points of the Gaussian quadrature. Young’s modulus and Poisson’s ratio were set to be 210 GPa and 0.3, respectively. The tolerance of the convergence criterion was set to be $10^{-3}$.

The convergence histories of the iteration of the present method are plotted in Fig. 25. The horizontal axis represents the iteration step, whereas the vertical axis represents the relative residual norm. As discussed in the previous subsection, the straightforward Gaussian quadrature (1×1×1 element subdivision) for the global and/or local meshes could not achieve convergence. In contrast, 2×2×2 element subdivisions for both the global and local meshes produced a converged solution. The number of iteration steps was 263.

The distribution of equivalent stress normalized by the remote tensile stress is visualized in Figs. 26 and 27. Finite elements of the global mesh inside the holes are not depicted in the visualizations. The present method is shown to be able to deal with s-FEM analysis with many local meshes that overlap each other. The holes near the top edge of the turbine blade in Fig. 27 produce higher stress values than the other holes. Such stress characteristics can be evaluated tractably by the present method.

**Fig. 25** Convergence histories of the coupling-matrix-free iterative s-FEM in the turbine blade problem.

![Convergence histories of the coupling-matrix-free iterative s-FEM in the turbine blade problem.](image)

**Fig. 26** Normalized equivalent stress of the turbine blade problem.

**Fig. 27** Enlarged view of Fig. 26.
4. Conclusion

In the present paper, the coupling-matrix-free iterative s-FEM (Yumoto et al., 2016a, 2016b; Yusa et al., 2018) was extended to perform analysis with multiple local meshes. The present method does not require the generation of coupling stiffness matrices between the global and local meshes or between the local meshes. The formulation of the present method with multiple local meshes was newly derived from that of the original s-FEM with multiple local meshes (Okada et al., 2004; Tanaka et al., 2006; Kikuchi et al., 2008; Kamaya et al., 2010). The present formulation requires stress transfers between the global and local meshes and between pairs of local meshes. In order to perform these stress transfers, the combination of interpolation methods for multiple local meshes was presented as an extension of our previous studies with a single local mesh (Yumoto et al., 2016a, 2016b; Yusa et al., 2018). Several numerical results demonstrated the capability of the present method. Two-hole and many-hole problems were analyzed. First, the present method successfully passed the patch test, in which a uniform stress distribution was well represented. The basic performance of the present method was then investigated through an analysis of a two-hole problem. The present method exhibited the capability of representing the stress concentration due to holes that are located in the vicinity of each other. Moreover, numerical integration methods of the global mesh as well as the local meshes were found to have significant influences on achieving convergence. In this problem, the straightforward Gaussian quadrature could not achieve convergence, whereas the element subdivision technique could. This is a new finding of the present study. Further investigations are needed in order to seek a better treatment of the numerical integration for the overlapping region between the local meshes. Finally, a structure model with 30 circular holes was analyzed in order to demonstrate the capability of the present method with many local meshes for many local features. This many-hole problem was successfully analyzed by the present method using the element subdivision technique.

In the future, convergence acceleration of the coupling-matrix-free iterative s-FEM with multiple local meshes should be investigated. For a single local mesh, a speedup of 100 times was already established using a linear or nonlinear iterative solution method (Yusa et al., 2018). Moreover, the present method will be applied to nonlinear problems, such as large-deformation elastic–plastic problems. A nonlinear iterative solution method, such as the Newton–Raphson method, can be combined with the present method, because the present method for a linearly elastic material already involves an iterative algorithm based on force equilibrium.

Acknowledgment

The present study was supported by JSPS KAKENHI Grant Number JP16K05988.

References

Budyn, É., Zi, G., Moës, N. and Belytschko, T., A method for multiple crack growth in brittle materials without remeshing, International Journal for Numerical Methods in Engineering, Vol. 61, No. 10 (2004), pp. 1741–1770.

Chen, X., Li, Z. and Wang, H., Progressive failure analysis of an open-hole composite laminate by using the s-version finite-element method, Mechanics of Composite Materials, Vol. 50, No. 3 (2014), pp. 279–294.

Fish, J., The s-version of the finite element method, Computers and Structures, Vol. 43, No. 3 (1992), pp. 539–547.

Gerasimov, T., Noi, N., Allix, O. and De Lorenzis, L., A non-intrusive global/local approach applied to phase-field modeling of brittle fracture, Advanced Modeling and Simulation in Engineering Sciences, Vol. 5 (2018), No. 1, Article 14.

Kamaya, M., Miyokawa, E. and Kikuchi, M., Growth prediction of two interacting surface cracks of dissimilar sizes, Engineering Fracture Mechanics, Vol. 77, No. 16 (2010), pp. 3120–3131.

Kaneko, S., Okada, H. and Kawai, H., Development of automated crack propagation analysis system (multiple cracks and their coalescence), Journal of Computational Science and Technology, Vol. 6, No. 3 (2012), pp. 97–112.

Kikuchi, M., Wada, Y. and Takahashi M., Fatigue crack growth simulation using S-FEM, Transactions of the Japan Society of Mechanical Engineers, Series A, Vol. 74, No. 742 (2008), pp. 812–818 (in Japanese).

Lee, S.-H., Song, J.-H., Yoon, Y.-C., Zi, G. and Belytschko, T., Combined extended and superimposed finite element method for cracks, International Journal for Numerical Methods in Engineering, Vol. 59, No. 8 (2004), pp. 1119–1136.
Loehnert, S. and Belytschko, T., A multiscale projection method for macro/microcrack simulations, International Journal for Numerical Methods in Engineering, Vol. 71, No. 12 (2007), pp. 1466–1482.
Moës, N., Dolbow, J. and Belytschko, T., A finite element method for crack growth without remeshing, International Journal for Numerical Methods in Engineering, Vol. 46, No. 1 (1999), pp. 131–150.
Nakasumi, S., Suzuki, K. and Ohtsubo, H., Crack growth analysis using mesh superposition technique and X-FEM, International Journal for Numerical Methods in Engineering, Vol. 75, No. 3 (2008), pp. 291–304.
Okada, H., Liu, C. T., Ninomiya, T., Fukui, Y. and Kumazawa, N., Analysis of particulate composite materials using an element overlay technique, Computer Modeling in Engineering and Sciences, Vol. 6, No. 4 (2004), pp. 333–348.
Ooya, T., Tanaka, S. and Okada, H., On the linear dependencies of interpolation functions in s-version finite element method, Journal of Computational Science and Technology, Vol. 3, No. 1 (2009), pp. 124–135.
Pyo, C. R., Okada, H. and Atluri, S. N., An elastic–plastic finite element alternating method for analyzing wide-spread fatigue damage in aircraft structures, Computational Mechanics, Vol. 16, No. 1 (1995a), pp. 62–68.
Pyo, C. R., Okada, H. and Atluri, S. N., Residual strength prediction for aircraft panels with Multiple Site Damage, using the “EPFEAM” for stable crack growth analysis, Computational Mechanics, Vol. 16, No. 3 (1995b), pp. 190–196.
Tanaka, S., Okada, H., Watanabe, Y. and Wakatsuki, T., Applications of s-FEM to the problems of composite materials with initial strain-like terms, International Journal for Multiscale Computational Engineering, Vol. 4, No. 4 (2006), pp. 411–428.
Xu, Q., Chen, J., Yue, H. and Li, J., A study on the s-version FEM for a dynamic damage model, International Journal for Numerical Methods in Engineering, Vol. 115, No. 4 (2018), pp. 427–444.
Yumoto, Y., Yusa, Y. and Okada, H., An s-version finite element method without generation of coupling stiffness matrix by using iterative technique. Mechanical Engineering Journal, Vol. 3, No. 5 (2016a), Paper No. 16-00001.
Yumoto, Y., Yusa, Y. and Okada, H., Element subdivision technique for coupling-matrix-free iterative s-version FEM and investigation of sufficient element subdivision, Mechanical Engineering Journal, Vol. 3, No. 5 (2016b), Paper No. 16-00361.
Yusa, Y., Okada, H. and Yumoto, Y., Three-dimensional elastic analysis of a structure with holes using accelerated coupling-matrix-free iterative s-version FEM, International Journal of Computational Methods, Vol. 15, No. 5 (2018), 1850036 (35 pages).