 EEMAI 2023
Volume-5-(2023)

Superposed element method for the numerical simulation of structure crack

Jianxin Ding[1],Qingzhou Yang[2]

[1]Changjiang Institute of Survey, Planning, Design, and Research, Wuhan, China

[2]PowerChina Hubei Electric Engineering Corporation Limited, Wuhan, China

*Corresponding author e-mail: whudjx@126.com

Abstract. Crack is a key factor for the normal operation of structures. In this paper, a new method called Superposed Element Method (SEM) is proposed for the numerical simulation of crack analysis. Firstly, the structure containing crack is divided into two independent meshes. One is the global mesh without crack, the other is the local mesh of the crack and its vicinity. Then, the two independent meshes are coupled together, finite element method (FEM) is used for the local mesh and the complete elements outside of the coverage of the local mesh, and numerical manifold technique is applied to the incomplete elements. Finally, in order to couple the global mesh with the local mesh, the penalty stiffness is imposed on the boundaries of local mesh. The results of the numerical examples show that the pre-processing of SEM is quite simple, and the accuracy of the solution is relatively high. The method proposed is expected to be widely used for dynamic simulation of crack propagation.

Keywords: numerical analysis; superposed element method (SEM); crack; stress intensity factor.

1. Introduction

The existence of cracks could affect the normal operation of structures and may lead to their instability and failure. Thus, it is necessary to specifically simulate the mechanical properties and propagation process of cracks. The geometric shape of cracks in engineering practices tend to be irregular, and the change of boundary conditions may occur with the propagation process of cracks, therefore, it is difficult for the numerical analysis of cracks.

At present, there are several numerical methods for crack analysis, among which the finite element method (FEM) has an obvious advantage with a mature theory and higher commercialization level of the software. Thus, it is widely used in engineering practices, especially in the analysis of static cracks [1]. However, it is difficult in pre-processing with remeshing operation and of low efficiency by using FEM. In view of this, many scholars devote to the research of extended finite element method (XFEM), element-free method (EFM), numerical manifold method (NMM) [2-6], etc. The common advantage of these new methods is the convenient process of pre-processing, there is no need for remeshing operation or just some points needs to be added in local. Based on the simulation of discontinuity of the crack surface and singularity of crack tips, the dynamic simulation of crack propagation is carried out by establishing complex shape functions or cover functions in local crack. However, the theories of these methods are not mature enough, the calculation is of low efficiency, and it is difficult to establish shape functions or cover functions especially in 3D crack problems. Therefore, it still remains further research.

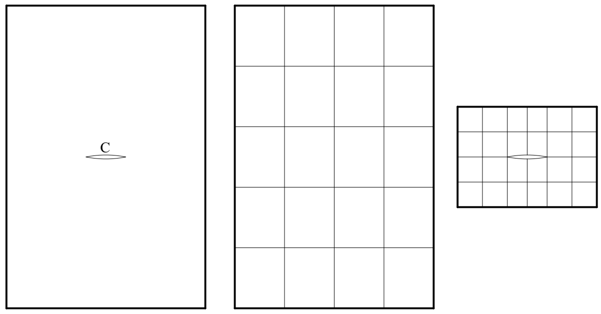
Based on the idea of superposed element method (SEM) for stress-strain and seepage analysis [7-11], SEM for the analysis of crack is proposed in this paper. Firstly, and the structure containing crack is divided into two independent meshes. One is the global mesh without crack, the other is the local mesh of crack and its vicinity. Then, the two independent meshes are coupled together, FEM is used for the local mesh and the complete elements outside of the coverage of the local mesh, and numerical manifold technique [12] is applied to the incomplete elements. Finally, in order to couple the local mesh with the global mesh, the penalty stiffness is imposed on the boundaries of local mesh. This method has two main characteristics: first, we can take advantage of the theoretical maturity of FEM which is applied to the simulation of local crack; second, for the problems of dynamic crack propagation, the modeling tasks are greatly simplified with only the local crack with free boundary shapes needs to be remeshed by adopting the technique of automatic mesh generation.

2. Basic principles

2.1 Structural discretization

Typical global and local mesh discretizations of SEM are given in Figure 1. The structure containing crack is divided into two independent meshes, one is the global mesh without crack (Figure 1(b)), and the other is the local mesh of crack with its vicinity (Figure 1(c)).

2.2 Displacement model



(a) Structure with crack (b) Global mesh (c) Local mesh

Figure 1. Mesh discretization in SEM

Shown in Figure 2 is the superposed mesh after the local and global mesh is coupled together. The part from global mesh which is covered by local mesh (complete and incomplete elements included) is treated as ineffective elements. The remaining effective elements can be divided into the following 3 types:

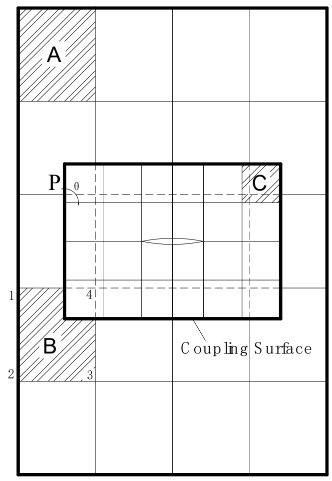


Figure 2. Local mesh superposed into global mesh

(1) Type A: the complete elements from global mesh which are not covered by local mesh;
(2) Type B: the incomplete elements from global mesh which are partly covered by local mesh;
(3) Type C: the complete elements from local mesh.

For the complete elements of type A and C where all the nodes are real, the displacement model is consistent with the ordinary finite element format:

$$\{\Delta\mu\}_e = [N]\{\Delta\delta\}_e \tag{1}$$

Where is the shape function of ordinary finite element, andis the node displacement increment.

For the incomplete elements of type B, the nodes out of local mesh are treated as real nodes, and the other nodes in local mesh are treated as virtual nodes. Nodes 1, 2, 3 are real nodes and node 4 is a virtual node, as shown in Figure 2. The displacement increment of an arbitrary node in this region is calculated by interpolation between the displacement of real nodes and virtual nodes:

$$\{\Delta\mu\}_e = [N]\begin{Bmatrix}\Delta\delta_{\mathrm{A}} \\ \Delta\delta_{\mathrm{V}}\end{Bmatrix}_e \tag{2}$$

Where $\{\Delta\delta_{\mathrm{A}}\}$ and $\{\Delta\delta_V\}$ are the displacement increment of real nodes and virtual nodes, respectively.

2.3 Deformation equation of coupling surface

The displacement of an arbitrary node in global or local mesh is determined by Eqs (1) and (2). However, the two independent meshes(global mesh and local mesh) need to be coupled together through the coupling surface of local mesh.

The displacement increment $\{\Delta\mu_{\mathrm{g}}\}$ of an arbitrary point P(Figure 2) in global mesh which is on the boundary of local mesh can be calculated by Eq.(1); the displacement increment in local mesh can be calculated by Eq.(2).

Thus, the relative increment of deformation of point P in the global and local mesh is

$$\{\Delta\delta\} = [L]_k(\{\Delta\mu_{\mathrm{g}}\} - \{\Delta\mu_{\mathrm{l}}\}) \tag{3}$$

Where $[L]_k$ is the matrix for local coordinate transform, which is the function of the angle θ of the boundary line and horizontal line in local mesh.

Theoretically, the deformation of point P in global mesh is supposed to be exactly the same as that in local mesh, so $\{\Delta\delta\}$ should be zero. The boundary surface of local mesh is treated as coupling surface and the stiffness is assumed to be huge, therefore, the relative increment $\{\Delta\delta\}$ of deformation is approximately to be zero.

2.4 Constitutive equation

In a preliminary research, only linear elastic constitutive relation is considered, therefore, the relationship between stress increment $\Delta\sigma$ and strain increment $\Delta\varepsilon$ of elements of type A, B, C can be expressed by:

$$\{\Delta\delta\}_i = [D]_i\{\Delta\varepsilon\}_i \quad (\text{i = A, B, C}) \tag{4}$$

Where $[D]_i$ is the elastic matrix of elements, which is the function of the modulus of elasticity and Poisson's ratio.

The relationship between stress increment and relative deformation increment of coupling surface k is shown in the same manner.

$$\{\Delta\delta\}_k = [D]_k\{\Delta\delta\}_k \tag{5}$$

Where $[D]_i$ is the elastic matrix of coupling surface, which is the function of the normal stiffness coefficient k_{n} and tangential stiffness coefficient k_{s} of the coupling surface.

2.5 Equilibrium equations

Based on the virtual work principle, the virtual work function of system can be expressed by:

�W_A + �W_B + �W_C + �$W_k = W_F$ (6)

Where W_{A}, W_{B}, W_{C} represent the virtual work of elements of type A, B, C, respectively. W_k is the virtual work of coupling surface k; W_F is the virtual work of the external load. W_{A}, W_{B}, W_{C} are given by:

$$W_i^e = \iiint_{\Omega i}\left\{\Delta\varepsilon^{*e}\right\}_i^{\mathrm{T}}\left\{\Delta\sigma^e\right\}_i \mathrm{d}\Omega \quad (\text{i = A, B, C}) \tag{7}$$

Where the vector with superscript “*” is a virtual vector. W_k is given by:

$$W_k = \iint_{Sk} \{\Delta\delta^*\}_k^T \{\Delta\sigma\}_k dS \tag{8}$$

Where sk is the surface integral region of coupling surface k.

The virtual work W_k of the external load is given by:

$$W_F = \sum_{i=1}^{nf} \{\Delta U^*\}_i^T \{\Delta F\}_i \tag{9}$$

Where nf is the vector number of external load.

By substituting the displacement functions (1), (2), (3) and the constitutive relations (4), (5) in (6) ~ (9), we have:

$$[K]\{\Delta U\} = \{\Delta F\} \tag{10}$$

Where $[K]$, $\{\Delta U\}$, $\{\Delta F\}$ are the whole generalized stiffness matrix, incremental vector of the whole generalized degree of freedom(DOF) and incremental vector of the whole generalized load, respectively.

The whole generalized stiffness matrix $[K]$ contains 3 different types of submatrix, they are stiffness matrix $[K]_{\mathrm{ABC}}$ of ordinary elements of type A, B, C and generalized stiffness matrix $[K]_{\mathrm{gg}}$, $[K]_{\mathrm{ll}}$ and $[K]_{\mathrm{gl}}$, $[K]_{\mathrm{lg}}$ of coupling surface.

For elements of type A and C, the ordinary gauss integration is applicable as the domain of integration is the complete ordinary finite elements. For the elements of type B, since the domain of integration is only a part of elements, an approximate numerical integration result can be obtained through gauss integration with dense points. Generally, the expression of $[K]_{\mathrm{ABC}}$ is the same as that of ordinary FEM.

$$[K]_{\mathrm{ABC}} = \iiint_{\Omega} [B]^{\mathrm{T}}[D][B] d\Omega \tag{11}$$

Let

$$[M]_k = [L]_k^T [D]_k [L]_k \tag{12}$$

Then

$$[K]_{gg} = \iint_{Sk} [N]_g^T [M]_k [N]_g dS \tag{13}$$

$$[K]_{ll} = \iint_{Sk} [N]_l^T [M]_k [N]_l dS \tag{14}$$

Based on the principle of symmetry, we have:

$$[K]_{gl} = [K]_{\mathrm{lg}} = -\iint_{Sk} [N]_i^T [M]_k [N]_j dS \tag{15}$$

The numerical integration described in Eqs. (13)~(15) needs to be done on the interface of global and local mesh. The shape of local mesh can be determined directly by topological analysis as the interface is just the outer surface of the local mesh. For the global mesh, the results will be obtained through geometry computation when the elements are cut to be incomplete by the interface.

The global incremental vectors of DOF and loads are calculated by:

$$\{\Delta U\} = \{\{\Delta U\}_1^T \cdots \{\Delta U\}_{nn}^T\}^T \tag{16}$$

$$\{\Delta F\} = \{\{\Delta F\}_1^T \cdots \{\Delta F\}_{nn}^T\}^T \tag{17}$$

Where nn is the number of effective finite element nodes (virtual nodes included); $\{\Delta U\}_i$, $\{\Delta F\}_i$ are the incremental vectors of DOF(real and virtual nodes included) and loads, respectively. Thus it can be seen that $\{\Delta U\}$, $\{\Delta F\}$ is formally consistent with that of ordinary FEM.

3. Verification examples

Based on the theory of SEM for crack analysis, an interrelated computer program was developed and 2 numerical examples were analysed. It is studied in example 1 that the size of local mesh region has an important role in the accuracy of solutions. According to the results of example 1, the stress intensity factor of a elliptic crack is calculated in the example 2. Every numerical example is calculated by both FEM and SEM, and the results are compared with analytical solutions.

3.1 Center cracked tension panel

A typical center cracked panel which is subject to tension is shown in Figure 3. The length of the horizontal crack is $2a$, 2H = 200mm, 2W = 100mm, modulus of elasticity E = 104 MPa, Poisson's ratio μ = 0.25, tensile stress σ = 9.0 MPa.

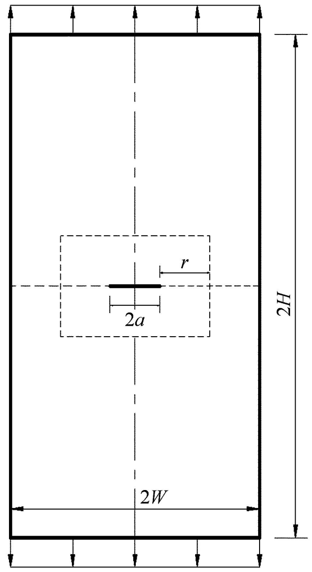


Figure 3. Center cracked panel subject to tension

The stress intensity factor is calculated when $a/W = 0.1, 0.3, 0.5$ with different ratio of the size r of local mesh region to the size Lg of global mesh, and the results are compared with analytical [13] and FEM solutions. The superposed element mesh of the panel when $a/W = 0.3$ is shown in Figure 4.

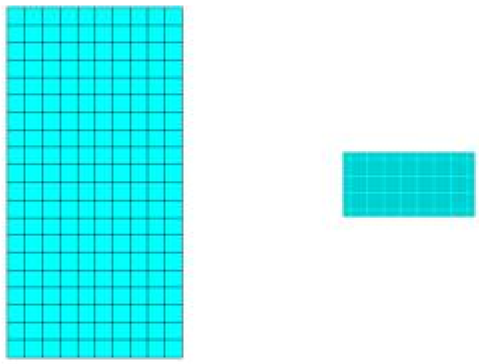


Figure 4. Superposed element mesh of a center cracked tension panel

Figure. 5 shows the relationship between the stress intensity factor and values of r/Lg with different a/W as 0.1, 0.3 and 0.5. It is seen that the results obtained by SEM agree well with the results obtained by analytical and FEM solutions. The relative error between the analytical results and results calculated by FEM, SEM is presented in Table 1.

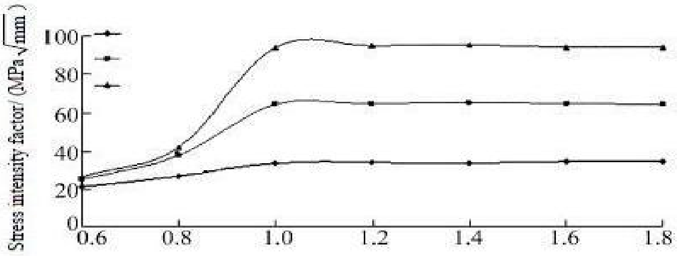


Figure 5. Stress intensity factor of center cracked tension panel

Table 1. Relative error between analytical and FEM, SEM solutions

a/W	Relative error with analytical solution/%	
	FEM	SEM
0.1	3.32	4.72
0.3	1.31	1.53
0.5	0.89	0.91

3.2 Elliptic crack

A typical elliptic crack is shown in Figure.6. Modulus of elasticity E, Poisson's ratio μ, tensile stress σ is the same as described example 1. W = 100 mm, T = 100 mm, 2H =100mm, a = 10mm, c = 20 mm.

Both the handbook of stress intensity factor [13] and Xie de et al. [14] provide an analytical solution to this problem: The stress intensity factor of the point A on the fracture front with coordinate $x = a\cos\theta$, $y = c\sin\theta$:

$$K_{IA} = K_I(\theta) = \frac{\sigma\sqrt{\pi b}}{E(\kappa)}\Theta \; ; \; K_{IIA} = K_{III}(\theta) = 0 \tag{18}$$

$$K_{IA} = K_I(\theta) = \frac{\sigma\sqrt{\pi b}}{E(\kappa)}\Theta = (\sin^2\theta + \frac{b^2}{a^2}\cos^2\theta)^{1/4}$$

Where:

$E(\kappa) = \int_0^{\pi/2}\sqrt{1-\kappa^2\sin\varphi}\,\mathrm{d}\varphi$ is the second kind of complete ellipse integral, modulus $\kappa = \sqrt{1-\frac{b^2}{a^2}}$, particular value: $E(1)=1$; $E(0)=\pi/2$.

When $\theta = \pm\frac{\pi}{2}$ (Equivalent to the ends of the short semi-axial of the ellipse), K_I take the maximum value:

$$K_{Imax} = K_I(\theta = \pm\frac{\pi}{2}) = \frac{\sigma\sqrt{\pi b}}{E(\kappa)} \tag{19}$$

For elliptic crack, the stress intensity factor along the upper edge of cracks is not a constant any more. The stress intensity factor is calculated with $r/Lg = 1.6$, and the results are compared with analytical [13] and FEM solutions. Figure 7 shows the superposed element mesh with elliptic crack.

Figure 8 shows the non-dimensionless stress intensity factor of the elliptical crack which is treated as follow:

$$F = \frac{K_I}{2\sigma\sqrt{\frac{c}{\pi}}} \tag{20}$$

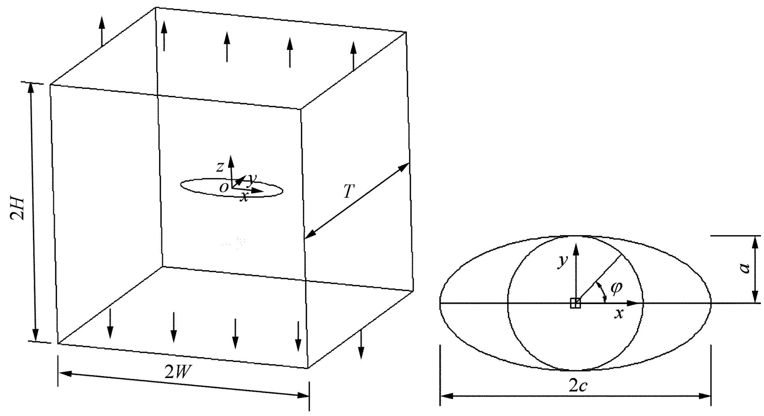


Figure 6. Elliptical crack

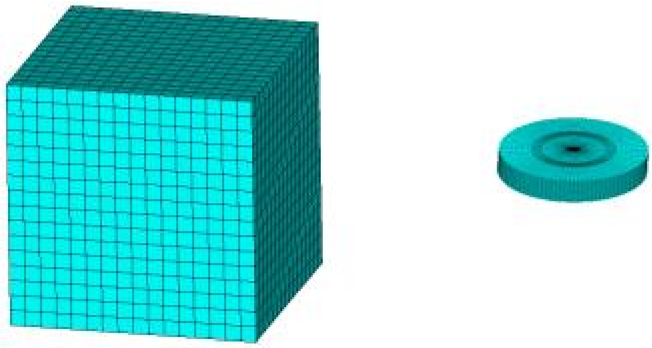


Figure 7. SEM mesh of elliptical crack

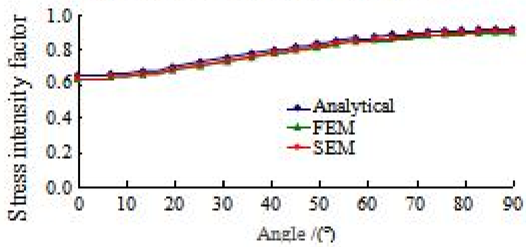


Figure 8. Non-dimensional stress intensity factor of elliptical crack

For elliptic crack, the largest magnitude of error of FEM and SEM is 3.23% and 2.54% respectively. The comparison shows that the distribution law of stress intensity calculated by FEM and SEM is basically the same, and agrees well with the analytical solution.

4. Conclusion

This paper proposes and studies SEM for crack analysis. The method has the following advantages:

(1) We can take advantage of the theoretical maturity of FEM which is applied to the simulation of local cracks;

(2) The global and local mesh of SEM are independent from each other, which makes the mesh generation and control of element morphology simpler. With a coarse grid in global mesh and a fine grid in local mesh, an accurate result can be obtained with less amount of calculation;

(3) For the problems of dynamic crack propagation, the modeling tasks are greatly simplified with only the local mesh of crack with free boundary shapes needs to be remeshed by adopting the technique of automatic mesh generation.

References

[1] Wang W.M Xu M.Y, Chen S.H. Finite element analysis of 3D crack propagation without remeshing[J]. Chinese Journal of Rock Mechanics and Engineering, 2001, 20(Supp.2):1 608–1 622.

[2] Kou X.D, Zhou W.Y. Using element-free method to trace crack propagation [J]. Chinese Journal of Rock Mechanics and Engineering, 2000, 19(1):18–23.

[3] Lu B, Ge X.R, Kong X.L. Comparative study on finite element method, meshless method and natural element method [J]. Chinese Journal of Rock Mechanics and Engineering, 2005, 24(5):780–786.

[4] Wang S.L, Ge X.R. Application of manifold method in simulating crack propagation [J]. Chinese Journal of Rock Mechanics and Engineering, 1997, 16(5):405–410.

[5] Yu T.T. Extended finite element method analysis of discontinuities [J]. Journal of Ship Mechanics, 2005, 24(5):780–786.

[6] Sukumar N, Prevost J H. Modeling quasi-static crack growth with the extended finite element method. Part I: computer implementation [J]. International Journal of Solids and Structures, 2003, 40:7 513–7 537.

[7] Belytschko T, Fish J, Bayliss A. The spectral overlay on finite elements for problems with high gradients [J]. Computer Methods in Applied Mechanics and Engineering, 1990, 81(1):71–89.

[8] Fish J. The s-version of the finite element method [J]. Computers and Structures, 1992, 43(3):539–547.

[9] Fish J, Markolefas S, Guttal R, et al. On adaptive multilevel superposition of finite element meshes for linear elastostatics[J]. Applied Numerical Mathematics, 1994, 14(1/3):135–164.

[10] Wang W.M, Chen S.H. A novel numerical analysis method for rock masses—superposed element method [J]. Chinese Journal of Rock Mechanics and Engineering, 2005, 24(7):1 093–1 098.

[11] Wang W.M, Liu Y.L, Ding J.X, et al. Superposed element method for simulating of drainage holes in seepage analysis [J]. Rock and Soil Mechanics, 2011, 32(Supp.1):674–679.

[12] Shi G.H. Numerical manifold method and discontinuous deformation analysis [M]. Beijing: Tsinghua University Press, 1997:53–61.

[13] Chinese aeronautical establishment. Stress intensity factors handbook [M]. Beijing: Science Press, 1981:164–322.

[14] Xie D, Qian Q, Li C.A. Numerical Calculation Method and Engineering Application in Fracture Mechanics [M]. Beijing: Science Press, 2009.