FATIGUE CRACK GROWTH ANALYSIS OF SS400 USING VIRTUAL ELEMENT METHOD

Jongyeop Kim[1], Nastaran Movahedi[1], Nam-Su Huh[2], Chang-Young Oh[3], Kyoungsoo Park[4]

[1] Graduate student, Yonsei University, Seoul, Korea
[2] Professor, Seoul National University of Science and Technology, Seoul, Korea
[3] Principal research engineer, Korea Institute of Materials Science, Kyungnam Changwon, Korea
[4] Professor, Yonsei University, Seoul, Korea (k-park@yonsei.ac.kr)

ABSTRACT

For the accurate numerical analysis of the fatigue crack growth of SS400, the use of a polygonal mesh and virtual element method is presented in this study. A graded polygonal mesh with a crack is generated with a mesh attaching process associated with an element split with node insertion and duplication. To evaluate the incremental length of the crack growth, the Paris law is utilized. The stress intensity factor in the Paris law equation is evaluated using the virtual grid-based stress recovery and interaction integral. The numerical result of the example agrees well with the analytical solution and experimental result. Furthermore, the results demonstrate that the virtual element method is suitable for handling polygonal elements and simulating fatigue crack growth behavior.

INTRODUCTION

Fatigue fracture can lead to failure of a critical structure such as a reactor vessel in nuclear power plant, which results in a significant risk in the safety of our community. Therefore, the accurate analysis and prediction of the fatigue crack growth is one of essential aspects in the safety analysis of critical structures. Numerical analysis of a fatigue crack usually uses the finite element method (FEM), and the analysis with FEM of fatigue crack behavior requires a high-quality re-meshing process according to the crack growth. However, the mesh generation is highly a time-consuming task, especially for a complex crack geometry. To effectively discretize a complex crack geometry, a polygonal element based on the virtual element method (VEM) (Beirão da Veiga et al., 2013) can be utilized because of the flexibility of the element shape.

In this study, a single-edge notched tension (SENT) test with SS400 material is performed to assess the suitability of VEM for the fatigue crack analysis. A mesh with polygonal elements is generated by simply creating a crack with the node duplication and the element split. The VEM is utilized to solve a boundary value problem and evaluate the nodal displacement. Based on the nodal displacement, the stress intensity factor (SIF) is accurately evaluated using the virtual gird-based stress recovery (VGSR) in a mesh with polygonal element (Choi & Park, 2019; Choi et al., 2022; Kim at el., 2023), and the incremental crack length is evaluated using the Paris law (Paris & Erdogan, 1963).

PROBLEM DESCRIPTION

The geometry, material property, and boundary condition of the SENT test with SS400 material under a remote tensile stress (σ) are illustrated in Figure 1, where an initial crack length (a) is 3 mm with the depth (t) of 3.5 mm. The elastic modulus (E) and Poisson's ratio (ν) are 206 GPa and 0.3, respectively. The cyclic load is applied at the left and right side of the specimen. The maximum traction (σ_{max}) is 130.5 MPa and the minimum traction (σ_{min}) is 13.05 MPa. The plane stress condition is assumed for the two-dimensional numerical analysis. The domain is discretized with arbitrary shaped polygonal elements. For the evaluation of the incremental crack length, the Paris law equation (Paris & Erdogan, 1963) is given as:

$$da/dN = C\,(\Delta K)^a \tag{1}$$

where da is the increment of crack length, dN is change of loading cycle, ΔK is the range of SIF. The constants of $C = 10^{-21.6}$ and $m = 6.12$ are experimentally obtained as shown in Figure 2 (Lee et al., 2022).

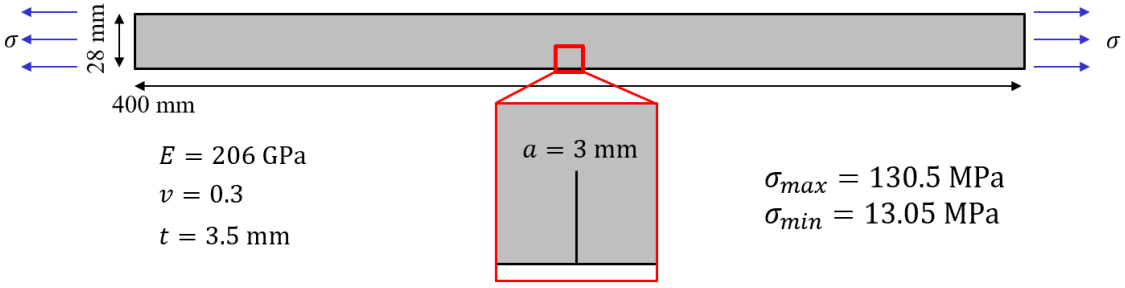

Figure 1. Single edge notched tension test (SENT) specimen.

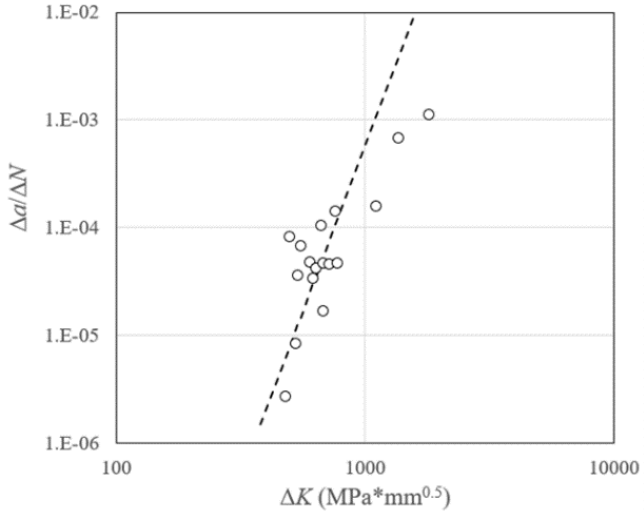

Figure 2. Relation between the SIF and the fatigue crack growth rate (Lee et al., 2022).

The relation between the loading cycle and crack length is evaluated using the analytical expression (Al Laham & Branch, 1999). The analytical expression of the SIF (K_I) for SENT is expressed as:

$$K_I = \sigma\sqrt{\pi a}\,\frac{\left(1+\frac{2a}{W}\right)}{\left(1-\frac{a}{W}\right)}\left[\begin{array}{c}1.12078 - 3.68220\left(\frac{a}{W}\right) + 11.95434\left(\frac{a}{W}\right)^2 - 25.85210\left(\frac{a}{W}\right)^3 \\ +33.09762\left(\frac{a}{W}\right)^4 - 22.4422\left(\frac{a}{W}\right)^5 + 6.17836\left(\frac{a}{W}\right)^6\end{array}\right] \tag{2}$$

where the width of the specimen (W) is 28 mm in this example. Figure 3 shows the comparison between the analytical solution and the experimental result for the relation between the loading cycle and crack length.

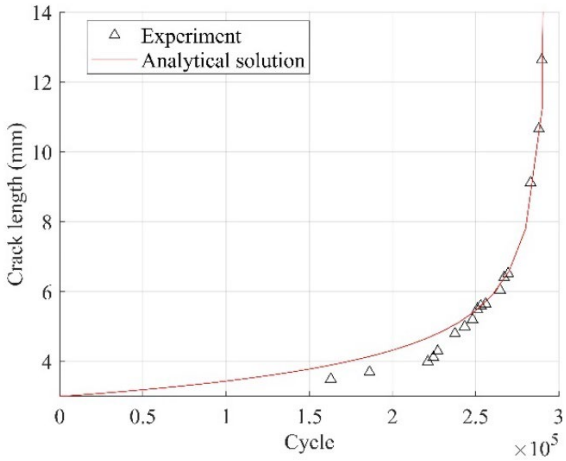

Figure 3. Analytical and experimental relation between loading cycle and crack length.

ANALYSIS METHOD

The analysis procedure for the SENT example is presented. The procedure consists of three steps, i.e., generation of a polygonal mesh, computation of the nodal displacement using VEM, and evaluation of the stress intensity factor and crack increment length. These three steps are repeated for the selected loading cycle until a crack tip reaches around a domain boundary.

Generation of polygonal mesh

A graded polygonal mesh with a crack is generated for the SENT example. Thanks to the flexibility of shape and type of elements in VEM, a simple mesh attachment algorithm, and element split algorithm are introduced to generate a graded mesh and insert the crack on it. The generation of polygonal elements starts with the generation of a mesh with polygonal elements. PolyMesher (Talischi et at., 2012) which is a generator of a mesh with polygonal elements written in Matlab is employed. Since most of the stress concentration occurs at a significantly small area around the crack tip compared with the entire domain, a graded mesh is required to avoid an excessive number of nodes of the discretization and the excessive use of computational resources. As presented in Figure 4, meshes with different sizes of elements in a rectangular domain are generated. They have overlapped edges at the same position. These meshes are attached by changing the node connectivity of elements on overlapped edges of each mesh as illustrated in Figure 5. For each element of mesh 1, new nodes which are possessed by mesh 2 are inserted and the node connectivity of the given element of mesh 1 is changed. After finishing the update of node connectivity for all the elements in mesh 1, the same process is repeated in mesh 2. A graded mesh is generated by attaching a series of mesh with different sizes of elements with the algorithm.

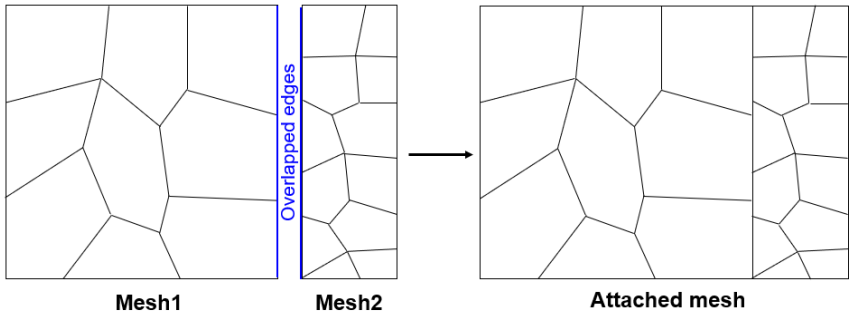

Figure 4. Meshes with polygonal elements and mesh attachment.

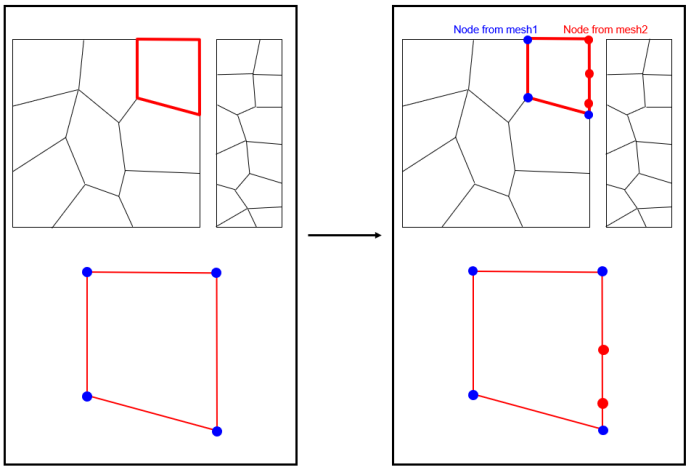

Figure 5. Change of the node connectivity of a polygonal element for the mesh attachment.

The crack is inserted into the graded polygonal mesh. A crack line is drawn on the graded polygonal mesh as shown in Figure 6. New nodes which are an intersection of an edge of mesh and crack line are inserted into the element crossed by the crack line. The new nodes are duplicated by splitting the element. A similar algorithm is used for the element that has a crack tip inside, the difference is that there is only one node inserted and the element is not split into two elements but cut by a crack line. An example of this process is shown in Figure 7 and Figure 8.

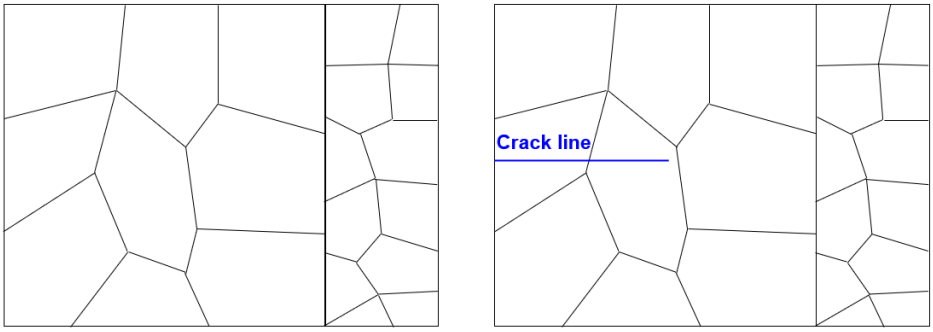

Figure 6. Example of insertion of crack line to the mesh.

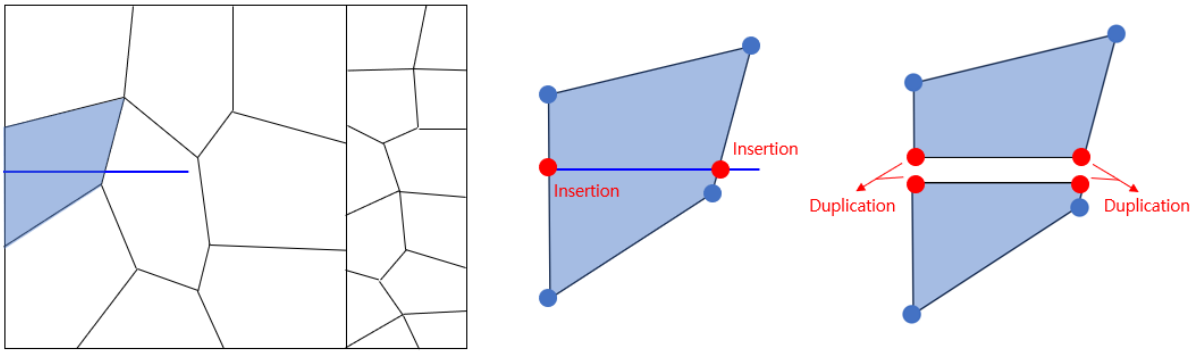

Figure 7. Element split and node duplication for the crack-crossed element.

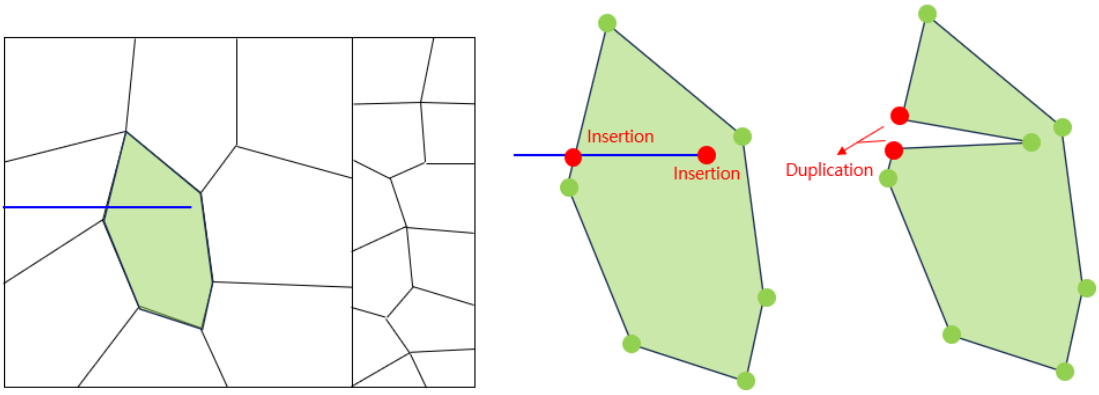

Figure 8. Element cutting and node duplication for the crack tip element.

Evaluation of nodal displacement with virtual element method

The VEM is proposed to consistently handle the flexibility in the shape of polygonal elements with implicitly defined shape function (Beirão da Veiga et al., 2013). In this study, two-dimensional first order VEM is utilized. A linear elastic solid domain (Ω) has displacement boundary conditions on the displacement boundary ($\partial\Omega_{\boldsymbol{u}}$) and the traction boundary condition on the traction boundary ($\partial\Omega_{\boldsymbol{t}}$) with traction ($\boldsymbol{t}$). The weak form of the linear elastic solid is obtained by the principle of virtual work, and it is given as:

$$\int_{\Omega} \boldsymbol{\varepsilon}(\boldsymbol{v}) : \boldsymbol{\sigma}(\boldsymbol{u}) d\boldsymbol{x} = \int_{\Omega} \boldsymbol{b} \cdot \boldsymbol{v}\, d\boldsymbol{x} + \int_{\partial\Omega_{\boldsymbol{t}}} \boldsymbol{t} \cdot \boldsymbol{v}\, d\boldsymbol{x} \quad \forall \boldsymbol{v} \in \mathcal{K}_0 \tag{3}$$

where $\boldsymbol{u}$ is the displacement field, $\boldsymbol{\varepsilon}$ is the strain field, $\boldsymbol{\sigma}$ is the stress field, $\boldsymbol{b}$ is the body force and, $\boldsymbol{v}$ is the virtual displacement field. $\mathcal{K}_0$ is the set of kinematically admissible displacements that satisfies compatibility and homogeneous displacement boundary conditions The global virtual displacement space ($\mathcal{K}_h$) is defined as:

$$\mathcal{K}_h = \{\boldsymbol{v}_h \in \mathcal{K}_0 : \boldsymbol{v}_h|_E \in [\mathcal{V}(E)]^2, \ \forall E \in \Omega_h\} \tag{4}$$

where Ω_h is a tessellation with polygonal elements of the domain, E is an element of the tessellation. $\mathcal{V}(E)$ is the space of local virtual displacement defined as:

$$\mathcal{V}(E) = \{\boldsymbol{v} \in \mathcal{H}^1(E) : \Delta \boldsymbol{v} = 0 \ in \ E, \ \boldsymbol{v}|_e \in [\mathcal{P}_1(e)]^2 \ \ \forall e \in \partial E\} \tag{5}$$

where Δ denotes Laplacian operator, e is an edge of an element, $\mathcal{H}^1$ is Sobolev space, and $\mathcal{P}_1$ is the space of first-order polynomials. A projection operator is required to handle implicitly defined shape functions and displacement fields in VEM. The operator denoted by $\Pi_E^{\nabla} : \mathcal{V} \to \mathcal{P}_1$ is defined as:

$$\int_E \nabla\left(\Pi_E^{\nabla} \boldsymbol{v}\right) \cdot \nabla \boldsymbol{p}\, d\boldsymbol{x} = \int_E \nabla \boldsymbol{v} \cdot \nabla \boldsymbol{p}\, d\boldsymbol{x} \quad \forall \boldsymbol{v} \in \mathcal{K}_0 \ \ and \ \ \forall \boldsymbol{p} \in [\mathcal{P}_1(E)]^2 \tag{6}$$

and

$$\sum_{\boldsymbol{x}_v \in E} \Pi_E^{\nabla} \boldsymbol{v}(\boldsymbol{x}_v) = \sum_{\boldsymbol{x}_v \in E} \boldsymbol{v}(\boldsymbol{x}_v) \tag{7}$$

where $\boldsymbol{x_v}$ is a position vector of a vertex in E. The right-hand side of equation 6 can be calculated exactly with the divergence theorem because $\Delta \boldsymbol{p}$ is always zero in E and $\boldsymbol{v}$ is explicitly known on ∂E. Therefore, the projection $\Pi_E^{\nabla} \boldsymbol{v}$ can be calculated exactly. Since the virtual displacement field is the sum of the projection term and the remaining term and the derivative of two terms are orthogonal, the local element stiffness matrix ($\boldsymbol{K}_E$) consists of two terms. The local (element) stiffness matrix is constructed by combining equation 3 and the projection $\Pi_E^{\nabla} \boldsymbol{v}$ and denoted as:

$$\boldsymbol{K}_E \approx \boldsymbol{K}_{E,c} + \boldsymbol{K}_{E,s} \tag{8}$$

where $\boldsymbol{K}_{E,c}$ is the consistency term and $\boldsymbol{K}_{E,s}$ is the stability term. The consistency term is expressed as:

$$\boldsymbol{K}_{E,c} = \int_E \left[\boldsymbol{b}\left(\Pi_E^{\nabla}\phi_1\right) \quad \cdots \quad \boldsymbol{b}\left(\Pi_E^{\nabla}\phi_n\right)\right]^{\mathrm{T}} \mathbf{C} \left[\boldsymbol{b}\left(\Pi_E^{\nabla}\phi_1\right) \quad \cdots \quad \boldsymbol{b}\left(\Pi_E^{\nabla}\phi_n\right)\right] d\boldsymbol{x} \tag{9}$$

where $\mathbf{C}$ is the elastic modulus matrix, ϕ_i is the i-th shape function of E, $\boldsymbol{b}$ is

$$\boldsymbol{b}\left(\Pi_E^{\nabla}\phi_i\right) = \begin{bmatrix} \frac{\partial \Pi_E^{\nabla}\phi_i}{\partial x} & 0 \\ 0 & \frac{\partial \Pi_E^{\nabla}\phi_i}{\partial y} \\ \frac{\partial \Pi_E^{\nabla}\phi_i}{\partial x} & \frac{\partial \Pi_E^{\nabla}\phi_i}{\partial y} \end{bmatrix} \tag{10}$$

In this study, the diagonal matrix-based stability term is used (Dassi & Mascotto, 2018; Chi et al., 2017). The nodal displacement for the mesh with polygonal elements is obtained from the loading term in Equation 3 and the stiffness matrix.

Evaluation of stress intensity factor and crack length

Evaluation of the SIF consists of two steps, i.e., (i) evaluation of stress using VGSR and (ii) evaluation of the SIF on the virtual grid with domain J-integral. VGSR is a post-processing stress evaluation technique for given mesh and nodal displacement (Choi et al., 2022). A virtual grid consists of rectangular elements in uniform shape and size with crack is generated and inserted to the position where the SIF is evaluated in the domain of VEM mesh. For each node of the virtual grid, a polygonal element of VEM mesh which contains a given node of the virtual grid inside is searched. After searching an element of VEM mesh, the nodal displacement for the node of the virtual grid is evaluated. The searched element of VEM mesh has nodal displacement from the VEM solution, and the nodal displacement for a given node of the virtual grid is mapped in the bilinear function (Kim et al., 2023). The stress field in the virtual grid is evaluated using the bilinear Lagrange basis for the quadrilateral element. An example process of VGSR is illustrated in Figure 9.

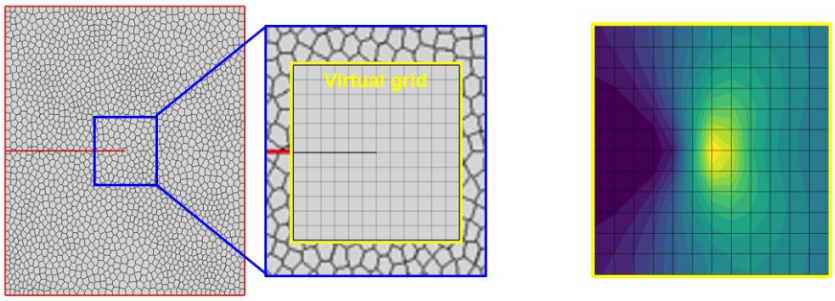

Figure 9. Virtual grid-based stress recovery (a) generation of the virtual grid, (b) evaluation of stress.

The SIF is evaluated with J-integral (Rice et al., 1968) defined as:

$$J = \int_{\Gamma} \left(W n_1 - \sigma_{ij} \frac{\partial u_i}{\partial x_1} n_j \right) d\Gamma \tag{11}$$

where W is the strain energy density, σ_{ij} is the stress component, u_i is the displacement component, and n_j is the components of unit normal vector along contour Γ. The contour integral form of J-integral is transformed into the form of domain integral by divergence theorem, expressed as:

$$J = -\int_{\Omega} \left(W \delta_{1i} - \sigma_{ij} \frac{\partial u_i}{\partial x_1} \right) \frac{\partial q}{\partial x_i} \, d\Omega \tag{12}$$

where q is a function which is zero on Γ and non-zero value inside Ω. Since there are all the required terms in the virtual grid for equation 12, the virtual grid domain can be used as an integral domain for J-integral. The SIF is evaluated using interaction integral (Yau et al., 1980) which is a method evaluating mixed-mode SIFs based on J-integral. The interaction integral (I) is defined as:

$$I = -\int_{\Omega} \left(\frac{1}{2} \left(\sigma_{ij} \varepsilon_{ij}^{aux} + \sigma_{ij}^{aux} \varepsilon_{ij} \right) \delta_{1j} - \sigma_{ij}^{aux} u_{i,1} - \sigma_{ij} u_{i,1}^{aux} \right) \frac{\partial q}{\partial x_j} \, d\Omega \tag{13}$$

where σ_{ij}^{aux} is the auxiliary stress component, ε_{ij}^{aux} is the auxiliary strain components, and u_i^{aux} is the auxiliary displacement component. These auxiliary fields are chosen with the asymptotic field around the crack tip. The SIF (K_I) for plane stress condition is evaluated as:

$$K_I = \frac{E}{2} I \tag{14}$$

The incremental length of the crack is gained from Equation 1. The maximum SIF (K_{max}) and the minimum SIF (K_{min}) are evaluated for maximum traction and minimum traction respectively. The range of SIF (ΔK) is computed as:

$$\Delta K = K_{max} - K_{min} \tag{15}$$

COMPUTATIONAL RESULTS

The graded polygonal mesh with crack for the SS400 SENT specimen at the first loading cycle is generated, as shown in Figure 10. The number of elements is 186,018 and the number of nodes is 365,938. The relation between the loading cycle and the crack length obtained from the VEM analysis is compared with the analytical solution and experimental results (see Figure 11). The numerical result shows the good agreement with the reference solutions. Therefore, the given numerical solving scheme with the VEM is an effective scheme for two-dimensional fatigue crack growth analysis of SS400 material.

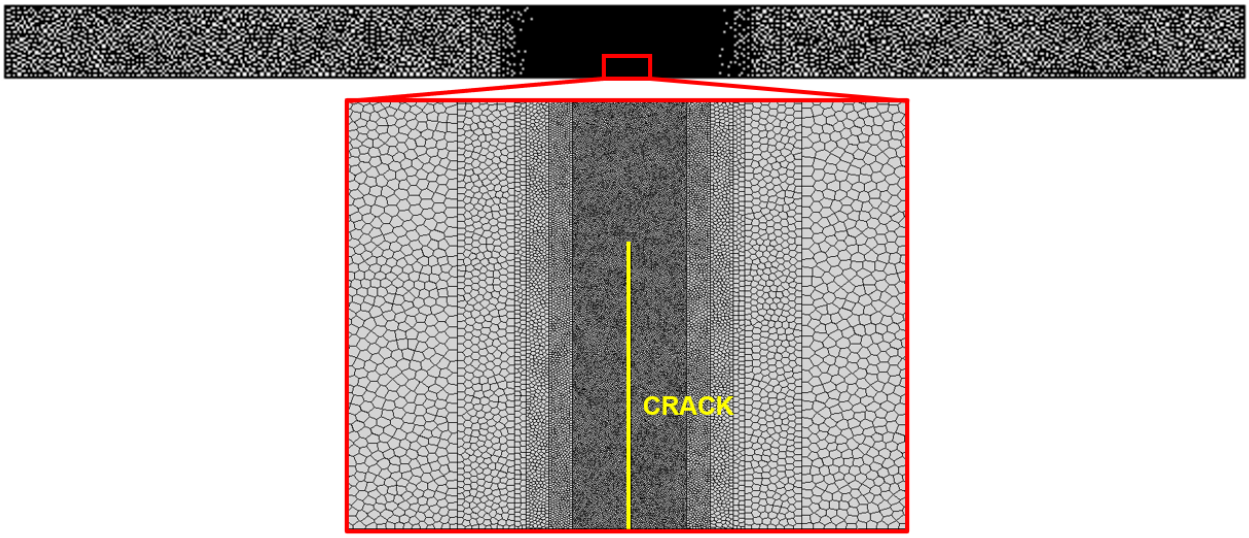

Figure 10. Graded polygonal mesh for the SENT specimen.

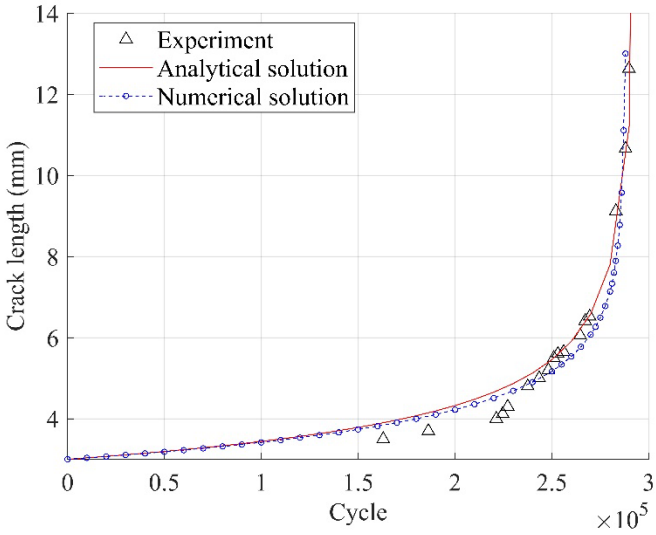

Figure 11. Relation between loading cycle and crack length.

CONCLUSION

In this study, a fatigue crack growth analysis is performed for SS400 with a SENT specimen. A graded mesh is generated with polygonal elements. A simple and intuitive algorithm is employed to generate a relatively fine element along a crack tip region, which demonstrates the flexibility of polygonal elements for the analysis of fatigue crack behavior. An integrated method with the mesh generation algorithm, the VGSR technique and the interaction integral method for the numerical analysis are utilized in this study, and the computational result agrees well with reference solutions for the fatigue crack growth.

ACKNOWLEDGMENT

This work was supported by KOREA HYDRO & NUCLEAR POWER CO., LTD (No. 2022-TECH-14).

REFERENCES

Al Laham, S., & Branch, S. I. (1998). *Stress Intensity Factor and Limit Load Handbook (Vol. 3)*., Gloucester, UK: British Energy Generation Limited.

Beirão da Veiga, L., Brezzi, F., Cangiani, A., Manzini, G., Marini, L. D., & Russo, A. (2013). "Basic principles of virtual element methods," *Mathematical Models and Methods in Applied Sciences*, *23*(01), 199-214.

Choi, H., Cui, H., & Park, K. (2022). "Evaluation of stress intensity factor for arbitrary and low-quality meshes using virtual grid-based stress recovery (VGSR)," *Engineering Fracture Mechanics*, *263*, 108172.

Choi, H., & Park, K. (2019). "Removing mesh bias in mixed-mode cohesive fracture simulation with stress recovery and domain integral," *International Journal for Numerical Methods in Engineering*, *120*(9), 1047-1070.

Dassi, F., & Mascotto, L. (2018). "Exploring high-order three dimensional virtual elements: bases and stabilizations," *Computers & Mathematics with Applications*, *75*(9), 3379-3401.

Kim, J.Y., Movahedi, N., Park, K. (2023), "Accurate evaluation of stress intensity factor in polygonal/polyhedral discretization using virtual grid-based stress recovery (VGSR) and interaction integral methods," (in preparation).

Lee, G. B., Jang, Y. Y., Huh, N. S., Park, S. H., Park, N. H., Park, J., & Park, K. (2022). "Crack growth simulation using iterative crack-Tip modeling technique," *Pressure Vessels and Piping Conference* (Vol. 86175, p. V04AT06A001). American Society of Mechanical Engineers.

Paris, P., & Erdogan, F. (1963). "A critical analysis of crack propagation laws, *Journal of basic engineering, 85(4)*, 528-533.

Park, K., Chi, H., & Paulino, G. H. (2019). "On nonconvex meshes for elastodynamics using virtual element methods with explicit time integration," *Computer Methods in Applied Mechanics and Engineering*, *356*, 669-684.

Rice, J. R. (1968). "A path independent integral and the approximate analysis of strain concentration by notches and cracks," *journal of applied mechanics, 35(2),* 379-386.

Talischi, C., Paulino, G. H., Pereira, A., & Menezes, I. F. (2012). "PolyMesher: a general-purpose mesh generator for polygonal elements written in Matlab," *Structural and Multidisciplinary Optimization*, *45*, 309-328.

Yau, J. F., Wang, S. S., & Corten, H. (1980). "A mixed-mode crack analysis of isotropic solids using conservation laws of elasticity," *Journal of applied mechanics, 47(2)*, 335-341.