International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 14, Issue 2, February 2025)

Analysis of Surface Cracks Using Knowledge Processing Techniques

Jin-A Choi[1], Joon-Seong Lee[2*]

[1]*Department of Mechanical Engineering, Graduate School, Kyonggi University, 154-42 Gwangkyosan-ro, Yeongtong-gu, Suwan, 16227, Korea*
[2*]*Professor, Department of Mechanical System Engineering, Kyonggi University*

[1]superwoman26@naver.com
[2*]jslee1@kyonggi.ac.kr

***Abstract*— In this paper, we developed a system that can perform automated analysis using an element decomposition system, which is a preprocessing step for 3D crack analysis, and easily perform J-integral calculations. In particular, we made it possible for users to select and apply a finite element decomposition system using a knowledge processing technique in various ways. That is, users can select Delaunay Triangular technique, bubble meshing technique, etc. After a geometry model, i.e., a solid containing one or more 3D cracks, is defined, and when the node density by region is selected, the elements are automatically generated and linked to the analysis code. Stress analysis is performed using the constructed finite element model. In this system, a 3D crack is introduced, and fracture mechanics parameters can be selected depending on the analyst. This paper explains the methodology for realizing this function, and also demonstrates the utility of the system.**

***Keywords*—Automatic Mesh Generation, Delaunay Triangulation, J-Integral, Knowledge Processing, Stress Intensity Factor, Surface Crack,**

I. INTRODUCTION

Natural cracks commonly found in structures usually appear as cracks with three-dimensional characteristics called surface cracks. In order to predict the growth rate and fracture strength of surface cracks, an accurate analysis of the stress intensity factor K of surface cracks is required. When applying the finite element method to fracture mechanics analysis, it has not reached a level that can be applied to everyday design due to the difficulty of numerical analysis and the complexity of experimental variables, considering the importance of practical problems. The mesh generation technology of the past, which had to be dependent on manual work, has been recently developed by several researchers[1-3]. However, it has been reported that various technologies related to the existing mesh generation method have many limitations when applied to actual problems, and especially, they do not provide adequate support when applying the three-dimensional crack problem of structures[4].

In 3D crack analysis, large-scale analysis is easy and requires enormous computing power. In addition, since special element division is required near the crack tip, which is a stress singularity, a lot of time and effort are required for element division and input data creation for 3D analysis. Therefore, an algorithm that automatically and quickly creates a large-scale finite element model with minimal manipulation is required. The author has developed an automatic element generation system using fuzzy theory from the information after inputting the shape definition, load, and boundary conditions of the structure using a 3D solid modeler with the main goal of improving the efficiency of large-scale analysis[5,6]. In addition, an automatic K-analysis system for 3D cracks in structures was developed by developing a system that automatically creates and analyzes input files of a general-purpose finite element analysis code[7].

However, the conventional system had a large limitation on the shape of the analysis model because it targeted a symmetrically existing crack in the structure. In addition, it was a system built only targeting the K value as a fracture mechanics parameter. Therefore, in this study, even in the case where a three-dimensional crack exists at an arbitrary location other than the plane of symmetry, a finite element model was automatically generated with minimal manipulation, and then the calculation of the J-integral, a nonlinear fracture mechanics parameter of the crack, was made possible.

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 14, Issue 2, February 2025)

II. OUTLINE OF THE SYSTEM

Even in complex-shaped real structural devices, it is possible to completely automatically generate large-scale finite element models with tens or hundreds of thousands of nodes with controlled element sizes by performing simple and easy manipulations on geometric models. The processing of this system consists of (a) definition of shape (geometric model), (b) definition and registration of crack shape, (c) specification of boundary conditions and material properties for geometric models, (d) specification of node density information, (e) node generation, (f) element generation and smoothing, (g) addition of boundary conditions and material properties to elements, (h) FE analysis, and (i) calculation of fracture mechanics parameters (J-integral calculation). Among these, (a) to (d) are interactive tasks that the user directly performs on the geometric model, and the rest are all tasks that are processed automatically.

A. Definition of geometry model

The geometric model was created using the geometric model definition function of Designbase[8], a general-purpose CAD system that can express free-form surfaces such as rational Bezier surfaces and rational Gregory patches, and has abundant library functions for shape processing. In subsequent processing, necessary information about shape data can be obtained by using the library functions.

B. Definition of 3D crack and registration

In order to define a 3D crack shape at an arbitrary location, the user needs to input the following information.
(1) Center coordinates of the ellipsoid representing the 3D crack (x, y, z)
(2) Radius of the ellipsoid (rx, ry, rz)
(3) Parameters required when creating an ellipse from an ellipsoid
Enter the number of cracks you want to define, such as the above information.

C. Enter boundary conditions and material properties for the geometric model

Specify the vertices, edges, and loops that make up the geometric model by clicking them with the mouse. Also, enter the type and value of the boundary conditions added to the components of the specified geometric model.

D. Designation of node density information

The node density is prepared in advance as multiple node density distribution functions within the system, such as a) a distribution corresponding to local stress concentration, b) a distribution that divides a finite area evenly, and c) a distribution that divides the entire area evenly. As a result, when the user selects these according to the analysis target and designates their location, the node density distribution over the entire area of the geometric model is automatically calculated by the Fuzzy theory[9].

E. Node and element generation

The After obtaining the node density distribution of the entire interpretation domain above, the bucket method[10], which is one of the computational geometric methods, is used to automatically generate nodes. Since this process generates nodes on the surface and inside of the geometric model, it uses the shape calculation library of Designbase. The generated nodes are quickly generated as tetrahedral elements using the well-known Delaunay triangulation method[11]. When using this Delaunay method, elements are generated outside the shape of the concave geometric model, so the elements generated outside are removed by checking the inside/outside judgment with the center of the element. In addition, since distorted elements are sometimes generated near the joints and boundaries of the node pattern, the Laplacian smoothing[12] method is introduced to modify the element shape.

F. Adding boundary conditions and material properties to elements and element generation

In this system, the data structure preserves the original shape elements (vertices, edges, and surfaces) to which the generated elements and nodes each belong as information. Therefore, the boundary conditions and material properties directly assigned to the shape model by the user in the work of section 2.3 are automatically assigned to the vertices (nodes), edges, surfaces, and interiors of the elements immediately after the elements are generated. As a result, the finite element model (element + material properties + boundary conditions), which is the input file of the finite element analysis code, is automatically generated.

This automatic analysis system can be applied to any FE analysis system in principle, but the current version is designed to output data corresponding to the tetrahedral 1st and 2nd elements used in the general analysis code ANSYS[13].

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 14, Issue 2, February 2025)

III. INTRODUCTION OF 3D CRACK

Assuming that the three-dimensional crack surface is a semi-elliptical shape parallel to the x-y plane, this system uses the Delaunay method as an element generation method. However, since the Delaunay method randomly divides the elements by the node distribution from which they are generated, it is necessary to adjust the node distribution in advance so as not to connect the nodes located above and below the crack surface. Therefore, we propose a crack introduction method that carves out an ellipsoid based on the similarity of the crack topology to an ellipsoid. Specifically, when defining the shape of a structure, the upper and lower surfaces of the crack are not connected by cutting out an ellipsoid at the location where you want to insert a crack. After the elements are generated, the crack part is expressed by crushing the ellipsoid whose node coordinates are moved in the z direction, as shown in Fig. 1. Here, rx is the semi-major axis of the crack, ry is the depth of the crack, and rz is the radius of the ellipsoid in the z direction. Also, when creating an ellipse from the ellipsoid entered in section 2.2, h is a parameter that indicates the range of movement of the nodes. When defining a shape using Designbase, the user can express the desired crack by only entering the location of the crack center and the rx, ry, rz, and h above.

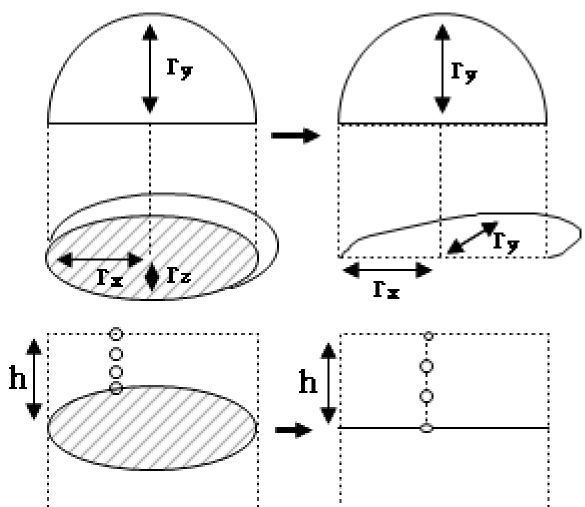

Figure 1 Mapping of an ellipsoid

After entering the parameters, the crack is automatically generated, but the following three processes are performed in this automatic analysis system. First, there are cases where points on the surface of the ellipsoid are combined into inappropriate elements as shown in Fig. 2. As explained in Section 2.5, this is because the element center is judged inside and outside, and the elements on the outside are eliminated, but Fig. 2 shows a state where the element center is on the inside of the model, so it remains as it is.

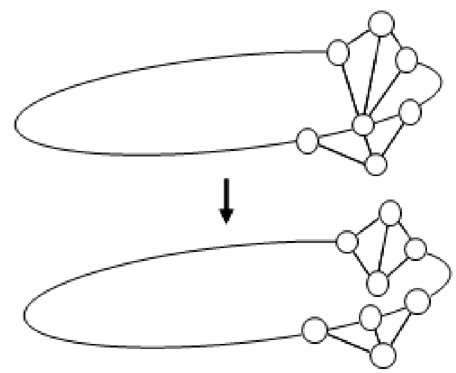

Figure 2 Example of mis-match elements

In this case, a node addition algorithm[9] is introduced to eliminate this combination before the ellipsoid is crushed. Second, there are cases where the element is deformed along with the node movement, and the volume becomes close to 0 or negative. In this case, after converting the ellipsoid to an ellipse, one node is added above and below to restore the volume so that it does not deviate from the crack shape. Finally, Barsoum's singular element[14] is placed on the crack tip. That is, the location of the middle node is moved 1/4 toward the crack tip as shown in Fig. 3.

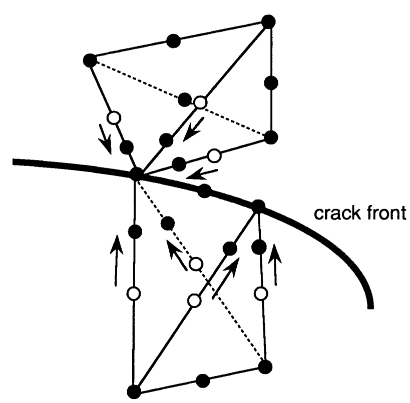

Figure 3 Conversion of quadratic tetrahedral elements along crack front into singular elements

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 14, Issue 2, February 2025)

By introducing the above treatment, it is now possible to analyze 3D cracks symmetrically or not on the plane of symmetry. This method can be applied to the cracks on the plane of symmetry that were analyzed in the previous system, and it is also possible to analyze the inherent cracks. Fig. 4 shows an example of element generation for surface crack in a cylindrical pressure tube. Fig. 5 shows an example of element division for a 1/4 model of a semi-elliptical surface crack that exists symmetrically on a flat plate. In addition, Fig. 6 shows an example of element division for a flat plate with three surface cracks.

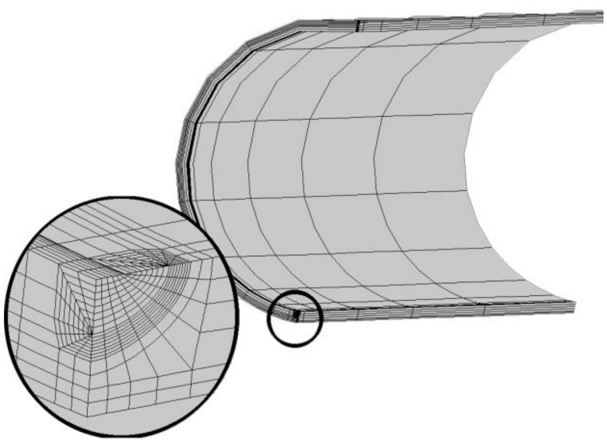

Figure 4 A typical mesh with a semi-elliptical surface crack in pressure tube

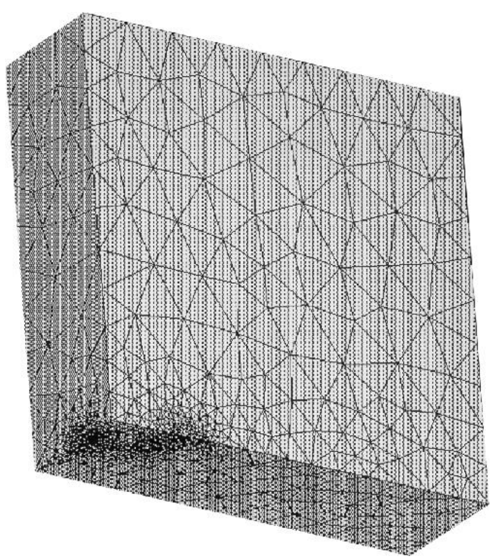

Figure 5 A typical mesh with a semi-elliptical surface crack

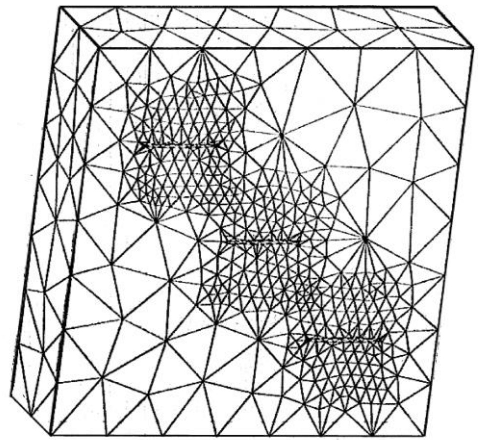

Figure 6 A typical mesh of plate with semi-elliptical cracks

IV. EVALUATION OF J-INTEGRAL AND VERIFICATION OF THE SYSTEM

This system can use the virtual crack propagation method and the Lorenzi method, which is an extended J-integral calculation method that can be analyzed in ANSYS as a calculation method for the J-integral. The user selects the J-integral calculation method and angle Φ for multiple cracks. Φ is the angle defined in Fig. 7, and when calculating the J-integral, the crack is virtually propagated in the normal direction to the point p4 line. Therefore, it can be seen that the J-integral is a function of Φ. In addition, the user inputs the parameters r, h1, h2, and h3 as shown in the figure. Here, r is the ratio of the crack depth to the distance from p1 to the path. The effect due to the stiffness change can be achieved by moving the nodes in a certain area. Therefore, the system calculates the change in strain energy that occurs by moving the nodes in the rectangular solid specified in this way by a small distance, and calculates the J-integral value by dividing it by the crack propagation area expressed by the moving distance. The input work of these parameters is performed when defining the shape using Designbase. However, care must be taken because the number of nodes moving in the crack section changes depending on the h3 value. This is because the amount of change in strain energy and the area of crack propagation change greatly. It is expected that it will be better if two or more nodes are moved than if one node is moved, but if too many nodes are moved, there are cases where the J integral of the specified position cannot be obtained correctly because of the angle dependence.

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 14, Issue 2, February 2025)

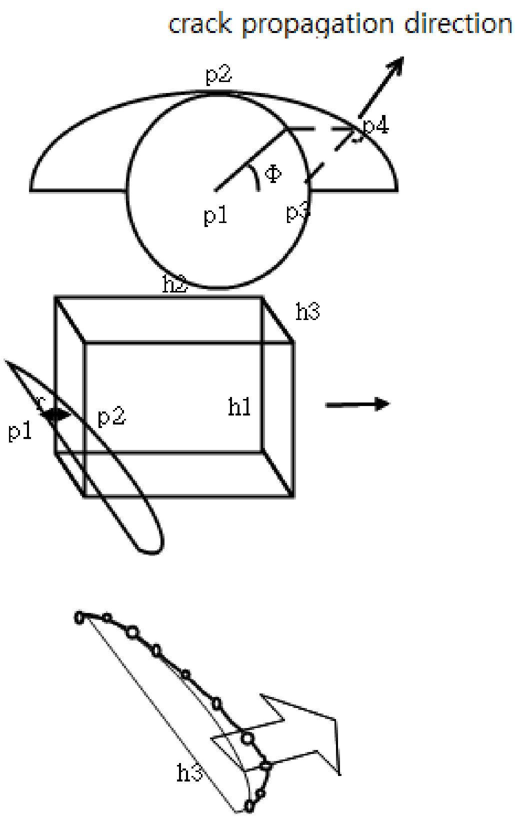

Figure 7 Evaluation of J-integral value

To verify the effectiveness of the system, an analysis was performed on a case where a semi-elliptical surface crack with a shape ratio of a/c=0.6 and a/t=0.2 of the crack depth (a) and the surface semi-length (c) on a tensile plate exists. The finite element model used here is as shown in Fig. 5, with 3,120 elements and 5,980 nodes. Since the J integral can be related to K in elastic analysis, it was compared with the non-dimensional value obtained by Raju-Newman's equation[15]. The calculation was performed for the movement of one node (narrow pass) and the movement of two or more nodes (broad pass), changing the node movement respectively. Fig. 8 shows the analysis results, and it can be seen that the values of the narrow pass and broad pass are similar.

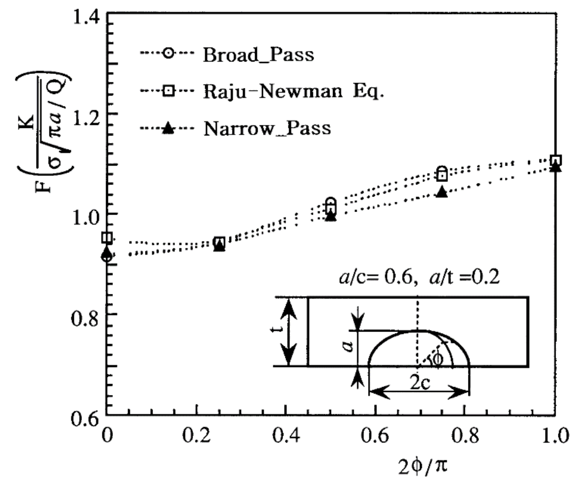

Figure 8 Comparison of present stress intensity factor with Raju-Newman equation

V. CONCLUSIONS

In this paper, we developed a system that allows the analyst to easily obtain the J integral value for a three-dimensional crack with minimal interactive manipulation. In particular, by adding the introduction function and analysis function of a three-dimensional crack existing at an arbitrary location, we developed an automatic system that performs a series of inputs to outputs for nonlinear analysis. In addition, to verify the effectiveness of this system, we performed an analysis on a flat plate with a semi-elliptical surface crack and compared the results with those of the Raju-Newman equation, and found good agreement.

ACKNOWLEDGEMENT

This work was supported by Kyonggi University's Graduate Research Assistantship 2025.

REFERENCES

[1] Sezer, L. and Zeid, I. 1991 Automatic Quadrilateral/Triangular Free-Form Mesh Generation For Planar Region, International Journal for Numerical Methods in Engineering, 32, 1441-1438.

[2] Pourazady, M. and Radhakrishnan, M. 1991 Optimization of a Triangular Mesh, Computers and Structures, 40(3), 795-804.

[3] Talbert, J. A. and Parkinson, A. R 1990 Development of an Automatic, Two-Dimensional Finite Element Mesh Generator Using Quadrilateral Elements and Bezier Curve Boundary Definition, International Journal of Numerical Methods in Engineering, 29, 1551-1567.

[4] Sugioka, K., Lee, J. S., Yoshimura, S. and Yagawa, G. 1994 Probabilistic Fracture Mechanics Analysis of Multiple Surface Cracks Considering their Interaction Effects, Proceedings of the '94 Annual Meeting of JSME/MMD, No. 940-37, 257-258.

[5] J.S. Lee, Y. Genki, M.W. Park 1996 Automatic element segmentation for 3D structures composed of free-form surfaces, Int. J. of CAD/CAM, 1(1), 65-75.

[6] Yoshimura, S., Lee, J. S. and Yagawa, G. 1993 FEM Modeler for 3D Solid Geometry with Free-Form Surface: Automatic Attachment of Boundary Conditions to 3D Mesh, The 6th Computational Mechanics Conference, No. 930-71, 505-506.

[7] Lee, J. S., Yoshimura, S. and Yagawa, G.1995 Automation of Analysis for Stress Intensity Factors of Three-Dimensional Cracks, Transact- ions of the Japan Society of Mechanical Engineers A, 61-587(94-2051), 1580-1586.

[8] Chiyokura, H. 1988 Solid Modeling with Designbase : Theory and Implementation,, Addition –Wesley.

[9] Lee, J. S., 1995 Automated CAE System for Three-Dimensional Complex Geometry, The Doctoral Thesis, The University of Tokyo

[10] Asano, T. 1985 Practical Use of Bucketing Techniques in Computational Geometry, Computational Geometry, North-Holland, 153-195.

[11] Sloan, S. W. 1987 A Fast Algorithm for Constructing Delaunay Triangulation in the Plane, Advances in Engineering Software, 9, 34-55.

[12] Cavendish, J. C., 1984 Automatic Triangulation of Arbitrary Planar Domains for the Finite Element Method, International Journal of Numerical Methods in Engineering, 8, 679-696.

[13] ANSYS User's Guide2021, ANSYS Inc.

[14] Barsoum, R. S., 1984 Application of Quadratic Isoparametric Finite Elements in Linear Fracture Mechanics, International Journal of Fracture, 10, 603-605.

[15] Newman, J. C. and Raju, I. S. 1981 An Empirical Stress-Intensity Factor Equation for the Surface Crack, Engineering Fracture Mechanics, 15, 185-192.