Development of an Automated System for Three-Dimensional J-Integral Analysis

Joon-Seong Lee
Dept. of Mechanical System Engineering, Kyonggi University, 154-42 Gwanggyosan-ro, Yeongtong-gu, Suwon, Gyeonggi-do, 16227 Korea
jslee1@kyonggi.ac.kr

Abstract. By integrating the 3D solid modeler with the universal FEM code, an automatic nonlinear analysis system for 3D crack problems has been developed. A geometry model, that is, a solid containing one or several 3D cracks is defined. Multiple distributions of local node density are selected and automatically overlapped with each other over the geometry model using fuzzy knowledge processing. Nodes are created by the bucketing method, and tetrahedral square tetrahedral solid elements are generated by the Delaunay triangulation technique. A complete finite element (FE) model is created and stress analysis is performed. In this system, analysts can significantly reduce the burden of introducing 3D cracks into FE models and estimating fracture mechanics parameters. This paper describes a methodology for realizing such a function and demonstrates the validity of this system.

1. Introduction
Natural cracks often found in structures usually appear as cracks with three-dimensional properties called surface cracks. In order to predict the growth rate and fracture strength of surface cracks, it is necessary to analyze the stress intensity factor K of surface cracks accurately. When the finite element method is applied to the analysis of fracture mechanics, the difficulty of the numerical analysis and the complexity of the experimental parameters have not reached the level that can be applied to everyday design in view of the importance of practical problems. The previous generation of mesh generation techniques, which had to be manually dependent, have been developed by several researchers in recent years [1-3]. However, many of the existing techniques for element generation have been reported to be limited when applied to real problems, and are not adequately supported when applying the 3-D cracking problem [4]. Three-dimensional crack analysis is likely to be a large-scale analysis and requires a tremendous amount of calculator power. In addition, special element partitioning is required near the crack tip, which is a stress-specific element, and therefore it takes a great deal of time and effort to divide the element for three-dimensional analysis and to prepare input data. Therefore, there is a need for an algorithm that automatically and rapidly generates a large-scale finite element model with minimal manipulation. The authors have developed the automatic generation system of elements using fuzzy theory from the information of shape, load, and boundary conditions of structures by using 3D solid modeler with the aim of efficiency of large scale analysis so far [5,6].

In addition, a system for automatically creating and analyzing input files of general-purpose finite element analysis codes has been developed and an automatic K analysis system for three-dimensional cracks of structures has been developed [7]. However, the conventional system has a limitation on the shape of the analytical model with respect to the symmetrically existing cracks in the structure. Also, it
was a system constructed only for K value as the fracture mechanics parameter. Therefore, in this study, the finite element model is automatically generated with minimal manipulation even when there is a three-dimensional crack at an arbitrary position other than the symmetric plane, and then the J integral, which is a nonlinear fracture mechanics parameter of the crack, can be calculated.

2. Outline of system
It is possible to fully and automatically generate a large number of tens or hundreds of thousands of finite element models with controlled element sizes by performing simple and easy manipulation of geometric models in complex geometrical actual structural devices. The process of this system is to define the shape (geometry model), (b) to define and register the shape of the crack, (c) to specify the boundary conditions and material properties for the geometry model, (d) designation of node density, (e) generation of node, (f) generation of element and smoothing, (g) boundary conditions and material property values to elements, (h) FE analysis, (i) calculation of fracture mechanics parameters (J integral calculations). Among them, (a) to (d) are interactive tasks that the user directly performs on the geometric model, and all others are tasks that are automatically processed.

2.1. Geometric modeling
The geometric model is based on the geometric model definition function of Designbase [8], which is a universal CAD system capable of expressing free curved surfaces such as glass Bezier curved surface and glass Gregory patch, and has abundant library functions for shape processing. In the subsequent processing, necessary information on the shape data can be obtained by using the library function.

2.2. Define the shape of 3D crack
In order to define the shape of a three-dimensional crack at an arbitrary position, the user needs to input the following information.

(1) The center coordinates (x, y, z) of the ellipsoid representing the 3D crack
(2) Radius of the ellipsoid (rx, ry, rz)
(3) Parameters required to create an ellipse from an ellipsoid

Enter the above information for the number of cracks you want to define. A more detailed explanation is given in section 3.

2.3. Enter B.C and material properties into geometric model
Specify vertices, ridges, and loops that make up the geometric model by clicking with the mouse. Also input the type and numerical value of the boundary condition to be added to the component of the specified geometry model.

2.4. Designation of node density
The node density is a function of a) distribution corresponding to local stress concentration, b) distribution that uniformly divides within the finite domain, and c) distribution that uniformly divides the entire area. As a result, the user chooses them according to the object to be analyzed, and if the position is specified, the fuzzy theory automatically calculates the distribution of the nodal density over the entire region of the geometric model [9].

2.5. Generation of node and element
After obtaining the distribution of the density over the entire analysis area, the nodal point is automatically generated using the bucket method [10], which is one of the computational geometric methods. This process uses nodes in the geometry library of the Designbase because it also creates joints on the surface and inside of the geometry model. The generated nodes are well-known to produce a tetrahedron element at high speed using Delaunay's triangulation method [11,12]. When using the Delaunay method, the concave geometry model will generate an element outside the shape
as well, so check the inner / outer determination as the center of the element to remove the externally generated element. In addition, a distorted element may sometimes be generated in the vicinity of the junction between the node pattern and the boundary, so that the element shape is modified by introducing the Laplacian smoothing technique [13].

2.6. Boundary condition and property value to element
In this system, the original shape elements (vertices, ridges, and planes) to which the created elements and the nodes belong to a data structure that preserves them as information. Therefore, boundary conditions and material properties directly assigned to the shape model by the user in the work of 2.3 are automatically assigned to the vertices (nodes), sides, faces and inside of the element automatically after the element is created. As a result, the finite element model (element + material property + boundary condition), 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 able to output data corresponding to the tetrahedron primary and secondary elements used in the general purpose analysis code MARC [14].

3. Introduction of 3D cracks
Assuming that the three-dimensional crack plane is a semi-elliptical shape parallel to the x-y plane, the present system uses the Delaunay method as an element generation method. However, since the Delaunay method is arbitrarily divided by the distribution of the nodes where the elements are generated, it is necessary to adjust the nodal distributions in advance so as not to connect the nodes located above and below the crack plane. Therefore, we propose a cracking method that cuts off the ellipsoid by focusing on the similarity of the crack topology as the ellipsoid. Specifically, when defining the shape of the structure, the ellipsoid is cut at the position where the crack is to be inserted so that the upper and lower surfaces of the crack are not connected. After the element is generated as shown in Fig. 1, the ellipsoid shifted in the z-direction of the nodal point is collapsed to realize the expression of the cracked portion. Where rx is the major radius of the crack, ry is the depth of the crack, and rz is the radius of the ellipsoid in the z direction. It is also necessary to generate an ellipse from the ellipsoid input in Section 2.2 and h is a parameter indicating a range for moving the node.

Figure 1. Mapping of an ellipsoid

Figure 2. Example of mis-match elements

When defining the shape using the design base, the user can express the desired crack by inputting only the location of the crack center and the rx, ry, rz, and h above.
After inputting the parameters, the cracks are automatically generated, and the following three processes are performed in the automatic analysis system. First, the point on the surface of the ellipsoid is shown in Fig. 2. In some cases, it may be combined with an inappropriate element. This is because, as described in Section 2.5, the elements inside and outside are determined by the inner and outer determinations of the element center, but figure 2 shows the state where the element center remains inside the model. In this case, we introduced a node addition algorithm [9] that eliminates this coupling before collapsing the ellipsoid. Secondly, along with the node movement, the element may be deformed so that the volume is close to zero or minus. In this case, after the ellipsoid is converted into an ellipse, the volume is restored by adding one node vertically so as not to deviate from the crack shape. Finally, Barsoum's singular element [15] is placed at the tip of the crack. That is move the position of the intermediate node to the crack tip side by 1/4 as shown in Fig. 3.

By introducing the process as described above, it is possible to analyze even when the three-dimensional crack is not symmetrically or on a symmetric plane. This technique is applicable to the cracks in the symmetric plane that were analyzed in the previous system, and the inherent cracks can also be analyzed. Fig. 4 shows an example of element partitioning for a 1/4 model of semi-elliptical surface cracks symmetrically to the plate. Also, Fig. 5 shows an example of element partitioning of a flat plate with three surface cracks.
4. Evaluation of J-integral analysis

In this system, the virtual crack propagation method which can be interpreted by MARC and the Lorenzi method which is the calculation method of extension J integral can be used as the J integral calculation method. The user selects the calculation method of the J integral for a plurality of cracks, angle $\Phi$ is shown in Fig. 6, and the J-integral computes the crack virtually in the normal direction of the point p4.

Therefore, we can see that J integral is a function of $\Phi$. The user also inputs parameters $r$, $h_1$, $h_2$ and $h_3$ as shown in the figure. Where $r$ is the ratio of the crack depth to the distance from $p_1$ to the path. The effect of stiffness change can be achieved by moving the node of a certain region. Therefore, in the system, the value of the J integral is calculated by obtaining the change of the strain energy generated by micro-distance movement of the node in the specified rectangular parallelepiped and dividing by the advancement area of the crack represented by the travel distance. The input of these parameters is performed when defining the shape using the Designbase.

However, care must be taken because the number of movements of the nodal point of the crack varies depending on the $h_3$ value. This is because the amount of change of the strain energy and the advancement area of the crack vary greatly. It is better to move two or more joints than one joint movement. However, if too many joints are moved, the J integral at the specified position may not be obtained correctly due to the angle dependency.

In order to verify the effectiveness of the system, it is necessary to analyze the case where semi-elliptical surface cracks with the aspect ratios $a/c = 0.6$ and $a/t = 0.2$ exist on the plate under tension load (a) respectively. The finite element model used here is shown in Fig. 4, the number of elements used is 2,980, and the number of nodes is 5,840. The J integral is related to $K$ in the elastic analysis, so it is compared with the non-dimensionalized value obtained by Raju-Newman’s equation [16]. The calculations were made by varying the nodal shift by narrow pass for 1 node movement and broad pass over 2 nodes. Fig. 7 shows the results of the analysis. It is found that the values of narrow pass and broad pass are similar.

![Figure 7. Comparison of present stress intensity factor with Raju-Newman equation](image)

5. Conclusions

In this paper, we constructed a system that enables the analyst to easily find the J integral value for the three-dimensional cracks with minimal interactive manipulation. In particular, we have developed an automatic system that performs a series of input to output operations for nonlinear analysis by adding the introduction function and the analysis function of a three-dimensional crack existing in an arbitrary position. In order to verify the effectiveness of this system, the analysis of the plate with semi-elliptical surface cracks was performed and compared with the results of Raju-Newman equation.
6. Acknowledgments
This work was supported by the GRRC program of Gyeonggi province. [2018-0261, Research on Innovative Intelligent Manufacturing system]

7. 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, Vol. 32, pp 1441-1438
[2] Pourazady, M. and Radhakrishnan, M. 1991 Optimization of a Triangular Mesh, Computers and Structures, Vol. 40, No. 3, pp. 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, Vol. 29, pp. 1551-1567
[4] Sugiooka, K., Lee, J. S., Yoshimura, S. and Yagawa, G., 2004 A Probabilistic Fracture Mechanics Analysis of Multiple Surface Cracks Considering their Interaction Effects, Proceedings of the '94 Annual Meeting of JSME/MMD, No. 940-37, p 257
[5] J.S. Lee, Y. Genki and M.W. Park 2006 Automatic Mesh Generation of Three-dimensional Structures, Transactions of CAD/CAM Engineers, Vol. 1, pp. 65-75
[6] Yoshimura, S., Lee, J. S. and Yagawa, G. 2003 FEM Modeler for 3D Solid Geometry with Free-Form Surface, The 6th Computational Mechanics Conference, No. 930-71, pp. 505
[7] Lee, J. S., Yoshimura, S. and Yagawa, G. 2005 Automation of Analysis for Stress Intensity Factors of Three-Dimensional Cracks, Transactions of the Japan Society of Mechanical Engineers A, Vol. 61-587, pp. 1580-1586
[8] Chiyokura, H.2008 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. 1995 Practical Use of Bucketing Techniques in Computational Geometry, Computational Geometry, North-Holland, pp. 153-195
[11] Watson, D. F. 2008 Computing the n-Dimensional Delaunay Tessellation with Application to Voronoi Polytopes, The Computer Journal, Vol. 24, pp. 162-172
[12] Sloan, S. W., 1997 A Fast Algorithm for Constructing Delaunay Triangulation in the Plane, Advances in Engineering Software, Vol. 9, pp. 34-55
[13] Cavendish, J. C. 1994 Automatic Triangulation of Arbitrary Planar Domains for the Finite Element Method, International Journal of Numerical Methods in Engineering, Vol. 8, pp. 679-696
[14] MARC Manual k7.2, MARC Analysis Research Corporation, Vol. A-E.
[15] Barsoum, R. S. 1984 Application of Quadratic Isoparametric Finite Elements in Linear Fracture Mechanics, International Journal of Fracture, Vol. 10., pp. 603-605
[16] Newman, J. C. and Raju, I. S. 1982 An Empirical Stress-Intensity Factor Equation for the Surface Crack, Engineering Fracture Mechanics, Vol. 15, pp. 185-192