BOUNDARY ELEMENT MESH GENERATION FOR MODELING CRACK PROBLEMS

G. Creci[a], **E. L. Albuquerque**[b], **P. Sollero**[c], **J. V. Ferreira**[d] and **R. Pavanello**[e]

Department of Mechanical Computing
State University of Campinas
PO 6122, Campinas, SP, Brazil
[a]gcreci@fem.unicamp.br, [b]edelim@fem.unicamp.br, [c]sollero@fem.unicamp.br,
[d]janito@fem.unicamp.br, [e]pava@fem.unicamp.br

This work presents the computational aspects related to the generation and refinement of unstructured meshes that will be used for the analysis of crack problems using the boundary element method. Fracture mechanics problems are particularly interesting because they have steep solution gradients in elastic bodies, since stresses tend to infinity at the crack tip. High stresses and strains increase the difficulty of obtaining accurate approximated solutions for the physical behavior of the crack using the finite element method, unless extremely refined meshes are used. Another important point is that continuous re-meshing requirement for crack growth simulation causes the finite element method to be less competitive when compared to the boundary element method. Mesh refinement in the region of the crack is also particularly important when using the boundary element method because, the quality of the mesh, and an appropriate refinement in the region of the crack, are responsible for obtaining well-conditioned matrices and, consequently, accurate solutions. In this context, a program with a friendly user interface, which allows an easy automatic mesh generation and element refinement in the region of the crack, is considered a very important tool. In this work the implementation is performed using the object-oriented approach and the Delaunay method. The triangular mesh elements are generated from a set of points, which is obtained from the boundary geometric description of the problem. In problems involving the presence of crack, a special attention is taken for the generation of this set of points in order to allow a larger generation of points near the crack and a more refined mesh in this region. Examples are presented to demonstrate the capabilities of the mesh generation program in modeling fracture mechanics problems evaluated by the boundary element method.

References

[1] Jin-Fa Lee, R. Dyczij-Edlinger, “Automatic Mesh generation using a modified Delaunay tessellation”, *IEEE Antennas and Propagation Magazine*, v. 39, n. 1, p. 34-45, Feb. 1997.

[2] H. Tsuboi, T. Asahara, F. Kobayashi, T. Misaki, “Adaptive triangular mesh generation for boundary element method in three-dimensional electrostatic problems”, *IEEE Transactions on Magnetics*, v. 34, n. 5, p. 3379-3382, September 1998.

[3] A. Nicolas, L. Nicolas, “3D Automatic mesh generation for boundary integral method”, *IEEE Transactions on Magnetics*, v. 26, n. 2, p. 767-770, March 1990.

[4] Y. Mi, “Three-Dimensional analysis of crack growth”, *Computational Mechanics Publications – Topics in Engineering*, vol. 28, 184p, 1996.