

BOUNDARY ELEMENT METHOD WITH AUTOMATIC PROGRESSIVE CELL GENERATION FOR ELASTOPLASTIC ANALYSIS

Tatiana S. A. Ribeiro, Christian Duenser and Gernot Beer

Institute for Structural Analysis (IFB)
Graz University of Technology (TUG)
Lessingstrasse 25/ II, A-8010 Graz, Austria
e-mail: ribeiro@ifb.tu-graz.ac.at, web page: <http://www.ifb.tugraz.at>

Key words: Boundary Element Method, Automatic Cell Generation, Plasticity.

Summary. *An adaptive automatic cell generation for elasto-plastic problems has been developed which eliminates the need of user definition of the cells. These are progressively generated only on those points where plasticity occurs. Unnecessary domain computations are avoided, which leads to a gain in efficiency. The new method has been tested on examples and the accuracy of the results is in agreement with the solution from a finite element calculation and from a calculation with the boundary element method using predefined cells.*

1 INTRODUCTION

The boundary element method (BEM) is a well known numerical method and has been shown to be a more efficient alternative to domain methods such as the finite element method (FEM) in the analysis of many physical problems. An appealing feature of the method is the reduction of the dimension of the problem in one order. In the case of performing a BEM analysis of infinite domains, the reduction of input data as well as the effort for storing the result data in comparison to the FEM, for instance, is considerable.

However, when dealing with plasticity, not only boundary integrals but also domain integrals have to be computed. There are some approaches, such as the multiple reciprocity BEM¹, which transforms the domain integral into an equivalent boundary integral and preserves the advantage of the boundary-only discretization. However, this approach introduces additional approximations and has not yet been applied to infinite domains. The commonly used approach to compute the domain integrals, valid also for infinite domains, is to adopt internal cells^{2,3}. These cells are like volume elements located in the plastic zone, and do not introduce any additional degree of freedom to the problem. The drawback here is the requirement of a domain discretization. Even though the discretization can be optimized, in a sense that the cells are located only where plasticity will occur, the amount of input data is still larger than for a boundary-only discretization and the computational cost can be higher than necessary depending on the definition of the cells. Also in many cases it may be difficult or impossible to predict zones where plasticity is likely to occur.

In the current work we intend to avoid the need to input the internal cells as well as

unnecessary domain integral evaluations by developing a procedure, valid for both finite and infinite problems, to automatically generate the domain discretization of the plastic zones during the analysis. The user does not need to know a priori where the plastic zone will occur as the discretization will be progressively generated. There is a gain in efficiency, since unnecessary domain computations can be avoided.

2 BOUNDARY ELEMENT METHOD FOR PLASTICITY

For a region Ω bounded by a boundary Γ , the direct boundary element equation for an initial stress formulation is

$$c_{ij}(P)\dot{u}_j(P) = \int_{\Gamma} U_{ij}(P, Q) i_j(Q) d\Gamma(Q) - \int_{\Gamma} T_{ij}(P, Q) \dot{u}_j(Q) d\Gamma(Q) + \int_{\Omega} E_{ijk}(P, q) \dot{\sigma}_{jk}^p(q) d\Omega(q) \quad (1)$$

where P, Q are points on the boundary and p, q are points on the domain; $\dot{u}_j(Q)$ is the displacement field, $i_j(Q)$ the traction field, $\dot{\sigma}_{jk}^p(q)$ is the initial stress field and $c_{ij}(P)$ is the jump term arising from the treatment of the improper integral involving $T_{ij}(P, Q)$ and is related to the geometry of the boundary. $U_{ij}(P, Q)$, $T_{ij}(P, Q)$ and $E_{ijk}(P, q)$ are the fundamental solutions for displacements, tractions and strains, respectively, due to a unit point load at P . Although plasticity is a time-independent phenomenon, it may be associated to a time-like parameter, the loading factor, and that is the reason why the rate notation is used.

Stress rates within the region can be computed from eq.2 which is obtained by differentiating eq.1 with respect to the source point p and using Hooke's law.

$$\dot{\sigma}_{ij}(p) = \int_{\Gamma} D_{ijk}(p, Q) i_k(Q) d\Gamma(Q) - \int_{\Gamma} S_{ijk}(p, Q) \dot{u}_k(Q) d\Gamma(Q) + \int_{\Omega} W_{ijkl}(p, q) \dot{\sigma}_{kl}^p(q) d\Omega(q) + F_{ij}(\dot{\sigma}_{kl}^p) \quad (2)$$

where $F_{ij}(\dot{\sigma}_{kl}^p)$ are the free terms arising from the singularity at p . This equation was derived for points located inside the body. In case the point is located at the boundary, the limit of the integral when the source point approaches the boundary can be taken. However, this procedure is computationally expensive, due to the occurrence of hyper-singular integrands in the boundary integrals. Therefore, an alternative and relatively simple way of evaluating boundary stresses from tractions and displacement tangential derivatives is used here. This is the so-called stress recovery method^{3,4}.

3 ADAPTIVE CELL GENERATION ALGORITHM

The main steps of the proposed method are as follows: an elastic solution is obtained for the boundary and the yield condition is checked for each boundary node. For each node where this condition is violated, a corresponding internal point is created. This point is located at a user defined distance from the boundary, in the direction of the normal to the boundary node. Points inside the domain corresponding to the other element nodes are created as well. From these points cells surrounding the plastic nodes of the boundary can be created. A pseudo boundary is defined, which contains the boundary of the cells. The stresses are evaluated for the generated internal points using the classical internal stress integral equation for elasto-plasticity^{3, 4, 5} (eq.2). The yield condition is checked for the pseudo boundary points and if

violated, new points and cells are created and the pseudo-boundary is updated. Cell creation is stopped when no point at the pseudo boundary violates the yield condition. The elasto-plastic algorithm is then applied as in the classical approach^{3, 4, 5}. However, after each iteration the stresses at the pseudo-boundary points are checked against the yield condition and if necessary new cells are generated such as explained above.

4 NUMERICAL EXAMPLES

4.1 Perforated Plate Under Tension

The perforated plate shown in Fig. 1a is subjected to a mean applied stress $\sigma_0 = 22$ MPa. Due to the symmetry of the problem only one half of the problem was analyzed. Only the boundary of the problem was discretized. The following material parameters are assumed: Young's modulus $E = 7000$ MPa, Poisson ratio $\nu = 0.2$, yield stress $Y = 50$ MPa. The material is considered to be elastic perfectly plastic and the von Mises yield criteria is applied. The problem was analyzed with linear and quadratic isoparametric approximation for boundary elements as well as internal cells.

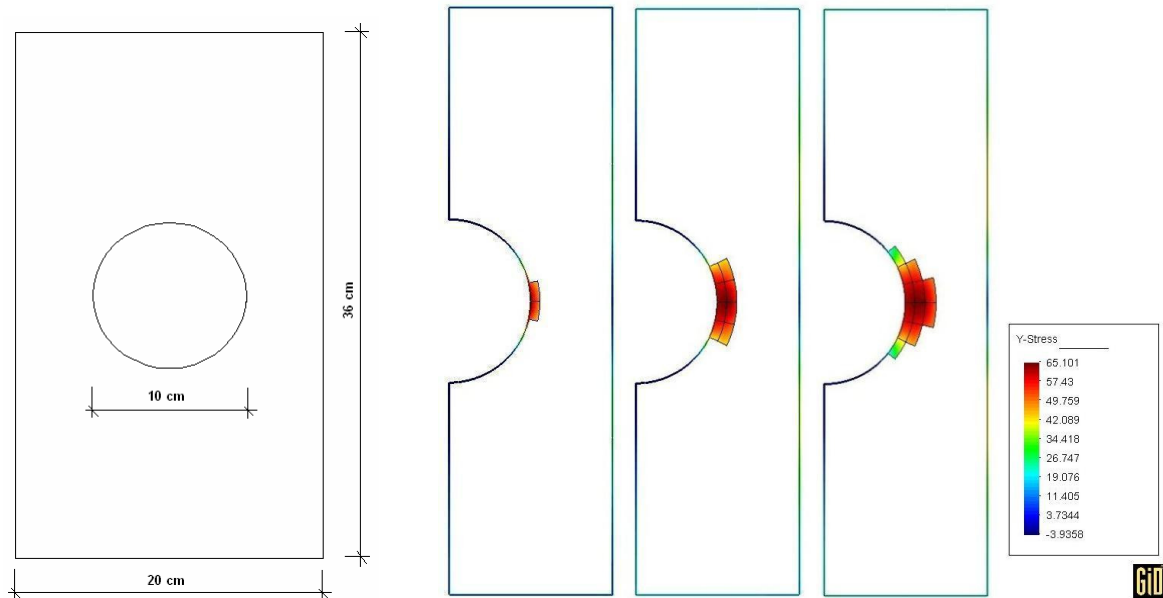


Figure 1: (a) Geometry of the problem; (b) Some stages of the cell generation with quadratic interpolation

Figure 1b shows some stages of the cell generation process for the quadratic approximation as well as the final results for the stress component σ_y . These results were compared to finite element and also boundary element results with pre-defined cells and are in very good agreement with them for both quadratic and linear approximation.

4.2 Notched Plate

A notched plate in tension ($\sigma_0 = 667$ N/mm²) was analyzed with a boundary discretisation only, elastic-perfectly plastic material was assumed, with Young's modulus $E = 210000$

N/mm^2 , Poisson ratio $\nu = 0.3$ and yield stress $Y = 1500 \text{ N/mm}^2$. Von Mises yield criteria is applied and again linear and quadratic isoparametric boundary elements and internal cells were used. Figure 2a shows the discretization of the problem for linear interpolation.

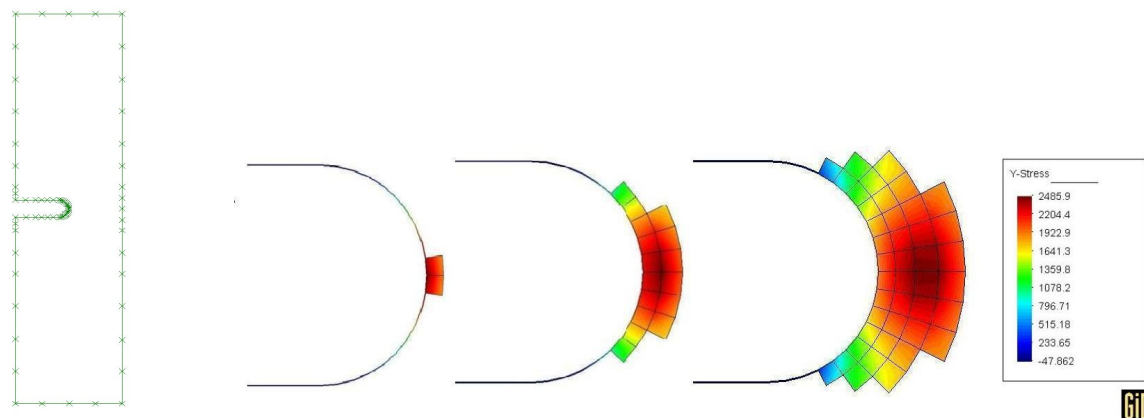


Figure 2: (a) Boundary element discretization; (b) Some stages of cell generation – zoom on the notch

Some stages of the cell generation process on the notch are presented in Fig. 2b, where the final results for the stress component σ_y are also plotted. These results are in very good agreement with results from a boundary element calculation with pre-defined cells and results from a finite element calculation, for both linear and quadratic interpolation.

5 CONCLUSIONS

A new approach to automatically generate volume discretization in the plastic zone has been briefly presented. The proposed approach retains the accuracy of the classical approach and has shown a significant gain in efficiency, especially for infinite problems or finite problems with stress concentration. Even though only 2D problems have been presented here, the approach can be easily extended to three-dimensional problems.

6 ACKNOWLEDGMENTS

The authors would like to acknowledge the Austria Science fund (FWF), project number P15523, for the financial support.

REFERENCES

- [1] A.J. Nowak and A.C. Neves, eds., *The multiple reciprocity boundary element method*, Computational Mechanics Publications, (1994).
- [2] G. Beer, *Programming the boundary element method*, Wiley, (2001).
- [3] X.W. Gao and T.G. Davies, *Boundary element programming in mechanics*, Cambridge University Press, (2002).
- [4] A.P. Csilino and M.H. Aliabadi, “A boundary element method for three-dimensional elastoplastic problems”, *Engineering Computations*, **15**, 1011-1030 (1998).
- [5] W.S. Venturini, *Boundary element method in Geomechanics*, Springer, (1983).