Asociación Argentina de Mecánica Computacional

Mecánica Computacional Vol XXIX, págs. 8187-8196 (artículo completo)
Eduardo Dvorkin, Marcela Goldschmit, Mario Storti (Eds.)
Buenos Aires, Argentina, 15-18 Noviembre 2010

MULTI-REGION BOUNDARY ELEMENT METHOD IMPLEMENTATION FOR VISCOPLASTIC ANALYSIS

Filipe E. S. Anacleto, Gabriel O. Ribeiro and Tatiana S. A. Ribeiro

Departamento de Engenharia de Estruturas, Universidade Federal de Minas Gerais, Avenida Antônio Carlos 6627, Escola de Engenharia, Pampulha, Belo Horizonte - MG, Brasil, filipe@dees.ufmg.br, gabriel@dees.ufmg.br

Keywords: Boundary Element Method, Viscoplasticity, Time-dependent behavior, Multi-region domain

Abstract. This work deals with the implementation of the boundary element method (BEM) for multi-region domain problems, under the hypothesis that the domain is composed of elasto/viscoplastic materials. The Perzyna's flow rule is adopted in order to achieve viscoplastic responses. The technique adopted to solve the multi-region problem consists of applying the classical BEM to solve separately each region with interfacial nodes restrained against displacements and the other nodes with the specified boundary conditions. Following this, a stiffness matrix is evaluated for each subregion by successively applying the BEM with a unit displacement value prescribed to each degree of freedom correspondent to each interfacial node and direction. By enforcing the compatibility and equilibrium conditions at the interfacial nodes and assembling a global stiffness matrix one can evaluate the displacements at the interfaces, and after this, the remaining unknowns can be obtained. This technique can also be used for coupling BEM and finite element method. The efficiency and accuracy of the BEM implementation are analyzed by means of numerical examples.

1 INTRODUCTION

The domain of some important engineering problems like excavations and tunnels present non-homogeneous material properties, which can be approximated by a zoned domain, *i.e.*, the domain is supposed to be made up by distinct subregions, each one with specific material properties. These subregions with their specific boundaries are modeled independently and then coupled along interfaces. Besides this, many materials, such as concrete, polymers and soils, exhibit a time-dependent behavior, *i.e.*, for a constant load different deformation and stress patterns are developed within the body as time passes.

This work deals with the implementation of the boundary element method (BEM) for two-dimensional multi-region domain problems, under the hypothesis that the domain is composed of elasto/viscoplastic materials. In order to simulate the viscoplastic flow the approach presented by Venturini (1983) is adopted, and to solve the multi-region problem, the stiffness matrix method presented by Beer et al. (2008) is used.

2 CONSTITUTIVE RELATIONS

The uniaxial representation of the elasto/viscoplastic mechanical model is shown in Fig. 1. This model simulates a material which behaves linearly elastic up to reach a limit value $Y(h)$ when irreversible time-dependent strains appear. In Fig. 1 $\Delta\sigma$ is a stress increment, $\Delta\varepsilon^e$ is an

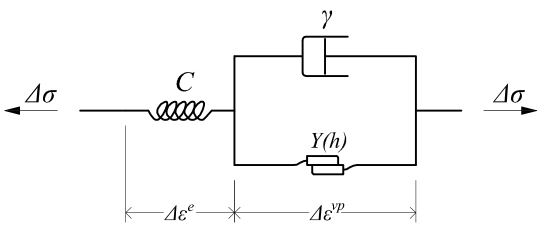


Figure 1: Elasto/Viscoplastic Model

elastic strain increment, $\Delta\varepsilon^{vp}$ is a viscoplastic strain increment, C is an elastic parameter, γ is a viscous parameter and Y is the yield stress which is an escalar function of the hardening parameter h.

The constitutive equation for the elasto/viscoplastic model generalized for tridimensional problems can be written as (Venturini, 1983)

$$\Delta\sigma_{ij} = C_{ijkl}(\Delta\varepsilon_{kl} - \Delta\varepsilon_{kl}^{vp}) \tag{1}$$

where $\Delta\sigma_{ij}$, $\Delta\varepsilon_{kl}$, $\Delta\varepsilon_{kl}^{vp}$, and C_{ijkl} stands for the components of the total stress increment, the total strain increment, the viscoplastic strain increment and the constitutive elastic tensors, respectively. It is important to note that Eq. (1) is defined in an initial stress sense and is valid only for a small increment of time Δt.

In a similar way as in the elastoplastic analysis, a yield function and a flow rule are needed. The von Mises yield criterion and Perzyna's viscoplastic flow rule are adopted in order to perform the analysis and can be written, respectively, as

$$F = \sqrt{3J_2} - Y(h) = 0 \tag{2}$$

$$\Delta\varepsilon_{kl}^{vp} = \frac{1}{\gamma}\left\langle \frac{F}{F_o} \right\rangle \frac{\partial F}{\partial \sigma_{kl}} \Delta t \tag{3}$$

where J_2 is the second invariant of the stress deviatoric tensor and F_o denotes any convenient reference value of F for the dimensionless representation of $\Delta\varepsilon_{kl}^{vp}$. In Eq. (3) the $\langle\rangle$ notation implies that

$$\left\langle \frac{F}{F_o} \right\rangle = \begin{cases} \frac{F}{F_o} & \text{for } F > 0 \\ 0 & \text{for } F \leq 0 \end{cases} \tag{4}$$

Then, for an increment of time Δt, an increment of the initial stress tensor $\Delta\sigma_{ij}^o$ can be expressed as

$$\Delta\sigma_{ij}^o = C_{ijkl}\Delta\varepsilon_{kl}^{vp} \tag{5}$$

The time increment value cannot be chosen freely due to numerical instability. In the current work, the time step length limit proposed by Cormeau (1975) for the associative von Mises flow rule is used which can be written as

$$\Delta t \leq \frac{4\gamma(1+\nu)F_o}{3E} \tag{6}$$

where ν is the Poisson coefficient and E is the Young's modulus. The time step limit is evaluated for each region based on its specific material properties and from these time steps values, the smallest value is adopted.

3 BOUNDARY INTEGRAL EQUATIONS

The elasto/viscoplastic displacement boundary integral equation (BIE) can be developed in a similar fashion as the classical displacement BIE, but in this case the BIE must be expressed in an incremental form taking into account the incremental constitutive equation (Eq. 1). The incremental displacement BIE based on the infinitesimal strain tensor, taking into account initial stresses, can be written as (Venturini, 1983)

$$\begin{aligned} C_{ij}(P)\Delta u_j(P) &= \int_\Gamma U_{ij}^*(P,Q)\Delta p_j(Q)d\Gamma \\ &- \int_\Gamma T_{ij}^*(P,Q)\Delta u_j(Q)d\Gamma + \int_\Omega E_{ijk}^*(P,q)\Delta\sigma_{jk}^o(q)d\Omega \end{aligned} \tag{7}$$

where U_{ij}^*, T_{ij}^* and E_{ijk}^* represents, respectively, the displacement, traction and strain Kelvin fundamental solutions, and Δu_j, Δp_j and $\Delta\sigma_{jk}^o$ denote, respectively, the displacement, traction and initial stress components. In Eq. (7) the letters P, Q and q represents, respectively, the source point, a field point at the boundary, and a field point in the domain. This is the same BIE of the elastoplastic formulation based on initial stress, the main difference is that an elastoplastic analysis proceeds in load increments while a viscoplastic analysis proceeds in time increments (Beer et al., 2008).

4 BOUNDARY ELEMENT METHOD

By considering the difficulties involved in solving Eq. (7) in closed form solution, the boundary Γ is then discretized into boundary elements defined by nodes, following the usual procedures of the BEM. The boundary values (displacement and tractions) are approximated using polynomial functions based on the respective boundary nodal values. The domain Ω is discretized into internal cells defined by internal nodes. In the current work continuous linear and quadratic boundary elements and cells are used for the approximations.

4.1 Viscoplastic formulation

By applying the discretization and the respective approximations in the viscoplastic BIE (Eq. (7)), leads to an incremental system of equations is obtained, which is expressed as

$$[T]\{\Delta u\} = [U]\{\Delta p\} + [E]\{\Delta\sigma^o\} \tag{8}$$

where $\{\Delta u\}$, $\{\Delta p\}$ and $\{\Delta\sigma^o\}$ denote vectors containing, respectively, nodal incremental values of displacements, tractions and initial stresses. The integrations coefficients of Kelvin fundamental solutions are held in matrices $[U]$, $[T]$ and $[E]$.

4.2 Multi-region method

Eq. (8) is valid only for one homogenous domain. To solve a multi-region problem using the stiffness method procedure, first each region N has to be solved separately considering the real applied load but also enforcing to the interface coupled nodes between regions zero displacement. This will result in a vector of tractions along the interface for each region $\{\Delta t\}_{c0}^N$ and in a solution vector for the free nodes that are not coupled to other regions $\{\Delta x\}_{f0}^N$. This phase can be written as

$$\left\{ \begin{array}{c} \{\Delta t\}_{c0}^N \\ \{\Delta x\}_{f0}^N \end{array} \right\} = [A]^{-1}(\{B\} + [E]\{\Delta\sigma^o\}) \tag{9}$$

where Matrix $[A]$ contains the integration coefficients correspondent to the unknown boundary values while vector $\{B\}$ gives the influence of the prescribed boundary values.

Then a stiffness matrix $[K]$ is assembled for each region N. In order to do this, the problem is solved N_{gl} times for each region (see Eq. (10)), where N_{gl} stands for the number of degrees of freedom of all its coupled nodes. In each solution a unit value displacement is applied for each degree of freedom of the interface, one at a time. In this phase matrix $[A]$ is constant and only vector B has to be revaluated for each solution.

$$\left\{ \begin{array}{c} \{\Delta t\}_{cn}^N \\ \{\Delta x\}_{fn}^N \end{array} \right\} = [A]^{-1}\{B\} \qquad n = 1, 2, \dots, N_{gl} \tag{10}$$

After assembling the stiffness matrix, the real solution at the boundary free nodes $\{\Delta x\}_f^N$ and the real tractions at the boundary coupled nodes $\{\Delta t\}_c^N$ can be written as a function of the real displacements at the boundary coupled nodes $\{\Delta u\}_c^N$, as can be seen in Eq. (11).

$$\left\{ \begin{array}{c} \{\Delta t\}_c^N \\ \{\Delta x\}_f^N \end{array} \right\} = \left\{ \begin{array}{c} \{\Delta t\}_{c0}^N \\ \{\Delta x\}_{f0}^N \end{array} \right\} + \left[\begin{array}{c} \{\Delta t\}_{c1}^N, \{\Delta t\}_{c2}^N, \dots, \{\Delta t\}_{cN_{gl}}^N \\ \{\Delta x\}_{f1}^N, \{\Delta x\}_{f2}^N, \dots, \{\Delta x\}_{fN_{gl}}^N \end{array} \right] \{\Delta u\}_c^N \tag{11}$$

Assembling the stiffness matrices for all regions and considering the equilibrium and compatibility relations at the interfaces (see Eq. (12)) it is possible to evaluate the real displacements $\{\Delta u\}_c^N$ at the interface using Eq. (13).

$$\begin{array}{c} \{\Delta t\}_c^I + \{\Delta t\}_c^{II} = 0 \\ \{\Delta u\}_c^I = \{\Delta u\}_c^{II} \end{array} \tag{12}$$

$$\{\Delta t\}_{c0} + \left[\{\Delta t\}_{c1}, \{\Delta t\}_{c2}, \dots, \{\Delta t\}_{cN_{gl}}\right] \{\Delta u\}_c = 0 \tag{13}$$

After the evaluation of the real displacements at the coupled nodes $\{\Delta u\}_c$, the real solution for the free nodes are evaluated using Eq. (11).

This multi-region approach works very well if interfaces between regions are smooth. But in order to apply it to generalized problems where the interfaces between regions are not smooth specials technics are needed, the more adequate being the use of discontinuous elements (Beer et al., 2008).

4.3 Automatic domain discretization

The BEM implementation in this work requires *a priori* only the boundary discretization of the problem. The domain discretization in cells during the viscoplastic analysis is performed only in the yielded regions by applying the algorithm proposed by Ribeiro et al. (2008). For each time step the yield criteria is checked at all nodes and then cells are created around each yielded node until all these nodes are surrounded by nodes yet in an elastic stress state. This scheme is illustrated in Fig. (2) from (a) to (d).

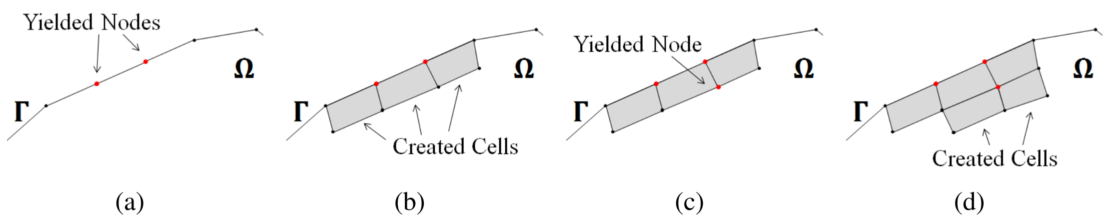


Figure 2: Update and evolution of the discretized region.

4.4 Singular integrations and stress evaluation

At the boundary, the weakly singular integrals are evaluated by means of the logarithmic Gaussian quadrature, while the strongly singular integrals are treated using a numerical technique proposed by Giggiani and Casalini (1987) to directly calculate the Cauchy principal value. Regarding domain integration, weakly singular integrals are evaluated using cell subdivision (Lachat and Watson, 1976) and strongly singular integrals are computed using subtraction of singularity and semi analytical integration as presented by Gao and Davies (2002).

The stresses at interior points are evaluated using Somigliana stress identity with initial stress. While at the boundary, the stresses are evaluated using a methodology called 'stress recovery method' (Gao and Davies, 2002) avoiding the evaluations of hyper singular integrals.

5 COMPUTATIONAL ROUTINE

The procedure to solve the viscoplastic multi-region problem consists mainly of the following steps:

(i) The matrices $[U]$ and $[T]$ (see Eq. (8)) are evaluated for each region.

(ii) Using the applied boundary conditions, and the multi-region technique exposed in section 4.2, the system is solved and a solution increment $\Delta\{x\}$ is obtained. In this phase there is no contribution of $\{\Delta\sigma^o\}$.

(iii) The final solution is updated, *i.e.*, $\{x\} = \Delta\{x\}$

(iv) Then a loop over time increments starts, where in each time increment:

- The stress is evaluated in each node and the yield criterion is checked for all regions.
- Cells are created around all yielded nodes and the correspondent $[E]$ matrix is evaluated for each region.
- An $\{\Delta\sigma^o\}$ vector is evaluated using Eq. (5) for each region.

- Using the multi-region technique exposed in section 4.2, the system is solved and a solution increment $\Delta\{x\}$ is obtained. In this phase it is considered only the contribution of the $\{\Delta\sigma^o\}$ evaluated for the current time step.

- The final solution is updated, *i.e.*, $\{x\} = \{x\} + \Delta\{x\}$

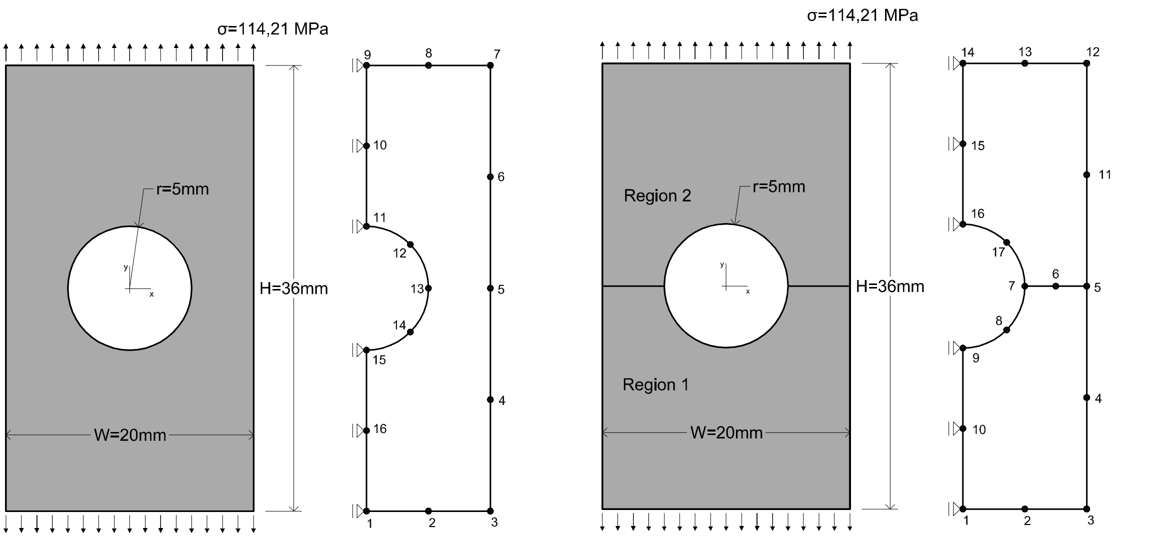
This loops continues until the total time interval is reached.

6 NUMERICAL EXAMPLES

In this section two bidimensional (2D) examples analyzed using the algorithm implemented are shown. First to demonstrate the efficiency of the multi-region approach a perforated plate under uniform tensile load is considered and the results are compared with a finite element solution. Secondly a lined cavity under surface load due to initial stress field is considered. This last example is mainly presented to show the convergence of the implemented algorithm. For both cases the von Mises yield criterion is used together with the hypothesis of perfectly viscoplastic material, *i.e.*, no hardening behavior is considered. Comparisons are based on the final viscoplastic results, *i.e.*, $t \to \infty$, when stability is reached (Cormeau and Zienkiewicz, 1974).

6.1 Perforated plate under uniform tensile load

Due to the symmetry of this problem only half of the problem is analyzed. The following material parameters are assumed: Young's modulus $E = 70GPa$, Poisson's ratio $\nu = 0.2$, viscosity $\gamma = 5days$ and yield stress $Y = 243MPa$. The geometries and the basic meshes for this problem can be seen in Fig. (3).



(a) 1 Region (b) 2 Regions

Figure 3: Geometries and Meshs

Initially the problem is solved considering only one region (1R), see Fig. (3(a)). For this case two boundary element meshes are considered, one with 128 nodes (BEM 128 1R), and one with 256 nodes (BEM 256 1R), both using linear elements. The results obtained are then compared with a finite element solution obtained using ANSYS® for a mesh with 4500 finite elements and 13831 nodes (FEM 13831 1R). The final maximum values obtained for each mesh considering only one region can be compared in Table 1, where the values inside the parentheses are the relative error considering the FEM 13831 1R mesh as the reference.

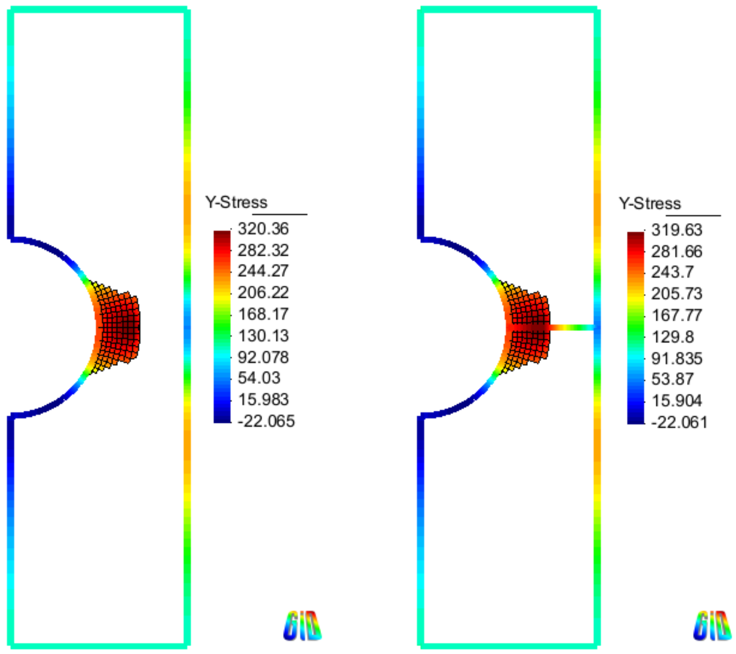
Mesh	u_y [mm]	σ_x [MPa]	σ_y [MPa]
FEM 13831 1R	0.0459017	-213.689	325.533
BEM 128 1R	0.046252 (-0.76%)	-211.12 (1.20%)	317.62 (2.43%)
BEM 256 1R	0.046444 (-1.18%)	-215.46 (-0.83%)	320.36 (1.59%)

Table 1: Comparison between finite elements and boundary elements results considering one region.

As can be seen in Table 1 the final maximum results for both meshes show good agreement. Considering now two regions (2R), see Fig (3(b)), the problem is solved using two new boundary meshes derived from the two previous ones (BEM 128 1R and BEM 256 1R). The first mesh with 136 nodes (BEM 136 2R) and the second one with 273 nodes (BEM 273 2R). In Table 2 a comparison between the one region and two regions analysis is shown. As can be seen, the algorithm for multi-region presented very good results. In Fig. (4) the final distribution for σ_y for the meshes BEM 256 1R and BEM 256 1R are shown.

Mesh	u_y [mm]	σ_x [MPa]	σ_y [MPa]
BEM 128 1R	0.046252	-211.12	317.62
BEM 136 2R	0.04625 (0.01%)	-211.52 (-0.19%)	314.06 (1.12%)
BEM 256 1R	0.046444	-215.46	320.36
BEM 273 2R	0.046449 (-0.01%)	-215.65 (-0.09%)	319.63 (0.23%)

Table 2: Comparison between boundary elements results considering one and two regions.



(a) 1 Region (b) 2 Regions

Figure 4: Final Solution for σ_y

6.2 Lined cavity under surface load due to initial stress field

The geometry and the BEM mesh for the lined cavity can be seen in Fig. (5). It is considered the lining to behave as a linear elastic material, while the rock as a elasto/viscoplastic material. For the rock the following material parameters are assumed: Young's modulus $E = 3500MPa$, Poisson's ratio $\nu = 0.2$, viscosity $\gamma = 2days$ and yield stress $Y = 580MPa$. And for the lining: Young's modulus $E = 21100MPa$, Poisson's ratio $\nu = 0.15$. The cavity is subjected to vertical, as well as horizontal load. The vertical load applied to the cavity surface is taken to be $\sigma_y = -680MPa$ whereas the horizontal load is assumed to be $\sigma_x = -272MPa$.In this problem linear and quadratic elements are used. In Fig. (6), (7), (8) and (9) the results obtained for each mesh are plotted. As can be seen from this results a very good convergence is reached.

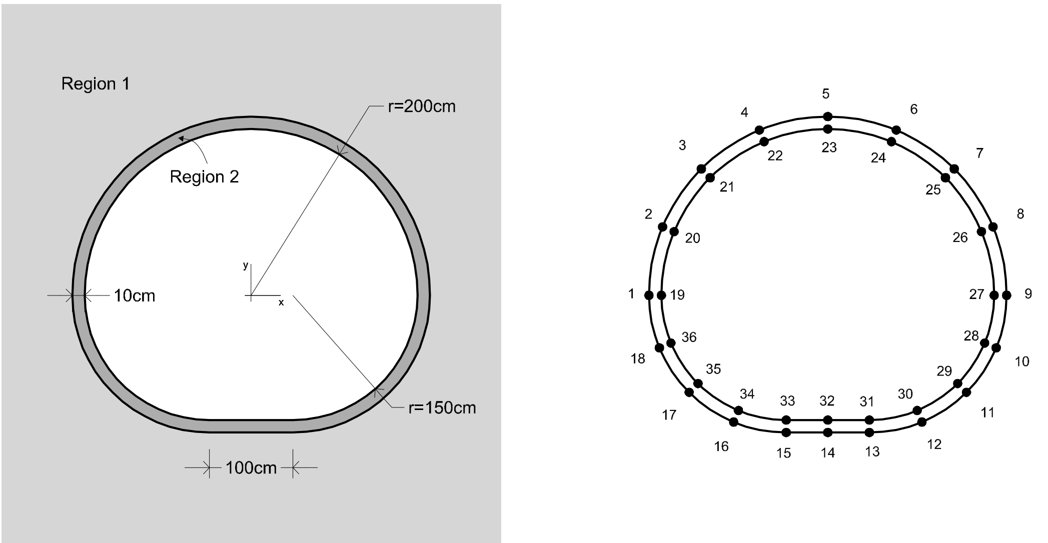


Figure 5: Geometry and mesh

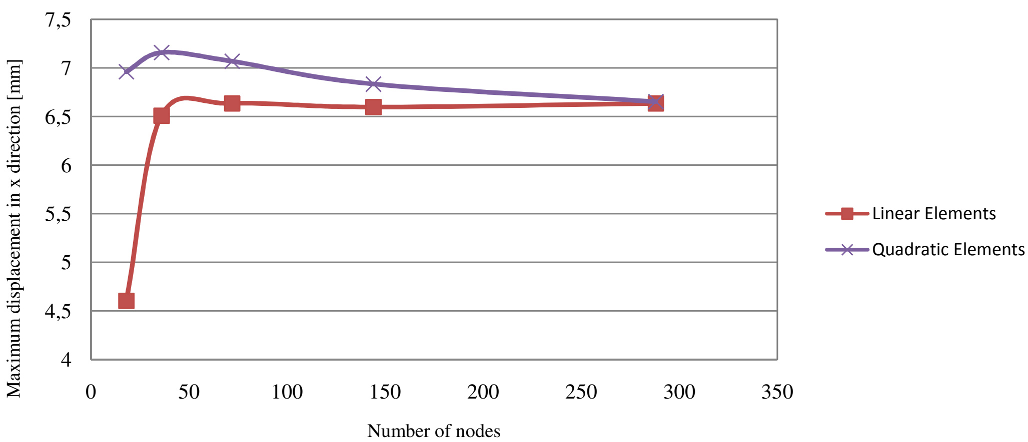


Figure 6: Convergence of the boundary element solution for the maximum displacement in the x direction

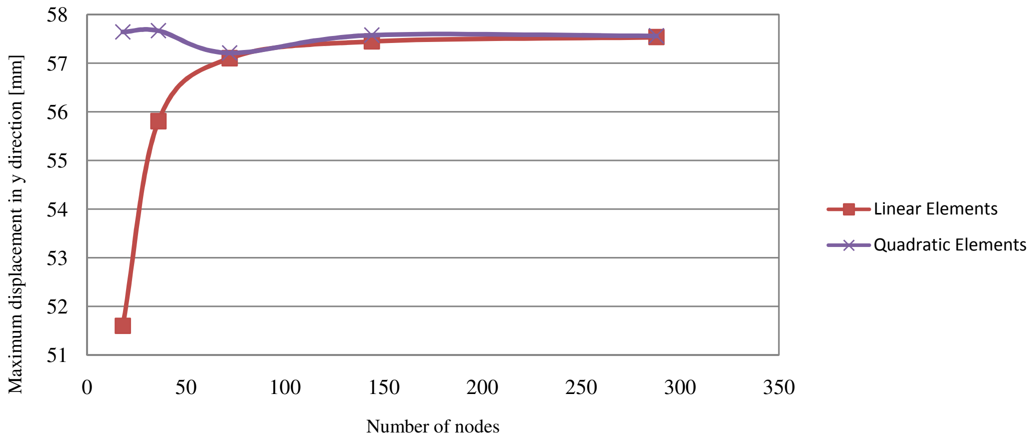


Figure 7: Convergence of the boundary element solution for the maximum displacement in the y direction

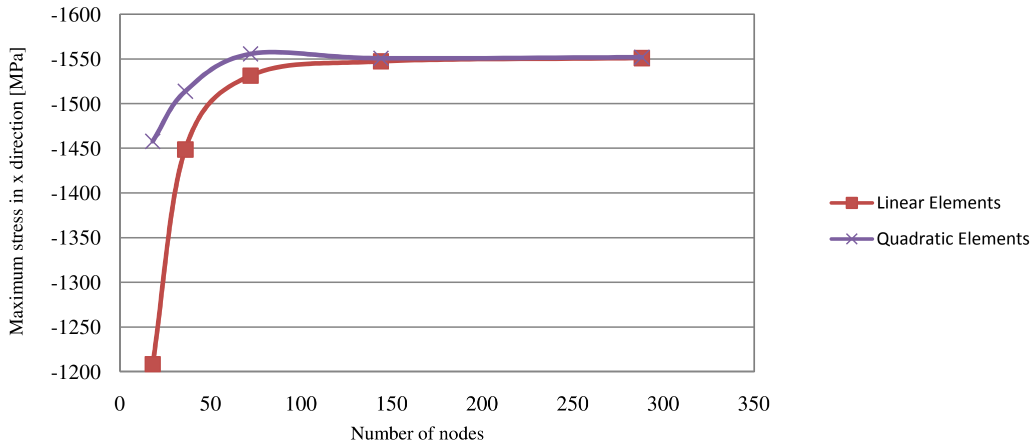


Figure 8: Convergence of the boundary element solution for the maximum stress in the x direction

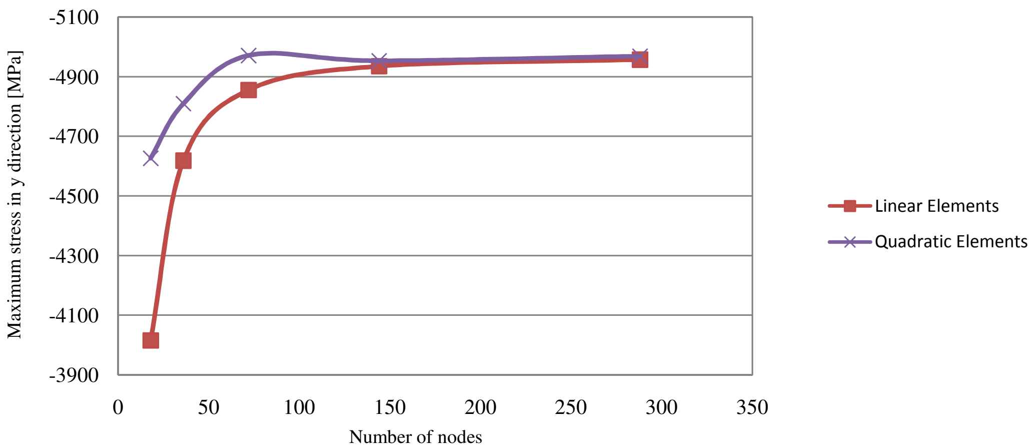


Figure 9: Convergence of the boundary element solution for the maximum stress in the y direction

7 CONCLUSIONS

The numerical BEM results obtained for this multi-region viscoplastic analysis have demonstrated to be valid and with a good accuracy. The algorithm to automatically generate internal cells has shown to be a good alternative to avoid the *a priori* domain discretization, saving end-user input and computational processing effort, since cells are only generated where needed. It is important to note that this multi-region approach works very well if interfaces between regions are smooth, but in order to apply it to generalized problems where the interfaces between regions are not smooth specials technics are needed, the more adequate being the use of discontinuous elements (Beer et al., 2008).

8 ACKNOWLEDGEMENTS

The authors gratefully acknowledge the support received from the following Brazilian agencies of research funding: CNPq - Conselho Nacional de Desenvolvimento Científico e Tecnológico; FAPEMIG - Fundação de Amparo à Pesquisa do Estado de Minas Gerais and CAPES - Coordenação de Aperfeiçoamento de Pessoal de Nível Superior.

REFERENCES

Beer G., Smith I., and Duenser C. *The boundary element method with programming for engineers and scientists*. Springer-Wien-NewYork, 2008.

Cormeau I. Numerical stability in quasi-static elasto-viscoplasticity. *International Journal for Numerical Methods in Engineering*, 9:109–127, 1975.

Cormeau I. and Zienkiewicz O. Visco-plasticity, plasticity and creep in elastic solids - a unified numerical approach. *International Journal for Numerical Methods in Engineering*, 8:821–845, 1974.

Gao X.W. and Davies T.G. *Boundary element programming in mechanics*. Cambridge University Press, 2002.

Giggiani M. and Casalini P. Direct computation of cauchy principal value integrals in advanced boundary elements. *International Journal for Numerical Methods in Engineering*, 24:1711–1720, 1987.

Lachat J.C. and Watson J.C. Effective numerical treatment of boundary integral equations: A formulation for three dimensional elastostatics. *International Journal for Numerical Methods in Engineering*, 10:991–1005, 1976.

Ribeiro T.S.A., Beer G., and Duenser C. Efficient elastoplastic analysis with the boundary element method. *Computational Mechanics*, 41:715–732, 2008.

Venturini W.S. *Boundary element method in geomechanics*. Springer-Verlag, Germany, 1983.