Transactions of the **13th International Conference on Structural Mechanics in Reactor Technology** (SMiRT 13), Escola de Engenharia - Universidade Federal do Rio Grande do Sul, Porto Alegre, Brazil, August 13-18, 1995

Elastoplastic finite element analysis of impact problems; study of a turbine accident

Bittencourt, E., Creus, G.J.
Universidade Federal do Rio Grande do Sul, CPGEC/CEMACO, Porto Alegre, RS, Brazil

ABSTRACT: A methodology for the analysis of impact problems among elastoplastic flexible bodies is studied. Emphasis is given to the description of the contact algorithm. In this paper, the procedure is used to study the consequences of an accident involving a turbine in a nuclear instalation. Comparisons of numerical results with tests performed by the U.S. Electric Power Research Institute (EPRI 1983), show the potential application of this numerical model for crashworthiness predictions.

1 INTRODUCTION

Numerical models are now an important tool for achieving better understanding of the behavior of structures during impact, as in crashworthiness analysis. Better algorithms and faster computers now allow the numerical modeling of dynamic effects, large deformations and contact effects, previously only possible through crash testing. Special attention has been given to the treatment of contact between flexible bodies, since it is so common in crashworthiness problems.

This paper presents a robust algorithm for two-dimensional (2D) treatment of contact between flexible bodies, able to identify and simulate post-buckling. The algorithm uses the penalty method for contact force calculation, and an explicit solution (time-integration using the Central Difference (CD) scheme). It was implemented into the finite element (FE) code METAFOR (Ponthot-Hogge 1991) that models large deformations processes.

In order to clearly demonstrate the efficacy and potential applications of the algorithm, a simulation was performed of a crash test for nuclear power plant turbines, developed by EPRI and proposed to provide benchmark data on the energy absorbed by turbine casings in stoping rotor fragments.

2 BASIC FORMULATION

The balance equations in a FE formulation for a body under impact conditions, can be written as

$$\underset{\sim}{M}\,\underset{\sim}{A} = \underset{\sim}{F}^{int} - \underset{\sim}{F}^{ext} \tag{1}$$

where "$\underset{\sim}{A}$" represents nodal aceleration, "$\underset{\sim}{M}$" is the mass matrix, "$\underset{\sim}{F}^{ext}$" are the external nodal forces and "$\underset{\sim}{F}^{int}$"are the internal nodal forces calculated as

$$F_i^{K,int} = \int \sigma_{ji} \frac{\partial \Phi^K}{\partial x_j} dV^{el} \tag{2}$$

where "σ_{ij} " is the Cauchy stress, "Φ^K" is the interpolation function of the FE for the node "K". (In this paper, isoparametric linear 2D FEs are used). The integration in (2) is performed in the FE volume "V^{el}". Here, only contact forces are included as external force. Its calculation will be seen in the next section.

For solution in time of (1), the CD scheme is used. The basic steps for one time increment are shown below

$$\begin{aligned}
&\underset{\sim}{A}^{(n)} = \underset{\sim}{M}^{-1}(\underset{\sim}{F}^{int} - \underset{\sim}{F}^{ext})^{(n)} \\
&\dot{\underset{\sim}{U}}^{(n+1/2)} = \dot{\underset{\sim}{U}}^{(n-1/2)} + \Delta t^{(n)}\,\underset{\sim}{A}^{(n)} \\
&\underset{\sim}{U}^{n+1} = \underset{\sim}{U}^{(n)} + \dot{\underset{\sim}{U}}^{(n+1/2)}\,\Delta t^{(n)} \\
&n = n+1
\end{aligned} \tag{3}$$

where "$\underset{\sim}{U}$" is the nodal displacement, "$\dot{\underset{\sim}{U}}$" is the nodal velocity, superindex (n) indicates the time step and "Δt" is the time interval for integration, determined according to the element size and material constants (Stainier 1993). A diagonal mass matrix is used, caculated by sum-row. Integration of the Cauchy stress in time is done using the last equilibrated configuration of the incremental solution (Updated Lagrangian formulation). More details about integration of the Cauchy stress can be found in Ponthot-Hogge (1991).

3 CONTACT FORCE CALCULATION

When two surfaces contact each other, one is defined as the "slave" and the other as the "target". Linear one-dimensional contact elements are defined on the surfaces, as we can see in Figure 1a. To determine contact forces, it is necessary to first project the nodes of the slave surface (slave nodes) on the target surface. Single projection can lead to error when target elements are smaller than slave elements because interpenetrations can be left undetected. To avoid this problem, we use a double-pass calculation; i.e. elements that were slaves in the first calculation are targets in the second, and vice-versa.

The first step in the algorithm is to find which target elements are candidates to contact a pre-defined slave node. This search is done by defining boxes around each target element using the maximum and minimum nodal coordinates of the element plus a tolerance. If the box contains a slave node, the target element is a candidate to contact the node. After this search, we have, for each slave node, a list of elements candidates for contact.

The next step is to orthogonally project the slave node onto the target element. To do so, we first define, for a contact element (see Figure 1a)

$$\underset{\sim}{X}^S = \begin{bmatrix} x^S \\ y^S \end{bmatrix}; \quad \underset{\sim}{X}^{TI} = \begin{bmatrix} x^{TI} \\ y^{TI} \end{bmatrix}; \quad \underset{\sim}{t} = \begin{bmatrix} t_1 \\ t_2 \end{bmatrix} \tag{4}$$

where "$\underset{\sim}{X}^S$" defines the coordinates of slave node "S"; "$\underset{\sim}{X}^{TI}$" defines coordinates of target element nodes "TI" (I=1,2), and "$\underset{\sim}{t}$" is the tangent vector to the target element.

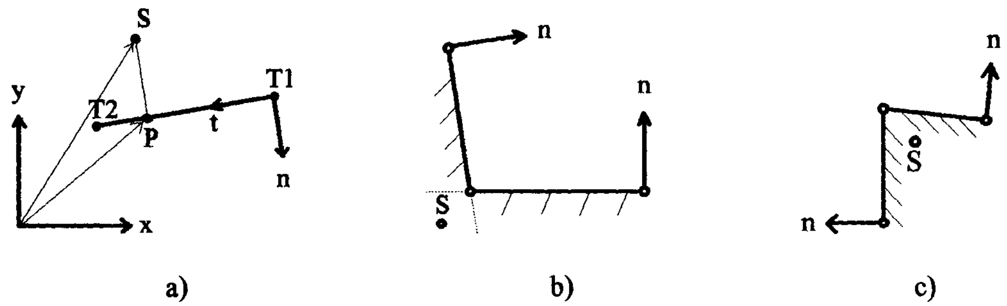


Figure 1. a) Contact element definition; b) Slave node "S" in a convex target region; c) Slave node "S" in a concav target region.

The orthogonal projection is defined by "$\underset{\sim}{P}$" and must satisfy the condition

$$(\underset{\sim}{P} - \underset{\sim}{X}^S).\underset{\sim}{t} = 0 \tag{5}$$

where $\underset{\sim}{P} = \Phi^{TI}\underset{\sim}{X}^{TI}$, and Φ^{TI} is the shape function associated with a target node

$$\Phi^{TI} = \frac{1}{2}(1 + \xi\xi^{TI}) \tag{6}$$

For the first target element node, we have $\xi^{T1} = -1$. For the second, $\xi^{T2} = 1$. "ξ" is the reduced coordinate of the orthogonal projection "$\underset{\sim}{P}$" onto the target element.

Using (5) and the definitions above, we can say that the reduced coordinate "ξ" is

$$\xi = \frac{t_1(2x^S - x^{T1} - x^{T2}) + t_2(2y^S - y^{T1} - y^{T2})}{-x^{T1}t_1 + x^{T2}t_1 - y^{T1}t_2 + y^{T2}t_2} \tag{7}$$

If $-1 \le \xi \le 1$, then the slave node "S" can be projected onto the element. This is the first contact condition.

We must still verify whether or not the slave node is inside the target body. If the node has penetrated, the condition (8) given below is true. This is the second contact condition.

$$(\underset{\sim}{P} - \underset{\sim}{X}^S).\underset{\sim}{n} > 0 \tag{8}$$

where "n" is the external normal vector to the target element (see Figure 1a).

There are still two special cases that must be considered:

First special case: The slave node is inside a concave zone (see Figure 1b). Here, although the node has penetrated the target body, the first contact condition is not satisfied. The solution used here is to project the slave node onto the nearest target node. This is more precise than projecting nodes onto prolongations of target elements, although it is more expensive in terms of computational time.

Second special case: The slave node is inside a convex zone (see Figure 1c). Here, there are two options for the orthogonal projection: it is possible to project the slave node onto either surface shown in the figure. In this algorithm, we chose to project the node on the surface closest to the node (see also Benson-Hallquist (1990)).

Once contact is verified, it is necessary to calculate the corresponding forces. To do so, we employ the penalty method so that the nodal forces on a slave node "S" are given by

$$F_n^{S} = k_n G_n \tag{9}$$

where "k_n" is the penalty factor and "G_n" is the normal distance between the slave node and the target element, calculated as

$$G_n = \left\| \underset{\sim}{X}^{S} - \underset{\sim}{P} \right\| \tag{10}$$

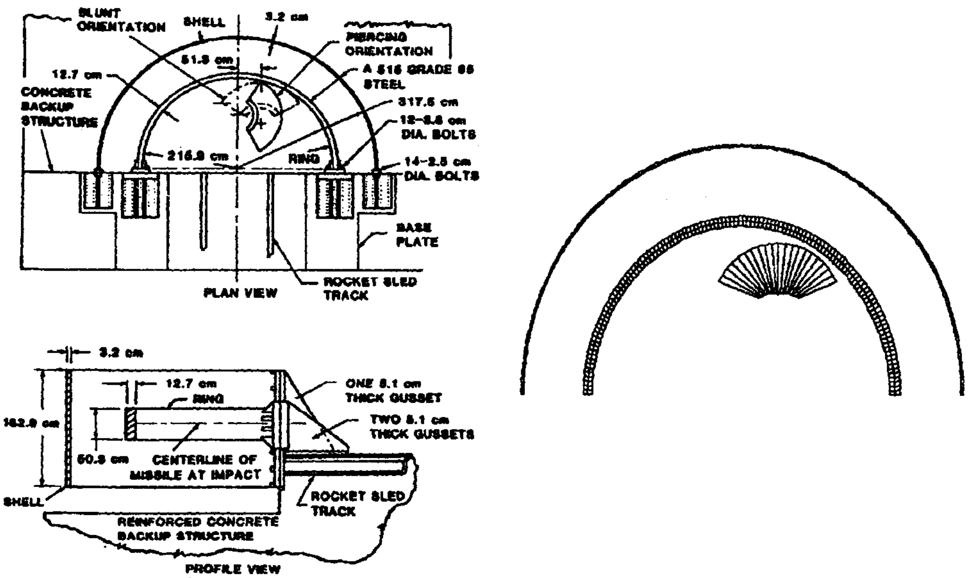
Reaction on the target element node "TI", in the normal direction, is given by

$$F_n^{TI} = -\frac{1}{2} F_n^{S} (1 + \xi\xi^{TI}) \tag{11}$$

It is important to note that the normal contact force calculation is done instantly and in the local axes. That means that this formulation is objective and thus precise, even for large deformation problems.

4 EXAMPLE

In this paper, we present an application of the algorithm to a problem of some importance in nuclear reactor structures: a crashworthiness test in a steam turbine, in which the rotor is broken during operation and hits the casing shells surrounding the turbine. The most critical case - rotor rupture in three pieces - is analysed here. To check results, comparisons are made with the test proposed by EPRI (1983) and performed at Sandia National Laboratories. The actual turbine geometry in EPRI's test was simplified as shown in Figure 2a. In our model, an even more simplified 2D (plain strain) FE mesh was used, as we can see in Figure 2b. In the crash test, impact was simulated with a jet-driven mass, disregarding the rotational movement. The same was done in the numerical simulation (the translational rotor velocity before impact used was 151 m/s), even though rotation could easily be introduced. The most conservative impact condition (the "blunt" orientation test) was modeled.



a) b)

Figure 2: a) Test geometry and b) FE mesh.

Casing shell material is ASTM A515 Gr65 cold-worked steel, and its properties are $\sigma_{yield} = 241$ MPa, $\sigma_{rupture} = 448$ MPa with elongation of 26 % at room temperature. The experimental data indicates that at time 0.0095 sec., the bolts that fix the internal shell on the right side broke. This information was explicitly introduced into the numerical modeling to change boudary conditions.

Figures 3a-3c show calculated configurations given by the present algorithm, at different times. They look fairly close to the configutations recorded during the tests. Figure 3d shows the missil trajectory in time as given by the numerical analysis and the test. We observe that, although the numerical model has a somewhat less stiff response, the result seems quite acceptable in light of the simplifications and approximations.

5 FINAL REMARKS

Until now, numerical simulations have been, in most cases, strongly dependent upon results gleaned from experiments. Here, for instance, we used time of bolt rupture data from the EPRI crash test in our calculation for defining change in boundary conditions. To make a pure numerical analysis using the algorithm presented here, it would be necessary to include fracture mechanics capabilities. On the other hand, given the favorable comparison between results achieved using the algorithm and those of actual crash-test, we may conclude that numerical simulation can be a safe way of making crashworthiness predictions with costs substantially lower than those of actual crash test.

Extensions to 3D contact and the consideration of friction can be found in Bittencourt (1994).

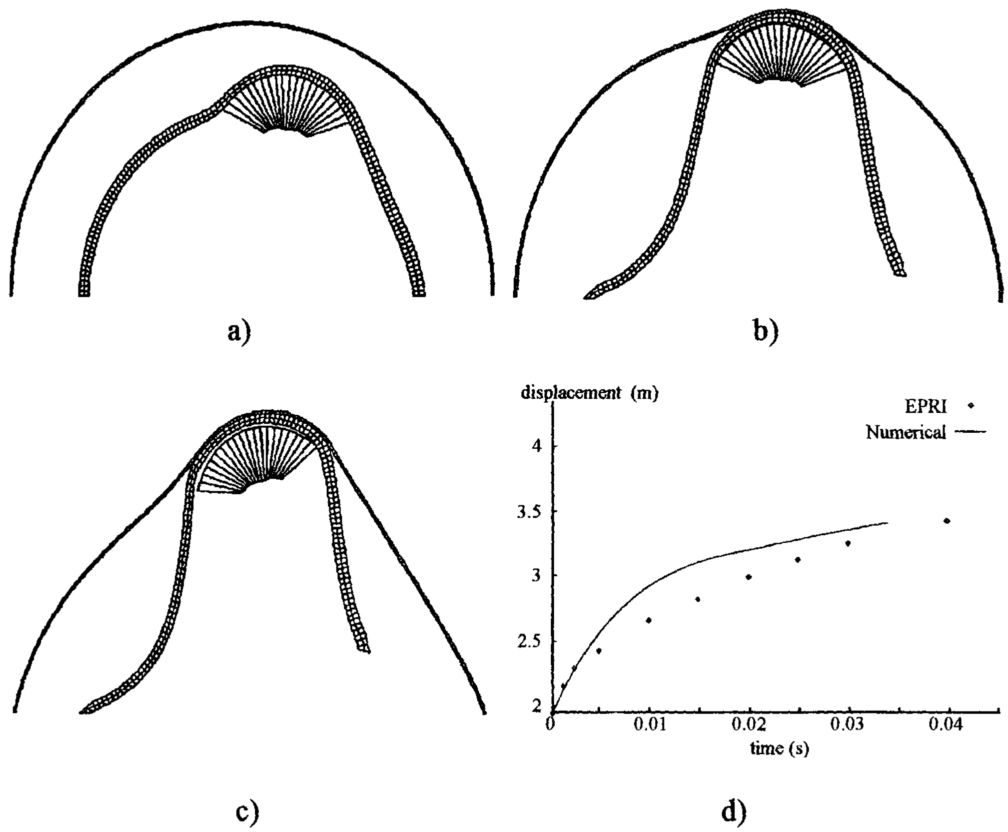


Figure 3. Configurations at time a) 0.008 sec.; b) 0.024 sec.; c) 0.04 sec. and d) comparasion between experimental (EPRI 1983) and the present numerical results.

REFERENCES

Benson, D. J.& Hallquist, J. O. 1990. A Single surface contactalgorithm for the post buckling analysis of shell structures. *Computer Methods in Applied Mechanics Eng.* 78: 141-163.

Bittencourt, E. 1994. Treatment of contact-impact problems in large deformations by the FEM, *Doctoral thesis.* Curso de Pós-Graduação em Engenharia Civil, Universidade Federal do Rio Grande do Sul, Brazil (in portuguese).

Electric Power Research Institute (EPRI) 1983. Full-scale turbine missile casing tests. *Final Report NP-2741.* Palo Alto, USA.

Ponthot, J. P. & Hogge, M. 1991. The use of Eulerian-Lagrangian FEM metal forming including contact and adaptive mesh. *Proc. ASME Winter Annual Meeting, dec. 1-6*: 1-16. Atlanta, USA.

Stainier, L. 1993. Optimal calculation of FE matrixes for non-compressible medium in large deformations. *Internal Report.* Laboratoire de Techniques Aéronautiques et Spatiales, Université de Liège, Belgium (in french).