G 2/4

Determination of Stress Intensity Factors K_I, K_{II} and K_{III}, by Use of Special Three Dimensional Finite Elements

G. Adda, T. Charras, P. Lafore
C.E.A., CEN Saclay, DEMT/SMTS/LAMS, F-91191 Gif-sur-Yvette Cedex, France
H. Tibi
C.E.A., IPSN/DAS/SASCEL, B.P. No. 6, F-92260 Fontenay-aux-Roses, France

Abstract

Two singular finite elements (20 nodes and 35 nodes) were developped and implemented within the BILBO code of the CEASEMT Finite Element System. This paper describes their formulation, which takes into account the exact solutions to the singular field problem. Validation has been performed on elementary cases. Finaly, a part through semi-elliptical crack, at the inner wall of a pressurized cylinder was considered. It was shown that such industrial applications are possible, since the necessity for mesh refinement in the vicinity of the discontinuity is significantly reduced, while values of the stress intensity factors remain accurate.

1. Introduction

Safety analysis of nuclear plants implies an evaluation of the risk of propagation of existing defects, within pressurized components. Such propagations might occur under normal conditions through fatigue loading, or in hypothetical accidental situations, leading to stresses exceeding the nominal design values. On the other hand, such studies can often be carried out within the frame of linear elastic fracture mechanics [1]. Many computational tools have already been derived in order to provide solutions for two dimensional problems. However, the geometrical complexity of some massive components as well as the necessity of considering the worst cases for defect position, sometimes imply the use of three dimensional analysis. This is particularly the case for nozzles, and junctions between pipes.

Several numerical methods can be used in order to solve this problem :

- Methods based on integral equations give good results, but they usually imply an important effort to modelize the geometry, which often has to be quite simplified.

- The finite element method is more practical to use [2], however its application to fracture mechanics also leads to difficulties : important mesh refinement is usually required around the crack tip, while usual elements cannot provide a mathematically correct formulation to the problem. On the other hand, determination of stress intensity factors has to be carried out in a post processor, as it is the case with displacements extrapolation. Methods based on an energy approach, such as the virtual crack extension method partially reduce these difficulties, however, they cannot simply lead to a separate determination of K_1, K_2 and K_3 in a complex loading case.

In order to solve this problem, special 3D, 35 nodes and 20 nodes elements were developed within the CEASEMT Finite Element System.

2. Definition of the special elements

2.1 - General approach

The two special elements are derived from one, or from two adjacent regular isoparametric elements. Each special element includes a portion of crack. This portion is supposed to be plane, its front is straight or circular. The two sides of the crack front correspond to two nodes of the element (see figures 1 and 2).

2.2 - Natural parameters

The series or the finite developments, corresponding to the exact mathematical solutions, taking into account the singularities at the crack front, have been used as singular shape function [3], [4]. 15 parameters corresponding to 5 exact singular solutions for each mode (1, 2, 3) have thus been added to the usual parameters corresponding to the regular element without crack.

2.3 - Final parameters

The final parameters interesting the user are the nodal displacements of the element and the values of K_1, K_2, and K_3 at each extremity of the crack front. Hence, these last two nodes have six degrees of freedom. For numerical reasons, K_1, K_2 and K_3 are not directly used. These quantities are replaced by proportional variables which are homogeneous to displacements and which are in fact of the same order of magnitude than the relative displacements of the crack lips. After the finite element computation, it is easy to use the values of the additionnal variables to compute K_1, K_2 and K_3.

The number of natural paramaters is equal to the number of final parameters under parametrisation is thus avoided.

2.4 - Relations between the two types of parameters

The final parameters can be computed from the natural parameters, using the isoparametric and the singular shape functions. A matrix is then obtained which is inverted. As a consequence the singular elements are not conform. The coincidence between displacements of adjacent elements is only achieved at nodal points, even for the generalized displacements corresponding to K_1, K_2 and K_3.

2.5 - Stiffness matrix computation

The stiffness matrix corresponding to the natural parameters is computed through volume integration. This implies a precise definition of the integration domain, i.e., of the physical boundaries of the element. The peripherical faces are defined by the coordinates of their 8 nodes, and by the isoparametric shape functions.

The other faces are assumed to the plane. The deformations of the element are the sum of the deformations of a non-cracked element, and of the deformations corresponding to the singular solutions.

The first ones are treated according to the usual isoparametric method ; the natural parameters are then the nodal point displacements. The singular solutions are considererd as shape functions for the second type of deformations. Theses solutions are obtained by exact limited developments for straight crack fronts, and series expansion for circular fronts.

They are classified according to the 3 modes, and 5 solutions are considered within the program, for each mode.

Preliminary trials have shown the danger of under-parametrisation, since artificial displacements can be introduced, which correspond to zero strain-energy. Therefore, the number of natural and final parameters was kept equal.

For the 20 nodes element, it is then necessary to use 5 singular solutions corresponding to the mode 1 : there are 20 x 30 = 60 final parameters corresponding to the nodal displacements of the elements, and two final parameters corresponding to the values of K_1 at each extremity of the crack front. On the other hand, because the element can only be used when the crack surface corresponds to a plane of symmetry, the displacements of nodes 1, 2 and 3, perpendicular to the crack, do not count as natural parameters. In order to compensate, it is then necessary to take into account 5 singular solutions, and the 5 corresponding parameters.

For the 35 nodes element, 15 singular solutions have to be taken into account ; i.e. 5 for each mode : the natural displacements of points 1 and 31, 2 and 32 and 3 and 33 are equal, 9 natural parameters are therefore substracted. In the other hand, the values of K_1, K_2 and K_3 at each extremity of the crack front must be added as final parameters. Hence, 15 additional natural parameters are needed.

The computation, of the natural stiffness matrix can be performed using Gauss method, in the following way : subroutines give, at any point, using the isoparametric and singular shape functions, the displacements and deformations, for each natural solution. The deformation can increase as $r^{-\frac{1}{2}}$, when approaching the crack front. In order to take this singularity into account, the integration is performed using cylindrical or torical coordinates. It is first performed on the **R** axis, starting from the crack front, up to the element surface. This method finaly leads to surface integrals. If the lateral faces are perpendicular to the crack front, there is only integration over the 3 or 6 peripherical faces. Using the relations already mentionned between natural and final parameters, the final stiffness matrix can be computed.

2.6 - Influence of a pressure on the crack lips

In order to take into account the influence of a pressure on the crack lips, a vector is computed, which dimension is equal to the number of final parameters of the elements. This vector is then incorporated in the second member of the final system, which solution leads to the "displacements".

This vector is first computed for the natural parameters. A unitary pressure is supposed to be applied on the crack lips. It is then necessary to determine the induced potential energy, for each elementary singular solution. This is achieved through integration of the displacement perpendicular to the crack, over its lips, for each elementary solution (only singular solutions can lead to a non zero result). The final vector is obtained using the matrix already mentionned in chapter 2.4.

2.7 - Implementation within the CEASEMT Finite Element System

The 20 nodes and 35 nodes singular elements have been implemented into the 3D code BILBO of the CEASEMT Finite Element System [5].

3. Validation and Application

In order to validate the elements, elementary cases where values of K_1, K_2 and K_3 can be analytically computed [6] were first considered :

- Cylindrical bar, including a central circular crack, subjected to uniaxial tension.
- Plate including a central crack subjected to uniaxial tension.
- Plate including a central crack with a unit pressure applied on the crack lips.
- Plate including a central crack, with unit shear applied on the crack lips.
- Cylindrical bar including a circumferential crack, subjected to torsion.

In each of the above cases, the values of the stress intensity factors given by the singular elements coïncide within a few per cent with the analytical values.

The more industrial case of a part through semi-elliptical crack, at the inner wall of a pressurized cylinder was finaly considered for a benchmark computation. The mesh is represented by figures 3 and 4. It can be noticed that the elements are relatively large in the vicinity of the discontinuity. The computation allowed to determine the variation of K_1 along the crack front. The results were in good agreement with values obtained with other methods, while the computation cost was relatively low.

4. Conclusion

Two singular finite elements (20 nodes and 35 nodes) were developped and implemented in the BILBO code of the CEASEMT Finite Element System, for linear fracture mechanics applications. Exacts solutions taking into account the singularities in the vicinity of the crack front have been incorporated in their formulation. Under parametrization has been avoided. Elementary cases as well as a more realistic application of these elements gave good results. Industrial cases can now be considered at reasonnable computer cost, since the necessity for mesh refinement in the vicinity of the crack is reduced, while values of K_1, K_2 and K_3 remain accurate.

References

[1] H.D. BUI
Mécanique de la rupture fragile
Mason

[2] O.C. ZIENKIEWICZ
The Finite Element Method
3rd édition, 1977
Mc Graw Hill

[3] F. DUDON - P. LAFORE
Mécanique de la rupture
Solution U + M grad div U = 0, avec contraintes nulles sur un demi-plan
Rapport DEMT/SMTS/BAMS/81-68

[4] P. LAFORE
Mecanique de la rupture
Determination des solutions singulieres exactes pour l'elaboration d'un element special
Rapport DEMT/SMTS/LAMS/83-16

[5] F. JEANPIERRE and al
CEASEMT - System of Finite Element Computer Programs.
IAEA/IWGFR Specialists Meeting of Hight Temperature Structural Design Technology of CMFBRs - 1976, Southampton-Pa.

[6] D.D. ROOKE and D.J. CARTWRIGHT
Stress Intensity Factors
LONDON - "Her Majesty's Stationery Office"

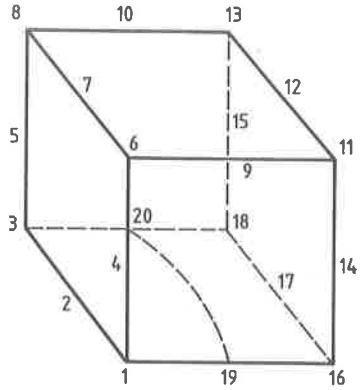

Fig. 1 - 20 MODES SINGULAR ELEMENT

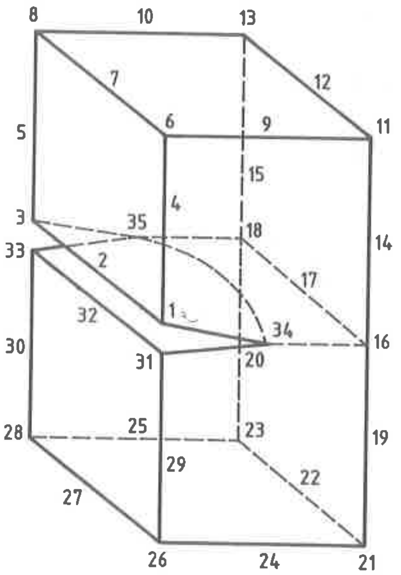

Fig. 2 - 35 MODES SINGULAR ELEMENT

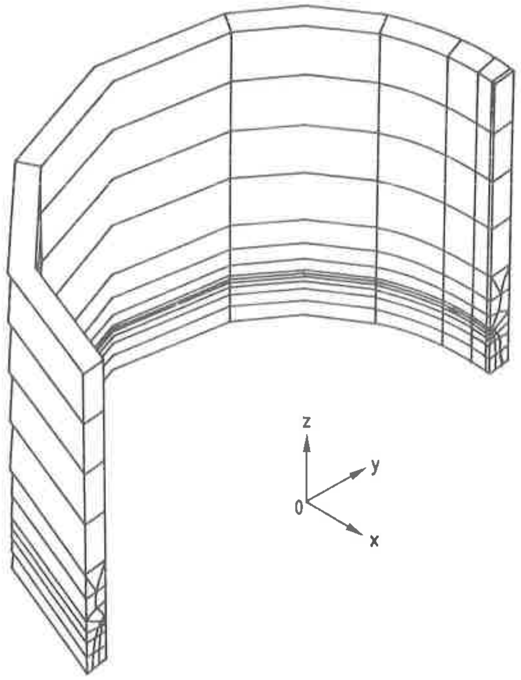

Fig. 3 - MESH USED FOR THE CASE OF A PART THROUGHT SEMI-ELLIPTICAL CRACK AT THE INNER WALL OF A PRESSURIZED CYLINDER

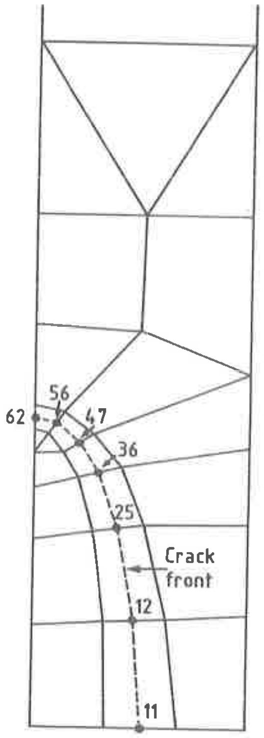

Fig. 4 - MESH REFINEMENT IN THE VICINITY OF THE CRACK