(APPLICATION OF THE FINITE ELEMENT METHOD FOR THE SAFETY EVALUATION OF REACTOR COMPONENTS)

M. MIKSCH, W. SCHMITT

Kraftwerk Union, Reaktortechnik, D-8520 Erlangen 2, Germany

SUMMARY

The finite element method is not only a powerful tool for the calculation of stresses and displacements in any structural component, but it is as well very useful for discussing several simple analytical methods. This will be shown by two examples:

1. *Calculation of hoop stresses due to temperature changes in a long hollow cylinder* (Young's modulus E and coefficient of thermal expansion α are temperature-dependent).

   For the calculation of stress-intensity-factors with analytical methods it is necessary to compute the thermal stresses normal to the plane of the crack. For constant E and α the displacements and stresses can be computed in a simple way. For E and α varying with temperature the differential equations for the radial displacements cannot be solved in a closed form. However, two different ways for an approximative analytical formulation seemed to be possible, but lead to different results. So two numerical methods, a finite element analysis and the application of a shell method have to show the accuracy of the analytical solution. In the range of application, both methods agree very well with one of the analytical methods.

2. *Calculation of the leakage area in a nuclear pressure vessel*

   The displacements of the crack surface for a given crack length (through crack) and from that the leakage area can be computed following the methods of linear elastic fracture mechanics with corrections for the platic zone. Since the presently available finite element codes do not include the effects of large strains, the adequate treatment of the crack-tip region is very difficult. A way to avoid these difficulties by an iterative elastic-plastic finite element analysis is discussed. It can be demonstrated that this method leads to conservative results.

Application of the Finite Element Method for the Safety Evaluation of Reactor Components

We consider an infinitely long hollow cylinder (pipe, pressure vessel) under internal pressure, having a through-crack of finite length 2b in the axial direction. The cylinder is subjected to an internal pressure p which causes a hoop stress σ_t = pr/s which comes close to the yield stress of the material. For safety evaluation it is important to know the leakage area of this crack (Figure 1).

To simplify the problem, a 2-D finite element analysis is chosen by taking into account the two planes of symmetry π_1 and π_2 and developing the remaining quarter of the pipe into a plane. The two planes of symmetry are defined as follows (see Figure 1): Plane π_1 goes through the centerline of the crack and includes the axis of the cylinder (π_1 is defined by the points A, B C in Figure 1). Plane π_2 is perpendicular to the axis of the cylinder and includes the center point of the crack (defined by points D, E, F in Figure 1).

As the wall thickness s is small compared to the radius R, plane stress conditions are assumed. Figure 2 gives the geometry and loading of the 2-D model which was finally chosen, the incremental plastic analysis was completed with the MARC-program using the plane stress quadrilateral element. The boundary conditions have to represent the lines of symmetry and the loading.

The line $\overline{AEB}$ in Figure 2 represents the line $\overline{DC}$ in Figure 1, and is a line of symmetry, therefore all nodal points lying between E and B have prescribed displacements v = 0. For the same reason all nodes on line $\overline{AD}$ have displacements u = 0.

The line $\overline{CD}$ represents the intersection of the plane π_1 with the cylinder wall opposite to the crack. For this line two different assumptions are made:

a) The cylinder is not allowed to buckle, the line DC remains straight, i.e. the v-displacements of all nodes lying on this line are the same (TYING-option in MARC).

b) This line is a line of constant tangential stress σ_t. This assumption is conservative, since the opening of the crack will decrease the hoop-stress above the crack.

The crack itself is simulated by force free nodes on line $\overline{AE}$. Since the applied tensile stress σ_t is very close to the yield stress σ_y (σ_t/σ_y= 6/7), from the fracture mechanics equations a comparatively large crack tip opening displacement δ_t has to be expected. The usually applied FE model for a crack, however, restricts δ_t to zero. To overcome this discrepancy, two different methods are discussed:

a) For the crack tip the focused isoparametric quadrilateral element is used.. This means that two nodes of the element have the same coordinates (crack-tip), but are completely independent degrees of freedom, so a 1/r-singularity in the strains is achieved, see refs. 1 and 2.

b) The nodes in the ligament $\overline{EB}$ in Figure 2 are successively released as soon as the adjacent elements yield. To explain this procedure, we look at an element (number 2 in Figure 3) which lies in the plastic zone near the crack tip, but does not have points on the crack surface or the ligament. The v-displacement of note 4 can be written as:

(Figure 3)

In reality, the strains near the crack tip exhibit a singularity of the order 1/r, whereas the used isoparametric elements are constant strain elements. Figure 4 gives an idea of the difference in the displacements.

The displacements are dependent on the size of the shaded areas below the curves ε v(1') in Figure 4. From this it is obvious that the commonly used method (fixing the crack tip opening displacement to zero and using elements which do not exhibit a strain-singularity) gives too small crack opening displacements, at least near the crack tip.

However, the contribution of element 1 to the stiffness of the total structure can be replaced by the stresses acting on the intersection lines to neighbouring elements. Thus, releasing node 1 after yielding of element 1 while the reaction forces accumulated in previous load increments still remain does not change the real stiffness of the structure , but allows a significantly larger v-displacement of nodes 1, 4, 6 etc. The v-displacements of these nodes obtained in this way are larger than in reality, because the restricting forces are underestimated as additional load increments do not increase these forces (the effect of strain-hardening are neglected).

Finite Element Analysis and Results:

To study the different approaches to this problem, a pipe was specified as follows:

Radius	$R = 445$ mm
Wall Thickness	$s = 44.5$ mm
Internal Pressure	$p = 30\ N/mm^2$
Half Crack Length	$b = 500$ mm
Yield Stress	$\sigma_y = 350\ N/mm^2$

For the given radius and wall thickness the pressure p would cause a nominal hoop stress of $\sigma_t = 300\ N/mm^2$ in an intact pipe. The length of the sides $\overline{AB}$ and $\overline{CD}$ in Figure 2 was taken as 5000 mm.

The analysis was completed with two different FE-meshes: a fine mesh with 189 elements and 192 nodes designed for performing method b and a coarse mesh with 73 elements and 85 nodes designed for the focused isoparametric elements (Figure 5 and 6 show both meshes). Both meshes were also used for the conventional method. The results for the crack opening displacements and the area differed very much with the assumed restrictions on line CD in Figure 2: keeping line CD straight (condition a) leads to significantly smaller crack opening displacements as are obtained with condition b, where general yielding above the crack occured before the full load could be applied.

The results for condition a are shown in Figure 7 and 8, Figure 7 showing the crack opening displacements for full load ($\sigma_t = 300\ N/mm^2$) obtained in 4 different ways as specified, Figure 8 showing the crack opening areas for different load increments. The coarse mesh gives smaller displacements ($\sim 5\%$). The influence of the different treatments of the crack tip (conventional, a, b) are not significant in this case.

For condition b, again the influence of the different meshes is obvious, whereas now the results differ significantly depending on the treating of the crack tip. The load-incrementations have been stopped after general yielding was observed, so in some cases no displacements for full load can be shown.

Figure 9 shows the crack opening areas vs. load for this case. It is remarkable that the data obtained with method b (fine mesh, releasing nodes) agree well with those obtained by using the focused isoparametric element at the crack tip.

Figure 10 combines all results for the crack opening areas obtained with different methods.

REFERENCES

1. Levy, N., Marcal, P.V., Ostergren, W.J. and Rice, J.R. (1971), "Small Scale Yielding Near a Crack in Plane Strain: A Finite Element Analysis", Int'l J.Fracture Mech., 7, pp. 143 - 156.

2. Tracey, D.M., "On the Fracture Mechanics Analysis of Elastic-Plastic Materials Using the Finite Element Method, " PHD-Thesis, Brown University, Providence, R.I. 1973.

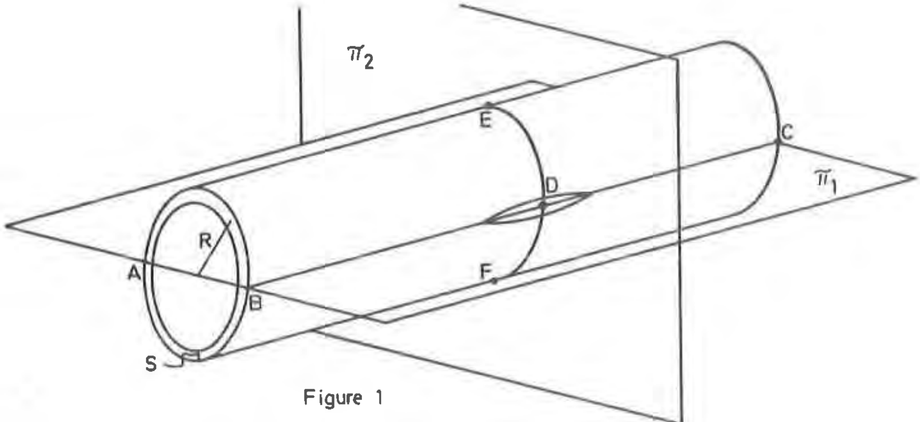


Figure 1

Model1 of the opening area

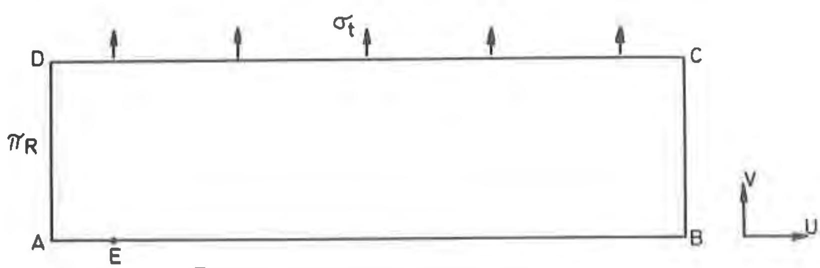
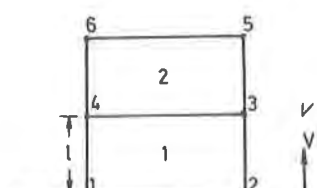


Figure 2 Boundary conditions



$$v = \int_0^l \varepsilon_v \, dl' + const.$$

Figure 3 Element in the region near the crack tip

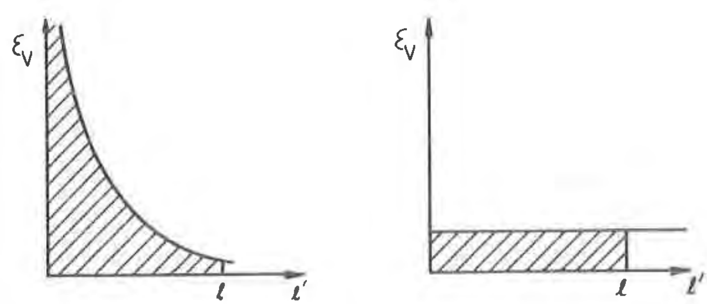


Figure 4 Displacements due to different elementproperties

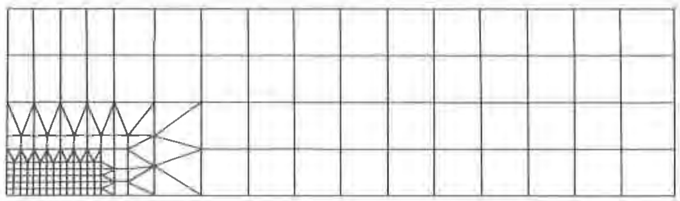


Figure 5 COARSE MESH
SCALE = 769.23 DATA UNITS/INCH

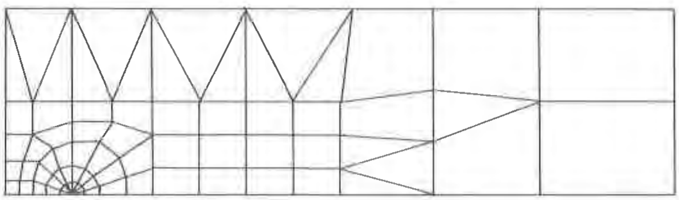


Figure 6 FINE MESH
SCALE = 769.23 DATA UNITS/INCH

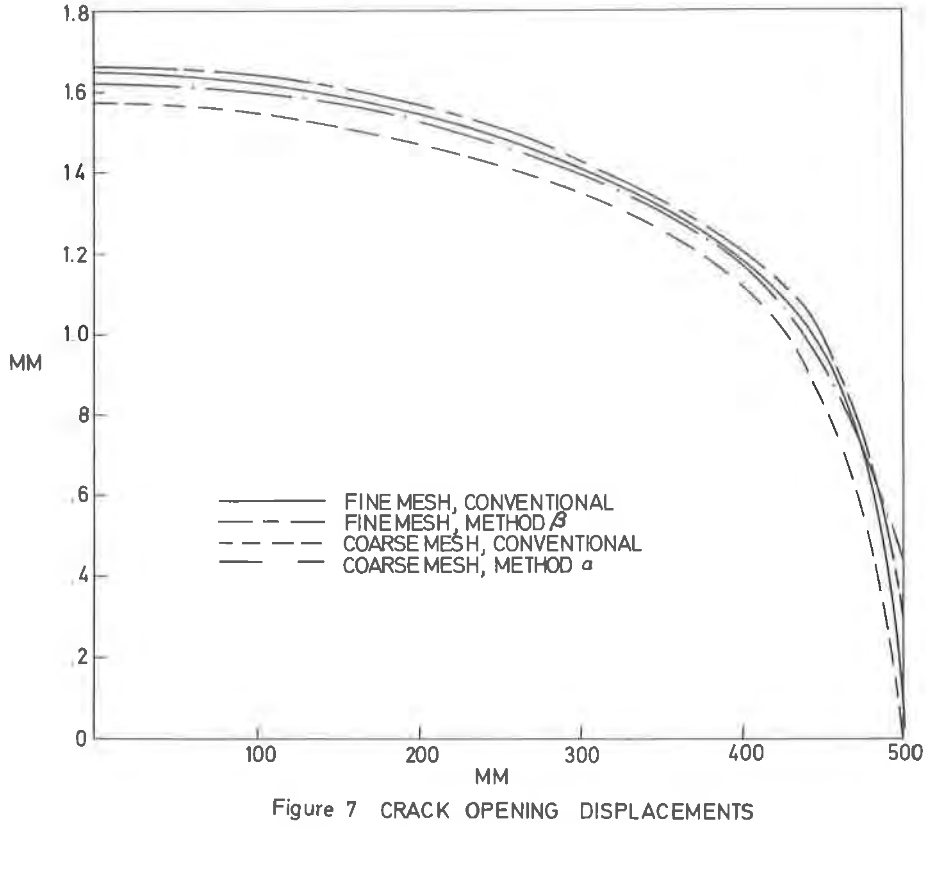


Figure 7 CRACK OPENING DISPLACEMENTS

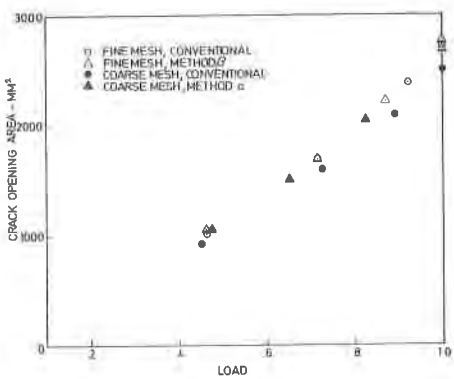


Figure 8 CRACK OPENING AREAS FOR BOUNDARY CONDITIONS a

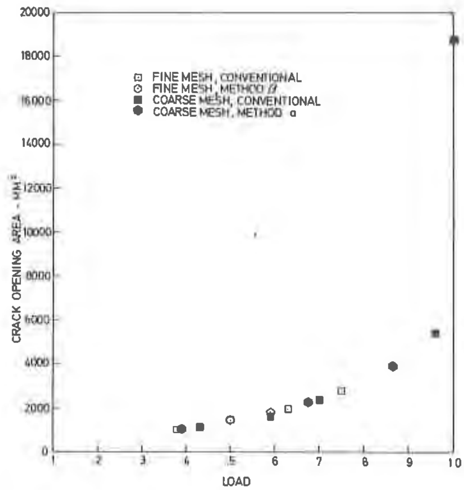


Figure 9 CRACK OPENING AREAS FOR BOUNDARY CONDITIONS b

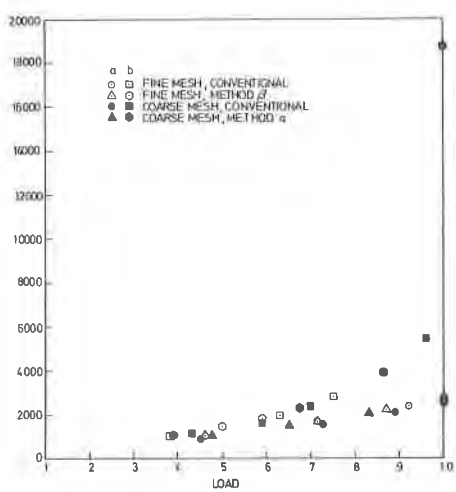


Figure 10 CRACK OPENING AREAS