ESIS RECOMMENDATIONS FOR USE OF FINITE ELEMENT METHOD IN FRACTURE MECHANICS

A. Sedmak*, N. Savović*, M. Pavišić*

ESIS recommendations for use of finite element method in fracture mechanics are presented. These recommendations are based mainly on the experience gathered during round robins held in Europe, (2,3) and meetings of Working Party on Numerical Fracture Mechanics. The recommendations concerned mainly finite element mesh, solution procedures, material law, type of analysis, crack tip modeling, fracture mechanics parameters and the reference solution.

INTRODUCTION

Technical Committee 8 (formerly Task Group for Elastic-Plastic Fracture Mechanics, Working Party on Numerical Fracture Mechanics) has formulated the recommendations for use of finite element method in fracture mechanics, published in ESIS Newsletter No 15, (1). These recommendations are based mainly on the experience gathered during round robins held in Europe under the charge of L.H.Larsson (2,3) and meetings organized in Freiburg in last three years.

In this paper more detailed analysis is given, including some additional results. The paper is organized in the same way as recommendations, emphasizing the problems of finite element mesh, solution procedures, material law, type of analysis, crack tip modeling, fracture mechanics parameters and the reference solution.

FINITE ELEMENT MESH

- If "new" problems are investigated, a convergence study with variation of mesh refinement is advisable. "Starting" point in such a study can be mesh similar to the one which is shown in Fig. 1.
- Isoparametric elements with quadratic shape function (8-noded for 2D, 20-noded for 3D) are recommended. Constant strain triangles are also reliable (in global sense), but not capable of crack blunting modeling. The 9-node u/p-element with three degrees-of-freedom (DOF) for hydrostatic pressure (3x3 Gauss integration) give practically identical results as standard 8-node elements, (4). The 8-node u/p elements with one DOF for hydrostatic pressure (default for 8-node elements in ADINA) cannot be recommended (4).

---

* Faculty of Mechanical Engineering, University of Belgrade

- The elements should be rectangular with a side ratio close to 1 in the regions of high strain gradients. Skewed elements should be avoided or, if unavoidable, integrated by 3x3 points.
- In transition region from small to larger elements the side of later one should not be larger more than twice the smaller side of neighboring elements.

SOLUTION PROCEDURE

- In the case of non-linear problems stiffness reformation and equilibrium iteration at any time step is recommended, as well as a convergence study concerning load/time step increments. Usually one increment should be done with not more than 10 iterations.
- There is a good experience with reduced integration (2x2) for the standard 8-node isoparametric elements. For special cases (large element deformations, curved, skewed or very fine elements) 3x3 integration is necessary.
- Nodal point loads often leads to inaccurate deformations.

MATERIAL LAW

- The material law should match the experimental stress-strain curve as closely as possible.
- Use the technical (engineering) stress-strain curve (related to the original cross-section) if material nonlinearities only (MNO) or in addition large deformations (Total Lagrange formulation - TL) are considered. If large strains and large deformations are taken into account by updating the geometry (Updated Lagrange formulation, UL), use the true stress-strain curve.

TYPE OF ANALYSIS

- 3D analysis gives accurate analysis (2D plane stress solution gives too soft, 2D plane strain too stiff results).
- In case of new problems first perform a simple analysis with MNO, and after that (if necessary) an analysis which considers the effects of large deformations (TL) and large strains (UL). Usually MNO gives satisfying results, specially if only global parameters, like J integral, are required.

CRACK TIP MODELING

- For elastic analysis use collapsed isoparametric, triangle crack tip elements with one crack-tip node and quarter-point midside nodes ($1/\sqrt{r}$ - singularity). For elastic-plastic analysis use triangle elements with independent crack-tip nodes and midside nodes lying in the middle ($1/r$ - singularity), see Figure 1.
- For simulation of crack growth rectangular elements without singularity can be used. It should be noted that the same element size as with collapsed elements leads to stiffer results.
- Do not constrain the midside nodes of crack-tip elements on straight lines.

FRACTURE MECHANICS PARAMETERS

- Use energy-parameter like J-integral, energy release rate G or stress intensity factor derived from G.
- Calculation of crack tip opening displacement is strongly mesh dependent. It should be done either by extrapolation of the crack face displacements or by means of 45^0 secant (see Figure 5).
- The J-integral could be path-dependent for large element deformations and small integration areas, as well as for non-proprotional loading and unloading.
- VCE is usually the better adapted technique for J evaluation as integration is performed over areas or volumes, and it can be easily extended to 3D structure.

REFERENCE SOLUTION

A reference solution for 2D fracture mechanics problem is the 3rd EGF numerical round robin. The problem is a CT specimen with the following data:

- Geometry: width 50 mm, height 30 mm, thickness 25.2 mm, crack length 29.78 mm - 2D plane strain analysis
- Material nonlinearity only (MNO) analysis
- Material law: Young's modulus 205 GPa, Yield stress 0.55 GPa, Hardening modulus 3 GPa, Poisson's ratio 0.3
- Isotropic hardening
- Independent crack tip nodes (1/r - singularity)
- Prescribed displacement at loading point (x,y)=(0,20) 1 mm or load F=2.55 kN

The Figures 2-6 show the finite element mesh, calculated force (load) and J-integral (for half of the specimen - J/2) versus load-line displacement, enlarged deformed mesh around crack tip with CTOD calculation, and crack opening stress radial distribution, respectively.

CONCLUSIONS

Although the recommendations for use of finite element method in fracture mechanics are well advanced now, many questions remain to be answered. Among others, we emphasize some crucial points:

- constraint effects, specially in regard to stable crack growth
- simulation of stable crack growth
- further development of energy concepts and calculation procedures for their evaluation (e.g. C^* and C_t integrals for creep, incremental integrals for non-proportional loading, influence of non-isothermal loading etc)

ACKNOWLEDGEMENTS

The work on this paper has been supported by the Scientific Fund and by the Fund for Technological Development of Republic of Serbia.

REFERENCES

1. ESIS TC8, "Recommendations for use of FEM in fracture mechanics", ESIS Newsletter No 15, 1991.
2. Larsson, L.H., "A Calculational Round Robin in Elastic-Plastic Fracture Mechanics", *Int.J.Pres.Ves.& Piping* **11** (1983) 207-228.
3. Larsson, L.H., "EGF Numerical Round Robins on EPFM", Proc. III Inter. Conf. NMFM, Swansea 1984, 775-814.
4. Brocks, W., Karakaya, V. and Noack, H.D, private communication

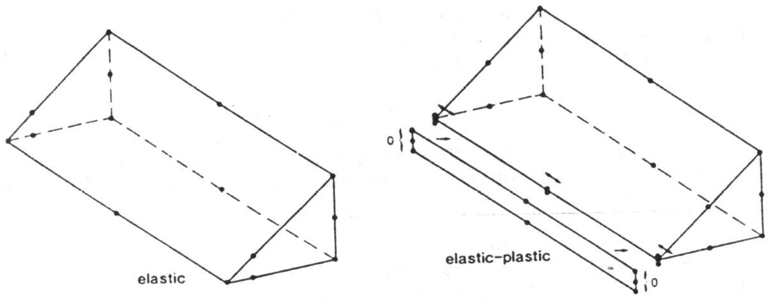

Fig. 1 Collapsed elements: a. elastic b. elastic-plastic analysis

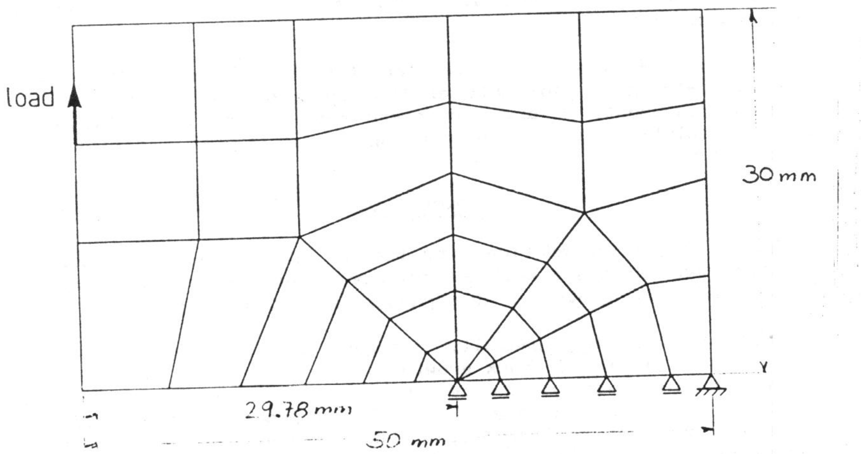

Fig. 2 Finite element mesh

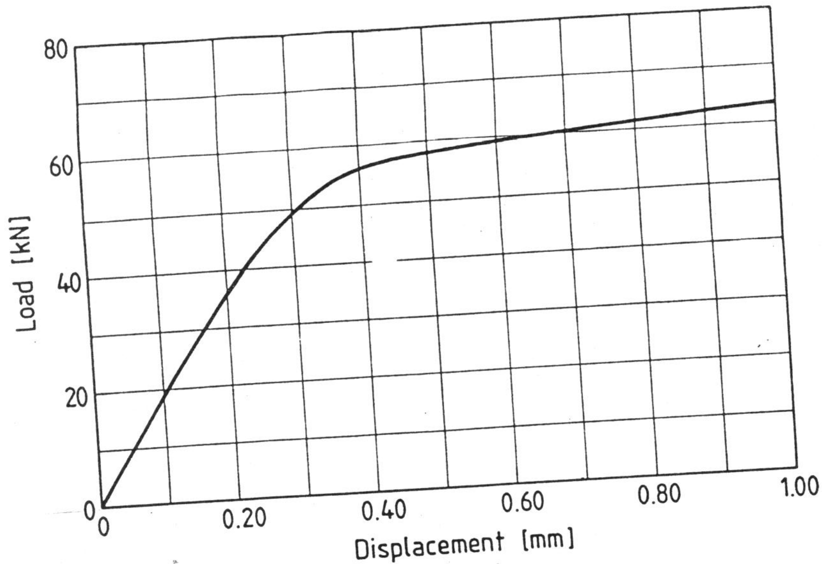

Fig. 3 Load versus load-line displacement

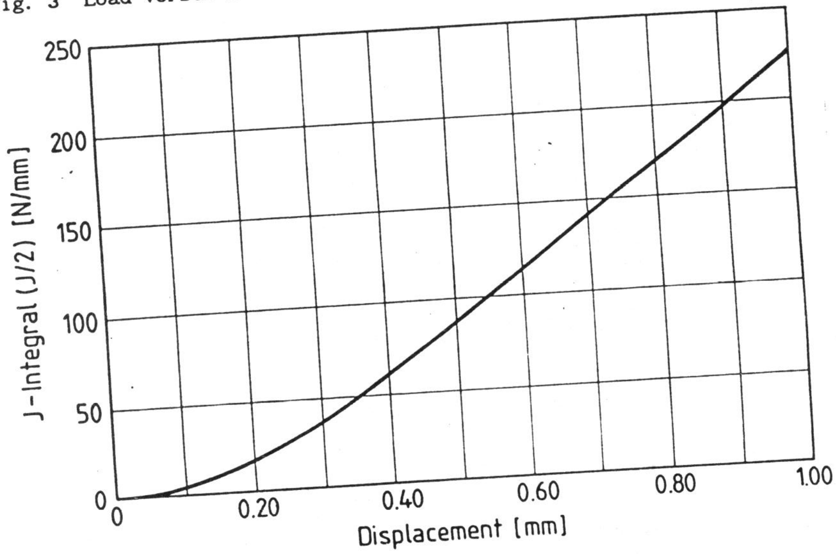

Fig. 4 J-integral versus load-line displacement

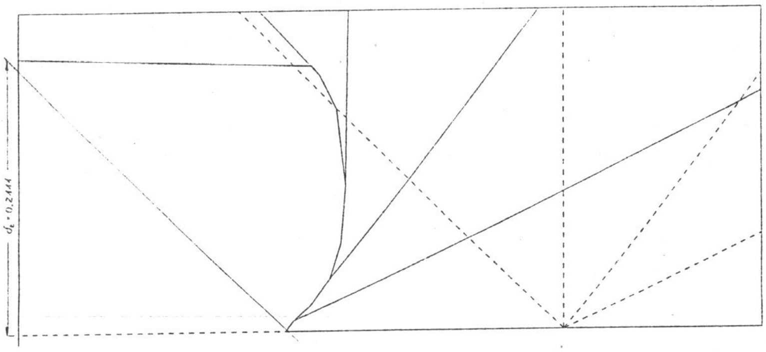

Fig. 5 Enlarged deformed crack tip elements and CTOD calculation

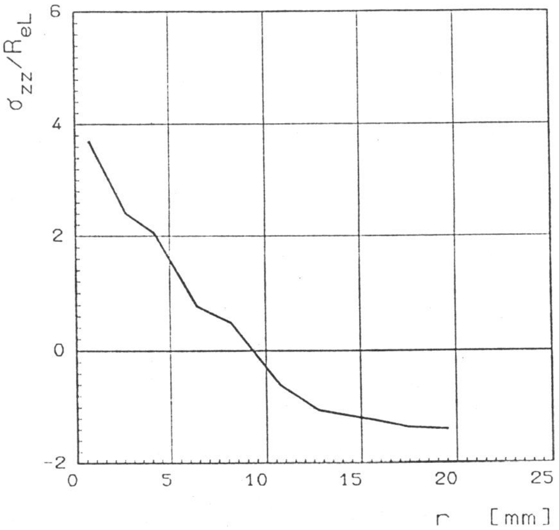

Fig. 6 Crack opening stress radial distribution