Development of an Automated Fracture Assessment System for Nuclear Structures

Timo P. J. MIKKOLA, Heikki RAIKO
Technical Research Centre of Finland, Vuorimiehentie, Finland

ABSTRACT

A program system for automated fracture mechanical analyses with three dimensional (3D) finite element (FE) models has been developed. The accuracy of the generated models is widely tested. The system is aimed at safety analyses of nuclear power plant components. Moreover, the results of the fracture mechanical FE-analyses can be implemented in an easy to use fracture assessment program based on the use of weight functions.

1 INTRODUCTION

The development of 3D FE-models of cracked structures has been automated [Leinonen et al. 1989]. The structures considered have mainly been pressurized components of nuclear power plants such as pressure vessels, pipes and pipe bends. The results are used in safety related fracture or fatigue analyses.

The quality of the automatically generated FE-mesh has been extensively tested comparing the results for the flawed plate under tension with other published results [Mikkola et al. 1990]. A large matrix of surface cracks has been analyzed and the accuracy has been found to be very good. For all the structures considered the same mesh design near the crack area is used and this should guarantee reasonable accuracy.

The present research activities are directed towards development of a quality procedure for the analyses. Several error estimates per element have been studied. The developed procedure is run after each analysis and the results are used in directing the mesh refinement. The FE-model refinement can be made by splitting the selected 3D brick elements into 8 new elements and using constraint equations.

The required effort in performing a full 3D FE-analysis with the developed program system has been substantially reduced. At the same time the quality of the analyses has been improved. The FE-analyses are made with the ADINA-program and a general purpose commercial FE-modeler program PATRAN is used in the mesh generation. Thus the system is well suited for industrial applications.

The automated system can be used for developing a database solution of the structure for an engineering fracture assessment program VTTSIF [Raiko et al. 1991]. Subsequent analyses with this program can be performed very easily and costly 3D FE-analyses can be avoided.

SMIRT 11 Transactions Vol. G (August 1991) Tokyo, Japan, © 1991

2 AUTOMATED FE-MODEL GENERATION

The FE-model is created with the ACR-program [Leinonen et al. 1989] in two phases. First a surface crack in a square plate is modeled and in the second phase this plate model is transformed into the final geometry. For pipes, elbows and pressure vessels the geometry transformation is a simple analytical function. For more complicated structures a very general transformation scheme is being developed using parametric representation of the geometry.

The initial plate FE-model is made by the ACRGEN-program module by writing an input file for the PATRAN-program [Users manual 1989], which is then used for generating FE-model for the plate. After the plate model is generated, the ACRMOD-program module is used for transforming the geometry, modifying the mesh and adding loads and boundary conditions.

The ACR-program is not intended for replacing an FE-modeler program. The PATRAN-program can be used for generating FE-model for other parts of the structure, adding loads and boundary conditions and eg. for mirroring the FE-model of the crack area to produce a more complete model of the crack.

2.1 Modeling of the crack area

The FE-mesh near the crack location developed by the ACR-program is well suited for the use of the virtual crack extension- (VCE) method in J-integral calculation, see Fig. 1. The elements form circular layers around the crack front, which are natural paths for J-integral evaluation. The nodes on the end-planes of these element layers are repositioned on planes perpendicular to the local crack front. The nodal points of the innermost element layer can be modified for producing $1/r$ - or $1/\sqrt{r}$ -singularity at the crack tip using the standard techniques for the isoparametric continuum elements. Also other modifications are performed.

Figure 1. Detail of the FE-mesh near the crack with the basic mesh density and a finer mesh density; an FE-model of a shallow crack with several element layers over thickness and separate element layer for cladding material.

2.2 Geometry transformations

The geometry transformations are used for mapping the initial plate to the final geometry of the structure. Transformations have been developed for cylindrical, toroidal and spherical geometries, see Fig. 2. These correspond to the geometries of several typical nuclear power plant components e.g. pipes, elbows and pressure vessels. The transformations do not need any information about the crack itself and thus they can be used for developing circumferential or axial part through cracks as well as inclined cracks or there may be several cracks in the model. The crack can also be transformed to the inner or outer surface of the final geometry.

The system is being extended to cracks in the nozzle area of a pressure vessel, see Fig. 2. This transformation is based on parametric description of the two surfaces of the structure. With this transformation, practically all thick walled geometries containing a surface crack can be modeled with the ACR-program.

Figure 2. Available transformation types in the ACR-program.

2.3 Generation of loads and boundary conditions

The ACR-program can generate loads and boundary conditions only for the model part formed of the initial plate geometry. The PATRAN-program can always be used for generating loads and boundary conditions for all parts of the FE-model including also the model near the crack.

3 ACCURACY CONTROL OF THE RESULTS

The modeling of the crack tip area has been developed to give accurate results for a broad range of crack geometries. The results for a plate with surface crack under tension from [Newman & Raju 1979] have been used as a basic test case when developing the FE-model.

3.1 Results for surface crack in a plate under tension

A large matrix of surface crack sizes in a plate under tension load has been analyzed [Mikkola et al. 1991] and the results are compared with the results of [Newman & Raju 1979]. The analyses were made with the 20-noded isoparametric solid element of the ADINA-program and the VTTVIRT-program [Talja 1989] is used for J-integral evaluation.

The FE-models were automatically created using the ACR-program. Typical results for the scaled stress intensity factor along the crack front are shown in Fig. 3 for the crack length ratio $a/c = 0.2$ for two crack depths. The stress intensity factor is scaled as

$$K = F \sigma_0 \sqrt{\frac{\pi a}{Q}} \quad (1)$$

where F is the shape factor, σ_0 is the nominal stress and Q is elliptical integral of the first kind. Also the FE-results and the results of an analytic formula given in [Newman & Raju 1979] are shown. The analytical formula is result of curve fitting to the FE-results with maximum difference of less than 5 % when compared to the maximum FE-result for corresponding crack size [Newman & Raju 1979]. The present results are in ± 5 % difference band everywhere along crack front for most of the crack sizes when referred to the FE-results.

The test matrix was analyzed using (3*3*3) and (2*2*2) integration and both $1/r$ - and $1/\sqrt{r}$ -singularity and the differences in the results were negligible.

Figure 3. Results for a plate with a surface crack under remote tension.

3.2 Local model refinement procedure

The h-version model refinement has been used for 2D-models together with an error measure to improve result accuracy [Mikkola et al. 1991]. The error measure was based on estimating the error in equilibrium equations for the FE-solution with equation

$$F_I^e = \int_{A_e} (\sigma_{ij}^{FE} n_j - t_i) dA \quad (2)$$

where t_i is true (unknown) traction force on element surface A_e , σ_{ij}^{FE} the finite-element approximation for the stresses and n_j the surface normal. This is equivalent to integrating the residual error in equilibrium equation over the element volume. The true traction is unknown, but the integral of it over the element surface is zero, while the structure has to be in balance.

The single edge cracked strip under tension was used as test case. The crack depths $a/t = 0.1, 0.2, 0.3, 0.5$ and 0.8 were analyzed. The unbalanced force was calculated for each element and minimized by mesh refinement. The 2D-analyses were made with the 8-noded isoparametric plane element and J-integrals were calculated from the FE-results.

The mesh refinement was made by sub-dividing the elements with highest unbalanced force. The results for the initial mesh and the mesh after refinement are shown in Fig. 4. The refined meshes are also shown for the three shallow crack models. The stress intensity factor is scaled with formula

$$K = H \sigma_0 \sqrt{\pi a} (1 - a/t)^{-1.5} \quad (3)$$

where σ_0 is nominal stress, a crack depth, t wall thickness and H the shape factor. The solution of [Tada, Paris, Irwin 1985] is used for comparison and it is drawn with the given accuracy limits of $\pm 0.5\%$.

Figure 4. Stress intensity factor results for a cracked strip with h-version mesh refinement.

The results for the two deep crack models, $a/t = 0.5$ and 0.8 were accurate already with the initial mesh and the error was large only for the shallow crack case with $a/t = 0.1$. For the three shallow cracks $a/t = 0.1, 0.2$ and 0.3 , the mesh was refined until the maximum unbalanced force occurred in the second circular layer of elements at the crack area. The elements in these two closest layers were not sub-divided.

At first, the unbalanced force weighted with the element strain energy was used to direct the

refinement. Only three and two refinements were required for the $a/t = 0.2$ and 0.3 cases, respectively, and the resulting meshes are shown in Fig. 4. For the shallow crack model with $a/t = 0.1$, mesh refinement was made also with unweighted unbalanced force. The final result was found with fewer steps without weighting. The resulting mesh refinement concentrated better to the stress concentration area as can be seen in Fig. 4. At first the stress intensity factor converged rapidly and then the rate of convergence slowed down. This indicated, that the mesh should also be refined at the crack tip area as was suggested by the unbalanced force.

CONCLUSIONS

The developed ACR-program significantly reduces the time and costs of performing a full 3D FE-analysis of flawed structures. At the same time the accuracy of the results has been increased. The system can easily be modified for new types of structures. An accuracy assessment procedure is being developed and results are promising. The error parameter was capable to direct the mesh refinement and yielded to accurate results. Further development is required before the error parameter can be used in accuracy assessment for other loading cases and geometries. The element sub-division procedure should be automated to 3D-models, too.

Acknowledgements - This work is a part of the Research Programme (Ra) of Technical Research Centre of Finland. The financial support of the two Finnish nuclear power companies (TVO and IVO) are gratefully acknowledged.

REFERENCES

- ADINA - A Finite Element Program for Automatic Dynamic Incremental Analysis. 1987. Report ARD 87-1, ADINA R&D, Inc., Watertown Massachusetts.
- Leinonen, M., Mikkola, T.P.J. 1989. The ACR-program for Automatic Finite Element Model Generation for Part Through Cracks. Espoo Finland: Technical Research Centre of Finland (VTT), Nuclear Engineering Laboratory, Research Reports 658: 80 p.
- Mikkola, T.P.J., Leinonen, M., Niemi, H., Talja, H. 1990. Automated Fracture Analysis with Three-Dimensional Finite Element Models. Proceedings of the 5th International Conference on Numerical Methods in Fracture Mechanics. Swansea Great Britain: Pineridge Press: pp. 545-556.
- Mikkola, T.P.J., Niemi, H. 1991. Quality Assurance for Fracture Mechanical Finite-Element Analyses. Paper presented in the 8th ADINA Conference in M.I.T on July 17.-19.1991, Cambridge Massachusetts.
- Newman, J.C., Raju, I.S. 1979. Analysis of Surface Cracks in Finite Plates Under Tension or Bending Loads. NASA-TP-1578. Hampton Virginia: NASA.
- PATRAN PLUS User manual. 1990. Release 2.5. Costa Mesa California: PDA Engineering.
- Raiko, H., Mikkola, T. P. J. 1991. Fracture Assessment Program System for Practical Engineering Applications. Proceedings of IV Suomen Mekaniikkapäivät, 5.-6.6.1991, Lappeenranta Finland: 8 p.
- Tada, H., Paris, P., Irwin, G. 1985. The Stress Analysis of Cracks Handbook, 2nd edition. St. Louis, Missouri: Paris Productions Inc.
- Talja, H. 1989. Elastic-Plastic Fracture Parameters and Their Calculation by Finite Element Method. Espoo Finland: Helsinki University of Technology, Licentiate's Thesis: 113 p. (In Finnish).