FINITE ELEMENT MODELING OF PRESSURE VESSEL WITH TWO INTERNAL SURFACE CRACK

E. Argoub[1], A. Sedmak[1], Z. Burzic[2] & G. Adziev[3]

[1] Faculty of Mechanical Engineering, University of Belgrade, Serbia and Montenegro
[2] Military Institute, Belgrade, Serbia and Montenegro
[3] Faculty of Mechanical Engineering, University of Skopje, Macedonia

ABSTRACT

This paper deals with the finite element modeling of the pressure vessel with two artificial internal surface cracks, one positioned in the vessel center and the other one close to the vessel cover. Cracks are made by electroerosion, with radius at the tip 0.1 mm. The pressure vessel has been experimentally investigated to determine strain distribution by suitably positioned strain gages along smooth portions of the path for J integral direct measurement, as the first phase of the structural integrity assessment procedure. In order to verify numerical procedure, different finite element models are used, one complex model with two cracks and two simple models, each with a single crack. Boundary conditions are identified as the most difficult problem in the finite element modeling process, requiring special attention. In order to verify the model validity, numerical and experimental results are compared. The equivalent (von Mises) stress and strain distribution indicated good agreement not only between the numerical and experimental results, but also between the results for different numerical models, proving that the simple finite element models can be used for a reliable prediction of strain and stress distribution. The J integral has been evaluated using the simplified procedure, based on the fact that the bending-traction term is negligible along the whole path for direct measurement, whereas the strain energy term can be neglected along the paths parallel to the crack plane. In this way the complicated integration along three dimensional path is reduced to the simple evaluation of two area integrals. Although not verified on the pressure vessel itslef, this procedure has been verified by similar investigation done on plates.

1 INTRODUCTION

The pressure vessel with two internal surface crack has been studied, one crack in the center of pressure vessel and the other in the upper side close to the edge, Fig. 1. The aim of this study has been to establish as simple as possible finite element model of the real structure. Toward this aim a complex finite element model of pressure vessel with two cracks should be designed and compared with two simple models, containing only one crack each. Besides this comparison, the experimental results will be used to verify numerical models. Finally, the finite element models will be used for J integral evaluation.

2 EXPERIMENTAL INVESTIGATION

The pressure vessel containing two inner surface cracks, one in the center (mid crack) and the other close to the cover (upper crack), is shown in Fig. 1. Both cracks are produced by electroerosion with the crack tip radius 0.1 mm, Argoub [1]. Pressure vessel has been instrumented with strain gages to determine strain distribution on the smooth side, Fig. 2. Manual water pump was used to pressurize the vessel. A personal computer collected the test data via multi channel data acquisitions. The results for the strain distribution are presented in Fig. 3 for mid crack and upper crack. The strain distribution is given along the smooth portion of a common path of J integral direct evalution, comprizing the cracked side of the vessel as well. The J integral direct evalution, explained in more details elsewhere, Read [2], is used for the structural integrity assessment procedure, which is the next step in this investigation as well.

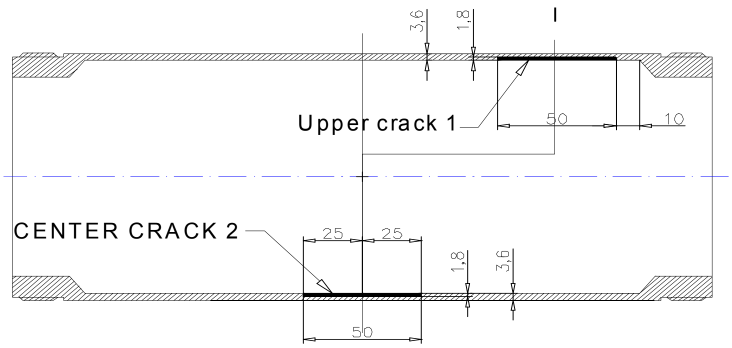

Figure 1. Pressure vessel dimensions

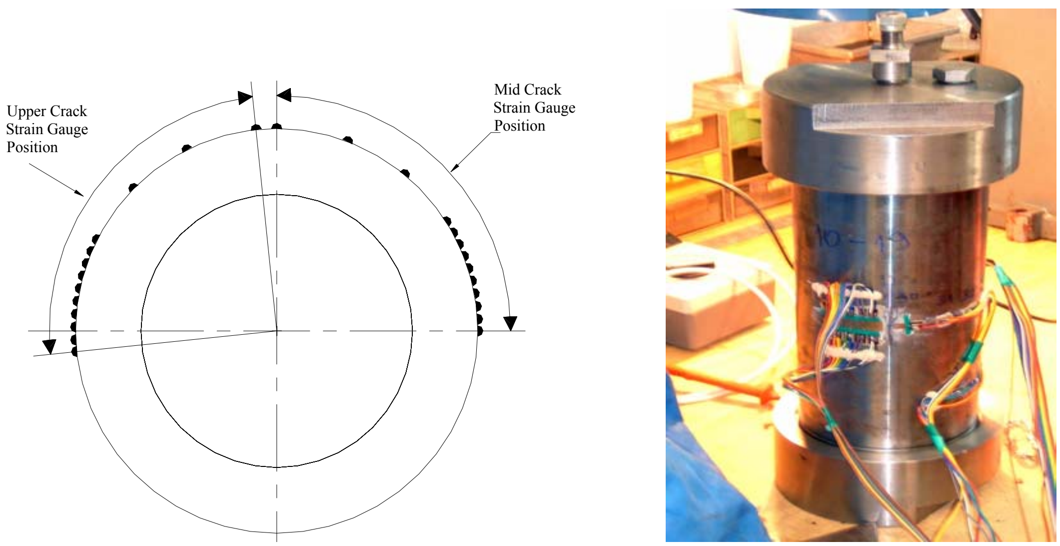

Figure 2. Strain gages distribution and pressure vessel with strain gages

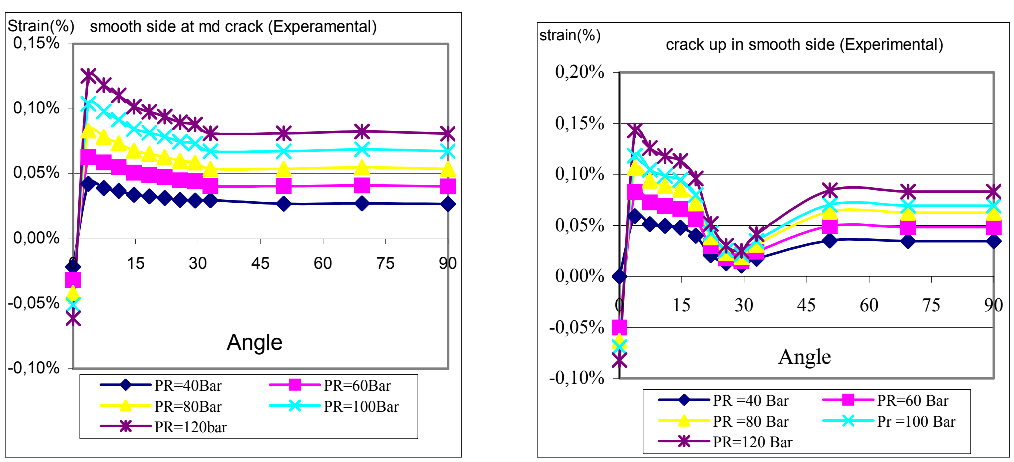

Figure 3. Strain distribution for smooth side for mid crack and upper crack

3 FINITE ELEMENT MODELING

Two different finite element models of pressure vessel have been used, as shown in Fig. 4 (1/8 of the vessel for the model with one crack) and Fig. 5 (1/4 of the vessel for the model with two cracks).

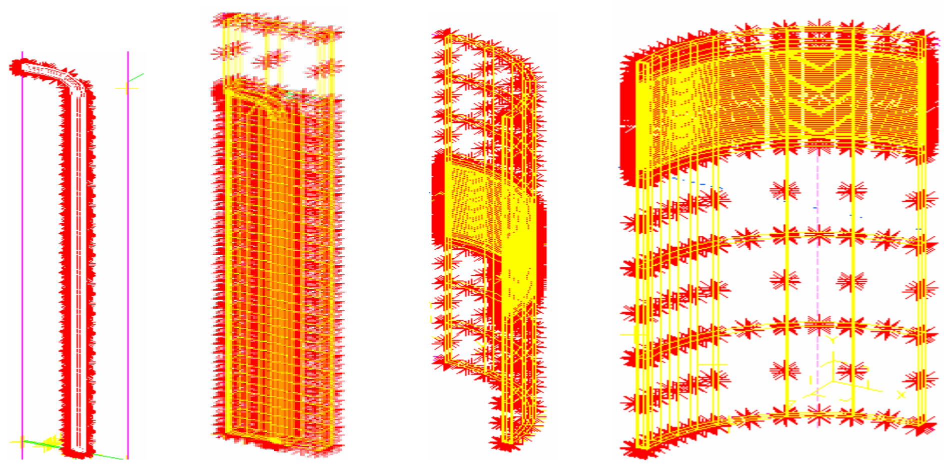

Figure 4. Construction of 3D mesh with singular elements (model with one crack)

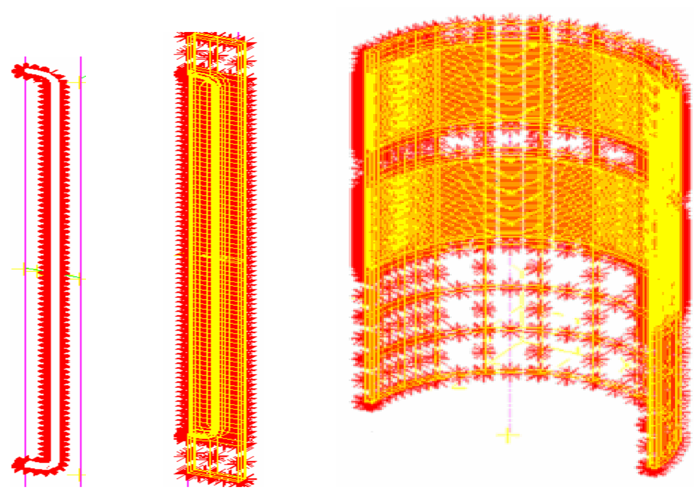

Figure 5. Construction of 3D mesh with singular elements (model with two crack)

In both models, after 3D singular elements formed the crack front, they are merged with the surrounding uniform mesh, consisting of hexaedric elements. For 3D problem boundary conditions are often the most difficult part of the problem. Here, the symmetry is taken into account and the appropriate boundary conditions defined as shown in Fig. 6 for two models with one crack.

The following material properties have been used: σ_Y=410 MPa, H'=6400 MPa, G=200 GPa, ν=0.3) for a QT steel. The stress-strain behaviour was modeled as bi-linear curve. More details about all aspects of finite element modeling is given by Argoub [1], including different variants of boundary conditions.

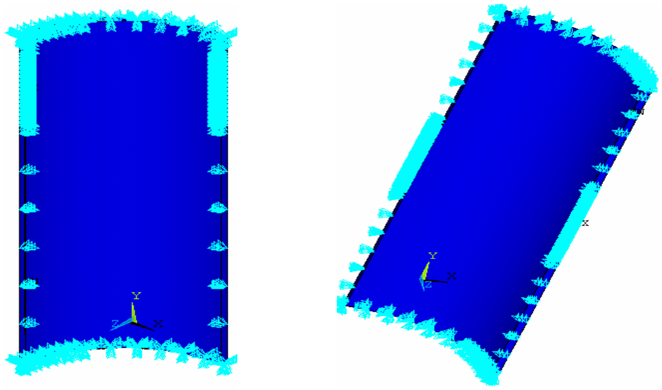

Figure 6. Boundary conditions for (a) upper crack, (b) mid crack

4 NUMERICAL RESULTS

The equivalent (von Mises) stress and strain distribution for both cracks are shown in Fig. 7 for both models. The strain distribution in the inner, cracked side of vessel is shown in Fig. 8 for both mid and upper crack, whereas the strain distribution in the outer, smooth side of vessel is shown in Fig. 9, together with the experimental results. One can see good agreement not only between the numerical and experimental results, but also between numerical results for two different models.

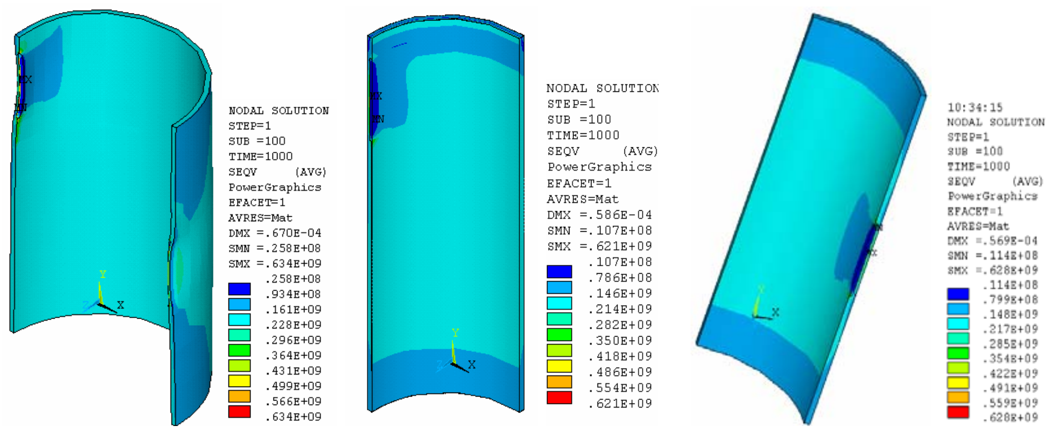

Figure 7. Equivalent stress (von Mises) for three model

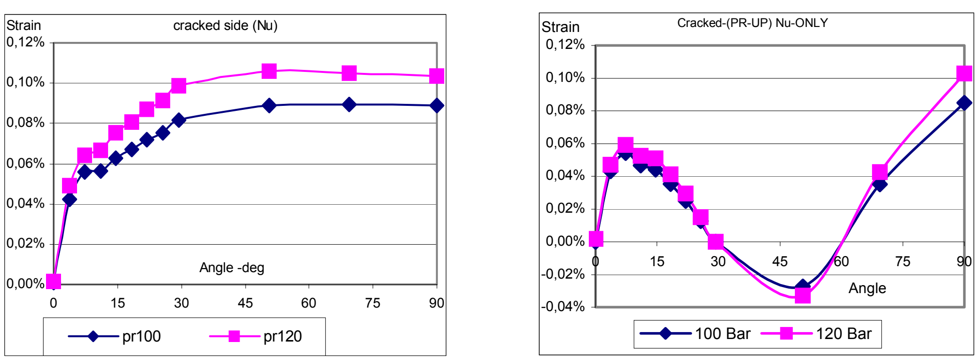

Figure 8. Strain distribution in cracked side for the mid crack and upper crack

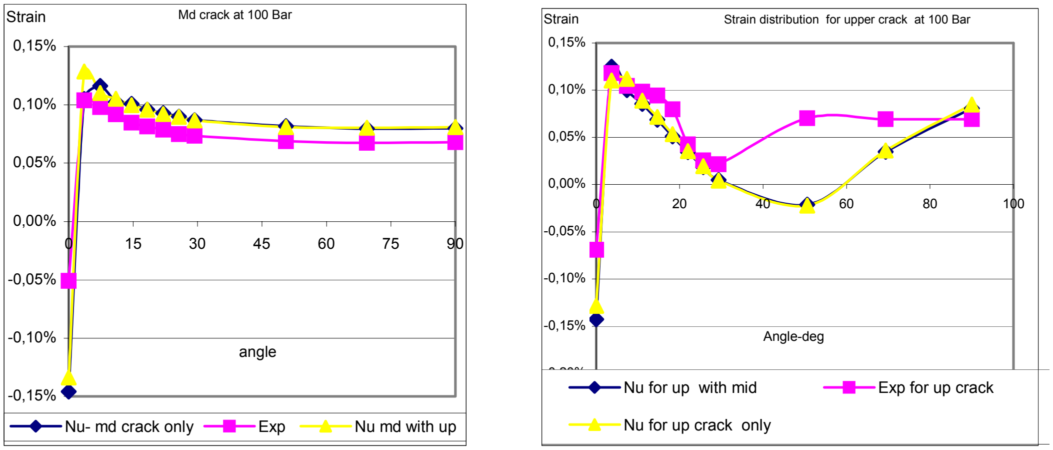

Figure 9. Comparison of strain distribution in smooth side (mid only and up only)

4.1. The J-INTEGRAL evaluation

The J integral evaluation is a complicated task, especially if line integration is applied for 3D problems. Anyhow, one should keep in mind the fact that the J integral, evaluated directly, can be presented as a sum of two terms, one being the consequence of the strain energy distribution, and the other one, being the consequence of combined effect of traction and bending:

$$J = \int W dy - \vec{T}\frac{\partial \vec{u}}{\partial x} ds \cdot$$

If J integral is calculated along the contour for its direct measurement, it is well known that the strain energy term exists only along paths normal to the crack plane, whereas the traction-bending term exists only along paths parallel to the crack plane, Read [2]. Furthermore, one should notice that the bending-traction term is practically zero for problems where bending is negligible. Pressurizing of a vessel is exactly this type of problem, providing the opportunity to present J integral as a difference between two area integrals. The results, obtained by this simple procedure are shown in Fig. 10 as J integral vs. pressure and in Fig. 11 as J intgeral vs. CMOD. Unfortunately, since no experimental results are yet available, the only verification of this procedure can be offered by Argoub [3], where the same procedure was applied to the similar investigation done on plates.

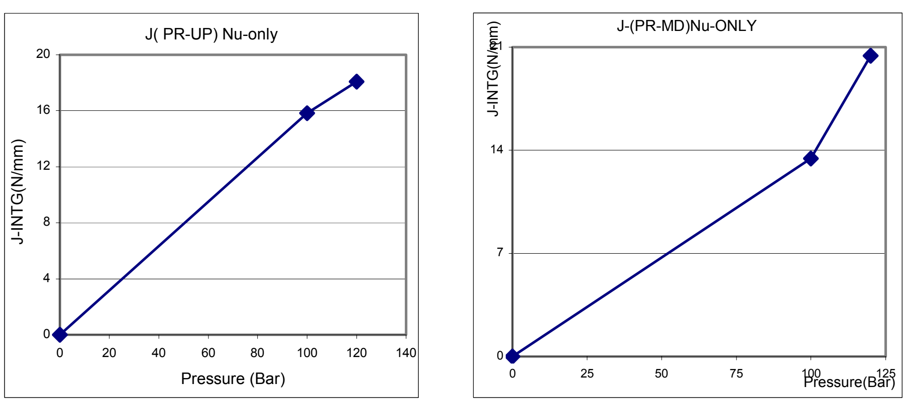

Figure 10. J integral vs. pressure

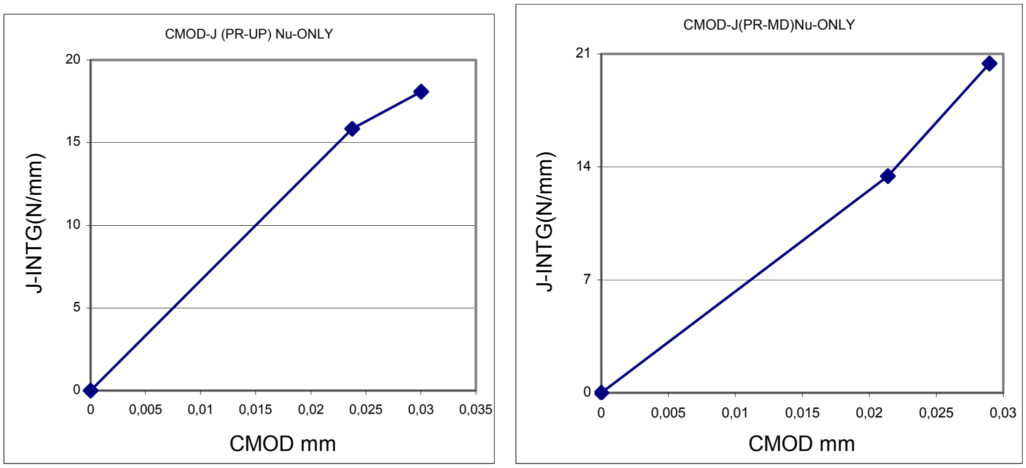

Figure 11. J integral vs. CMOD

5 CONCLUSIONS

Based on the presented results one can conclude that the capability of the finite element method to solve even the most complex problems has been clearly demonstrated. On the more specific base one can conclude the following:

- The finite element model of pressure vessel with one crack can be used as a reliable tool to solve complex problems like the pressure vessel with two cracks.
- The J integral can be efficiently evaluated using the simplified procedure, reducing integration along three dimensional path to the simple evaluation of two area integrals.

References

1. Argoub, E.O., *D.Sc. thesis*, Faculty of Mechanical Engineering, University of Belgrade, 2004
2. Read, R.B., *Experimental method for direct measurement of J-contour integral*, STP791, ASTM, Philadelphia, USA, pp. I-444, 1983
3.Argoub, E.O., Sedmak, A., *Numerical Simulation of Mismatching Behavior of Weldment with Surface Crack*, The First Conference on Computational Mechanics, Belgrade, 2004