The paper was presented at the Ninth Meeting
"New Trends in Fatigue and Fracture" (NT2F9)
Belgrade, Serbia, 12–14 October, 2009

Katarina Gerić[1], Aleksandar Sedmak[2]

NUMERICAL MODELLING OF CRACK TIP OPENING IN WELDED JOINT
NUMERIČKO MODELOVANJE OTVARANJA VRHA PRSLINE U ZAVARENOM SPOJU

Original scientific paper
UDC: 539.42:519.673
Paper received: 10.6.2010

Author's address:
[1)] University of Novi Sad, Faculty of Technical Sciences, ageric@ptt.rs
[2)] University of Belgrade, Faculty of Mechanical Engineering

Keywords
- numerical modelling
- crack
- heat-affected-zone (HAZ)
- simulation

Abstract

Welded joint is of heterogeneous structure with different microstructural regions. To determine the local behaviour of the region measuring crack opening, the problem is to locate the notch tip properly. Simulation of the welding is performed by heating specimens and programmed cooling on different temperatures so as to obtain the microstructure similar to the one in the heat affected zone.

Numerical model of stress–strain field around the crack tip based on the finite element method, enables to determine the load for the crack growth with satisfactory accuracy, as is verified by experimental results. The application of this model enables to avoid expensive experiments and instead use numerical simulation to determine the load level that causes crack growth.

Ključne reči
- numeričko modelovanje
- prslina
- zona uticaja toplote (HAZ)
- simulacija

Izvod

Zavareni spoj je heterogene strukture sa područjima različitih mikrostruktura. Da bi ispitali lokalno ponašanje područja prateći otvaranje prsline, problem je postaviti vrh zareza u željeni položaj. Izvedena je simulacija zavarivanja zagrevanjem i programiranim hlađenjem na različite temperature tako da se dobije mikrostruktura slična onoj u zoni uticaja toplote.

Numerički model naponsko-deformacijskog polja na vrhu prsline, zasnovan na metodi konačnih elemenata, omogućava da se dovoljno tačno dobije opterećenje za rast prsline, kao što je potvrđeno eksperimentalnim rezultatima. Primenom ovog modela mogu da se izbegnu skupi eksperimenti i umesto toga vrši numerička simulacija da se odredi vrednost opterećenja koje uzrokuje rast prsline.

INTRODUCTION

The welded joint and especially the heat affected zone (HAZ) is of heterogeneous microstructure, containing regions of markable differences in microstructure and mechanical properties. The continuous heterogeneity makes it difficult to define the position of crack critical microstructure. Determination of fracture mechanics parameters, like crack opening, in the welded joint is very important for structural integrity assessment, but it is difficult to locate properly the notch tip in a specimen. Applying welding simulation, by specimen heating and programmed cooling at different temperatures, it is possible to obtain a sufficiently large size of desired microstructure in HAZ, /1/.

Experimental investigation is expensive, and a numerical model of the stress and strain field ahead of the crack tip is developed to predict critical crack tip location in a welded joint, avoiding expensive metallographic testing. The basic idea is to compare the microstructure in welded joint samples and in specimens simulated at given temperature, in order to determine more accurately local mechanical properties of the material in HAZ, necessary for crack resistance evaluation, /2/.

The considered problem has an important practical implication regarding structural integrity in the manufacturing phase /3, 4/, as well as in the exploitation of welded structures, /5-8/.

EXPERIMENTAL PROCEDURE

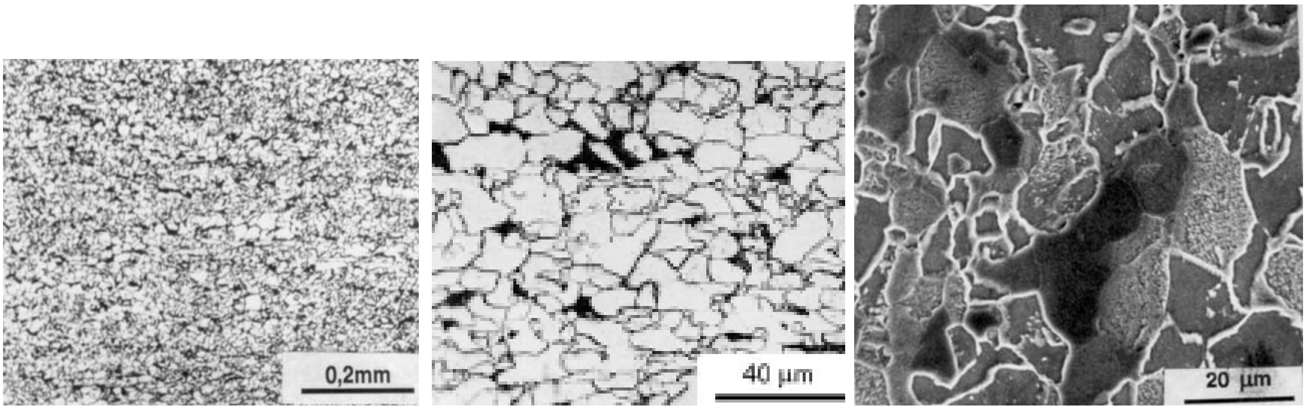
Testing was performed with thermomechanical controlled rolled steel, Nb and Ti microalloyed, of given chemical composition (Table 1) and mechanical properties (Table 2). The microstructure of parent steel is shown in Fig. 1.

Table 1. Chemical composition.
Tabela 1. Hemijski sastav

C	Nb	Cr	Ni	Mo	Ti
0.08	0.026	0.027	0.019	0.010	0.017

Table 2. Mechanical properties.
Tabela 2. Mehaničke osobine

Yield strength MPa	Ultimate strength MPa	Elongation %	Charpy toughness J	Hardness HV5
428	520	34.6	228.9	199



a. b. c.

Figure 1. Parent steel microstructure – optical microscopy (a, b) and scanning electron microscopy (SEM) (c).
Slika 1. Mikrostruktura osnovnog metala – optička mikroskopija (a, b) i skening elektronska mikroskopija (SEM) (c)

The microstructure of HAZ is simulated on Smitweld LS1402 device in the Institute of welding and materials testing, Timişoara (Romania). Samples 11 × 11 × 60 mm are heated to 1350, 1100, 950 and 850°C, typical for transformation to different HAZ microstructures (coarse grain–GZ, fine grain–FZ and partially transformed structures between points A_{c1} and A_c) and programmed cooled, in cooling time $\Delta t_{8/5} = 15$ s, Fig. 2. The microstructure of simulated samples is shown in Fig. 3a-d.

Welded samples are produced by manual arc welding (MAW) in 4 passes without preheating, due to small sample thickness, applying average heat input 1.5 kJ/mm.

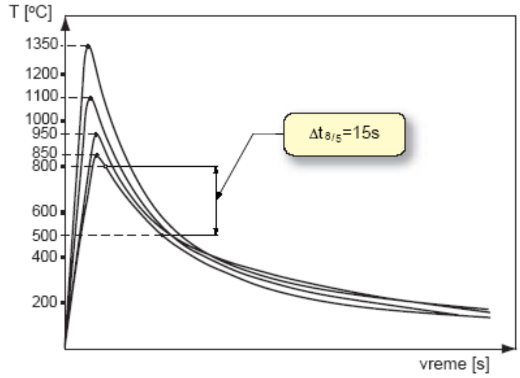


Figure 2. Simulation cycles.
Slika 2. Ciklusi simulacije

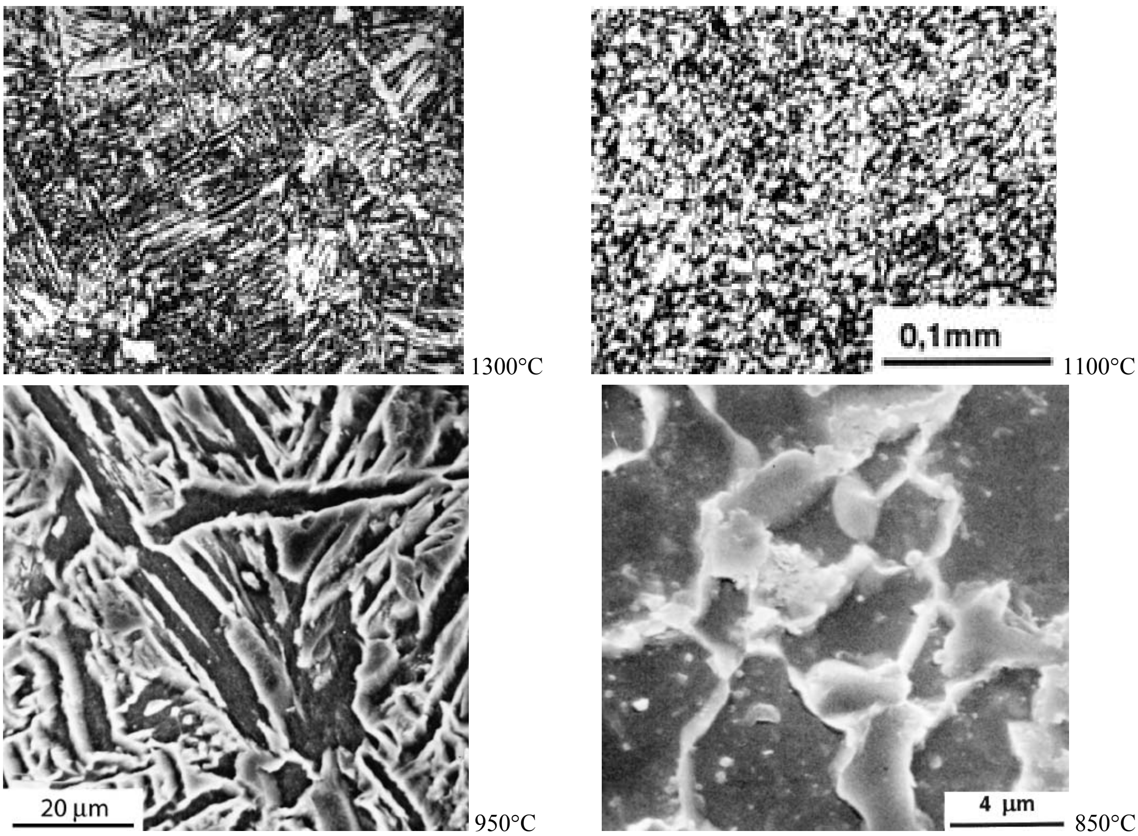


Figure 3. Microstructure simulated at different temperatures.
Slika 3. Mikrostruktura simulirana na različitim temperaturama

In welded joint specimens, notch tips are cut both on the fusion line (LS) and in locations in HAZ, closely corresponding to simulation temperatures, as shown by numbers 1–12 in Fig. 4. Position of crack tip, produced by specimen fatigue, is about 3 mm under the specimen surface.

Width of the heat affected zone is about 2.5 mm. Near the fusion line a coarse grain region is formed of bainite and ferrite microstructure, Fig. 4a (location GZ, 4), followed by a fine grain bainite–pearlite zone (location FZ, 6).

Microstructures of HAZ regions simulated at temperatures 1350, 1100, 950 and 850°C are presented in Fig. 5.

The results of crack tip opening displacement (CTOD) testing are presented in Fig. 6 for both types of specimens, i.e. taken from samples simulated at above given temperatures, and produced from welded joints with the crack tip located in a corresponding microstructure according to the locations in Fig. 4.

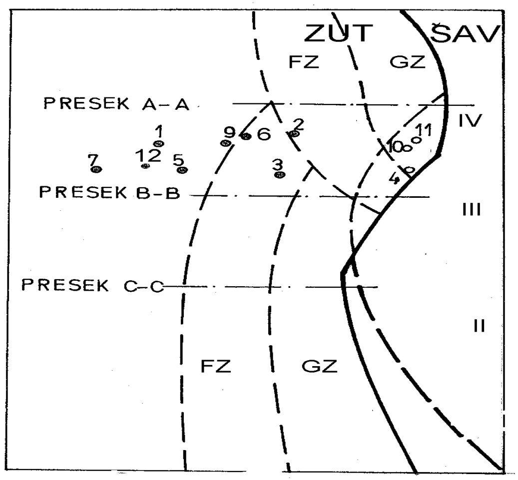


Figure 4. Locations of crack tip in the welded joint.
Slika 4. Položaji vrha prsline u zavarenom spoju

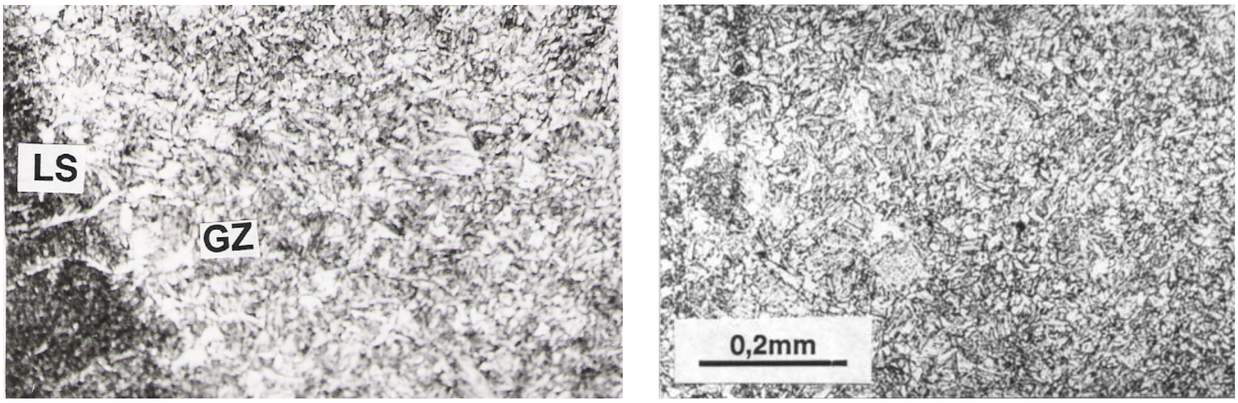


Figure 5. Microstructure in section A-A (see Fig. 4): coarse grain zone GZ near fusion line LS (a); fine grain zone (b).
Slika 5. Mikrostruktura u preseku A-A (vidi sl. 4): grubozrno područje GZ u blizini linije stapanja LS (a); finozrno područje (b)

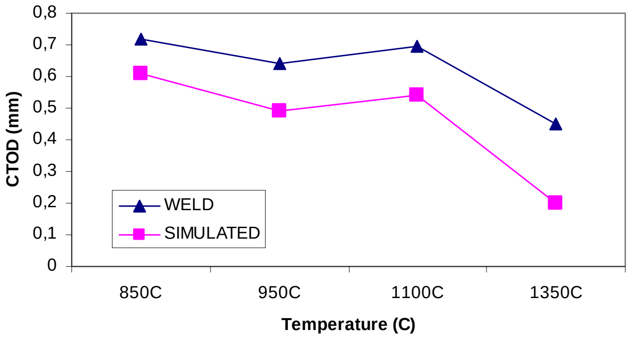


Figure 6. Crack tip opening displacement (CTOD) values for specimens of welded joint and simulated samples.
Slika 6. Vrednosti otvaranja vrha prsline (CTOD) za epruvete iz zavarenog spoja i iz simuliranih uzoraka

NUMERICAL MODELLING OF CRACK OPENING

Numerical analysis of crack opening displacement (COD) is performed using DPLAST programme which is a modification of the Owen and Fawkes' programme, /9-12/. This programme enables elastic-plastic analysis of solid bodies in plane stress and plane strain conditions, using the finite element method (FEM), /13, 14/. Two yielding criteria are inserted, Tresca and von Mises, as well as four algorithms for solving system equilibrium. The von Mises criteria, as proved to be the best for describing metal behaviour and the tangent stiffness algorithm with the best convergence properties are implemented, based on the well known Newton-Raphson iteration method.

FEM analysis is performed for two types of specimens: specimens simulated at different temperatures (1350°C – specimen S2; 1100°C – S4; 950°C – S5; 850°C – S6), and specimens from real welded joint with crack tip located in HAZ regions of different microstructure (specimens SZ1, SZ3, SZ4, SZ6 with locations of crack tip given in Fig. 4). Solving of elasto-plastic stress fracture mechanics problems (without crack growth) is performed using the recommendations of European Structural Integrity Society (ESIS), /15/. Rectangular (square) elements are chosen for FEM mesh, and a singularity near the crack tip is modelled with triangular finite elements, with three independent nodes at the crack tip. The mesh near the crack tip is shown in Fig. 7. Sufficiently fine mesh with 1431 nodes and 444 elements is used here for simulated specimens.

Having in mind the type of the problem and the simple shape of the domain, the used number of elements provided sufficient precision and no convergence check for the solution was necessary. Meshes for four simulated specimens differed only in crack length. It should be noted that the mesh used for simulated specimens represents one half of specimen, as a specimen for three point bending has one plane of symmetry (for homogenous materials), and was taken into account in the modelling. Such a simplification could not be used for welded joints, as the heterogeneity of the welded joint "spoils" the symmetry of the problem. For this reason, the FE mesh for specimens of the welded joint had to model the entire specimen (2873 nodes and 898 elements). In order to use such a mesh, it was necessary to adjust the existing FE programme, /16/.

The stiffness matrix is calculated for each load increment (time step). The choice of load increment had no influence on the solution itself and its correctness, but only on the efficiency of the procedure. As the mesh is relatively regular with no drastic differences in the size of elements, Gaussian 2×2 numerical integration is used.

The stress–strain curve is represented as bilinear, i.e. given by two straight lines. Slopes of the lines define the modulus of elasticity in linear elastic deformation field, and the hardening properties in the plastic field. The modulus of elasticity is taken as the same for all specimens in this research (E = 210 GPa), and hence the yield stress R_{eh} and hardening index H of each specimen are used as input data.

Data for yield stress are taken from previous results /1/, whereas data for the hardening index are estimated based on the values of yielding stress, ultimate tensile strength R_m and elongation A, using the expression

$$H \approx 3 \cdot (R_m - R_{eh})/A$$

The input data for simulated specimens are given in Table 3. This data also holds for the other type of specimen (the HAZ regions), whereby HAZ is divided into two subregions: fine grain (FZ) and coarse grain (GZ). Such a division is known in references, /5/. Yield stress R_{eh} and hardening index H values for FZ HAZ are taken as an average of the values for corresponding specimens S4, S5 and S6 (R_{eh} = 397 MPa, H = 1290 GPa), whereas the values for specimen S2 (R_{eh} = 472 MPa, H = 1500 MPa) are chosen as values for GZ HAZ. The data for the weld metal (WM) and for parent metal (PM) are given in Table 4.

Table 3. Yield stress R_{eh}, and hardening index H, for simulated specimens.
Tabela 3. Napon tečenja R_{eh}, i indeks ojačavanja H, za simulirane epruvete

	S2	S4	S5	S6
R_{eh}, MPa	472	428	408	365
H, GPa	1500	1200	1180	1500

Table 4. Yield stress R_{eh}, and hardening index H, for weld metal (WM) and parent metal (PM).
Tabela 4. Napon tečenja R_{eh}, i indeks ojačavanja H, za metal šava (WM) i osnovni metal (PM)

	WM	PM
R_{eh}, MPa	450	428
H, GPa	1500	1200

For here considered specimen, 10 mm thick, it can be assumed the plane stress condition holds. Such an assumption causes the values of all parameters to be smaller than real. A more correct solution would be obtained by 3D modelling of the specimen, but it is too expensive. In order to adjust the results, calculations of plane stress condition are performed, giving larger displacement values than real. Based on EPRI recommendation, /5/, 3D modelling is carried out, and the ratio between two extreme cases – plane stress (RSN) and plane strain (RSD) is found.

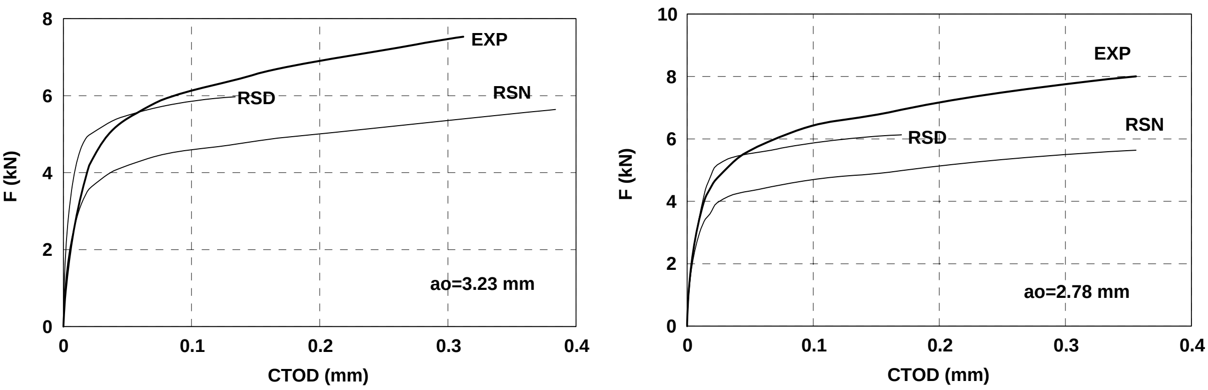


Figure 7. Calculated F-CTOD values for simulated specimens S5 (left) and S6 (right).
Slika 7. Sračunate vrednosti F-CTOD za simulirane epruvete S5 (levo) i S6 (desno)

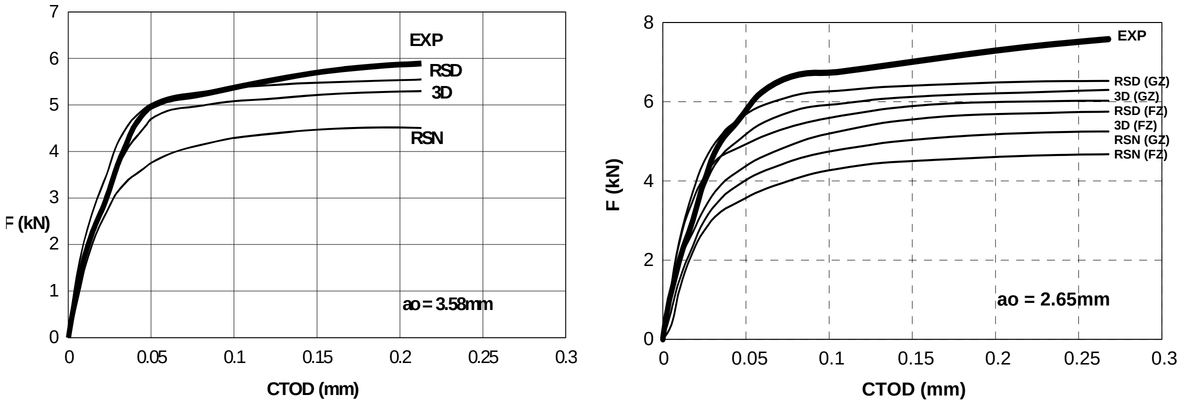


Figure 8. Calculated F-CTOD values for welded joint specimens SZ1 (left) and SZ6 (right).
Slika 8. Sračunate vrednosti F-CTOD za simulirane epruvete SZ1 (levo) i SZ6 (desno)

RESULTS OF NUMERICAL MODELLING

Estimated results for simulated specimens are given in Fig. 7, and for welded joint specimen in Fig. 8. Results are shown in the load vs. CTOD diagram for two different 3D calculations – plane strain (RSD) and plane stress (RSN), for estimated value based on 3D calculation and compared to experimentally obtained curves (EXP).

Estimation of fracture mechanics parameters

Estimation of fracture mechanics parameters is separated here in the elastic and plastic part. In the first part, solutions based on EPRI manual, published by Tada-Paris-Irwin, are applied, and in the second part, tabular solutions based on FEM are made for several simple specimen geometries, like single edge notched bending (SENB) specimen, /17/. Material behaviour is given with Ramberg-Osgood relation, the analytical form of which is:

$$\frac{\varepsilon}{\varepsilon_0}=\frac{\sigma}{\sigma_0}+\alpha\left(\frac{\sigma}{\sigma_0}\right)^n \quad (1)$$

Based on theoretical approach /17/, the expressions for determining crack mouth opening displacement (CMOD), CTOD (δ) and J are derived:

$$\text{CMOD}=\alpha\varepsilon_0 a h_2(a/b,n)(P/P_0)^n \quad (2)$$

$$\delta=\alpha\varepsilon_0 c h_4(a/b,n)(P/P_0)^{n+1} \quad (3)$$

$$J=\alpha\sigma_0\varepsilon_0 c h_1(a/b)(P/P_o)^{n+1} \quad (4)$$

where α, σ_0, ε_0 are material characteristics, a is crack length, b – specimen width, c – length of ligament, P is the applied load, P_0 critical load for crack growth.

Major contribution of the performed investigation is in determination of critical load for RSN and RSD conditions. For simulated specimens the data:

- specimen S2: P_0 = 6350 N (RSD), P_0 = 4850 N (RSN), P_0 = 6150 N (EXP),
- specimen S4: P_0 = 5925 N (RSD), P_0 = 4375 N (RSN), P_0 = 5725 N (EXP),
- specimen S5: P_0 = 5820 N (RSD), P_0 = 4200 N (RSN), P_0 = 5600 N (EXP),
- specimen S6: P_0 = 5400 N (RSD), P_0 = 4220 N (RSN), P_0 = 5280 N (EXP),

present the coefficients used for interpolation.

For welded specimens critical load for RSN and RSD conditions is determined:

- specimen SZ1: P_0 = 4880 N (RSD), P_0 = 3120 N (RSN), P_0 = 4200 N (EXP),
- specimen SZ3: P_0 = 4400 N (RSD), P_0 = 3275 N (RSN), P_0 = 4125 N (EXP),
- specimen SZ4: P_0 = 3820 N (RSD), P_0 = 2900 N (RSN), P_0 = 3600 N (EXP),
- specimen SZ6: P_0 = 4300 N (RSD), P_0 = 3220 N (RSN), P_0 = 3980 N (EXP) for CG HAZ,
- specimen SZ6: P_0 = 4130 N (RSD), P_0 = 2920 N (RSN), P_0 = 3980 N (EXP) for FG HAZ.

DISCUSSION

Results of FEM calculation for simulated specimens show good agreement with the experimental results, as long as load is below critical value. When load goes over critical value, i.e. when crack growth starts, values obtained by numerical simulation start to deviate from experimental values, which is a consequence of the fact that the case of crack growth is not addressed by the model. Based on the obtained results, one can claim that the numerical model is verified within the boundaries for which it was defined.

Results of FEM calculation for welded joint specimens show good agreement with experimental results, except in the case of SZ6 specimen for which the first version of calculation deviated significantly from experimentally obtained results, Fig. 8, results marked by FZ. Bearing in mind that the location of the crack tip is determined based on the micrograph, small deviations from actual location of crack tip are possible and even a small deviation can result in a change of microstructure of the crack tip location. In the next version of calculation it is assumed that for specimen SZ6 the crack tip lies in GZ HAZ, rather than in FZ HAZ. The obtained results (curves marked with GZ, Fig. 8) have entirely justified this assumption, as the agreement of the second calculation version produced results in good agreement with experimental results, i.e. on the same agreement level as for the other specimens. This has also shown that the numerical analysis can be used to validate experimental results.

Investigation of crack opening displacement demonstrated how difficult it is to find the locations of lowest toughness in HAZ. For final identification of notch location, microstructural investigations are necessary which are complicated and expensive, and still cannot always guaranty the proper location of the notch, since metallographic investigation usually takes only one intersection along the whole front of crack.

Numerical modelling makes it possible to obtain results on value of load that causes crack growth based on minimal investigation of mechanical properties (tensile tests). For specimens simulated by heating and for real welded joints, the values for opening loads obtained by numerical modelling and experimentally are very close, as long as the crack does not start to grow. This is expected as the model did not consider the case of crack propagation.

With this model it should be possible to determine the region of HAZ where the crack tip is located, by using the experimental diagram of load-opening and by comparing these loads with values of the load obtained by numerical modelling for several locations, and hence avoiding the metallographic analysis.

CONCLUSION

The precision of the results regarding the load that would cause crack growth in simulated samples, obtained using finite element method numerical modelling of stress strain field around crack tip, are of satisfactory accuracy, and are confirmed by experimental results. Applying this model, it would be possible to avoid expensive experiments and instead determine the load that causes crack growth using numerical simulation.

Numerical modelling of the stress strain field around the crack tip in welded joints indicated the possibility of predicting the notch and crack location in a real welded joint, if tensile properties of particular zones are known, as obtained from the investigation of simulated specimens.

REFERENCES

1. Gerić, K., Crack initiation and growth in high strength steel welded joints, Doctoral Thesis, University of Belgrade, Faculty of Technology and Metallurgy, 1997.
2. Burzić, Z., Sedmak, S., Manjgo, M., *Eksperimentalno određivanje parametara mehanike loma zavarenih spojeva*, Structural Integrity and Life, Vol.1, No2 (2001), pp.97-106.
3. Živković, A., Kutin, M., Sedmak, S., Milutinović, Z., Milanović, M., Arsić, M., *Mikroprsline u ZUT spojeva finozrnih čelika zavarenih u zaštiti CO_2*, Structural Integrity and Life, Vol.2, No.1-2 (2002), pp.43-50.
4. Milović, Lj., *Significance of cracks in the heat-affected-zone of steels for elevated temperature application*, Structural Integrity and Life, Vol.8, No1 (2008), pp.55-64.
5. Gerić, K., *Cracks and microstructure modelling in welded joints*, "The challenge of materials and weldments" – Structural Integrity and Life Assessments, Eds. S. Sedmak, Z. Radaković, J. Lozanović, MF, DIVK, TMF, GOŠA, Belgrade, 2008, pp.169-182.
6. Sedmak, S., *Crack problems in welded structures*, "The challenge of materials and weldments" – Structural Integrity and Life Assessments, Eds. S. Sedmak, Z. Radaković, J. Lozanović, MF, DIVK, TMF, GOŠA, Belgrade, 2008, pp.3-32.
7. Gerić, K., Cracks in welded joints, a monograph, University of Novi Sad, Faculty of Technical Sciences, 2005, (in Serbian).
8. Sedmak, S., Grabulov, V., Momčilović, D., *Chronology of lost structural integrity initiated from defects in welded structures*, Structural Integrity and Life, Vol.9, No1 (2009), pp.39-50.
9. Fawkes, A.J., Owen, D.J.R., Engineering Fracture Mechanics/ Numerical Methods and Applications, Pineridge Press Ltd., Swansea, 1983.
10. Berković, M., *Numerical Methods in Fracture Mechanics*, Structural Integrity and Life, Vol.4, No2 (2004), pp.63-67.
11. Berković, M., Maksimović, S., Sedmak, A., *Analysis of welded joints by applying the finite element method*, Struct. Int. and Life, Vol.4, No2 (2004), pp.75-84.
12. Sedmak, A., Berković, M., Savović, N., *Numerical analysis of surface crack problems in pressure vessels*, Struct. Int. and Life, Vol.4, No2 (2004), pp.91-100.
13. Berković, M., *Problems of plane and triaxial stress states in pressure vessels and pipelines*, Struct. Int. and Life, Vol.4, No2 (2004), pp.67-74.
14. Berković, M., Sedmak, A., *The application of finite element method in calculation of thin shell J integral*, Struct. Int. and Life, Vol.4, No2 (2004), pp.85-90.
15. ESIS Recommendations for Use of FEM in Fracture Mechanics. ESIS Newsletter No.15, p.3, 1991.
16. Savović, N., Doctoral Thesis, University of Belgrade Faculty of Mechanical Engineering, 1994.
17. Savović, N., Sedmak, A., *Numerical simulation of weldment heterogeneity*, Welding & Welded Structures, Journal of Yugoslav Welding Society, Vol.39, No4 (1994), pp.185-188.