Available online at www.sciencedirect.com

ScienceDirect

Procedia Materials Science 3 (2014) 1123 – 1128

Procedia Materials Science

www.elsevier.com/locate/procedia

20th European Conference on Fracture (ECF20)

Numerical modeling of austenite-ferrite weldment tensile test

Simon Sedmak[1], Uroš Tatić[1], Radomir Jovičić[1], Aleksandar Sedmak[2*], Miloš Milošević[1], Ramo Bakić[3], Stojan Sedmak[4]

[1] *Innovation Center of the Facultz of Mechanical Engineering, Kraljice Marije 16, 11120 Belgrade, Serbia*
[2] *Faculty of Mechanical Engineering, University of Belgrade, Serbia*
[3] *Technical School, Tutin, Serbia*
[4] *Faculty of Technology and Metallurgy, University of Belgrade, Serbia*

Abstract

The aim of this paper was to numerically simulate the plastic behavior of a three-material body specimen and compare the results with physical experiments performed on the same specimen. Both the physical and numerical models were subjected to tensile load which lead to the forming of a neck in the specimen due to significant plastic strain. Specimens were made by welding two materials, steel M (ferrite) and steel X (austenite) using filler metal with yield strength higher than the other two materials, producing the overmatched welded joint.

Plastic behavior of the specimen was simulated using the finite element method (FEM). As for the physical experiment, in addition to tensile tests of the three-material body specimen, each material was tested individually in order to obtain stress-strain diagrams. These diagrams were then used to determine the input parameters required for the numerical simulation of plastic strain. The results have confirmed that FEM can be used to effectively and accurately simulate the plastic behavior of a specimen made of three different materials.

Keywords: Numerical model, storage tank, austenite-ferrite weldment, tensile properties

---

1. Introduction

The research presented in this paper was inspired by the appearance of unexpected cracks in the fusion line of a welded joint in a cylindrical tank used for storing of liquefied CO_2. In order to better understand the forming and propagation of these cracks, a numerical model was developed, with material properties of the parent metals, as well

---

[*] Corresponding author. Tel.: +0-000-000-0000 ; fax: +0-000-000-0000 .
E-mail address: asedmak@mas.bg.ac.rs (A. Sedmak)

Selection and peer-review under responsibility of the Norwegian University of Science and Technology (NTNU), Department of Structural Engineering
doi:10.1016/j.mspro.2014.06.183

as the weld metal used in the cylindrical tank. The model, however, had to be verified with a physical experiment. For this purpose, specimens were made and tested, providing results for comparison with the numerical simulation.

Cylindrical storage tank for under pressure liquefied CO_2 (Fig. 1) consists of a pressure vessel, ferrite microalloyed structural steel M, 14 mm thick, ($R_{p\,0,2}$ - 450 MPa, R_m - 565 MPa) with welded manhole flange and outlet pipe connections made of high alloyed austenite steel X, 18/8 type ($R_{p\,0,2}$ - 394 MPa, R_m - 595 MPa). They were welded by INOX 29/9 consumable, with overmatched weld metal W ($R_{p\,0,2}$ - 550 MPa, R_m -750 MPa), as three-material body. In-service periodical inspection had been required. After longer service, unexpected pores and cracks were detected. Atypical cracks in steel M, parallel to the fusion line have 2 mm ligament (Fig. 2). Overmatching and residual stresses might have caused cracks [1].

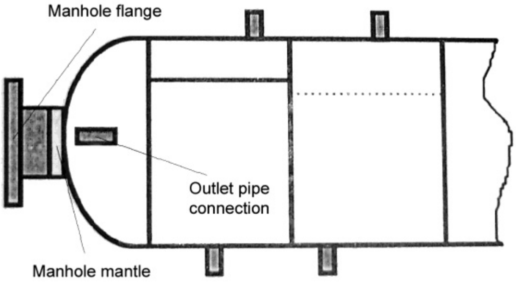


Figure 1. Storage tank with dissimilar weldments

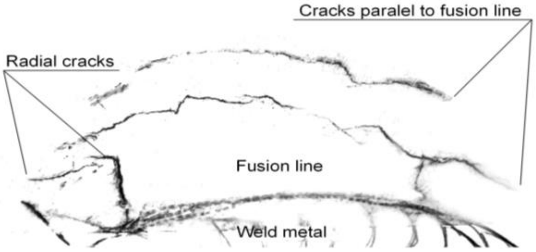


Figure 2. Cracks detected in ferrite steel

Recently, ABAQUS software has been used to develop numerical bi-material body model of an undermatched weldment, based on the results of previous experiments, [2]. The shape of deformation loop produced by loading and unloading agreed well with experimental data [3].

This stimulated the application of ABAQUS to develop numerical model for dissimilar weldment (steels M and X). To better understand deformation behaviour trial samples were welded with WPS for pressure vessels, tensile test specimens produced and tested to obtain initial data from stress-strain diagrams for steels M and X, and the weld metal W, and also for welded joint specimen. Numerical model exhibited the same strain distribution as obtained in the experimental test. Experimental results were compared with numerical model values for final verification.

2. Materials and method

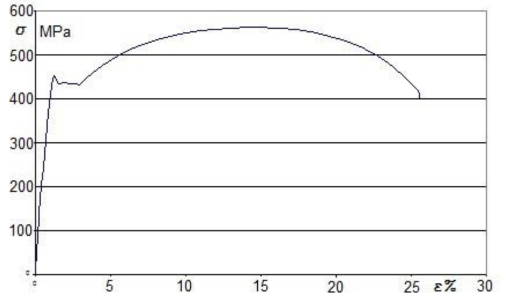
First step was to obtain the stress-strain diagrams for all three materials that were used. These diagrams were made by performing tensile tests of appropriate specimens. From the diagrams, values for yield strength were determined and used as the starting point for the defining of plasticity for each material.

ABAQUS software (Dassault Systèmes, France) was used for creating a 3D model that was used in the numerical method. Material properties required for plastic analysis include Young Modulus and Poisson's ratio, as well as true stress and strain for the plastic behaviour. ABAQUS requires the calculation of true stress and strain using values obtained from stress-strain diagrams. Properties of the three materials - steel M (ferrite), steel X (austenite) and weld metal W, are given in table 1 below. Table 2 shows real stress-strain diagrams and the ones used in the numerical simulation.

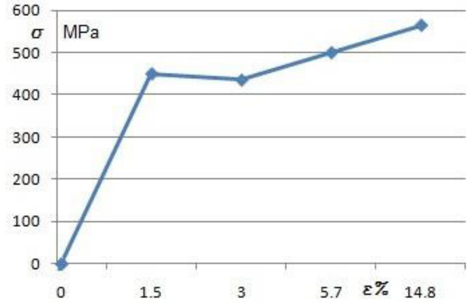
Table 1. Properties of materials used for the 3D model

Material	Yield strength, R_p (MPa)	Tensile Strength, R_m (MPa)	Young Modulus (GPa)	Poisson's Ratio
Ferrite (M)	450	565	210	0.3
Austenite (X)	294	595	210	0.3
Weld metal (M)	550	750	210	0.3

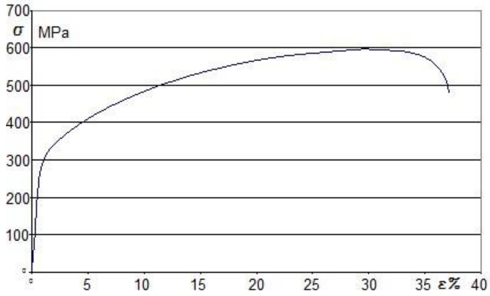
Table 2. Stress strain diagrams for all materials, obtained experimentally and numerically



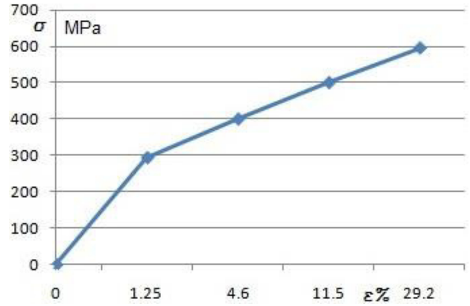
σ-ε diagram for steel M - experimental



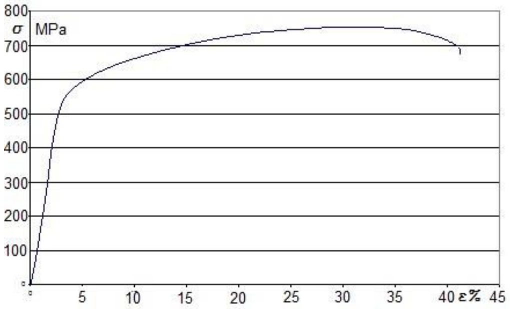
σ-ε diagram for steel M - numerical



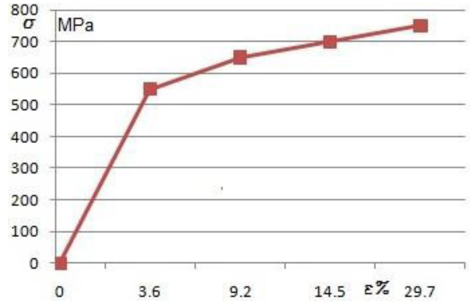
σ-ε diagram for steel X - experimental



σ-ε diagram for steel X - numerical



σ-ε diagram for the weld metal - experimental



σ-ε diagram for the weld metal - numerical

Stress-strain diagrams with true stress and strain were created using values from several characteristic points (shown on the right diagrams in the above table) on each corresponding experimental diagram. In ABAQUS, these values were given in form of a table which is used to define plastic behavior.

Geometry and mesh of the model are shown in figure 3. The model is symmetrical along the x axis, so one half of it was used in order to make the calculation less demanding in terms of time required to complete. Even though the geometry of the model itself is also symmetrical along y axis as well, it could not be taken into account, since the materials on the left and the right side of the welded joint are different. Due to simple geometry, the model was meshed using C3D8R elements. Boundary conditions are shown in figure 4. The front surface was fixed, in order to simulate the tensile test machine fixture and the side where the specimen was cut in half was constrained along the x axis.

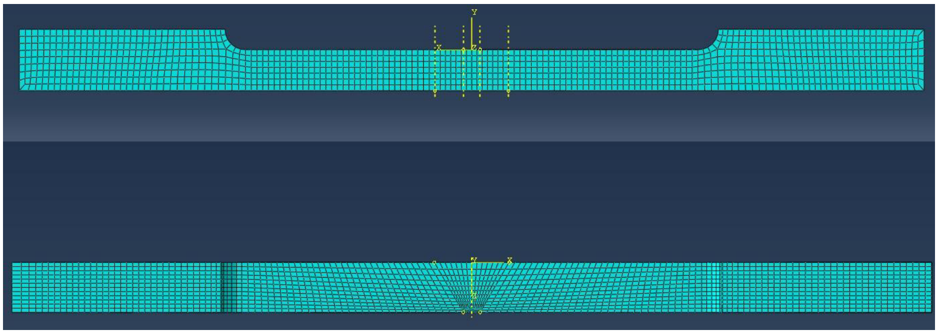


Figure 3. Specimen geometry and mesh

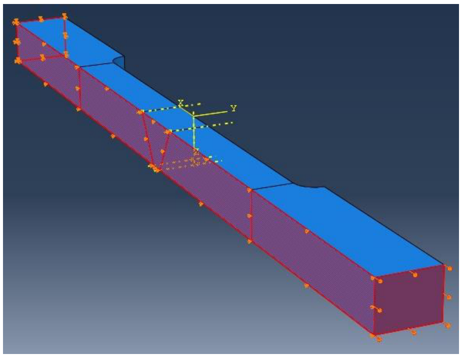


Figure 4. Boundary conditions

The load was given in form of a displacement and was applied on the back surface (in fig. 4), gradually increasing up to the value of 27 mm. This displacement lead to significant deforming of the specimen, which in turn resulted in plastic strain appearing and concentrating in an area referred to as the neck of the specimen.

Along with the numerical simulation, a physical experiment was performed with three-material body specimens to provide realistic results. The experiment was performed using a tensile test machine. Strain was measured using digital image correlation. This method based on creating images of a specimen subjected to load at different load stages and comparison of those images in order to determine the strain values [4].

3. Results

Presented in this section of the paper are the results, obtained both experimentally and numerically. In addition, the development of plastic strain over time is given as well, since it played an important part in determining the behavior of the specimen in terms of strain hardening.

Figure 5 shows the comparison of results for plastic strain in experimental and numerical specimens. As it can be seen from this figure, numerical results have shown excellent compliance with results obtained by the physical experiment. This was expected, since the displacement that was used as load in the numerical model was based on the total deformation of the physical model. However, the position and shape of the neck in both cases is also almost identical, which suggests that the model used for the simulation corresponds to the realistic conditions [5].

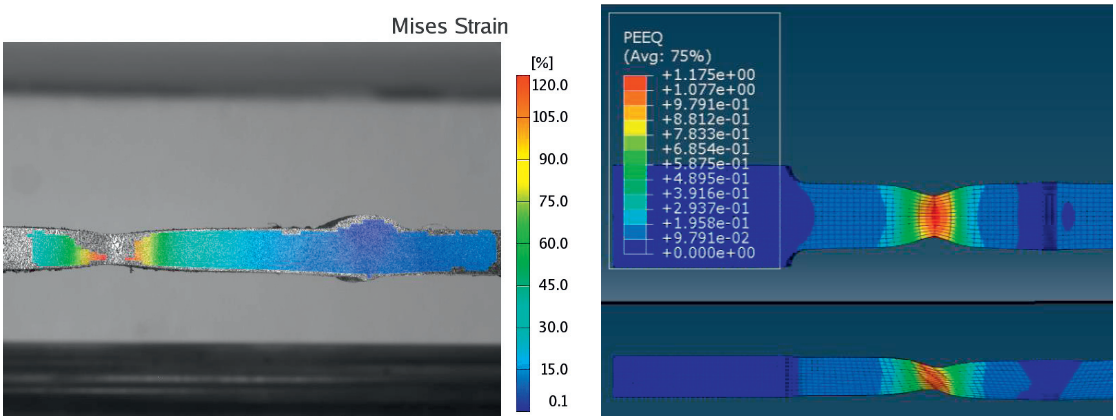


Figure 5. Experimental and numerical results

Figure 6 shows the strain distribution within the specimen at different stages of loading. Upper left image presents the specimen shortly after the load was initiated, the upper right image shows the strain at the point where the neck started forming and the bottom one shows the strain near the end of the loading time interval.

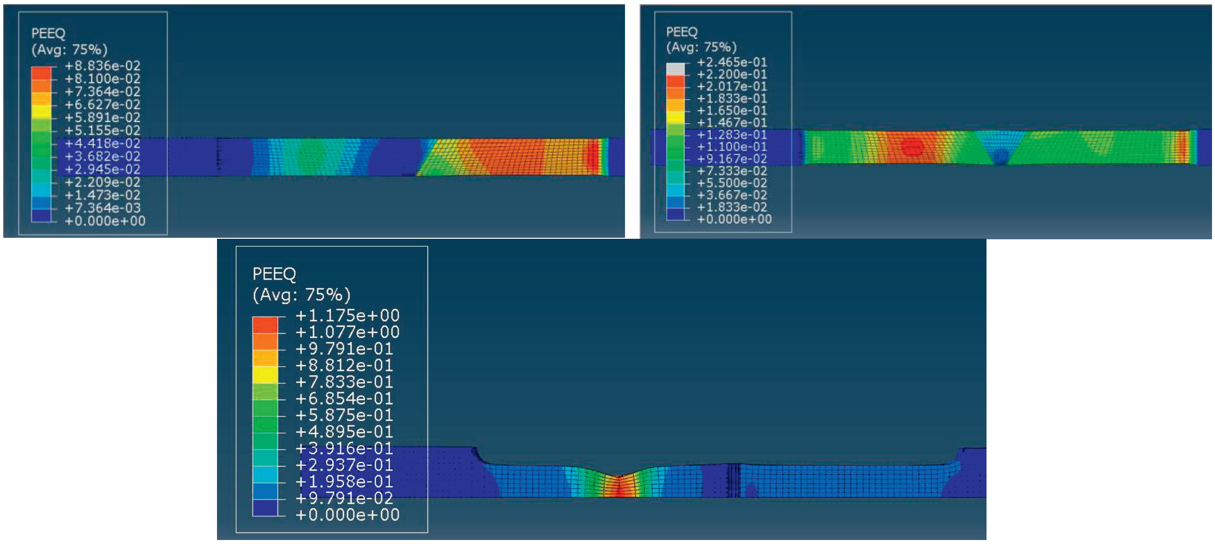


Figure 6. Strain distribution during various loading stages in the numerical model

4. Discussion

As seen in figure 5, both specimens show very similar behavior in terms of both plastic strain values and concentration (necks are located at the nearly same place). Maximum strain is 120% in the experiment and 117.5% in the numerical model, meaning that the difference between them was around 2%. As expected, strain concentration is present within the weakest of the three materials used, whereas the welded joint itself was only elastically deformed, due to overmatching. Initially, steel X showed higher levels of strain, since its yield strength was the lowest. However, at one point strain hardening started occurring in steel X and the strain started concentrating in steel M [6]. Beyond the point where the neck started forming, the strain in steel M became considerably higher than in steel X, since its tensile strength was lower. Additionally, at this point the plastic strain became significantly higher (~10 times) than in figure 6, upper left image.

Plastic strain concentration and distribution were similar in both models, suggesting that the numerical model provided a good representation of a physical problem. This required a detailed approach to creating the simulation, including the defining of boundary conditions, load, the mesh and especially the mechanical properties necessary for the calculation, along with approximations that would reduce the calculation time.

5. Conclusion

Based on the results shown in the paper, one can conclude:

Plastic behavior of the specimen was simulated using finite element method. As for the physical experiment, in addition to tensile tests of the three-material body specimen, each material was tested individually in order to obtain stress-strain diagrams. These diagrams were then used to determine the input parameters required for the numerical simulation of plastic strain. The results have confirmed that FEM can be used to effectively and accurately simulate the plastic behavior of a specimen made of three different materials.

The numerical simulation presented here should be used for developing of a more detailed approach to plastic behavior under tensile load of welded specimens. Results obtained during this analysis provide a sound base for further investigation of cracked weldment behaviour, which should also include the effects of heat affected zones.

6. Acknowledgements

The study was carried out within the Project TP – 35040, financed by the Ministry of Education, Science and Technological Development, Republic of Serbia. We would also like to thank Dr. Tarek Aburuga for the assistance provided with the numerical analysis.

7. References

[1] Milović, Lj.; Bulatović, S.; Aleksić, V.; Radaković, Z.; Marković, S.; Sedmak, St.; Assessment of the Behaviour of Fatigue Loaded HSLA Steel Welded Joint bz Applzing Fracture Mechanics Parameters, Structural Integrity and Life, Vol.12, No. 3, pp 175-179

[2] Algoul, M.; Sedmak, A.; Petrovski, B.; Tatić, U.; Sedmak, S.; Đurđević, A.; 2012 Quality Assurance of a Large Welded Penstock Manufacturing by Means of Full-scale Model Testing, 2nd International Conference Manufacturing Engineering & Management 2012, pp 107-108

[3] Tatić, U.; Sedmak, S.; Đurđević, A.;Sedmak, A.; Bakić, R.; Numerical Modelling of Full'Scale Penstock Model Testing, 3rd International Conference High Strength Steels for Hydropower Plants, 219-225

[4] Milošević N.; Milošević, M.; Sedmak, S.; Tatić, U.; Mitrović, N.; Hloch, S.; Jovičić, R.; Digital Image Correlation in Experimental Strain Analysis of Welded Joints, 5th International Scientific and Expert Conference TEAM 2013, Prešov, 2013, pp420-423

[5] Sedmak, S.; Tatić, U.; Mitrović, N.; Jovičić, R.; Maneski, T.; Sedmak, A.;, Milošević, M.; Modern Approach to Experimental Verification of Numerical Models for Tensile Tests of Welded Specimens, 30th Danubia - Adria Symposium on Advances in Experimental Mechanics, Primošten, 2013, pp 294-295.

[6] Ghosh, A.K.; Tensile Instability and Necking in Materials With Strain Hardening and Strain-rate Hardening, Acta Metallurgica, Vol. 25, pp 1413-1424, 1977