doi.org/10.15407/tpwj2017.09.10

# ENGINEERING APPROACH TO DETERMINATION OF STRESS INTENSITY FACTOR AND PARAMETERS OF GROWTH OF AXIAL CRACK IN CIRCUMFERENTIAL WELD OF PIPELINE

**V.V. KYRYCHOK** and **V.M. TOROP**
E.O. Paton Electric Welding Institute, NASU
11 Kazimir Malevich Str., 03150, Kiev, Ukraine. E-mail: office@paton.kiev.ua

The problem of evaluation of stress intensity factors in the zone of residual welding stresses was considered. The joint application of the software packages Ansys Workbench and Abaqus/CAE and the compiler Python for accurate determination of parameters of crack resistance of pressure pipelines and vessels was proposed. On the example of the pipe specimen 133×13 mm, the simulation of welding process and subsequent opening of semi-elliptic crack in the zone of residual stresses was carried out. The analysis of the possible further growth of defect from the action of cyclic loads in the weld region was performed and the most dangerous zones of location of axial cracks were found. 6 Ref. , 5 Figures.

**Keywords:** *stress intensity factor, finite element method, residual stresses, crack, script, weld*

In welds of pressure pipelines and vessels, which worked out a significant part of their life, crack-like defects are often revealed. The cause for their formation is mechanical, structural and chemical heterogeneity, corrosion, static and cyclic loads. In order to determine whether the pipeline can serve with a detected defect, it is necessary to evaluate the stress intensity factor (SIF) and J-integral or critical crack opening for the existing crack. For the area, which is located far from the weld, the methods were created which provide an accurate evaluation of these parameters. However, in the area of weld the plastic deformations are formed which cause residual stresses and have a non-uniform distribution. They depend on properties of the material, geometry of the pipe and welding mode. These factors do not enable the evaluation of SIF with minimal errors. At the present moment the standardized method [1] for determination of this parameter exists. It envisages the determination of $K_I$ by superposition of the working component $K_I$ and the component from residual stresses. The disadvantage of this algorithm is the fact that it does not envisage the peculiarities of redistribution of residual stresses after loading, does not take into account the welding parameters and does not provide accurate information of the area which should be considered as a weld.

In the given work the specified procedure is described, with the use of which SIF $K_I$ can be evaluated by the finite elements method. To do this, it is necessary to perform simulation of the thermal cycle of welding and to solve the problem of residual stress-strain state on the basis of it. After that it is necessary to evaluate how the stresses will be redistributed after inducing a crack. Analytical determination of the residual stress-strain state after welding is a complex problem, but today we can select software tools and even ready-made subprograms in the known software complexes of finite element analysis, with the help of which it can be calculated at a high accuracy. However, for simulation of a crack, which will open on the specimen after solution of thermo-mechanical problem, the use of special techniques is necessary. In general, it is possible to do this with the help of the software packages Ansys and Abaqus. Unfortunately, it is impossible to directly implement this task in the user-friendly interface Ansys Workbench or Abaqus/CAE, since the most commands, which should be at the same time broadcasted to the solver, are absent in them.

A direct study of the problem with a crack, which appears at the last step of the algorithm, can be realized using the parametric modeling in the APDL (Ansys Parametric Design Language) medium, the work in which assumes scripting in a language, which is similar to FORTRAN. Another way is the use of the solver Abaqus/Standard with a parallel parametric modeling in the language FORTRAN. The both methods envisage the need in textual programming of the model and due to this reason they are too complicated for realization for the majority of users. Therefore, the creation of special methods which would allow evaluating the process of crack formation with a high accuracy without parametric modeling represents a significant practical and scientific interest.

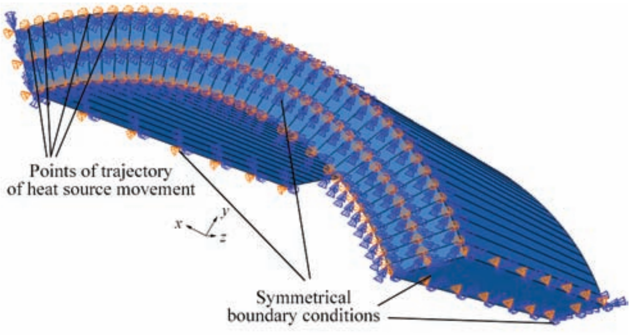


**Figure 1.** Calculation model of thermo-mechanical problem

The paper presents the results of successful combination of capabilities of Ansys Workbench and Abaqus/CAE for obtaining the required parameters of crack resistance. The problem was solved on the example of the pipe 133×13 mm made of steel 20. The mechanical and physical properties of the specimen material, as well as their dependence on temperature, were simulated with the help of the program «JMatPro» (this module allows obtaining a wide range of properties of the material depending on its chemical composition and structure). It was also taken into account that the characteristic of crack resistance (fracture toughness) $K_{1c}$ for hot-rolled tube specimens of steel 20 is about 160 MPa·m$^{0.5}$, and the threshold value $K_{\text{th}}$ was assumed as equal to 8 MPa·m$^{0.5}$ [2].

According to the work [3], the steel pipes with a wall thickness of up to 16 mm and the yield strength of up to 600 MPa are not prone to brittle fracture. However, if the stress intensity factor exceeds the value of $K_{\text{th}}$ in the field of variable loads, then one should expect the further growth of fatigue crack. The process of growth of fatigue crack is described by the Walker equation [2]:

$$\frac{da}{dN}=\frac{C(\Delta K)^{n}}{(1-R)^{n(1-\gamma)}}, \qquad (1)$$

where $da/dN$ is the rate of crack increment for 1 cycle; $\Delta K$ is the range of SIF of the component $K_I$; $R$ is the coefficient of cycle asymmetry according to SIF; $n$, $C$, $\gamma$ are the constants of material.

For simulation of thermal cycle of welding the Abaqus/CAE in combination with Pyton script [4] (in general, Python subprograms have the same capabilities as CAE interface, however their use is convenient for automated plotting of a model with a large number of operations) was applied. To reduce the calculation time, one eight part of the pipe was considered applying the symmetric boundary conditions to it (Figure 1). The thermal cycle was simulated for two-pass welding. The first pass was conditionally performed on the inner radius of pipe and the second one was done after cooling of the first one, i.e. on the outer radius. To simulate the arc movement and energy evolution from the arc process along the trajectory of welding in the arrays of points, the point heat sources were alternately activated, acting for some time and then deactivated. This method is described in details in the work [5]. With the help of the scrypt Python it was managed to create a dense automatic breakdown of the trajectory of arc movement in such a way that to minimize the «jerkiness» of this process.

The result of thermo-mechanical problem on the circumferential stresses is shown in Figure 2. After simulation of welding process, an operating pressure of 22.5 MPa was applied to the pipe.

During loading of welded pipe with the operating inner pressure, it was revealed that additional plastic deformation in the vicinity of a weld is almost absent, which allows applying the elastic model for this specimen at the next stage of finding SIF.

The introduction of a crack into the model was realized with the help of Ansys Workbench. To do this, the procedure of mapping the stress fields from the first stage was used. In addition, for the possibility of simulating a crack in its asymmetric location relative to the weld, a part of the pipe was plotted in the axial direction. If the area under consideration is not fixed, then the stresses induced with the help of mapping are redistributed and we will not be able to obtain an adequate picture. Therefore, a rigid fixation at the ends

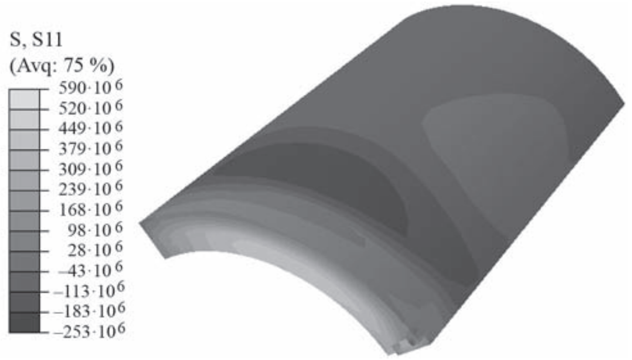


**Figure 2.** Distribution of circumferential residual stresses after welding (Pa)

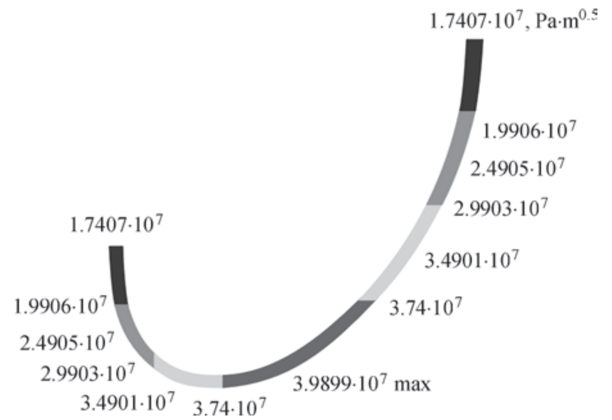


**Figure 3.** Distribution of SIF of a crack depending on residual stresses

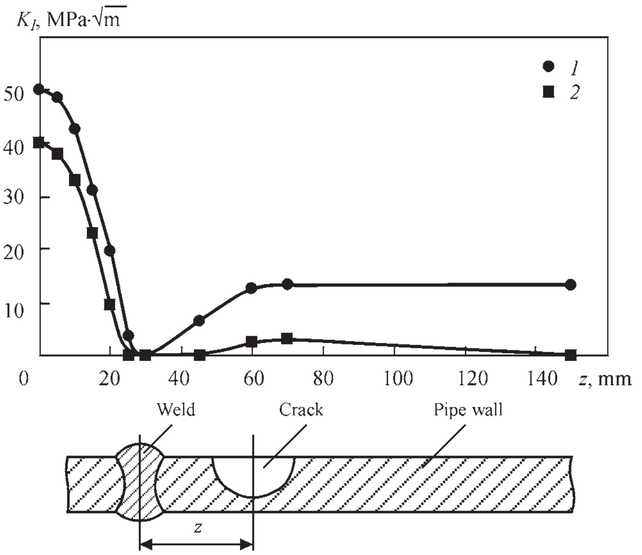


**Figure 4.** Distribution of maximum SIF in pipe axis in the vicinity of a weld: *1* — welding + operating pressure; *2* — welding

and a cylindrical fixation in the radial direction were applied to the specimens under consideration. This made it possible to leave the residual stresses and provide the possibility of opening the crack lips during its inducing. In this case, the error of transfer of component of circumferential residual stresses S11 from Abaqus to Ansys after balancing of stresses does not exceed 10 %.

In the work, the postulated axial defect of a semi-elliptical shape outside the pipe of 16 m length and 8 mm depth was considered. The crack was alternately located in the weld center and at different distances from it in the zone of residual stresses. For example, the results of SIF distribution from residual stresses in the weld center are presented in Figure 3.

At the some distance of crack from the weld, at first the maximum SIF decreases to 0 and then increases. The distribution of this value, depending on the distance to the center, is shown in Figure 4.

The next task was to evaluate how SIF will change in the heat-affected zone if operating pressure is applied to the pipe. To do this, a similar simulation was carried out with the difference that at the first stage in Abaqus the inner pressure was applied after the welding process. The distribution of SIF, depending on the distance to the weld center under the operating load, is shown in Figure 4.

For the pipeline, the evaluation of value of defect growing in a one cycle according to the formula (1) was performed, depending on the location of a crack relative to the weld axis (Figure 5). At the same time, the values *n*, *c*, and $\gamma$ were obtained from the operation diagrams [6]. According to the operation [6], these constants for different ferrite-pearlite steels almost do

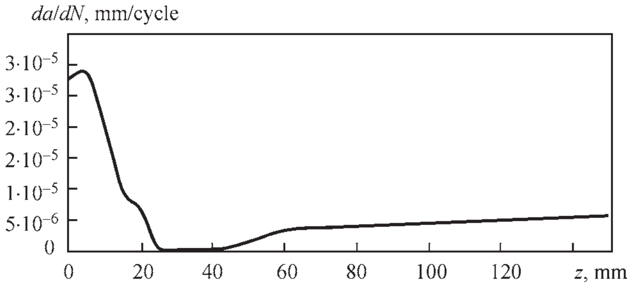


**Figure 5.** Distribution of crack growth rate depending on its location in the vicinity of a weld

not differ. For the specimen under consideration, the martensitic phase is mostly not formed during welding, therefore, the same parameters of crack resistance were taken for the weld and base metal. The results of calculations showed that in the heat-affected zone the amplitudes of $\Delta K_I$ are smaller than for the weld-free area of a pipe, but the cycle asymmetry contributes to the fact that the rate of growth of a defect in the weld zone several times exceeds the rate of growth in the pipe region, which is distant from the zone of plastic deformations.

## Conclusions

The methods for determination of stress intensity factors in the heat-affected zone are given using the finite element method. The capabilities of different software means for simulation of a welding process and determination of stress fields of pipe specimens with defects are presented. The given results of SIF distribution and the rates of growth of a defect are presented, depending on the distance of a crack to the circumferential weld. It was analytically shown that in case of assumed homogeneity in the mechanical properties of base metal, weld zone and heat-affected zone, the axial defects in the zone of welded joints of pressure pipelines and vessels are more dangerous than cracks at an arbitrary point of the pipe, especially in the cyclic types of loads.

1. *DSTU-N B.2.3-21:2008*: Determination of residual strength of main pipelines with defects.
2. Stephens, R., I., Ali Fatemi, Stephens, R. R., Fuchs, H. O. (2001) *Metal fatigue in engineering*. 2nd ed. New York, John Wiley & Sons.
3. *PNAEG-7-002–86 (1989)*: Codes of design on strength of equipment and pipelines of nuclear power plants. Moscow: Energoatomizdat.
4. (2010) *Abaqus Scripting Reference Manual*. USA, Providence, RI.
5. Yakhno, B.O. (2013) *Calculation of welded joints using program complex Abaqus: Guidelines*. Kyiv, NTUU KPI.
6. Suryanarayana, C. (2003) *Experimental techniques in material and mechanics*. Boca Rathon, USA, Taylor & Francis Group.

Received 25.04.2017