Determination of Residual Stresses in Welded Pipes Using a Simplified Heat Source

Dragi Stamenković[1)]
Mato Perić[2)]

In this paper, a numerical simulation based on the finite element method was used to determine residual stresses in welded pipes. The simulation consists of sequentially coupled thermal and structural analyses. In the thermal analysis, the heat source is introduced, using simplified heat flux. The temperature field, calculated in the thermal analysis, is used as the loading for the mechanical analysis. The welding simulation was conducted using the software packages Abaqus and Ansys, and the obtained results agree very well. This paper presents a methodology of determining residual stresses in butt welded pipes.

Key words: welding, residual stresses, welded joint, butt welding, pipe, thermal analysis, finite element method.

Introduction

A conventional means to calculate residual stress distribution in engineering structures is the finite element method. Many authors used the finite element method to perform welding simulations. There are two distinct methods in the analysis: sequential and direct. The sequential method consists of two or more sequental analyses. Performing the thermal and the mechanical analysis separately is more computationally efficient. After the thermal analysis, the determined temperature fields are used as input loads for the mechanical analysis. In literature, it is convenient to use the volumetric heat flux for the thermal analaysis proposed by Goldak [1]. The proposed method is widely accepted by many authors [2-6]. Today, widely used commercial software Abaqus and Ansys do not contain the built-in volumetric heat flux proposed by Goldak and in order to use it, it is necessary to write a special subroutine in FORTRAN. However, good results of residual stress and strain can be obtained using a uniform distribution of the volume heat flux at the weld bead [7-10]. The consequence of taking a uniform heat flux is a different time dependent temperature field compared to that when the heat flux entered as a volumetric heat flux proposed by Goldak. This difference is significantly expressed in the root of the weld and with the distance from the weld zone it disappears quickly. The magnitude of residual stresses in the case of modeling with a uniform heat flux as the load is slightly different from that in which the thermal analysis is performed with a volumetric heat flux proposed by Goldak.

This paper presents two numerical simulations of welding two steel pipes of dimensions Ø324 x 3.96 mm with the constant width of the weld of 3.84 mm throughout the whole pipe wall. The numerical simulations of welding processes of two pipes are performed using software cods Abaqus and Ansys. The tensile residual axial stresses are produced on the inner surface while for compressive residual axial stresses exist on the outer surface of pipes. Farther from the weld zone, residual stresses are of negligible values. Furthermore, magnitude of residual tensile stresses is very close to yield stress of materials.

Finite element analysis approach

The numerical simulations performed in this work use the sequential thermal-stress solution procedure in which the transient heat transfer analysis is followed by the thermal stress analysis. Temperatures predicted by the heat transfer analysis are used as the loading for thermal stress analysis.

Thermal model

The governing equation for the transient non-linear heat transfer analysis is:

$$\frac{\partial}{\partial x}\left(k_x \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k_y \frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k_z \frac{\partial T}{\partial z}\right) + Q = \rho C \frac{\partial T}{\partial t} \quad (1)$$

where, k_x, k_y, k_z thermal conductivity in the x, y and z respectively, T is the current temperature, Q is the heat generation, ρ is the density, C is the specific heat capacity and t is the time, respectively. The general solution of equation (1) is obtained introducing the initial and boundary conditions, as follows:

Initial condition:

$$T(x,y,z,0) = T_0(x,y,z) \quad (2)$$

Boundary conditions:

$$\left(k_x \frac{\partial T}{\partial x} N_x + k_y \frac{\partial T}{\partial y} N_y + k_z \frac{\partial T}{\partial z}\right) +$$

$$+ q_s + h_c(T - T_\infty) + h_r(T - T_r) = 0 \quad (3)$$

[1)] Termoelektro d.o.o. Uralska 9, 11060 Belgrade, SERBIA
[2)] Bureau of Energetics and Mechanical Engineering Ltd., Petrovaradinska 7, 10000 Zagreb, CROATIA

where, N_x, N_y, N_z are the direction cosine normal to the boundary, h_c, h_r are the convection and radiation heat transfer coefficients respectively, q_s is the boundary heat flux and T_r is the temperature of the radiation heat source and T_∞ is the surrounding temperature. Radiation heat losses are dominant near the weld and can be expressed by equations:

$$h_r = \sigma \varepsilon F (T^2 + T_r^2)(T + T_r) \qquad (4)$$

where, 5.67×10^{-8} J/(m^{2}K^4) is the Steffan-Boltzman constant, ε is the effective emissivity and F is the configuration factor. Farther from the weld zone, radiation heat losses decrease and increase the share of the convection heat loses h_c. The total heat input is given by:

$$q = \frac{\eta U I}{V_H} \qquad (5)$$

where, η is the arc efficiency, V is the volume of the weld bead, I is the current, U is the arc voltage and V_H is the volume of the activated weld bead element.

Mechanical model

The equilibrium equation can be expressed as:

$$\sigma_{ij,j} + \rho b_i = 0 \qquad (6)$$

$$\sigma_{ij} = \sigma_{ji} \qquad (7)$$

In equations (6) and (7) σ_{ij} is the stress tensor, ρ is the density and b_i is the body force. In the numerical simulation, thermal-plastic constitutive equations were used:

$$[d\sigma] = [D^{ep}][d\varepsilon] - [C^{th}]dT \qquad (8)$$

$$[D^{ep}] = [D^e] + [D^p] \qquad (9)$$

where $[D^e]$ is the elastic stiffness matrix, $[D^p]$ is the plastic stiffness matrix, C^{th} is the thermal stiffness matrix, $d\varepsilon$ is the strain increment and dT is the temperature increment.

Butt-weld of two pipes

Welding conditions, boundary conditions and material properties

The welding process of a butt-weld joint of two SAE 1020 steel pipes with the dimensions shown in Fig.1 was simulated. The thermal and mechanical properties are given in Figs. 2 and 3. The material is modelled as an elastic – ideally plastic.

Due to high temperature and stress gradients near the weld, the finite element model has a relatively fine mesh in both sides of the weld center line. The eight-node brick elements with linear shape functions are used in meshing the model. The three-dimensional mesh consisted of 14400 elements. The same mesh is used both for the thermal and the mechanical analysis. Using the Abaqus code [12], DC3D8 elements are used in the thermal analysis, while for the mechanical model C3D8 elements are used. In the analysis conducted using the Ansys code [13], the SOLID70element type is used in the thermal analysis, while for the mechanical model SOLID45 elements are used.

Elements with a single degree of freedom were used in the thermal analysis while for the mechanical analysis elements with three translational degress of freedom were used. To simulate the moving heat source it is necessary to model the heat source during each time increment. In this analysis the moving heat source is simplified by assuming that the welding arc stays at an element with a constant specific volume heat generation, and then moves to the next element at the end of the load step as the welding is finished. The heat input during welding is modeled in the Abaqus and Ansys by the equivalent heat input which includes the body heat flux. The amount of heat input was calculated using Eq. 5, is $q = 2.076 \times 10^{10}$ J/m^{3}s. The welding parameters chosen for this analysis are as follows: tungsten inert gas welding, welding current I=110 A, welding voltage U=20 V and welding speed v=5 mm/s. The following values are assumed: the convective heat transfer h_c=15 W/m^{2}K, the arc efficiency η =70% and the emissivity ε= 0.8. The welding of pipes is modelled in a single pass. The mechanical boundary conditions were prescribed for preventing rigid body motion. The detailed boundary conditions used in the finite element model are shown in Fig.1 by the triangles.
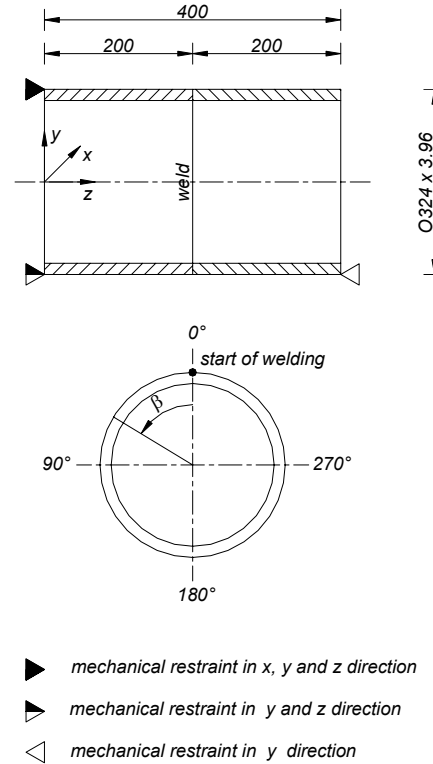

▶ *mechanical restraint in x, y and z direction*

▷ *mechanical restraint in y and z direction*

◁ *mechanical restraint in y direction*

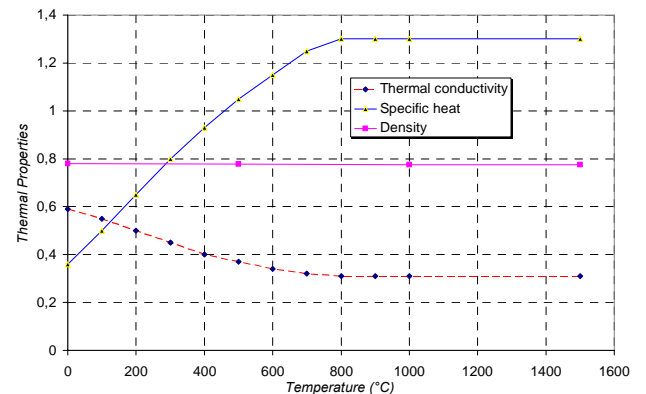**Figure 1**. Geometry of butt-welded pipes

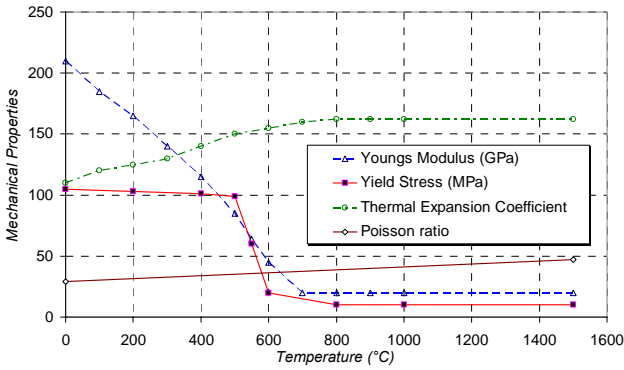**Figure 2.** Thermal properties of steel SAE 1020 [11].

Figure 3. Mechanical properties of steel SAE 1020 [11].

Results

Figures 4 and 5 show the temperature fields 110 *s* after the start of welding with a β =180° central angle, while Figures 6 and 7 show the temperature profiles on the outer surface of the pipe. The proposed method using a simplified heat source was validated by good agreement in the results.
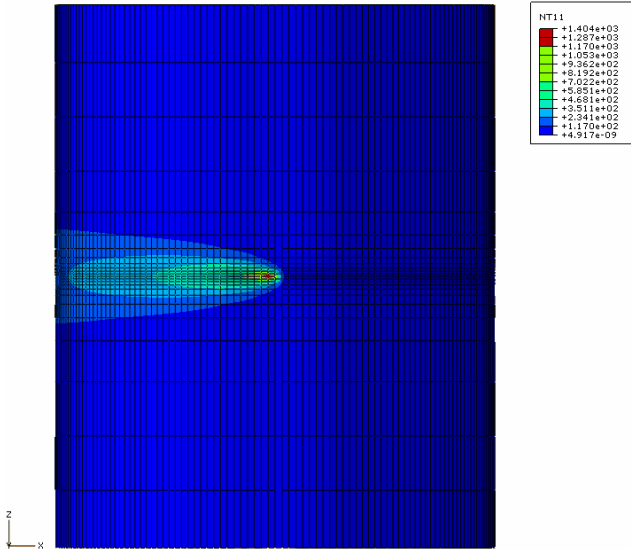

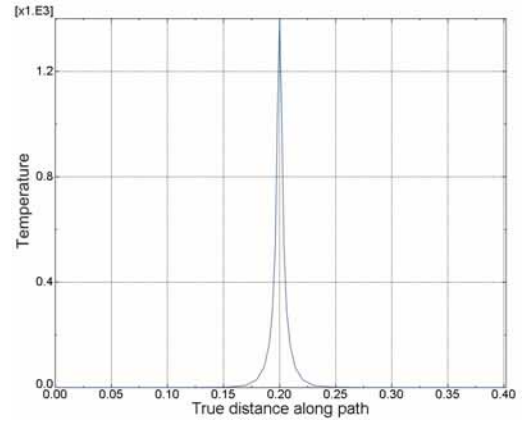**Figure 4.** Temperature field 100 s after the start of welding, β =180°, Abaqus

Figure 5. Temperature field 100 s after the start of welding, β =180°, Ansys

Figures 8 and 9 show the residual stresses field σ_z distribution in the axial direction computed by the finite element code Abaqus and Ansys. Figures 10 and 11 give the comparison of the axial residual stress distribution on the inside and the outer surface with a β =180° central angle. The predictions of the residual stresses have a similar distribution shapes near the weld zone. Significant discrepances between the peak values of residual stresses can be seen at the center of the weld. These values calculated using both software codes are very close to the yield stress of materials.

Figure 6. Temperature profiles t = 100 s and β =180°, Abaqus
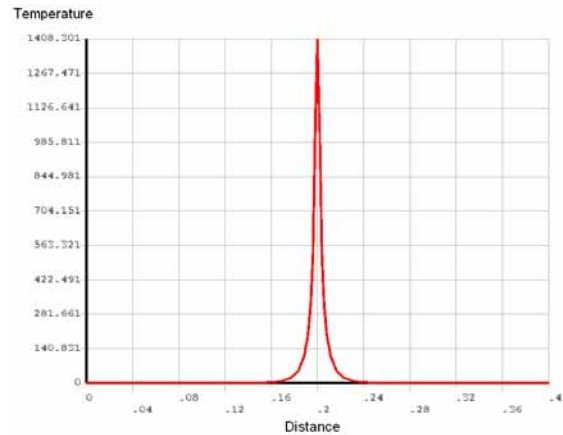
Figure 7. Temperature profiles t = 100 s and β =180°, Ansys

Figure 8. Residual stresses in σ_z in the axial direction, Abaqus

```
ANSYS 12.1
MAY  2 2011
18:17:10
        -.131E+09
        -.101E+09
        -.704E+08
        -.399E+08
        -.935E+07
         .212E+08
         .517E+08
         .822E+08
         .113E+09
         .143E+09
```
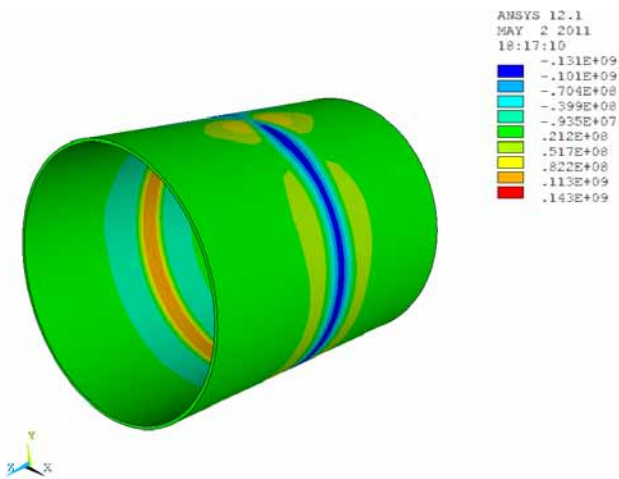
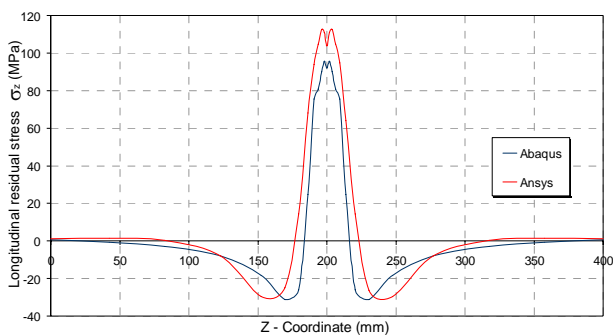**Figure 9**. Residual stresses in σ_z in the axial direction, Ansys

Figure 10. Comparison of residual stresses σ_z in the axial direction

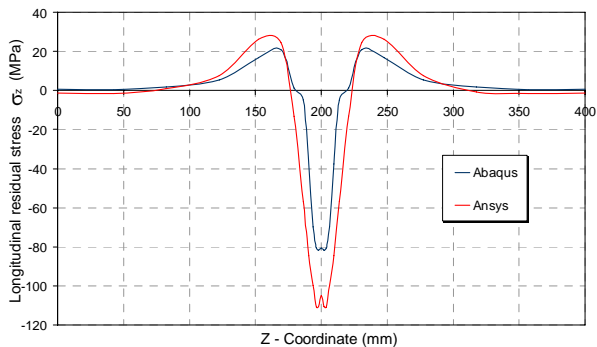on the inner surface, $\beta = 180°$, calculated in Abaqus and Ansys

Figure 11. Comparison of residual stresses σ_z in the axial direction

on the outer surface, $\beta = 180°$, calculated in Abaqus and Ansys

Conclusion

This work presents a methodology and the finite element method to determine residual stresses in high strength carbon steel pipes, joined by butt weld. The finite element method is an efficient technique in predicting residual stresses in welding processes. A three-dimensional finite element welding simulation was carried out on a one-pass welding. The welding simulation was considered as a sequential coupled thermo-mechanical analysis. The finite element analysis results of the residual stress distributions of two butt welded pipes in the axial directions are presented in Fig.10 and Fig.11. Thus the values of the axial residual stresses calculated by the finite element method, using two different software codes Abaqus and Ansys are very close.

This method implemented in welding simulation can be used in other analyses. It could consider different process parameters, for example welding speed, number and sequence of passes, filling material supplying, etc. Moreover, various geometrical constraints and material nonlinearities can be included in the analysis.

References

[1] GOLDAK,J., CHAKRAVARTI,A., BIBBY,M.: *Metalurgical Transactions B 15*, June, 1984, pp.299-305.

[2] DENG,D., MURAKAWA,H.: *Numerical simulation of temperature field and residual stress in multi-pass weld in staunless steel pipe and comparison with experimental results*, Computional Material Sciennce 37, 2006, pp.269-277.

[3] DENG,D.: *FEM prediction of welding residual stress and distortion in carbon steel considering phase transformation effects*, Materials and Design 30, 2009, pp.359-366.

[4] LONG,H., GERY,D., CARLIAER,A., MAROPOULOS,P.G.: *Predictions of welding distortion in butt joint of thin plates*, Materials and Design 30, 2009, pp.4126-4135.

[5] SURAJ,J., SEMETAY,C., PRICE JOHN W.H., NIED,H.F.: *Weld-induced residual stress in a prototype dragline cluster and comparison with design codes*, Thin-Waled Structures 48, 2010, pp.89-102

[6] KERMANPUR,A., SHANANIAN,M., YEGANEH,E.V.: Three-dimensional thermal simulation and experimental investigation of GTAW circumferentialy butt-welded Incoloy 800 pipes, Journal of Materials Processing Technology 199 (2008) 295-303.

[7] PERIC,M., TONKOVIC,Z., KARSAJ,I.: *Numerical analysis ofresidual stresses using a shell/3D modeling technique*, Proceedings of the International Conference on Advances in Welding Science and Technology for Construction, Energy and Transportation AWST-2010, pp.75-80

[8] STAMENKOVIC,D., VASOVIC,I.: *Finite Element Analysis of Residual Stresses in Butt Welding Two Similar Plates*, Scientific Technical Review, ISSN 1820-0206, 2009,Vol. LIX, No.1, pp.57-60.

[9] PERIC,M., STAMENKOVIC,D., MILKOVIC,V.: Comparison of residual stresses in butt-welded plates using software packages abaqus and ansys, Scientific Technical Review, ISSN 1820-0206, 2010,Vol.60,No.3-4, pp.22-26

[10] TENG,T.L., CHANG,P.H.: *Three-dimensional thermo mechanical analysis of circumferentially welded thin-walled pipes*. International Journal of Pressure Vessels and Piping 75, 1998, pp.237-247.

[11] ABAQUS/Standard, *User's guide and theoretical manual*, Version 6.8, Hibbit, Karlsson & Serensen, Inc. (2008).

[12] *Theory Reference for ANSYS and ANSYS Workbench*, ANSYS Inc. Release 12.1

Određivanje zaostalih napona u zavarenim cevima korišćenjem pojednostavljenog izvora toplote

U ovom radu korištena je numerička simulacija, koja se zasniva na metodi konačnih elemenata, za određivanje zaostalih napona u zavarenim cevima. Simulacija se sastoji od sekvencijalno spregnute termičke i mehaničke analize. U termičkoj analizi, unošenje toplote je izvedeno korišćenjem pojednostavljenog toplotnog fluksa. Temeraturno polje dobijeno u termičkoj analizi, uvedeno je kao opterećenje u mehaničkoj analizi. Simulacija zavarivanja je sprovedena korišćenjem dva softverska paketa Abaqus i Ansys. Ovaj rad predstavlja metodologiju određivanja zaostalih napona u sučeono zavarenim cevima.

Ključne reči: zavarivanje, zaostali naponi, zavareni spoj, suočeno zavarivanje, cev, termička analiza, metoda konačnih elemenata.

Определение остаточных напряжений в сварных трубах при использованием упрощённого источника теплоты

В настоящей работе использована цифровая симуляция, которая основывается на методе конечных элементов, для определения остаточных напряжений в сварных трубах. Симуляция состоит из последовательности пар термического и механического анализов. В термическом анализе внесение теплоты применено использованием упрощённого теплового потока. Температурное поле, получено в термическом анализе, введено в роли нагрузки в механическом анализе. Симуляция сварки проведена использованием двух коммерческих пакетов программного обеспечения компьютора Abaqus и Ansys. И так настоящая работа является методологией определения остаточных напряжений во встык сварных трубах.

Ключевые слова: сварка, остаточные напряжения, сварной стык, сварка притык, труба, термический анализ, метод конечных элементов.

Détermination des tensions résiduelles chez les tuyaux soudés à l'aide d'une source thermique simplifiée

La simulation numérique basée sur la méthode des éléments finis est utilisée pour la détermination des tensions résiduelles dans les tuyaux soudés et elle est présentée dans ce papier. Cette simulation est composée des analyses thermiques et mécaniques qui sont couplées l'une après l'autre. Au cours de l'analyse thermique la chaleur est introduite par le flux thermique simplifié. Le champ de température obtenu pendant l'analyse thermique est introduit comme la charge dans l'analyse mécanique. La simulation du soudage est effectuée au moyen de deux progiciels Abaqus et Ansys. Dans ce travail on a présenté la méthodologie pour la détermination des tensions résiduelles dans les tuyaux soudés en bout.

Mots clés: soudage, tensions résiduelles, jointe soudée, soudage en bout, tuyau, analyse thermique.