HEFAT2012
9th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
16 – 18 July 2012
Malta

THERMAL AND RESIDUAL STRESS ANLAYSES OF A WELDED STAINLESS STEEL PLATE

Kaman M.O. Celik N*. and Gokce I.
*Author for correspondence
Department of Mechanitronics Engineering
Firat University,
Elazig, 23119,
Turkey ,
E-mail: nevincelik23@gmail.com

ABSTRACT

In this study, a welding simulation was analyzed numerically by using finite element method. Two-dimensional model of butt welded plate was conducted in *ANSYS* finite element package software. Transient thermal distribution on *ST-37* plate was obtained for all time intervals until it reaches environment temperature. The initial temperature of *ST-37* was considered 300 K, whilst the temperature of welded zone was assumed to be 1773 K. The enthalpy and density of the welded zone material were assumed to have temperature dependent values, although the non-welded zone of the steel was considered to have constant values. In accordance to this, plastic stresses were presented base on the elasto-plastic bilinear material behavior using structural analysis. Finally, residual stresses were found out by removing loads on the butt welded plate. Changing of residual stresses and spread of plastic zones according to cooling time were investigated.

INTRODUCTION

Nowadays, there is wide application area of metallurgical joints made by welding in fabrication industry due to their advanced properties such as high joint efficiency, water and air tightness and low manufacturing cost. The types of welded joint can be classified into five basic categories; butt, fillet, corners, lap and edge [1]. Butt welds are commonly used in joining metal plates either to same metal or to different metal. Problem associated with welded joints is residual stress near the weld zone on the metal plate due to localized heating by the welding process. If residual stresses reach high values, this situation causes a reduction of load capacity and complex failure modes. Because of this reason, it is wanted more detailed design knowledge and production techniques about welded joints. For the prediction of the temperature and residual stress distribution, numerical studies on thermo-mechanical behaviour of welded structures have increased with the development of finite element method in addition to the experimental studies [2-4].

Teng et al. [1] described the thermal elasto-plastic analysis using finite element techniques to analyze the thermomechanical behavior and evaluate the residual stresses and angular distortions of the T-joints in fillet welds. The effects of flange thickness, welding penetration depth and restraint condition of welding on the residual stresses and distortions were discussed.

Barsoum and Lundback [5] carried out two and three dimensional finite element welding simulations in order to study the formation of the residual stresses due to 3D effect of the welding process. Residual stress measurements were carried out using X-ray diffraction technique on the manufactured T-welded structure. The 2D residual stress predictions showed good agreement with measurements.

Barsoum and Barsoum [6] developed a welding simulation procedure using the finite element software *ANSYS* in order to predict residual stresses. The procedure was verified with temperature and residual stress measurements found in the literature on multi-pass butt welded plates and T-fillet welds. The predictions showed qualitative good agreement with experiments. The welding simulation procedure was then employed on a welded ship engine frame box at *MAN B&W*. A subroutine for LEFM analysis was developed in 2D in order to predict the crack path of propagating fatigue cracks.

Teng et al. [7] analyzed the thermo-mechanical behaviour and evaluated the residual stresses with various types of welding sequence in single-pass, multi-pass butt-welded plates and circular patch welds. This was achieved by performing thermal elasto-plastic analysis using finite element techniques.

Lindgren [8-10] used finite element simulation in order to predict temperature fields, residual stresses and deformation due to welding in 2D and 3D.

Kong et al. [11] developed a model based on a double-ellipsoidal volume heat source to simulate the gas metal arc welding (GMAW) heat input and a cylindrical volume heat source to simulate the laser beam heat input to predict the temperature field and thermally induced residual stress in the hybrid laser–gas metal arc (GMA) welding process. Numerical simulation showed that higher residual stress distributed in the weld bead and surrounding heat-affected zone (HAZ).

Zain-ul-abdein et al [12] investigated the effect of metallurgical phase transformations upon the residual stresses and distortions induced by laser beam welding in a T-joint configuration using the finite element method. Two separate models were studied using different finite element codes, where the first one describes a thermo-mechanical analysis using *ABAQUS*; while the second one discusses a thermo-metallo-mechanical analysis using *SYSWELD*.

Long et al. [13] investigated distortions and residual stresses induced in butt joint of thin plates using metal inert gas welding. A moving distributed heat source model based on Goldak's double-ellipsoid heat flux distribution was implemented in finite element simulation of the welding process.

In this study, finite element simulation of butt welded plate is achieved using finite element software *ANSYS*. Initially temperature distribution on plate is obtained according to the initial conditions and saved for every load step. In the thermal analysis, the initial temperature of welded *ST-37* plate is considered 300 K and the temperature of welded zone is assumed to be 1773 K. Then structural analysis is performed base on the elasto-plastic bilinear material behaviour using structural analysis. At the end of the simulation, residual stresses are obtained by removing loads on numerical model.

PROBLEM DEFINITION

Residual stresses as a result of welding process for metal plate are predicted by use of software package *ANSYS*. It is assumed that electric arc welding has been applied to the *ST-37* plate in a single pass in the numerical analysis. Butt welded plate is modelled as 2D and its geometry and dimensions are given Fig. 1.

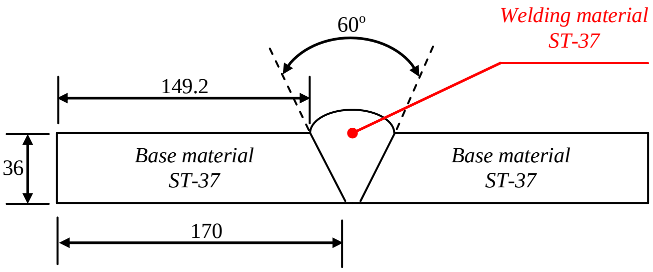


Figure 1 Geometry and dimensions of welded plate

Accurate material data in the high temperature region is in general difficult to obtain and becomes at best a reasonable approximation. However, the material model and relevant properties need only to represent the real material behaviour with sufficient accuracy [6].

The material of the weld metal and the base metal are assumed to be the same. The material considered is mild steel with the grade of steel *ST-37*. The thermal and mechanical properties of the weld material are assumed to have temperature dependent values, although the base material is considered to have constant values. All told values are listed in Table 1. In addition to this, the properties of weld material are illustrated as the function of temperature in Fig. 2 (a-c).

Table 1 Thermo-mechanical properties of base material [14]

Density	7841.72 kg/m^3
Thermal conductivity	52 W/mK
Specific heat	450 J/kgK
Young Modulus	2.1 10^{11} Pa
Poisson Ratio	0.276

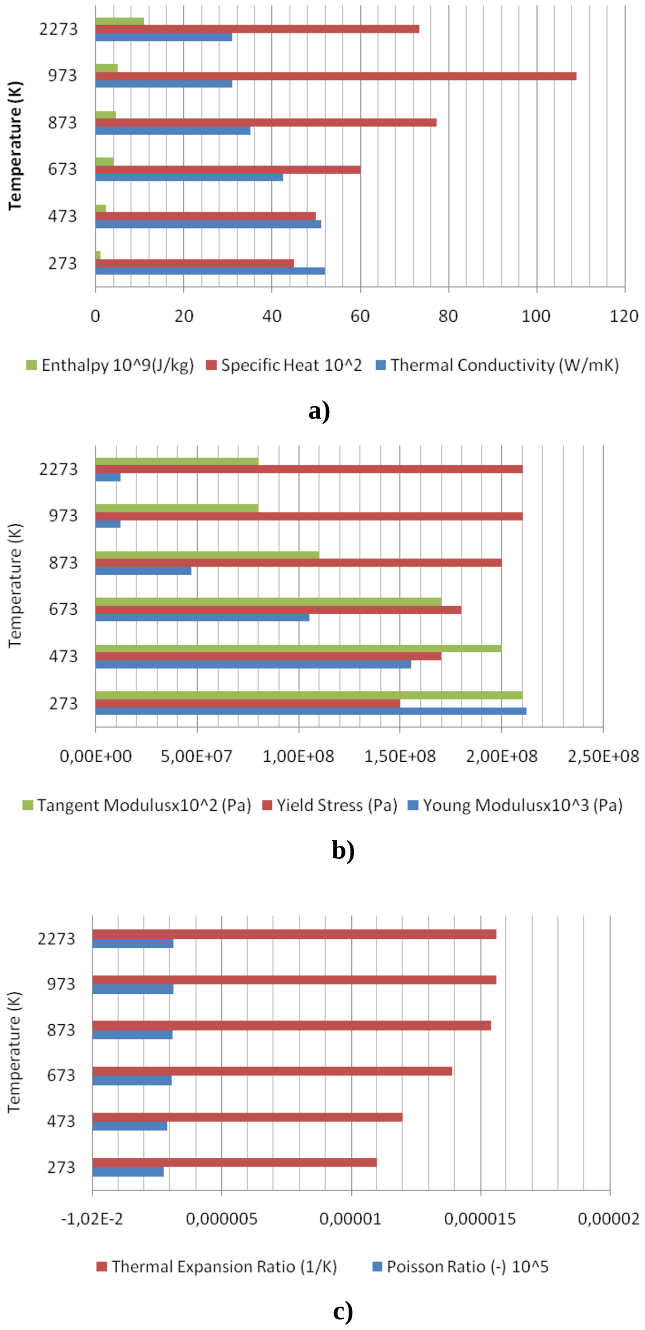


Figure 2 Variation of the thermal and mechanical properties of weld material with respect to temperature, **a)** Enthalpy, Specific Heat and Thermal Conductivity [6, 14-16], **b)** Tangent modulus, Yield stress, Young modulus, **c)** Thermal Expansion Coefficient and Poisson ratio [6, 14, 15]

THERMAL MODEL

Transient analysis is conducted in the thermal study. The initial temperature of *ST-37* is considered 300 K, whilst the temperature of welded zone is assumed to be 1773 K. The convective heat transfer coefficients on the surfaces are estimated by using engineering formulae for natural convection as 13 Wm^2/K (Fig. 2a). Environment temperature is assumed as 300 K and transient analysis is continued till plate temperature is reached. Thermal boundary conditions are exhibited in Fig. 3.

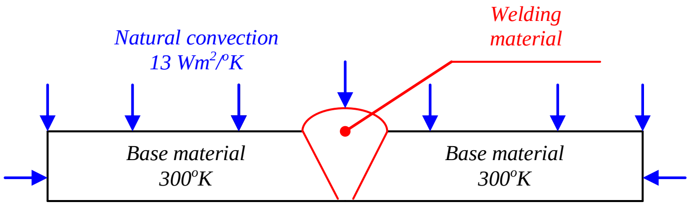


Figure 3 Boundary and initial conditions of the transient thermal analysis model

In the numerical model *Plane55* element type is used. Thermal model consists of 7959 elements and 8199 nodes. A view of whole mesh structure and an enlarged zoomed out mesh are shown in Fig. 4.

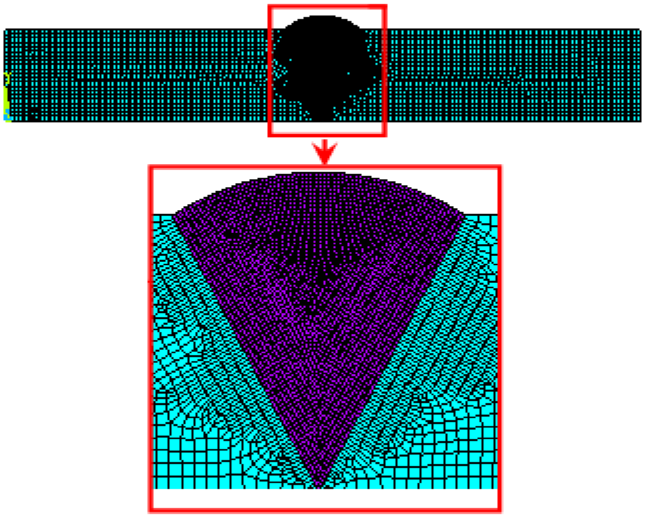


Figure 4 Finite element model of the welded plate

STRUCTURAL MODEL

Plane182 element type is used in structural model. The model consists of 7959 elements and 8199 nodes. Boundary conditions of the structural analysis are shown in Fig. 5.

Temperature distribution obtained from thermal analysis for all analysis time is saved and used structural analysis as boundary condition of plate. Variation of mechanical properties of the weld material with the temperature is given Fig. 2(b-c). Mechanical behaviour of weld material is assumed to be bilinear elasto-plastic.

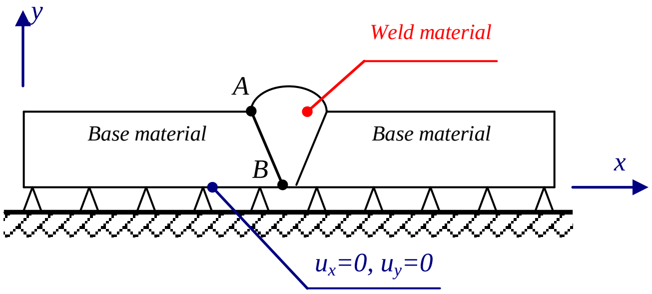


Figure 5 Boundary conditions of the structural analysis model

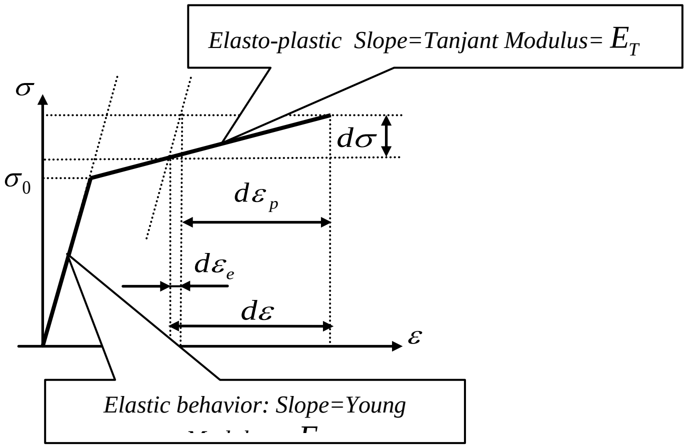


Figure 6 Bilinear elasto-plastic material

The Ludwick stress–strain relation in plastic region is given as,

$$\sigma = \sigma_0 + K\varepsilon_p^n \tag{1}$$

where σ_0, ε_p, n and K are the yield stress, the true strain, the strain hardening parameter and plastic constant, respectively. The initial slope of the curve is taken as the Young modulus of the material (Fig. 6). At the specified yield stress, the curve continues along the second slope defined by the tangent modulus, E_T . Plastic constant and total strain can be written as

$$K = \frac{d\sigma}{d\varepsilon_p} \tag{2}$$

$$d\varepsilon = d\varepsilon_e + d\varepsilon_p \tag{3}$$

where $d\varepsilon_e$ is elastic strain and $d\varepsilon_p$ is plastic strain. Using Eqs.(2) and (3) tangent modulus is obtained as [17]

$$E_T = \frac{KE}{K+E} \tag{4}$$

In structural study two separate analyses is performed. The first one is to find plastic stresses and the second one is to obtain the residual stresses. In the first analysis, temperature distributions as thermal load obtained from thermal analysis every time step have been applied to the plate surface and plastic strains are calculated when the stresses exceed to the yield stress. In the second analysis, load is removed consecutively in order to obtain residual stress after last time step where is obtained environment temperature on plate as shown in Fig. 7. This figure illustrates a typical load history for a nonlinear analysis. In the load steps 2 for Fig. 7, the Newton–Rhapson Iteration Method is used. On the other hand, high local displacements occurred in 2*D* finite element model near the weld material. Hence, to obtain more sensitive results, geometrically nonlinear analyses are carried out in this study.

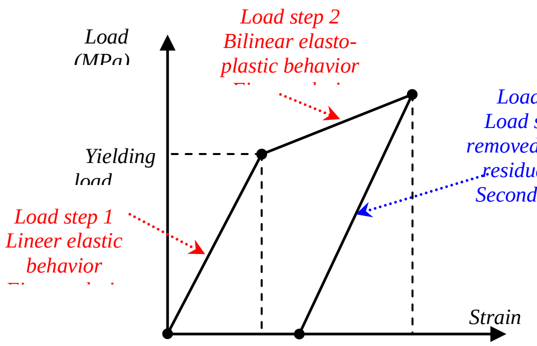
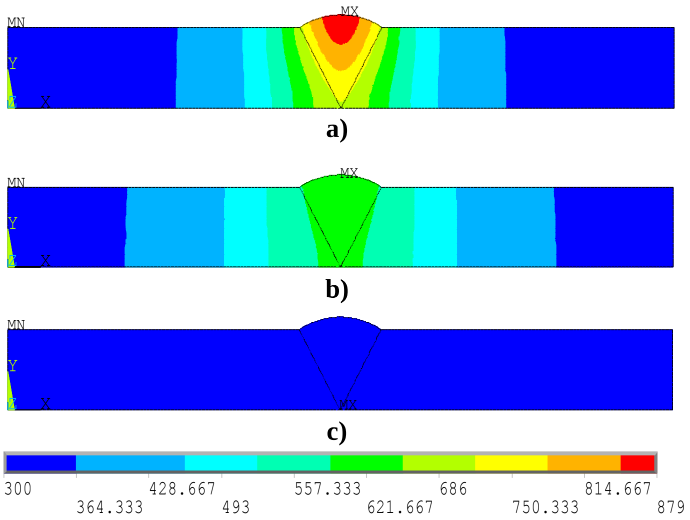


Figure 7 Typical load history for a nonlinear analysis [17]

RESULTS AND DISCUSSIONS

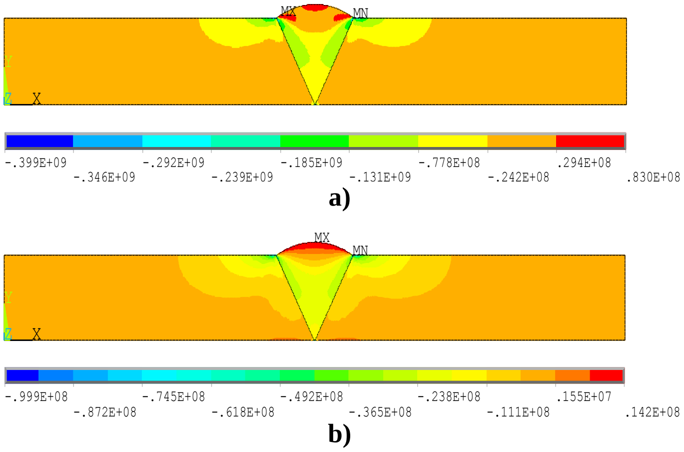
Temperature distribution on welded plate is given in Fig. 8 for various time steps. By checking the temperature contour plots, it can be evidently seen that, the plate temperature reaches to environment temperature 300 K in about 3800 s. However, there are still small differences between temperatures on the plate. The temperature on whole plate is equal to the 300 K when time step is 7000 s. This situation is the equilibrium condition.



a)
b)
c)

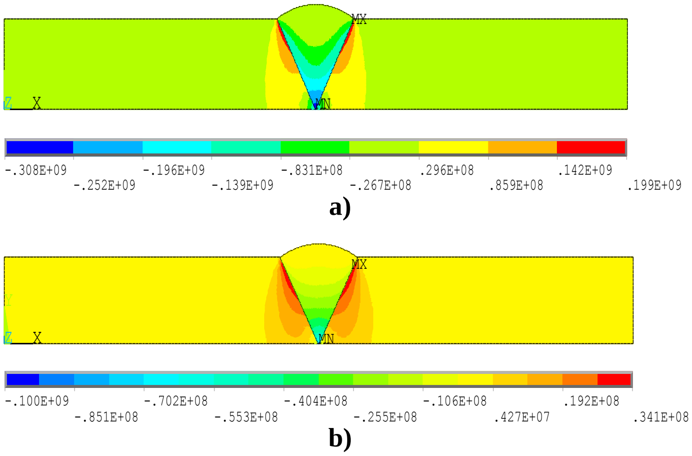
Figure 8 Temperature distributions on welded plate (K), **a)** 100 s, **b)** 200 s, **c)** 3800 s

Distribution of plastic and residual stresses in the last step is equal to the 7000 s and given in Figs. 9 to 11. Maximum tensile stress is found as 0.199 10^9 Pa, that is presented by σ_y on interface between weld and base material. However, maximum compressive stress is obtained as -0.399 10^9 Pa for σ_x top of the base material on interface (pls. check point A in Fig. 5).



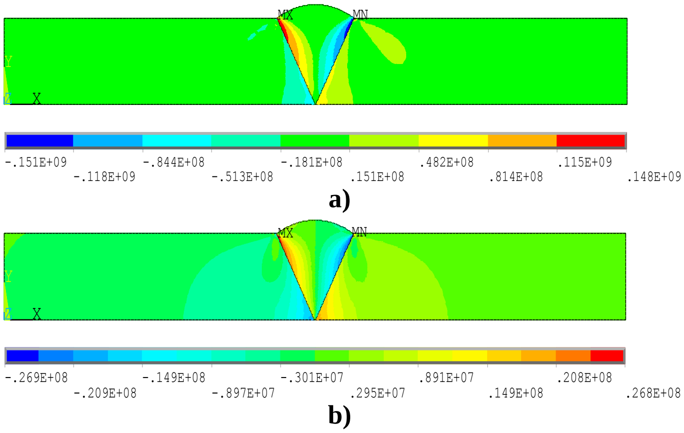
a)
b)

Figure 9 σ_x **a)** plastic, **b)** residual stress distribution on welded plate (Pa)



a)
b)

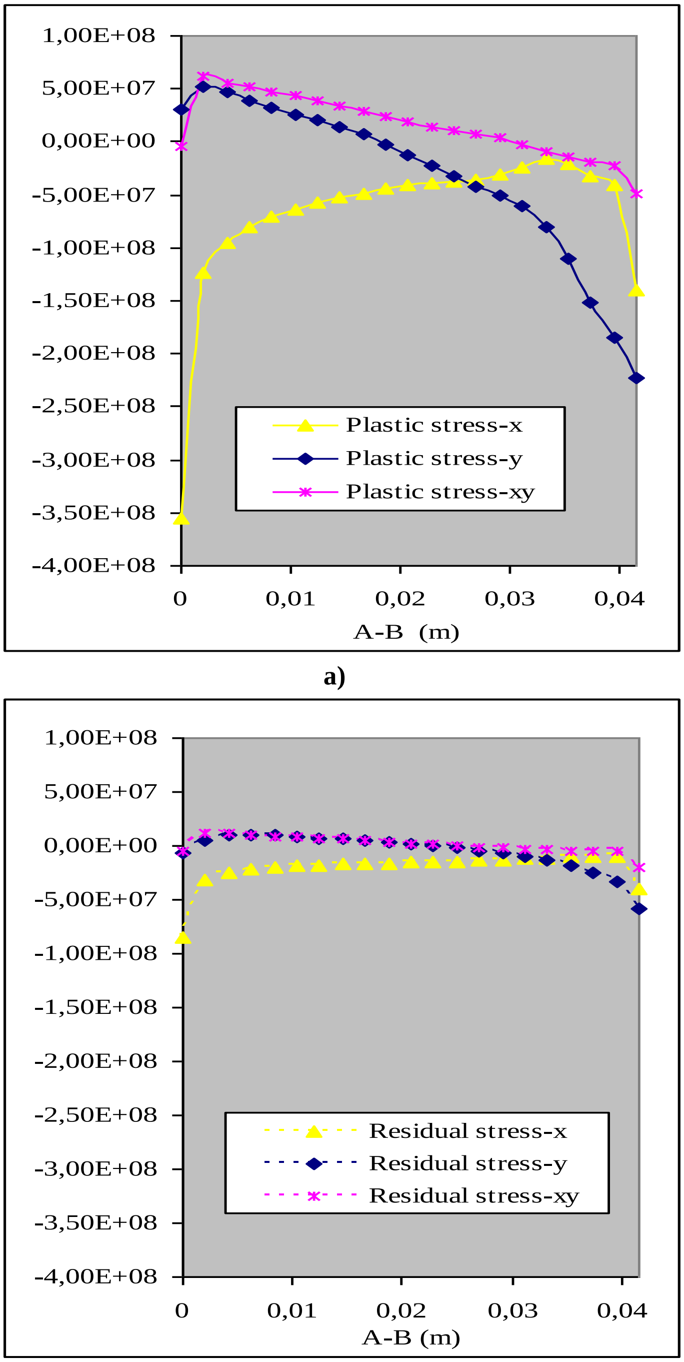
Figure 10 σ_y **a)** plastic, **b)** residual stress distribution on welded plate (Pa)



a)
b)

Figure 11τ_{xy} of **a)** plastic, **b)** residual stress distribution on welded plate (Pa)

Generally, it is observed that stress concentrations are occurred on interface between weld and base material. Because of this reason and in order to compare differences between stress components, values of plastic and residual stresses on A-B path are showed in Fig. 12. It can be seen from Fig. 12 that residual stresses represent similar characteristic behaviours as well as plastic stresses but their values are very small according to the plastic values.



a)

b)

Figure 12 Variation of **a)** plastic **b)** residual stress with A-B path

CONCLUSIONS

In this study, a simulation of electric arc welding is conducted numerically by using finite element method. Two-dimensional model of butt welded *ST-37* plate is analyzed in *ANSYS* finite element package software [18]. Transient thermal distribution on *ST-37* plate is obtained for all time intervals until it reaches 7000 s. In accordance to this, maximum residual stresses are calculated as σ_x in compression and σ_y in tensile on weld interface. Using this residual stress distribution, plastic zone dimensions can be plotted. Moreover, if this stresses remains small values, it can be caused to increase of joint strength.

REFERENCES

[1] Teng, T.L., Fung, C.P., Chang, P.H., Yang, W.C., Analysis of residual stress and distortions in T-joint filet welds, *International Journal of Pressure Vessels and Piping*, 2001,78, pp.523-538.

[2] Pang HL, Pukas SR, Residual stress measurements in a Cruci-form welded joint using hole drilling and strain gauges, *Strain*, 1989, pp.7-14.

[3] Cheng W, Finnie IA, Method for measurement of axisymmetric axial residual stresses in circumferentially welded thin-walled cylinders, *Journal of Engineering Material Technology*, 1985, pp.181-185.

[4] Masubuchi, K,., Martin, D.C., Investigation of residual stresses by use of hydrogen cracking, *Welding Journal*,1961, pp.553-563.

[5] Barsoum, Z. Lundbäck, A., Simplified FE welding simulation of fillet welds – 3D effects on the formation residual stresses, *Engineering Failure Analysis*, 2009, pp.2281–2289.

[6] Barsoum, Z. Barsoum, I. Residual stress effects on fatigue life of welded structures using LEFM, *Engineering Failure Analysis*, 2009, pp.449–467.

[7] Teng T.L., Chang, P.H., Tseng, W.C., Effect of welding sequences on residual stresses, *Computers and Structures*, 2003, pp.273–286.

[8] Lindgren, L.E., Finite element modelling and simulation of welding, Part1: increasing complexity, *Journal of Thermal Stresses*, 2001, pp.141–192.

[9] Lindgren, L.E., Finite element modelling and simulation of welding, Part2: improved material modelling, *Journal of Thermal Stresses*, 2001, pp.195–231.

[10] Lindgren, L.E., Finite element modelling and simulation of welding, Part3: efficiency and integration, *Journal of Thermal Stresses* 2001, pp.305–334.

[11] Kong, F., Ma, J. and Kovacevic, R., Numerical and experimental study of thermally induced residual stress in the hybrid laser–GMA welding process, *Journal of Materials Processing Technology*, 2011, pp. 1102–1111.

[12] Zain-ul-abdein, M., Nélias D., Jullien, J.F., Boitout, F., Dischert L., Noe, X. Finite element analysis of metallurgical phase transformations in AA 6056-T4 and their effects upon the residual stress and distortion states of a laser welded T-joint, *International Journal of Pressure Vessels and Piping*, 2011, pp. 45-56.

[13] H. Long , D. Gery, A. Carlier, P.G. Maropoulos, Prediction of welding distortion in butt joint of thin plates, *Materials and Design* 2009, pp. 4126–4135.

[14] Benli S., An investigation of residual stresses in welded parts, *Master Thesis, Dokuz Eylul University, Graduate School of Natural and Applied Sciences*, 2004.

[15] Teng, T.L., Chang, P.H., Tseng, W.C. Effect of welding sequences on residual stresses, *Computers and Structures*, 2002, pp.273-286.

[16] Celik, Y., The computer simulation of the welding processes that occurring residual stress and distortion, *Master Thesis, Sakarya University, Graduate School of Natural and Applied Sciences*, 2011.

[17] Karakuzu, R. Aslan Z., Okutan, B., The effect of ply number, orientation angle and bonding type on residual stresses of woven steel fiber reinforced thermoplastic laminated composite plates subjected to transverse uniform load, *Composites Science and Technology*, 2004, pp.1049–1056.

[18]ANSYS 12.1 Academic Teaching Introductory Help Menu, 2009.