

Finite Element Analysis of Residual Stress in Butt Welding Two Similar Plates

Dragi Stamenković, MSc (Eng)¹⁾
Ivana Vasović, BSc (Eng)²⁾

In this paper, Manual Metal Arc Welding of carbon steel plates was studied. The finite element analysis of residual stresses in butt welding of two similar plates is performed with the ANSYS software. This analysis includes a finite element model for the thermal and mechanical welding simulation. It also includes a moving heat source, material deposit, temperature dependant material properties, metal plasticity and elasticity, transient heat transfer and mechanical analysis. The welding simulation was considered as a sequential coupled thermo-mechanical analysis and the element birth and death technique was employed for the simulation of filler metal deposition. The residual stress distribution and magnitude in the axial direction was obtained. A good agreement between the computation and experimental results is obtained.

Key words: welding, welding joint, plate, residual stresses, stress analysis, finite element method.

Introduction

WELDING is widely used in automotive industries to assemble various products. It is well known that the welding process relies on an intensely localized heat input, which tends to generate undesired residual stresses and deformations in welded structures, especially in the case of thin plates. Therefore, estimating the magnitude of welding deformations and characterizing the effects of the welding conditions are deemed necessary. With modern computing facilities, the finite element (FE) technique has become an effective method for prediction and assessment of welding residual stress and distortions [1–12]. However, the welding deformations are various with production variations such as dimension, welding materials and welding process parameters. Therefore, rapidly and accurately predicting welding induced distortion for real engineering applications is more challenging.

In many high temperature applications, it is necessary to join together components of same or different chemical, physical and mechanical characteristics. Undoubtedly, the joining of dissimilar metals is more challenging than that of similar metals due to differences in the properties of the base metals welded.

Firstly, welded structures must meet the strength requirements and the probability of defect formation must be estimated and monitored. Moreover, residual stresses may exist in any welded structures. Welding residual stresses may be very complex and their distribution is very difficult to predict.

Many techniques have been used for measuring residual stresses in metals including stress relaxation techniques, diffraction techniques, cracking techniques and techniques by use of stress sensitive properties. These techniques cannot obtain complete stress distribution and most of them are costly and time consuming and some of them are

destructive. In recent years, numerical analyses are established to solve the complex engineering problems and among them evaluation weld-induced residual stresses. The finite element method is the conventional means of calculating residual stresses.

In this paper, the finite element analysis is used to perform welding simulation and to predict weld-induced residual stresses in butt welding of two similar carbon steel plates. This analysis includes moving heat source, material deposit, temperature dependant material properties, metal plasticity and elasticity, transient heat transfer and mechanical analysis.

Theoretical consideration

Theoretically the welding process can be considered either using a thermal or a mechanical analysis. Thermal stresses induced during a welding process are obtained from the temperature distributions determined by the thermal model. The residual stresses from each temperature increment are added to the nodal point location to determine the updated behavior of the model before the next temperature increment.

There are two distinct methods: sequential and direct, in a coupled-field analysis. Which procedure for a coupled-field analysis will be used depends on which fields are being coupled. The sequential method involves two or more sequential analyses that belong to a different field. Adversely, the direct method usually involves just one analysis that uses a coupled-field element type containing all necessary degrees of freedom.

In this work, the process of welding is simulated by the FE method. The welding process computation can be split into two solution steps: thermal and mechanical analyses. First, the temperature and phase evolution are determined as a function of time in the thermal analysis. Then, the mechanical analysis

¹⁾ Termoelektro a.d., Uralska 9, 11060 Belgrade, SERBIA

²⁾ GOŠA Institut, Milana Rakića 35, 11000 Belgrade, SERBIA

employs the previous results to get displacements at nodes and stresses at integration points. Since the thermal field has a strong influence on the stress field with little inverse influence, a sequentially coupled analysis works very well. Moreover, a 3-D FE analysis is the optimum method of ascertaining the thermal cycle of welding. Therefore, in this paper, the welding process is simulated using a sequentially coupled 3-D thermo-mechanical FE formulation based on the ANSYS code. For both the thermal and mechanical analyses, temperature-dependent thermo-physical and mechanical properties of the materials are incorporated.

To simplify the welding simulation, it is computationally efficient to perform thermal and mechanical analyses separately. It is assumed that changes in the mechanical state do not cause a change in the thermal state. But a change in the thermal state causes a change in the mechanical state. Firstly, the computation of the temperature history during welding and subsequent cooling is completed and this temperature field is applied to the mechanical model as a body force to perform the residual stress analysis.

This work includes FE models for the thermal and mechanical welding simulation. To develop suitable welding numerical models it necessary to consider the process parameter (welding speed, number and sequence of passes, filling material supplying, etc.), the geometrical constraints, the material nonlinearities and all physical phenomena involved in welding. Therefore it is a great challenge to consider all factors at the same time; so generally the models include some approximations: in the works [1–12] we can find some attempts to reduce modeling efforts and computer time. This work deals with the following main assumptions and features about the thermal model:

- (i) the displacements of the parts, during the welding, do not affect the thermal distribution of the parts themselves;
- (ii) all the material properties are described till to the liquid phase of metal;
- (iii) convection and radiation effects are considered;
- (iv) the element birth and death procedure is used.

The thermal analysis is the first step and during this phase the distributions of the temperatures are calculated and saved for every load step. Here it is assumed that the thermal calculation at a given time is independent from the structural results obtained at a previous time according to the point (i): so the thermal and the mechanical analysis can be uncoupled.

Numerical example

The welding process of a butt-weld joint of two ASTM 36 steel plates with the dimensions shown in Fig.1 was simulated.

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. 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 the welding arc stayed at an element with a constant specific volume heat generation, and then moved to the next element at the end of the load step as the welding was finished, see Fig.2. The element type SOLID70, which has a single degree of freedom, was used for the thermal analysis. For the structural analysis the element type SOLID45, with three translational degrees of freedom at each node, was used. Fig.3 shows the meshed model used in the analysis.

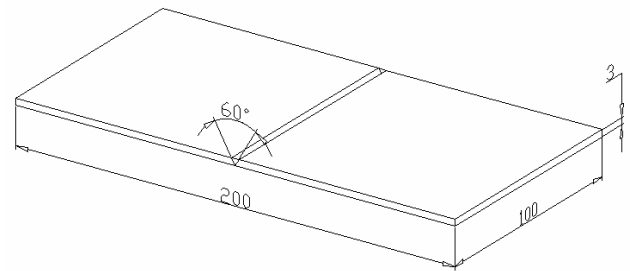


Figure 1. Geometry of the model used in the analysis

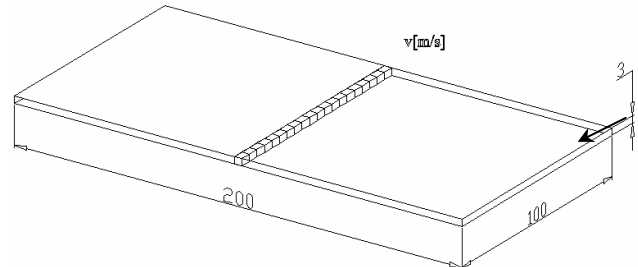


Figure 2. Time simplification of a moving heat source in 3D welding simulation

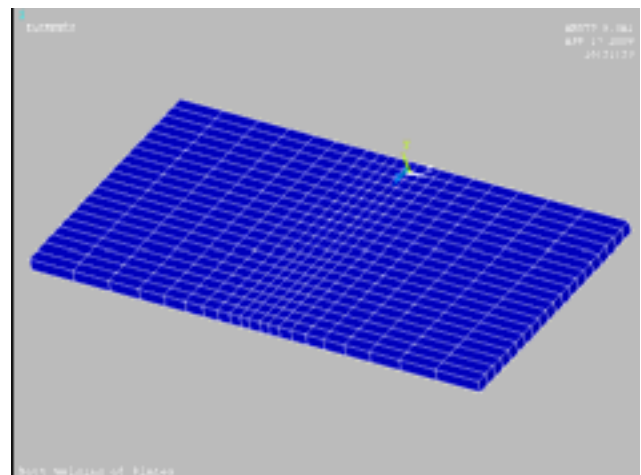


Figure 3. Meshed model used in the analysis

The temperature dependant thermal material properties for the plates, heat affected zone (HAZ) and the filler weld material were assumed to be the same, see Table 1. For the mechanical material properties, same material models were used for the weld beads and the base materials according to the yield strength. The plasticity material model used was von Mises rate-independent isotropic bilinear hardening.

Table 1. Material properties

Temperature (°C)	Specific heat (J/kg°C)	Conductivity (W/m°C)	Density (kgm-3)	Yield stress (MPa)	Thermal expansion coefficient (10-5/°C)	Young's modulus (GPa)	Poisson's ratio
0	480	60	7880	380	1.15	210	0.3
100	500	50	7880	340	1.2	200	0.3
200	520	45	7800	315	1.3	200	0.3
400	650	38	7760	230	1.42	170	0.3
600	750	30	7600	110	1.45	80	0.3
800	1000	25	7520	30	1.45	35	0.3
1000	1200	26	7390	25	1.45	20	0.3
1200	1400	28	7300	20	1.45	15	0.3
1400	1600	37	7250	18	1.45	10	0.3
1550	1700	37	7180	15	1.45	10	0.3

The heat input during welding is modeled in the ANSYS by the equivalent heat input which includes body heat flux. The amount of heat input was calculated as follows:

$$Q = \eta \frac{UI}{V} \quad (1)$$

where: η is the arc efficiency, V is the travel speed, U and I are the arc voltage and the current, respectively. In this analysis, the current was 180A, the voltage was 24V and the welding speed was 5 mm (sec)^{-1} , while the arc efficiency of the process considered to be 85%.

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 stayed at an element with constant specific volume heat flux, and then moved to the next element at the end of the load step as the welding was finished.

Figures 4 and 5 illustrate the temperature fields of the butt welding plates at $t=5.15 \text{ sec}$ after the start of welding, respectively.

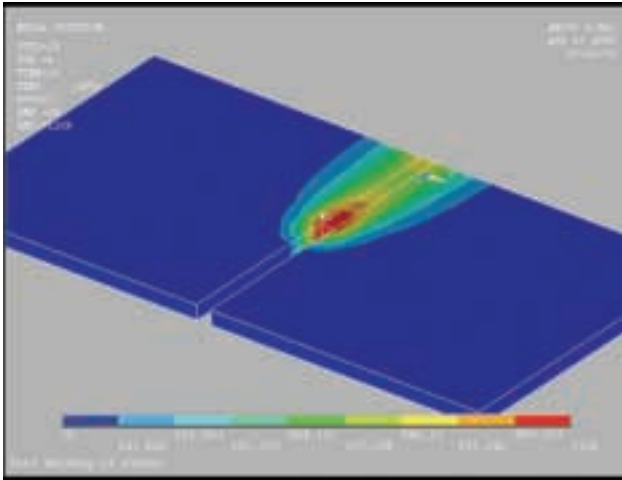


Figure 4. Temperature distribution at $t = 5 \text{ sec}$

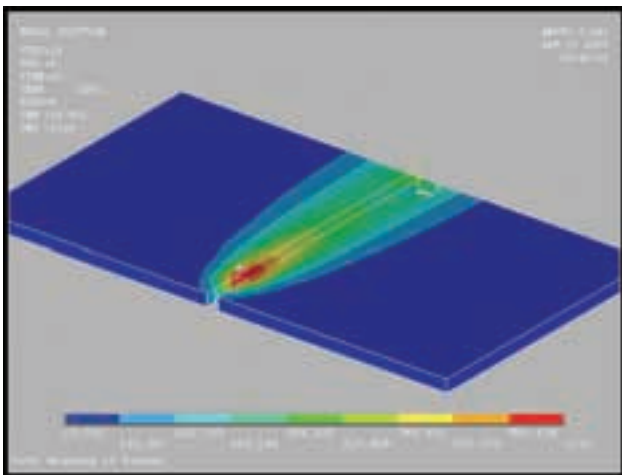


Figure 5. Temperature distribution at $t = 15 \text{ sec}$

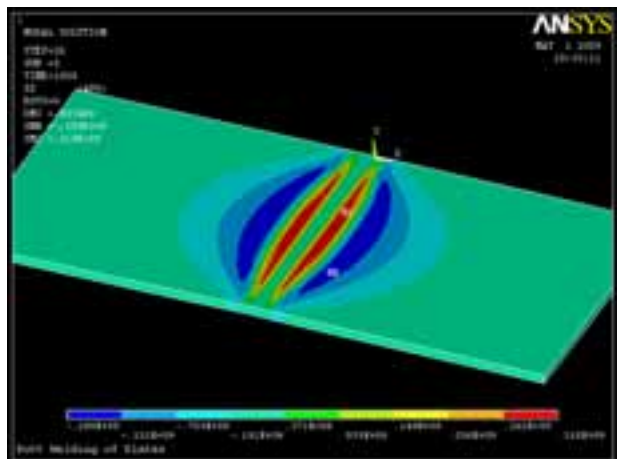


Figure 6. Residual stresses in the axial directions

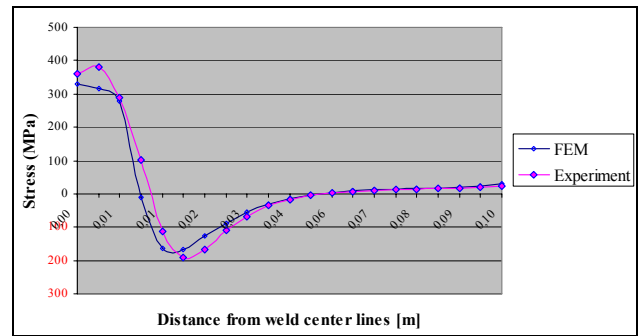


Figure 7. Comparison of the FEM with experimental results for axial residual stresses

The values of axial residual stresses calculated by the finite element method are compared with the experiments, Fig.7. These results show that the computation finite element results are very close to the experimental results.

Conclusions

This work presents the FE model for numerical simulation of welding residual stresses in high strength carbon steel butt welds. The finite element method is an efficient technique in analyzing residual stresses in welding processes. A three-dimensional finite element welding simulation was carried out on a one-pass Manual Metal Arc Welding plain joint structure. The welding simulation was considered as a sequential coupled thermo-mechanical analysis and the element birth and death technique was employed for the simulation of filler metal deposition. The finite element analysis results of the residual stress distributions of two butt welded plates in the axial directions are presented in Fig.6. Thus the values of the axial residual stresses calculated by the finite element method and experiment are shown in Fig.7. These results show that the present results obtained by the finite element method are very close to the experimental results.

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. In order to obtain results using the finite element method very close to the results obtained in experiments, different mesh sizes could be taken into consideration.

References

- [1] MEDVEDEV,S.V.: *Computer modeling of residual welding strains in technological design of welded structures*, Welding International. - 2002, Vol.16(1), pp.59-65.
- [2] BONIFAZ,E.A.: *Finite element analysis of heat flow in single-pass arc welds*, Welding Res. Supplement, 2000, pp:121-125.
- [3] DENG,D., MURAKAWA,H.: *Numerical simulation of temperature field and residual stresses in multi-pass welds in stainless steel pipe and comparison with experimental measurements*, Comput. Mater. Sci., 2006, 37:269-277.
- [4] HANSEN,J.L.: *Numerical modeling of welding induced stresses*, PhD thesis, Technical University of Denmark, ISBN 87-90855-52-3, 2003.
- [5] JONSSON,M., KARLSSON,L., LINDGREN,L.E.: *Deformations and stresses in butt-welding of large plates with special reference to the mechanical material properties*, J. of Eng. Mat. and tech., 1985, Vol.107, pp.256-270.
- [6] WIKANDER,L., KARLSSON,L., NASTROM,M., WEBSTER,P.: *Finite element simulation and measurement of welding residual stresses*, Modelling Simul.mater.Sci.Eng.,1994, Vol.2, pp.845-864.

- [7] GOLDAK, J., BIBBY, M., MOORE, J., HOUSE, R., PATEL, B.: *Computer modeling of heat flow in welds*, Metallurgical transaction B, 1986, Vol.17B, pp.587-600.
- [8] FRICKE, S., E. keim and SCHMIDT, J.: *Numerical weld modeling a method for calculating weld-induced residual stresses*, Nucl.Eng.Des., 2001, 206:139-150.
- [9] HSIANG, P.C., LIANG, T.T.: *Numerical and experimental investigations on the residual stresses of the butt-welded joints*, Int. J. Comput. Mater.Sci., 2004, 29:511-522.
- [10] LINDGREN, L.E., KARLSSON, L.: *Deformations and stresses in welding of shell structures*, Int. J. Numerical Methods Eng., 1998, 25:635-655.
- [11] SARKANI, S., TRITCHKOV, V., MICHAELOV, G.: *An efficient approach for computing residual stresses in welded joints*, Finite Element Anal. Des., 2000, 35:247-268.
- [12] MURTHY, Y.V.L.N., RAO, G.V., IYER, P.K.: *Numerical simulation of welding and quenching processes using transient thermal and thermo-elasto-plastic formulations*, Comput. Struct., 1996, 60:131-154.

Received: 09.02.2009.

Primena metode konačnih elemenata u analizi zaostalih napona kod sućeono zavarenih ploča

U radu je razmatran problem zavarivanja ugljeničnih čelika. Određivanje zaostalih napona u zavarenom spoju je određena primenom metode konačnih elemenata koristeći softverski paket ANSYS. Ova analiza obuhvata kretanje toplotnog izvora, depozit materijala, promene karakteristika materijala sa temperaturom, elastično i plastično ponašanje materijala, prenos toplote i mehaničke analize. Ovom analizom su određeni zaostali naponi u aksijalnom i transverzalnou pravcu. Simulacija procesa zavarivanja je razmatrana kao sekvenca spregnutih termo-mehaničkih analiza kao i korišćenjem tehnike radanja i nestajanja za simulaciju zavara u spojnom žljebu. Zaostali naponi kao posledica procesa zavarivanja su određeni. Dobijeno je dobro slaganje između rezultata proračuna i eksperimenta.

Ključne reči: zavarivanje, zavareni spoj, ploča, zaostali naponi, analiza napona, metoda konačnih elemenata.

Применение метода конечных элементов в анализе остаточных напряжений у встык сварённых плит

В настоящей работе рассматривана проблема сваривания углеродистых сталей. Определение остаточных напряжений в сварном соединении установлено применением метода конечных элементов при пользовании коммерческого программного обеспечения АНСЮС. Этот анализ охватывает движение теплового источника, депонирование материала, изменения характеристик материала с температурой, адаптационную и пластическую способность материала, теплопередачу и механические анализы. Этим анализом определены остаточные напряжения в аксиальном и трансверсальном направлениях. Моделирование процесса сваривания рассматривано в роли последовательности соединённых термо-механических анализов, а в том числе и пользования техники *рождения* и *умирания* для моделирования сварки в соединительной канавке.

Ключевые слова: сварка, сварное соединение, плита, остаточные напряжения, анализ напряжений, метод конечных элементов.

Emploi de la méthode des éléments finis dans l'analyse des tensions résiduelles chez les plaques soudées en bout

Ce papier considère le problème du soudage de l'acier carbonique. La détermination des tensions résiduelles chez la soudure est effectuée par l'application de la méthode des éléments finis et en utilisant le progiciel commercial ANSYS. Cette analyse englobe le mouvement de la source de chaleur, le dépôt du matériel, les changements des caractéristiques des matériaux avec la température, le comportement plastique et élastique des matériaux, le transfert de la chaleur et les analyses mécaniques. On a déterminé les tensions résiduelles dans les directions axiales et transversales via cette analyse. La simulation du procédé du soudage est considérée comme la séquence des analyses thermomécaniques couplées ainsi que par l'emploi de la technique de la naissance et de la mort pour la simulation de la soudure chez la rainure de jointe.

Mots clés: soudage, soudure, plaque, tensions résiduelles, analyse de la tension, méthode des éléments finis.