THERMAL FIELD AND RESIDUAL STRESSES IN THE WELDED JOINT BETWEEN WEB GIRDER AND DECK PLATE

Dan Birsan[1]

[1] *"Dunarea de Jos" University of Galati, dbirsan@ugal.ro.*

Abstract: *The thermal field influences the phase transformations during welding and therefore the microstructure and mechanical properties of the welded joint. It is also responsible for the occurrence of residual stresses and deformations in welded joints. In the welding processes the shape of the thermal field and its variation in time depend on many factors, but admitting some simplifying hypotheses, calculation relations can be obtained for different practical situations. The simplifying hypotheses are related, first of all, to the homogeneity and isotropy of the bodies. A 3D FEM model has been modeled in order to simulate the welding process. It is assumed that the small cut-outs for the bulb profiles will not influence the stiffness of the web very much, and they have therefore been omitted from the model.*

Keywords: *weld, naval structure, finite element*

1. Introduction

The works published in the literature reveal important research and simulations of welding processes, through one or more passes, in combination with one or more heat sources. Most of the undesirable consequences of welding processes are caused by the nonlinear heat flow introduced by moving heat sources. Therefore, this paper presents the evolution in modeling of the T-welding processes with one heat source.

In the years 1960-1970, most of the research focused on the study of the thermo-physical behavior of materials during the welding process. Prior to 1970, finite difference methods (DEF) were developed for nonlinear analysis of heat transfer in welding. In the late 1970s, countless efforts were made to develop computer source codes for analyzing the complex mechanism of heat flux variation in welding processes. In recent years, Lindgren has published a detailed three-part analysis [5], [6], [7] that has studied the entire welding process in its complexity. Another recent and comprehensive study was conducted by Dr. Anas Yaghi and Adib Becker [8].

The main objective of this paper is to describe the major contributions of research in the field, especially to the welding of naval structures. The first step towards simulating the welding process was the model of the moving heat source, presented by Rosenthal [9], for the analytical solution of the temperature distribution for electric arc welding. In his work, the author presented the linear two-dimensional and three-dimensional heat flow, in solid state, of bodies of infinite size. The author validated the experimental model, by measuring the temperatures during welding on sheets with different geometries. Subsequent research has shown that Rosenthal's model gave a good approximation of temperatures in the area of the welded joint, although in the immediate vicinity of the heat source, the modeled temperature field recorded very high values. Subsequently Rybicki [10] and Debicarri [11] developed new models, which gave a better approximation of the temperature distribution in the transient stage of the process. Later, the research conducted by SEO, Yang and Jang [12] studied the linear heat source and the modeling of the heat developed by the electric arc. Goldak, in his paper [13], introduced the notion of predefined temperature for certain areas of the welded joint. A more complex model is the one developed by Goldak, representing the model of the heat source with Gaussian distribution [14], [15]. This model, known as the ellipsoidal heat source model, is by far the most widely used in modeling and simulating welding processes. Goldak also developed the model of the different distribution of heat flow introduced into the parts, which take into account parameters and sizes that other researchers did not take into account in the models previously developed.

2. Numerical Model Development

In the case of welding by fusion, temperature field profile depends on primary welding parameters - voltage, amperage, welding speed - thermo-physical properties of the base material - specific heat, mass density, thermal conductibility, thermal diffusivity and heat loss by convection and radiation. Two double-ellipsoidal heat sources are considered in simulation of the stiffeners on the bulkhead.

The 3D FEM model consist by a deck plate (150x100x1.5mm) and a web plate (150x22x1.5mm), has been modeled in order to simulate the welding process. The mesh is fine around these weld lines, and coarse in the far field, as shown in Fig. 1.

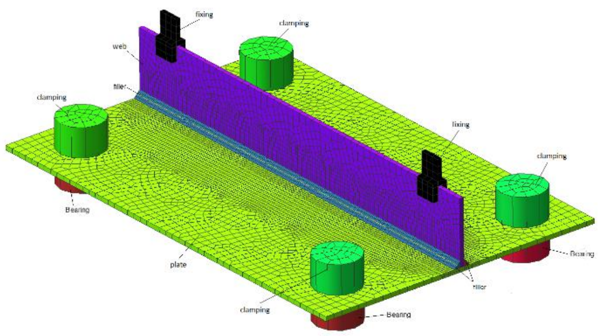

Figure 1: *3D FE model*

The deck plate is constraint with bearing and clamping and the web plate with fixing elements. The web plate is welded by deck plate through two weld fillets with throat 1.5mm that are alternate welded. The heat is applied using a moving Gaussian heat source. The size of the Gaussian surface is shown in Fig.2. The welding parameters are:

- source velocity: 2.5cm/s;
- current: 180A;
- voltage: 18V;
-efficiency: 0.85.

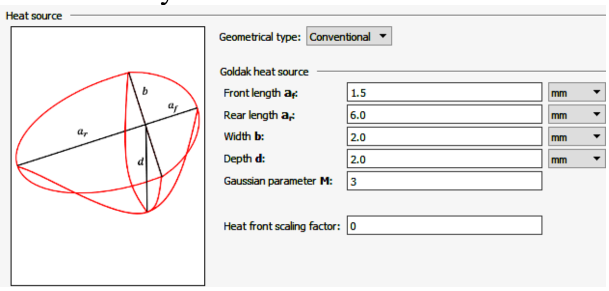

Figure 2: *Dimensions of the heat source*

Values of temperature and flux gradients are predicted in the entire welded joint. Still, they are particularly pursued in and around the fusion zone (FZ) and heat affected zone (HAZ), where metallurgical and mechanical properties are seriously affected. The heat transfer in the structure was modelled as 3D heat transfer problem using the MSC Marc Mentat code.

In this simulation the material used is S235-JMP-MPM low carbon steel. Table 1 show the S235 chemical composition.

Table 1: *S235 chemical composition (%)*

C	Cu	Fe	Mn	N	P	S
0.15	0.4	98.293	1.1	0.01	0.02	0.027

The numerical model has been developed, taking into consideration the following assumptions:

- isotropy of the base metal;
- thermo-physical properties dependent on temperature;
- convection and radiation losses;
- latent heat;
- phase transformation.

3. Results and Discussions

3.1. Temperature Distribution

Modeling, simulation and analysis of thermal fields were performed taking into account all the assumptions and conditions in chapter 2. The welding parameters were kept constant for both passes.

In the Fig. 3 to 6 there are presented the distributions of the thermal fields, generated by the welding source at 3.5 and 9.5 seconds from the beginning of the welding process. It is observed that the shapes of the thermal fields obtained in the molten metal pool are different for the two passes. The molten metal pool for the second fillet is significantly higher than the first, as the second bath benefits from the preheating temperature of the first (see Fig. 7).

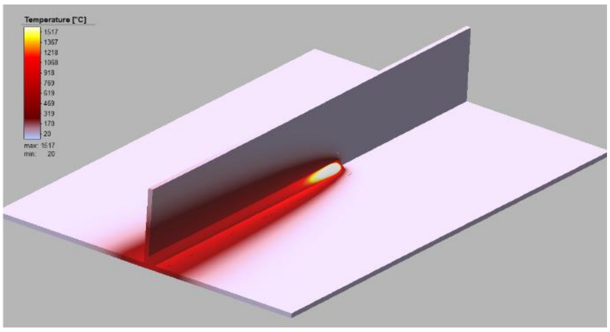

Figure 3: *Thermal field at 3.5s*

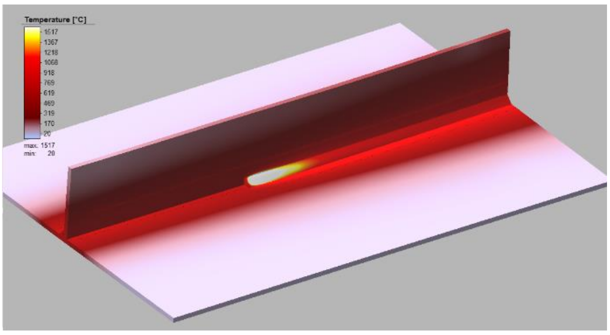

Figure 4: *Thermal field at 9.5s*

The simulation of the thermal transfer in the welded joints, for the case of the mobile thermal source, requires a fine mesh for an area wide enough to include the thermal influence, all along the length of the two parts to be welded. For austenitic steel, the isothermal lines for AC1 and AC3 temperatures and the phase fractions of ferrite, pearlite, austenite, bainite and martensite at a moment of the welding process, are shown in Fig. 7.

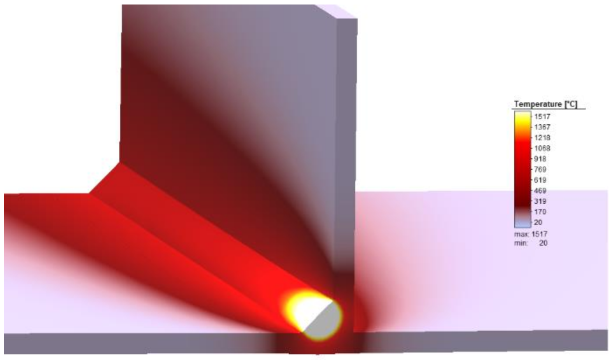

Figure 5: *Cross section through the molten metal pool at 3.5s*

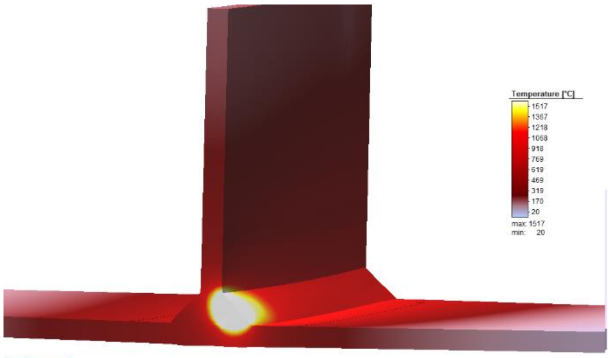

Figure 6: *Cross section through the molten metal pool at 9.5s*

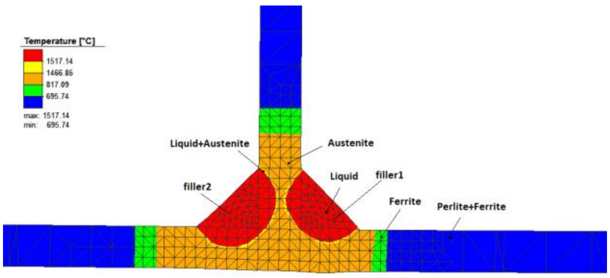

Figure 7: *Phase fractions in two moments during welding*

The temperature variation in time of the selected nodes (see Fig. 8) from the deck and web plates are shown in Figs. 9 and 10. Can be observed the maximum temperature reached during the welding process for both passes that are highlighted in these charts.

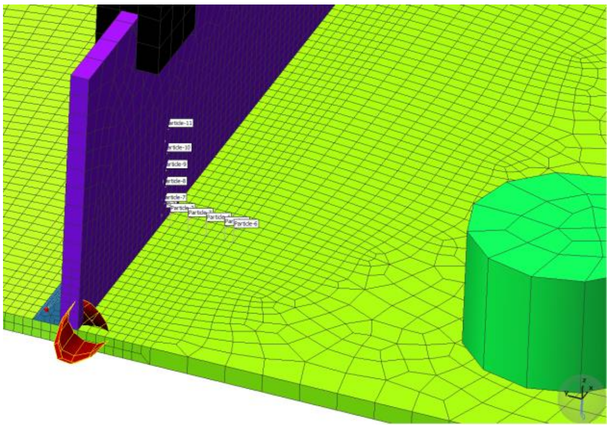

Figure 8: *Selected nodes in deck and web plates for monitoring the temperature and stresses*

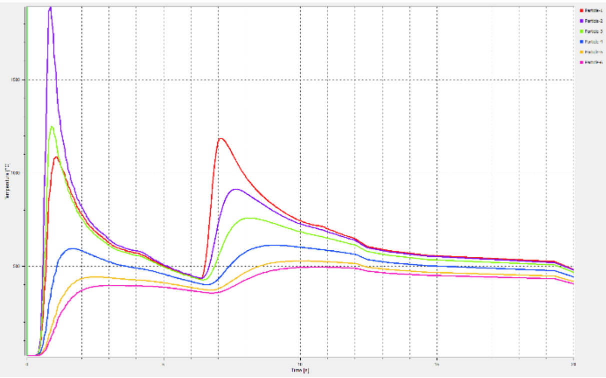

Figure 9: *Thermal cycle in nodes of the deck plate*

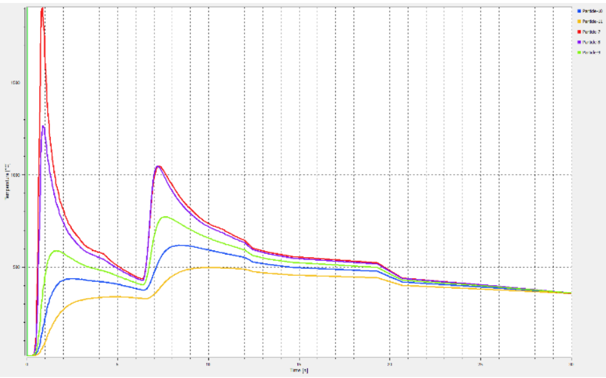

Figure 10: *Thermal cycle in nodes of the web plate*

3.2. Von Mises Stresses Distribution

Modeling stresses in welds includes all distortions that can be predicted by thermal stress analysis.

The vast majority of thermal stress analyses have used thermo-elasto-plastic constitutive models with rate independent plasticity.

In Figs. 11 and 12 can be observed the variation in time of effective stresses in the welded joint for both welding passes. The chosen nodes to be analyzed are situated at the cross direction of the weld pool at the given time of 3.5s from the beginning of the welding process, in longitudinal direction of the deck and web plates (see Fig. 7).

The effective stress of the welded joint depends by its temperature. It can be noticed that the maximum value of the stress it is found on area situated in the front of the weld pool and in the heat affected zone (HAZ). Can be seen in Figs. 13 and 14, that the maximum value of the equivalent stress is approximatively 250MPa.

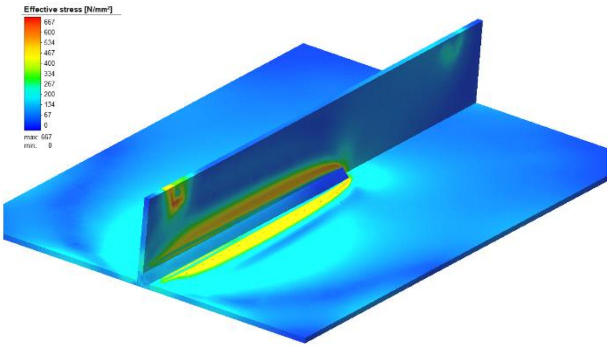

Figure 11: *The Von Mises stresses distribution in plate at 3.5s*

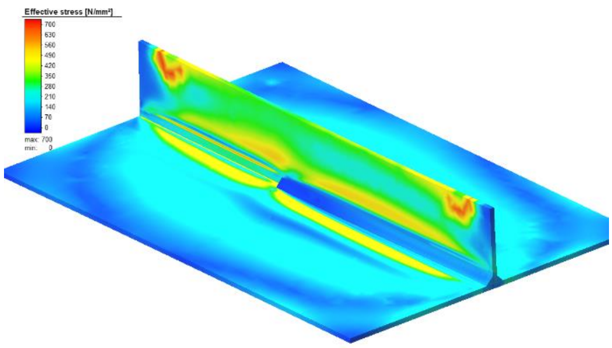

Figure 12: *The Von Mises stresses distribution in plate at 9.5s*

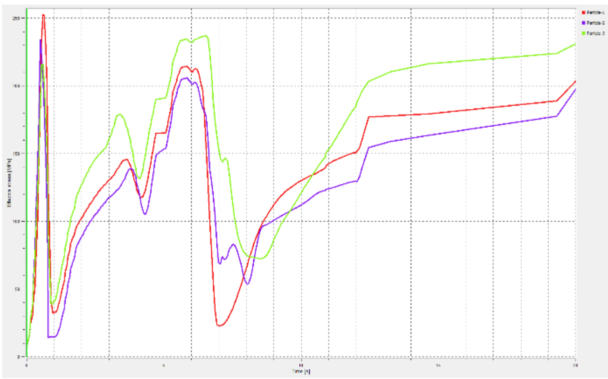

Figure 13: *Effective stress in nodes of the deck plate*

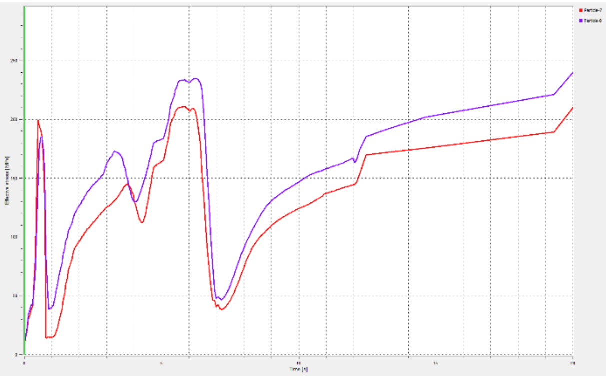

Figure 14: *Effective stress in nodes of the web plate*

4. Conclusions

The welding simulation takes into account the variation with temperature of the thermo-physical-mechanical characteristics of the S235 steel, fact that leads to obtaining of very precise solutions regarding the distribution and the values of the thermal field and the residual stresses.

In this welding simulation a major importance has been given to the phase transformation of the considered material.

The three-dimensional analysis gives us significant data regarding the shape of the welding pool and the isotherms of the thermal field.

The maximum value of the equivalent stress corresponds to the heat affected zone. The increase of the residual stress appears only after finishing the structural transformations.

References

[1] [Goldak, 1984], Goldak J., Chakravarti A. and Bibby M., *A new finite element model for heat sources*, Metallurgical Transactions B, 1984.

[1] [Yapp, 2007], Yapp D., *Management of Weld Quality*, Lecture Notes on Topic MAN1.5, WERC 2007, Cranfield University, Bedford, UK.

[2] [Blodgett], Blodgett O.W., Distortion: *How to minimize it with Sound Design Practices and Controlled Welding Procedures Plus Proven Methods for Straightening Distorted Members*, pp. 1-19, The Lincoln Electric Company, G261

[3] [Hibbitt 1973], Hibbitt H.D., P.V. Marcal, *A Numerical Thermo-mechanical Model for the Welding and Subsequent Loading of a Fabricated Structure*, Journal of Computers and Structures 1973; 3: 1153-1174.

[4] [Doan 2010], Doan T., Tran T.P., Vo T.Q., Kim I.S., *Finite Element Predictions of Residual Stresses and Distorsions in T Joint Fillet Welds*, The 11th Asia Pacific Industrial Engineering and Management Systems Conference, 2010

[5] [Lindgren 2001], Lindgren L.E., *Finite Element Modeling and Simulation. Part 1: Increased Complexity*, Journal of Thermal Stresses, vol. 24, pp. 141-192, 2001.

[6] [Lindgren 2001], Lindgren L.E., *Finite Element Modeling and Simulation. Part 2: Improved Material Modeling*, Journal of Thermal Stresses, vol. 24, pp. 195-231, 2001.

[7] [Lindgren 2001], Lindgren, L.E., *Finite Element Modeling and Simulation. Part 3: Efficiency and Integration*, Journal of Thermal Stresses, vol. 24, pp. 305-334, 2001.

[8] [Yaghi 2004], Yaghi A., Becker A., *State of the Review-Welding Simulations using Finite Element Methods*, Report No. FENET-UN0TT-DLE-08, School of Mechanical, Materials & Manufacturing Engineering, University of Nottingham, UK, 2004.

[9] [Rosenthal 1941], Rosenthal D., *Mathematical theory of heat distribution during welding and cutting*, Welding Journal 20(5), 1941, pag. 220s-234s

[10] [Rybicki 1978], Rybicki E., Schmueser D.W., Stonesifer R.W., Groom J. and Mishler, H.W., *A Finite Element Model for Residual Stresses and Deflections in Girth Butt Welded Pipes*, Journal of Pressure Vessel Technology, vol. 100, pp. 256-262, 1978.

[11] [Debiccari 1986], Debiccari A., *Control of Distortion and Residual Stresses in Girth Welded Pipes*, Ph.D. Thesis, Massachusetts Institute of Technology, USA, 1986.

[12] [Seo 2001], Seo S.I., Yang Y.H. and Jang C.D., *Development of New Finite Element Method to Analyze Deformation of Plate due to Line heating*, Journal of Ship Production, vol. 17, pp. 1-7, 2001.

[13] [Goldak 1996], Goldak J., Zhou J., Breiguine V., and Montoya F., *Thermal Stress Analysis of Welds: From melting Point to Room Temperature*, JWRI, vol. 25, No. 2, pp. 1851-89,1996.
[14] [Goldak 1984], Goldak J., Chakravarti A. and Bibby M., *A new Finite Element Model for Heat Sources*, Metallurgical Transactions B, vol. 15 B, pp. 299-305, 1984.
[15] [Goldak 1986], Goldak J., Bibby M., Moore J., House R., Patel B., *Computer Modeling of Heat Flow in Welds*, Metallurgical Transactions B, vol. 17 B, pp. 587-600, 1986.