FINITE ELEMENT ANALYSIS OF RESIDUAL STRESSES IN NAVAL STRUCTURES DUE TO MULTIARC WELDING PROCESS

Dan Birsan[1]

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

Abstract: *One aim of naval architects is to design structures that are strong enough and capable of absorbing impact energy. The ship plates are stiffened using different stiffeners to withstand compressive and other loads resulting from loads due to collisions, stranding, or grounding. Due to welding process of the stiffeners on the bulkhead, the in-plane shrinkage, may occur. A 3D FEM model, in real size (3.3m x 2.3m) dimensions, 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

Finite element analysis, based on numerical methods, is very useful to predict distortions and residual stresses, caused by welding, even from design stage of product. Some two-dimensional and three-dimensional numerical models were developed in order to simulate one of most complex technologic processes such as welding process. A complex model of the thermal source - often used in the modelling and simulation of the welding process - was developed by Goldak, who reported the heat source as an ellipsoid model [Goldak, 1984 and 1986]. As a particularity, Goldak modelled the heat flux in different ratios in front and in the rear of the thermal source. Other models for thermal sources were developed by Sabapathy et al. [Sabapathy, 2001], Ravichandran et al. [Ravichandran, 1996], Ueda and Yamakawa [Ueda, 1971]. Based on the coupled sequential analysis technique, Friedman [Friedman, 1975], Andersson [Andersson, 1978] developed a specific procedure applied in simulation of welding process. The analysis of the thermo-physical phenomena, conducted during butt welding of plates, was made by performing only a half numerical model and obtaining a great economy of computing and numerical analysing time.

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, in real size (3.3m x 2.3m) dimensions, has been modeled in order to simulate the welding process. The web plate is modeled with approximately 20000 linear, two-dimensional plane elements. The smallest size of the elements in the weld area is 10mm. The thickness is set to 10mm in order to take into account the lack of thermal gradients for these elements and hence be able to obtained the "right" temperature profile for the given heat input and source size. The mesh is fine around these weld lines, and coarse in the far field, as shown in Fig. 1.

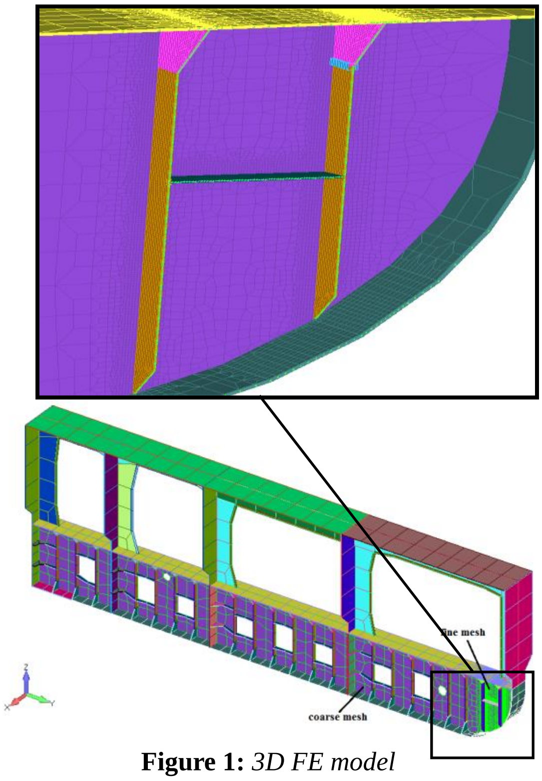

Figure 1: *3D FE model*

The model is pined constraint in the right edge, and in the upper left edge. The stiffeners are simultaneous welded. The problem is solved numerically in the same way as for the T-profile, first the thermal and then the mechanical analysis. The heat is applied using a moving Gaussian heat source. The size of the Gaussian surface is 10mm in front, 75mm in rear and 50mm in the transverse direction. The power of the heat source is considered to be 9400W with an efficiency of 90%.

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. 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 specific to the phase transformation.

3. Results and Discussions

3.1. Temperature Distribution

The simulation of the thermal transfer in the welded joints, for the case of the mobile thermal source, requires the fine mesh for an area wide enough to include the thermal influence, all along the length of the two parts to be welded.

For a complete analysis of the temperatures distribution in the welded joint as well as the tension state from the thermal influenced area, the fine mesh was considered for a width of 40mm all along the weld line.

Fig. 2 shows five heat sources occurring after 60s from the starting of the process. Five heat sources move along the surface in the X and Y-direction.

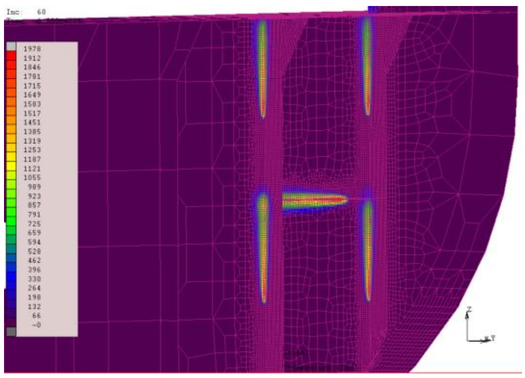

Figure 2: *Location of the five heat sources after the process starting (t=60s)*

When the heat sources reach the edge they automatically switch off and the temperature in the plate is only changed due to the radiation and convection boundary conditions.

Fig. 3 shows a zoomed view of the isotherms around the heat source travelling along the X and Y-axis. The extent of the heat source is relatively small compared with the dimensions of the web plate.

The temperature variation in three interest points of the web plate in time is shown in Fig. 4. The maximum temperature reached in the welding pool was of 1923°C.

Figure 3: *Zoom view of the heat source*

In Fig. 4, the ascendant slope, as well as the descendent one of the variation chart for temperature, in time, in the chosen measure point, for a single pass may have different values, a lot of influence factors existing. In this chart, of a great importance is the maximum value that the temperature reaches in that point. On the plate on which the thermal source moves, represented by the electrical arc, parallel and symmetrical to the movement axis of the source, are to be found points that have reached, successively, to the same maximum temperature.

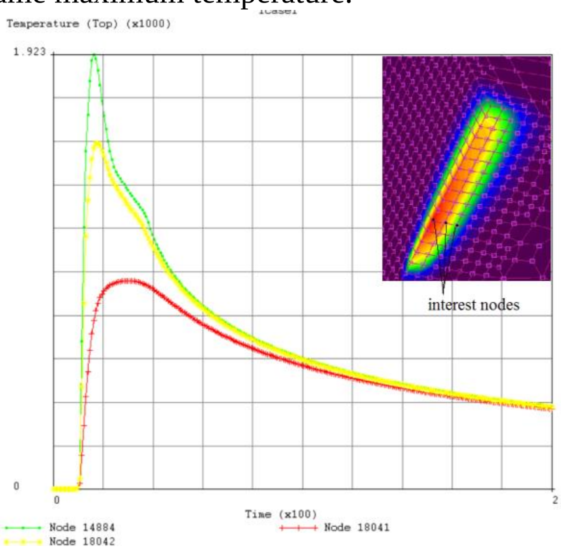

Figure 4: *The temperature profile in time in three nodes*

3.2. The Von Mises Stresses Distribution

Modeling stresses in welds includes all distortions that can be predicted by thermal stress analysis. Models of the mechanical properties of base metal, HAZ and weld metal, are needed as input data for thermal analysis. To evaluate the significance of a given defect, stress analysis is required.

Most thermal stress analyses have used thermo-elasto-plastic constitutive models with rate independent plasticity. Rate independent plasticity implies the viscosity is zero and therefore the relaxation time is zero. This means the stress relaxes instantly to the yield stress. The higher the temperature and the longer the time, the more important viscous deformation becomes. A rate independent model is certainly not valid in the liquid region and is suspect near the melting point where viscous effects are expected to be important; most analyses assume a cutoff temperature. They assume that the thermal strain, Young's modulus and yield strength do not change above a cutoff temperature.

Fig. 5 presents Von Mises stresses distribution in plate after t = 60s. It can be noticed that the maximum value of the stress state corresponds to the area from the front of the thermal source as well as to the corresponding areas of the thermal influenced area. The maximum value of the Von Mises stress is 250MPa.

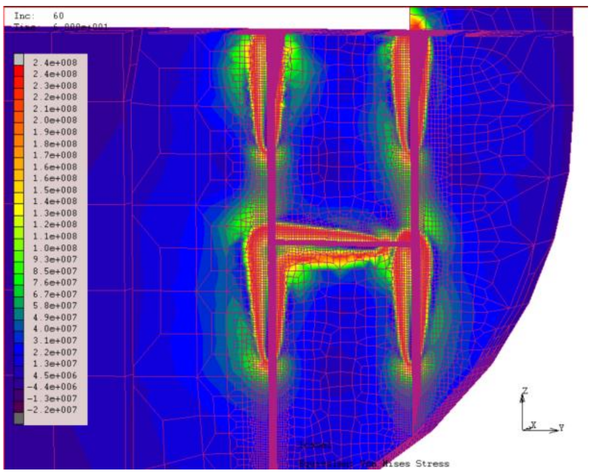

Figure 5: *The Von Mises stresses distribution in plate after 60s*

In Fig. 6 it can be noticed the shape of the variation curve in time of the Von Mises stress. The chosen points to be analyzed are situated at 120mm from the left end of the parts to be welded and in the welding axis.

The yield stress of the structure of the welded material depends very much upon the temperature that it has, while the developed stresses in any point of the welded assembly exceed the value of the yield stress. Thus, the thermal deformations produce plastic deformations distributed un-uniformly in the structure where the values of the yield stress are exceeded.

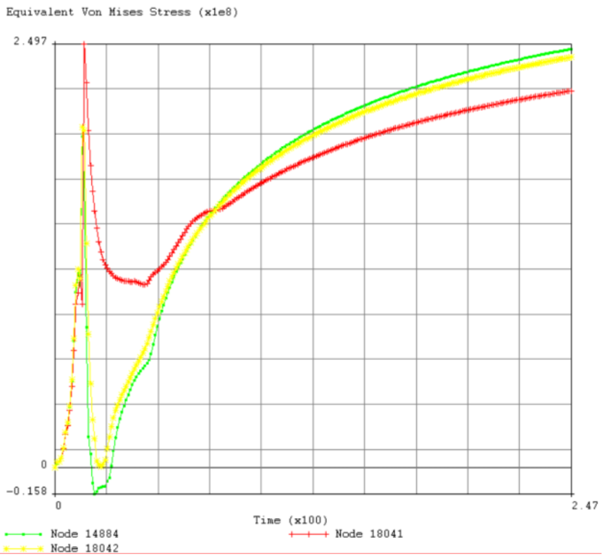

Figure 6: *The variation curves of the Von Mises stress in quasi-stationary thermal field, in time, for three nodes situated in the welding axis.*

4. Conclusions

The proposed modeling takes into account the variation of the physical-mechanical properties of the considered steel with the temperature, fact that leads to obtaining of very precise solutions regarding the distribution and the values of the thermal field, the remaining tensions and deformations.

A major role in the present simulation is played by the latent heat that in the area of the phase transformation makes the temperature variation in the parts to tend to zero.

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 tension state corresponds to the proper thermal influenced area, the maximum value recorded in this area being below the flow limit. The increase of the remaining tensions takes place, 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.

[2] [Goldak, 1986], Goldak J., Bibby M., Moore J., House R., Patel B., *Computer modelling of heat flow in welds*, Metallurgical Transactions B, 1986.

[3] [Sabapathy, 2001], Sabapathy P. N., Wahab M. A., Painter M. J., *Numerical models of in service welding of gas pipeline*, Journal of Materials Processing Technology, 2001.

[4] [Ravichandran, 1996], Ravichandran G., Raghupathy V. P., Ganesan N., *Analysis of temperature distribution during circumferential welding of cylindrical and spherical components using the finite element method*, Journal of Computer and Structures, 1996.

[5] [Ueda, 1971], Ueda, Y., Yamakawa T., *Analysis of thermal-elastic stress and strain during welding by finite element method*, JWRI, 1971.

[6] [Friedman, 1975], Friedman E., *Thermo-mechanical analysis of the welding process using the finite element method*, ASME Journal of Pressure Vessel Technology, 1975.

[7] [Andersson, 1978], Andersson B. A., *Thermal stresses in submerged arc welded joint considering phase transformations*, ASME Journal of Engineering Material and Technology, 1978.