ANALYSIS OF THE INFLUENCE OF COMPLEX MATERIAL BEHAVIOUR ON FUSION WELDING SIMULATIONS

MARINE 2013

JP. LEFEBVRE[*], N. POLETZ[†], L. D'ALVISE[♠], M. CAZUGUEL[♦], A. FRANCOIS[†] AND E. WYART[†]

[*] Cenaero France SASu
462 Rue Benjamin Délessert – BP 83
ZI de Moissy-Cramayel – 77554 Moissy-Cramayel - France
e-mail: jean-pierre.lefebvre@cenaero.fr, www.cenaero.fr

[†] Cenaero
Rue des Frères Wright 29
6041 Gosselies - Belgium
e-mail: nicolas.poletz@cenaero.be, arnaud.francois@cenaero.be, eric.wyart@cenaero.be, www.cenaero.be

[♠] GeonX
c/o SONACA, Route Nationale Cinq
6041 Gosselies - Belgium
e-mail: laurent.dalvise@geonx.com, www.geonx.com

[♦]DCNS Ingénierie Sous-Marins
CSE/CSB/Groupe Calcul – Rue Choiseul –
56311 Lorient Cedex - France
e-mail: mikael.cazuguel@dcnsgroup.com, www.dcnsgroup.com

Key words: Fusion welding, Metallurgy, Phase transformation

Abstract. Numerical simulation of welding processes allows the assessment of the residual stresses and deformations. Accurate results require the precise knowledge of the operating conditions and the materials. Because of the large temperature range the material overcomes during the process, complex phenomena may occur. The material can therefore be modeled using different levels of complexity. This paper presents a discussion of the influence of the material modeling, applied to a welding process, on the final results.

1 INTRODUCTION

Welding processes are among the most common techniques to assemble metallic parts together in the industry. However, the sequence of heating up and cooling down phases generates residual stresses and deformations that can have non negligible effects for further assembly or usage. In order to avoid manufacturing expensive prototypes, numerical simulations of manufacturing processes are helpful during the design phase.

Because of the temperature range the material is subjected to, its modeling must take into account all phenomena such as material flow and phase transformation effects. In this paper, the welding process assembling a stiffener onto a panel is numerically simulated. Different material models (with or without phase transformation) are analyzed, and their influence on the accuracy of the results is discussed. Numerical welding simulations are performed using the software Morfeo/Welding, which is dedicated to simulate manufacturing processes, such as machining and welding [3].

2 DESCRIPTION OF THE CASE

2.1 Geometry and Mesh

In order to improve the mechanical characteristics of the structures, assemblies made up of panels and stiffeners are commonly used. Shipbuilding industry encountering the same issues, the validation case is defined as a stiffener welded onto a panel through two T-welding joints. The panel dimensions are 500x500x4 mm, while the stiffener dimensions are 500x100x4 mm. Figure 1 presents an overview of the analyzed geometry, as well as the associated mesh. In order to capture in an accurate way the gradients occurring during the welding process, specific mesh refinements are required in the Heat Affected Zone (HAZ). Mesh size is therefore fixed to 1.0 mm in the volumes close to both welding beads. Outside of these areas, element size is increasing with the distance to the melting pool. However, in order to avoid over-estimation of the panel stiffness and to capture accurately potential bending, a minimum of two elements is kept across the plate thickness.

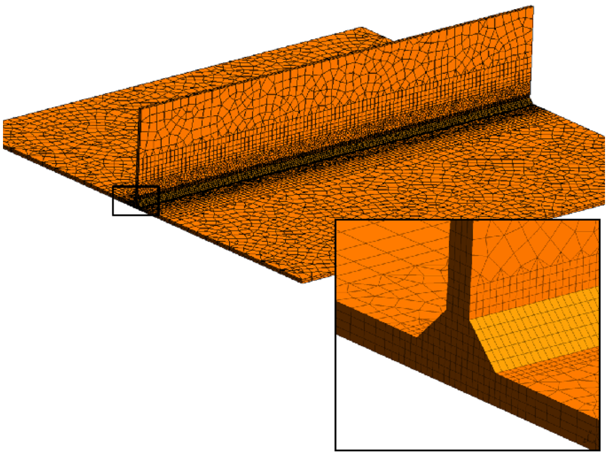


Figure 1: Mesh of the assembly Stiffener-Panel

The mesh is composed by degree 1 hexaedra and prisms elements. The presented mesh is constituted by 119 000 nodes and 152 000 elements.

2.2 Material behavior

Both parts of the assembly – stiffener and panel – are manufactured using the same steel alloy. The objective of the analysis is to examine the influence of the mechanical modeling of the material. In order to simplify the model, and to save computation time, the quantities characterizing the thermal behavior are kept constant independently of the thermal evolution. Common values for steel alloys are therefore considered for the density, the thermal conductivity and the specific heat.

The constitutive law governing the mechanical response of the material is more complex, since it includes viscous effects and metallurgy and phase transformation. The elasto-plastic behavior is defined through a pure isotropic hardening (Eq. 1). The J2 yield function defining the elasticity domain follows the same formulation whatever the current phase is.

$$f(\sigma,R) = J_2(\sigma) - \sigma_y - R \tag{1}$$

with J_2 the second invariant of the stress tensor, σ_y the initial yield stress, and R the hardening stress defined as

$$R = Q[1 - \exp(-b\,p)]^{\beta} \tag{2}$$

where p refers to the accumulated plastic strain.

The viscous effects are also taken into account in the mechanical law. The retained formulation is the Cowper-Symonds overstress power law [4], defined as follows

$$\dot{p} = D\left[\frac{|\sigma| - R}{R}\right]^{n} \tag{3}$$

where $\dot{p}$ refers to the plastic rate, D and n are respectively the modulus and exponent of viscosity.

In order to complete the description of the mechanical behavior, metallurgy and phase transformation can be integrated into the model. The considered steel alloy is defined by two different phases: the ferrite-pearlite and the austenite. Phase transformation is modeled thanks to the Leblond-Devaux modeling approach, which defines the phase transformation velocity as a function of the temperature T and the proportion of the phase z (eq. 4), z_{eq} being the phase proportion reached at the equilibrium for the given temperature T and τ a characteristic time constant:

$$\dot{z} = \frac{z_{eq}(T) - z(T)}{\tau(T)} \tag{4}$$

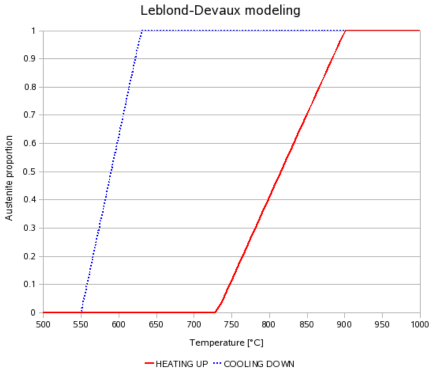


Figure 2 : Phase transformation : Leblond-Devaux modeling

These last two parameters can be deduced from dilatometric curves: using quasi-static heating-up velocity for z_{eq}, so that each state can be considered as equilibrate. τ is identified using faster heating-up velocities in order to get the right temperatures at the beginning and the end of the transformation.

Phase transformation integration induces changes in the expansion coefficient curve: during the heating-up phase, the transformation from ferrite-pearlite phase to austenite introduces the volumetric contraction (or expansion during the cooling down phase), effects being not considered for a single phase material description. The integration of the metallurgy allows taking into account the transformation plasticity induced. The formulation is given below (Eq. 5), where K is a material constant, z describes the progress of the new phase generation and σ_{dev} is the deviatoric stress tensor.

$$\begin{aligned} \dot{\varepsilon}^{pt} &= K\,\phi'(z)\,\dot{z}\,\sigma_{dev} \\ \phi(z) &= z(2-z) \end{aligned} \tag{5}$$

2.3 Operating conditions

The Metal Active Gas welding process is fully modeled. A phenomenological approach is used to model the heat input: the welding energy is integrated as an equivalent heat source with a volumetric power distribution through the workpiece thickness [1]. The sequence is also included in the model (torch trajectories, welding velocities, and cooling-down periods). Metal deposition is not taken into account in this case: elements modeling the weld joints are active from the beginning of the simulation.

The equivalent heat source is correlated to the experiment thanks to a metallographic analysis, which allows to determine easily the dimensions of the melted pool. Figure 3 compares the melted pools of the numerical simulation (defined by the isotherm 1500°C) and the experiment.

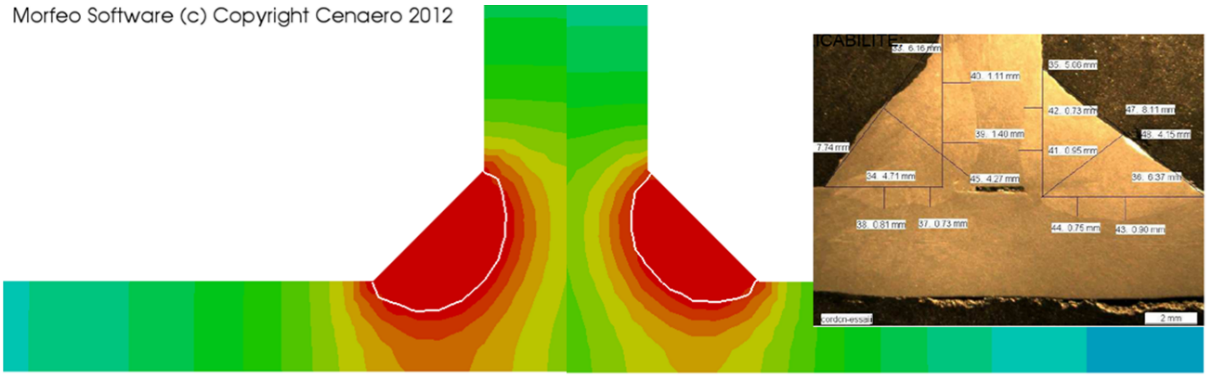


Figure 3 : Comparison between experimental and numerical melted pools

The welding power of 2440 W is applied into the model, and the weld torch is moving at the constant velocity of 5.30 mm/s.

2.4 Thermal and mechanical boundary conditions

Thermal exchange with the environment must be taken into account. It includes radiation and convection. Both contributions are gathered in one unique condition, applied on every free surface of the assembly, with an equivalent exchange coefficient evolving with the temperature.

Mechanical clamping is applied on surfaces on the thickness of the panel, as plotted in Figure 4. Each of these two surfaces is located at 24 mm from the plate side, and is 24 mm long.

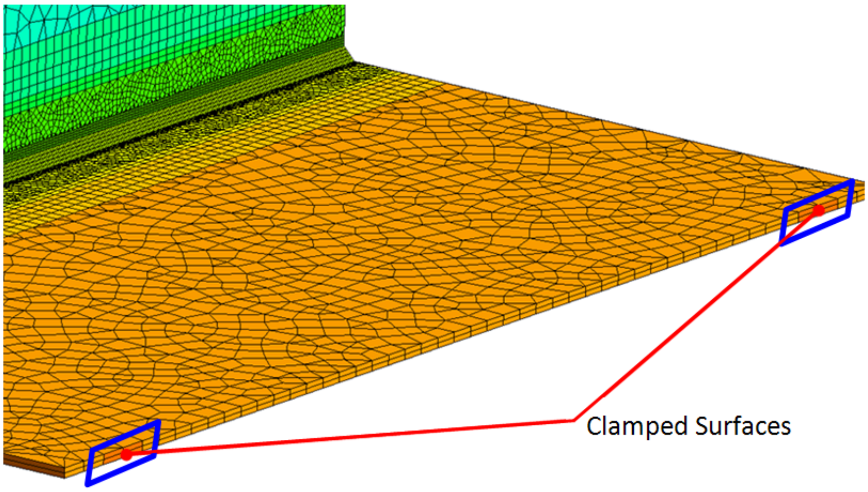


Figure 4 : Mechanical clamping system

3 NUMERICAL SIMULATION OF THE WELDING PROCESS

The numerical simulation of the fusion welding simulation is performed as a transient thermo-mechanical computation, using the dedicated software Morfeo/Welding. Resolution of the problem is performed using a weak coupling method between both thermal and mechanical computations. The thermal analysis is based on a transient non-linear conduction formulation as expressed in Eq. 6

$$\rho C \frac{\partial T}{\partial t} = D \Delta T + P \tag{6}$$

where ρ is the material density, C the specific heat, D the conductivity tensor (reduced to the thermal conductivity as isotropy is considered) and P the heat source density.

The variation of the temperature field induces the generation of thermal strain. The generalized Hooke's law (Eq. 7) is solved to compute the thermo-elasto-plastic response of the structure for each time step.

$$\sigma = C\, \varepsilon^{el} = C\left(\varepsilon^{total} - \varepsilon^{plastic} - \varepsilon^{thermal}\right) \tag{7}$$

where C is the elasticity tensor.

The time increment has to be chosen carefully, specifically during the welding phase: it depends mainly on the mesh size and on the operating conditions (equivalent heat source dimensions and welding velocity). A compromise must be found in order to get accurate results without spending too much computation time. In this particular case, the increment has been chosen as the duration requested for the torch to move for 2 mm (which is twice as the element length in the welding joints). During the cooling down phases, the time step increases as the temperature gradients become lower and lower.

Within Morfeo, two main solver categories are available: iterative or direct solver. The iterative solver has been applied to solve the thermal analysis, while the parallel direct one was chosen for the mechanical resolution. Main criteria in the solver choice are the results restitution time and the influence of the solver settings (which are more delicate to adjust in the case of the iterative solver).

Thanks to the parallelism of Morfeo, results can be exploited within a reasonable time. 16 processors, using each less than 2 Gbytes, were used to perform the simulations, and complete results were available after less than 24 hours of computation.

4 PRESENTATION OF THE RESULTS

Two different simulations are performed with an increasing level of complexity in the material modeling: in the first simulation, only the elasto-visco-plastic response is considered, without any metallurgy, while it is taken into account in the second simulation.

Results are presented for three different time steps, respectively corresponding to the end of both weld joints, and after cooling down. All quantities are normalized with respect to the maximal value, common for both computations.

4.1 Simulation without metallurgical effects

In this model, one single phase is considered all over the complete temperature range.

The evolution of the normalized Von Mises stress field is presented in Figure 5.

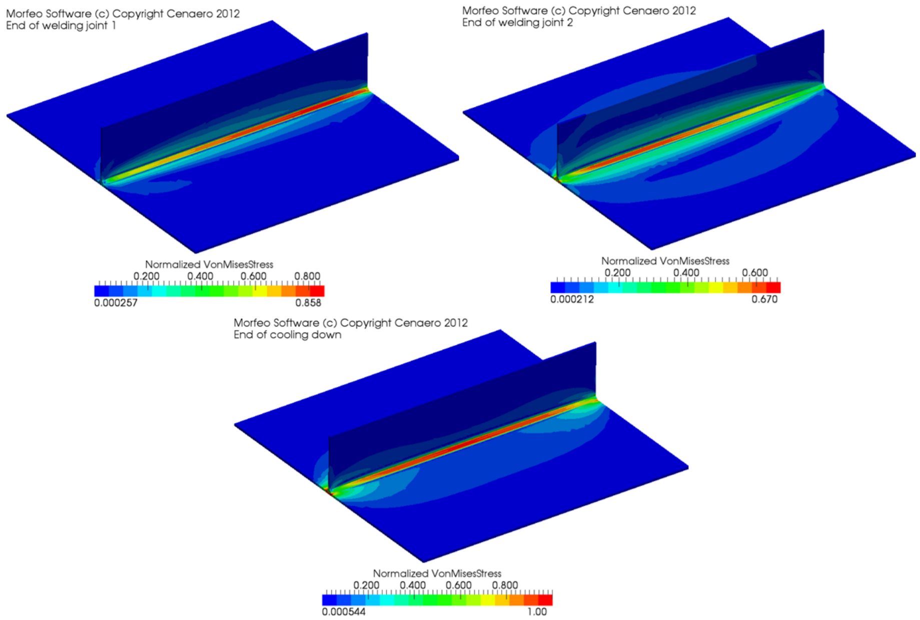


Figure 5 : Simulation without metallurgy - Von Mises stress map

The maximal stress values are located in the welding joints, areas where the thermal strain is the largest. It can be observed that maximal stress values at the end of the second weld are lower than those computed at the end of the first pass. This is can be explained by the temperature field that is higher in the welds because of the two heating-up phases, and tends to relax the stresses.

The analysis of the deflection of the assembly, and mainly of the panel is of great interest for further assembly in a larger structure. The evolution of the normalized vertical displacement is displayed in the following Figure 6.

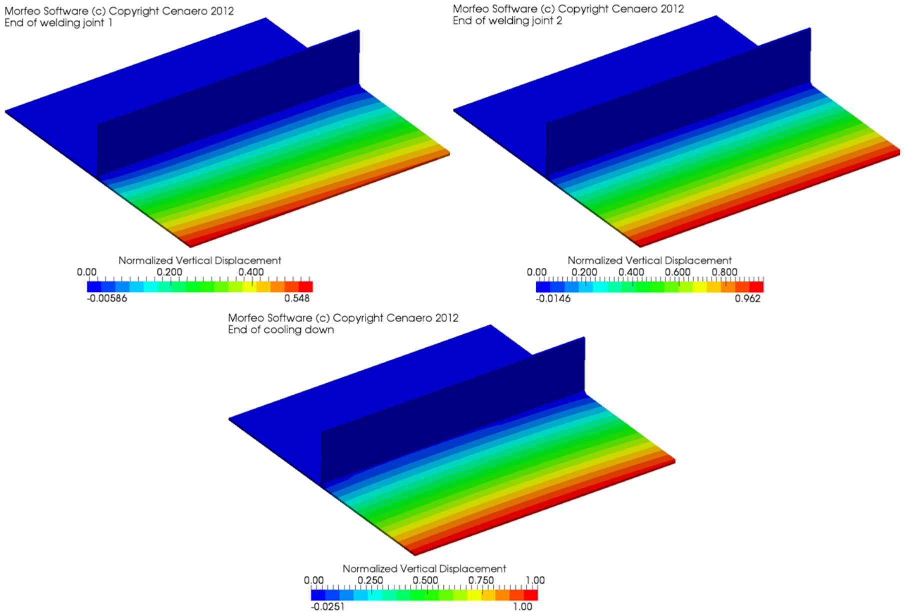


Figure 6 : Simulation without metallurgy - Vertical displacement

The cooling down phase induces volume contraction of the joint that generates bending of the panel. Figure 6 shows that the bending of the panel is uniform, and that no torsion occurs as the iso-lines are parallel to the welding trajectories.

Advanced phenomena such as the phase transformation, the transformation plasticity induced are not considered in this material model. They can however affect the final solution [2], and the influence has to be quantified. Metallurgy is therefore included in the material modeling, and the same process is analyzed

4.2 Simulation with metallurgical effects

Normalized Von Mises stresses and vertical displacement are post-processed for the same three time steps, as it was done previously. Results are presented below (Figure 7 and Figure 8).

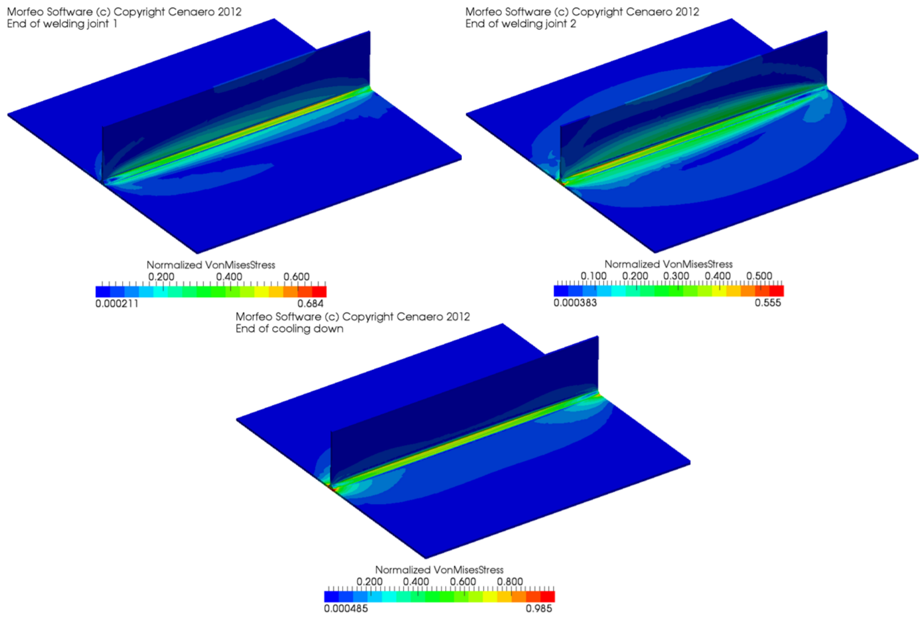


Figure 7 : Simulation including the metallurgy - Von Mises stress map

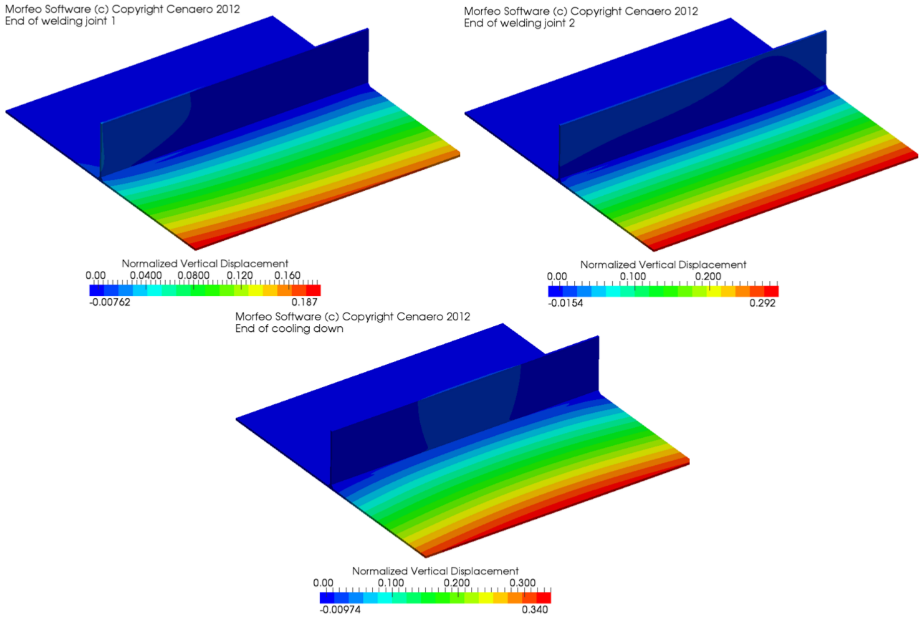


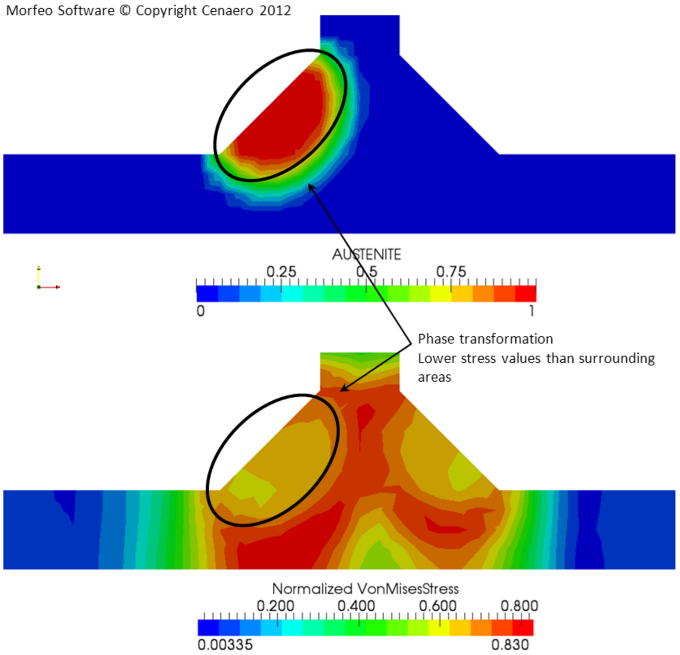
Figure 8 : Simulation including the metallurgy - Vertical displacement

The maximal stress values are about 15% lower than those computed without metallurgical effects. Maximal stress levels are located around the melting pool, where no phase transformation occured. The general shape of the deformed structure is similar to the one presented in Figure 6, although the final deflection is almost three times lower.

4.2 Influence of the metallurgy on the final results

Taking into account the metallurgy influences the results of the numerical simulation and the resources (CPU time and memory) usage. It induces an increase of the CPU Time in the mechanical task of 12% (there is no change in the thermal model, so that the computation time are identical), and requires 25% more memory: computations are completed respectively in less than 21 and 23 hours, using respectively 8.7 Gbytes and 10.8 Gbytes.

The maximal stress levels are not located exactly at the same place: without metallurgy, maximal values are located in the welding beads, where the thermal strain was the higher. This is not the case in the second model: during the cooling down phase, the phase transformation austenite->ferrite+pearlite induces some volume expansion, which relieves the stresses in and around the melting pool. Maximal stress values are therefore lower than those computed in the first model, and located in the surrounded areas, where no phase transformation occurred, as illustrated in Figure 9.


Morfeo Software © Copyright Cenaero 2012

Phase transformation
Lower stress values than surrounding areas

Figure 9 : Domain subjected to phase transformation - Location of the highest stress values

The structures overcome similar deformed shape: the panel's bending (Figure 6 and Figure 8). Nevertheless, the final deflection is three times larger when metallurgy is not considered. In this case, the bending is indeed due to the volume contraction during the cooling down phase. When the metallurgy is applied into the model, the transverse stress map is slightly different. Because of the volume expansion happening during the austenite to ferrite+pearlite transformation, the compressive stress becomes lower, and therefore the induced deflection of the panel.

Unfortunately, experimental measurements are not reliable for this application case. For a finer investigation of the benefit of taking into account of metallurgy within fusion welding simulations, a smaller and simpler instrumented case, made of two plates butt welded, could be analyzed numerically and experimentally.

The previous comparison clearly shows that material modeling plays a key role in the accuracy of the simulations. However, increasing the complexity of the material model requires an even more complex and precise characterization to return accurate results.

5 CONCLUSIONS

In order to analyze the influence of complex material modeling, the MAG fusion welding process is applied on a shipbuilding typical structure, made up of a stiffener and a panel. The constitutive law is defined as elasto-visco-plastic. Two different material models are analyzed in this paper : the first model considers the material as a single phase material, while the second one takes into account the metallurgical effects, such as phase transformation, transformation induced plasticity and thermal contraction/expansion during the phase changes.

The material model plays a key role in the assessment of the residual stresses: with metallurgical effects, residual stresses predicted are lower than those computed with a simpler material model. The higher stress values are located outside of the HAZ (where no phase transformation occurred), whereas the initial model predicts them in the welding beads. It also has a great influence on the deflection computation. On this application case, it has been proved that a factor 3 was applied to the final deflection.

The main conclusions of this analysis are relative to the crucial importance of the material characterization, and the reliability of the welding tests that could be performed to correlate and validate the model. The integration of advanced phenomena in the material behavior requires a precise characterization, and a strong validation on instrumented samples.

6 AKNOWLEDGMENT

The authors would like to show their gratitude to DCNS who provided the studied case and the financial support to perform these analyses.

REFERENCES

[1] Modelling and Simulation of Welding and Metal Deposition, A. Lundbäck, Department of Applied Physics, Mechanical and Materials Engineering, Luleà University of Technology, Luleå, Sweden, 2010

[2] Three-Dimensional Finite Element Simulation of Laser Welded Stainless Steel Plate, D. Berglund, L.E. Lindgren & A. Lundbäck, Computer Aided Design, Luleà University of Technology, Luleå, Sweden, 2001

[3] Morfeo 1.7 User's Manual, Cenaero, Belgium, 2012

[4] Abaqus 6.10 User's Manual, Dassault Systèmes Simulia Corp., Providence, RI, USA, 2010