Journal of Constructional Steel Research 132 (2017) 83–96

Contents lists available at ScienceDirect

Journal of Constructional Steel Research

Practical aspects of welding residual stress simulation

Peter Knoedel *, Stefanos Gkatzogiannis, Thomas Ummenhofer

Karlsruhe Institute of Technology, Steel & Lightweight Structures, Research Center for Steel, Timber & Masonry, Germany

ARTICLE INFO

Article history:
Received 13 September 2016
Received in revised form 31 December 2016
Accepted 4 January 2017

ABSTRACT

The interest on calculating welding residual stresses from the welding process, through finite element (FE) simulation, has increased in the last decades. The prediction of residual stresses and strains is important for the industry, as it can ensure more efficient mechanical design and manufacturing process, reducing as well the time of post weld treatments. More specifically, in structural engineering, the residual welding stresses may have detrimental effect on earthquake, fatigue or stability behaviour. Weld simulation is a complex subject and the application of simplified models, for calculating the residual stresses, is quite usual in practice. For this reason, the results quite often deviate significantly from the experimentally measured residual stresses on real welds. This paper reports on a straightforward but robust engineering approach for the calculation of the residual welding stress field. State of the art methods are applied in some parts, whereas a new engineering approach is proposed, for modelling the material behaviour during a thermal cycle. Aim of the current work is to provide a calculation method for the engineers, which is repeatable for any material and weld geometry, using only material data easily found in literature. The proposed approach is validated with experimental results, which were found in older published work. The approach was modelled in the finite element software ANSYS [1]. The calculated and the measured residual stresses show good agreement. The approach was validated for a single-pass weld with material HT-36 steel (Swedish steel grade HT-36 with yield stress 355 MPa) [2] and S355J2 [3].

1. Introduction

Simulation of weld processes is a challenging task, rapidly evolving in the last years, parallel to the constantly increasing computing efficiency. It is a complex subject, depending on numerous parameters, often interlinked with each other. Generally, the weld analysis can be divided in three fields, as presented in Fig. 1. Simplifications in all steps of the simulation are possible and quite often necessary, as there are many interactions, which cannot be described easily. Which of the coupled interactions will be taken into consideration in each weld simulation case, depends on the required level of accuracy. For example, ignoring the influence of metallurgical transformation on the thermal behaviour of steel is common strategy in most simulations [4,5].

The thermal field of the simulation includes the solution of the thermal transfer problem, from the weld pool to the rest of welded specimen through conduction. Boundary conditions are in this case the heat losses through convection and radiation from the surface of the specimen to the environment. Simpler models are solved on 2D or/and ignore the influence of the radiation heat losses. In the last years, due to the constantly increased computing resources, 3D models are widely applied. In this case the challenge is to simulate the moving weld heat source, whose shape differs for each weld type and varying weld parameters, to a sufficient degree. For the metal-arc welding the double-ellipsoidal weld source model presented by Goldak, Chakravarti and Bibby [6] is nowadays considered to be state of the art. For obtaining robust results, the temperature dependency of the material thermal parameters has to be known. More complicated models for the simulation of the weld pool shape or the interaction between arc and material were developed in the last years, but the existing models (heat conduction modelling with the double-ellipsoidal heat source) are considered adequate, for simulating the usual weld processes, such as the shielded metal arc welding (GMAW) process [7].

The mechanical field includes calculation of the residual stress, based on the calculated thermal history, using the mechanical behaviour, which is defined in the microstructural analysis field. The mechanical behaviour is highly dependent on the microstructural evolution of the simulated material. Steel is a multi-phase solid, with each phase exhibiting unique mechanical behaviour. During welding, the

* Corresponding author.
E-mail addresses: peter.knoedel@kit.edu (P. Knoedel), Stefanos.gkatzogiannis@kit.edu (S. Gkatzogiannis).

http://dx.doi.org/10.1016/j.jcsr.2017.01.010
0143-974X/

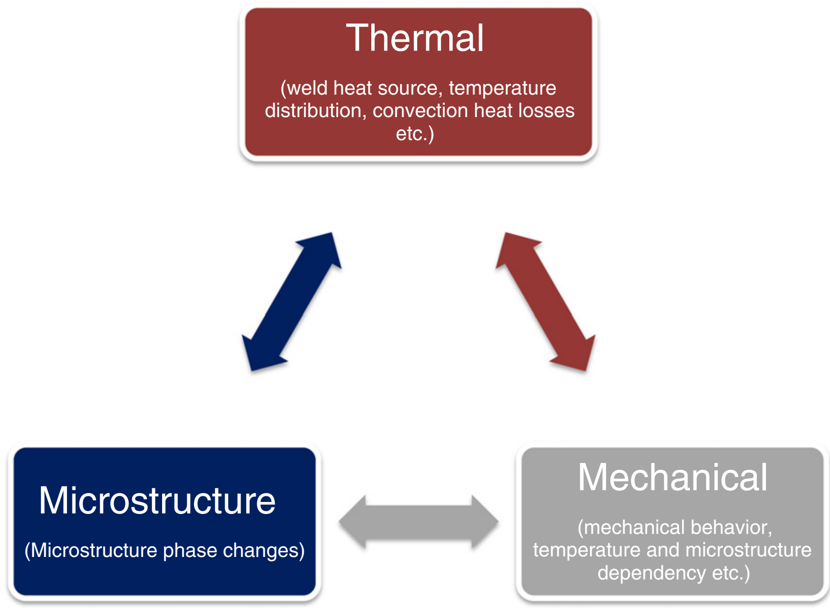

Fig. 1. Fields of the weld simulation – based on a figure found in [5].

microstructure evolves differently in each area of the welded component (fusion zone, heat-affected zone etc.), due to the differentiated thermal cycles. As a result, differences in the mechanical behaviour of these areas are observed. Therefore, knowing only the temperature dependency of the initial microstructure, cannot stand alone for a precise simulation, without taking into consideration the microstructural evolution.

The microstructural field covers microstructural transformations in the sense of the well-known phase changes. Several models over the years were presented, which described steel phase transformations under specific circumstances. According to the authors' knowledge, the models from Leblond [8] and Koistinen-Marburger [9] are the most widely applied until today, when calculating the microstructure changes during welding. Most FE weld-specialized software are applying these models, quite often without sufficient documentation, according to the authors' opinion. Whereas the Koistinen-Marburger equation is easily calibrated for each material, where just the beginning and finishing temperature of the martensitic transformation is needed, the calibration of the Leblond equation for the austenitic transformations requires the calculation of an equivalent balance fraction of the transformation at each temperature level, which is material dependent. Therefore, calibration of the model for each alloy should be carried out before application. Moreover, programming these equations in a general-purpose FE software and carrying out calibration and validation of the model is outside the working area of a structural engineer with limited metallurgy knowledge.

The selected complexity level to be used on each of the above mentioned fields, is usually dependant on the accuracy of output information that is sought. In structural engineering, the final residual stress field and hardness caused by phase transformations are of high importance, whereas an exact calculation of the final microstructure is not usually significant. Of course, the evolution of the microstructure is influencing the final mechanical behaviour and residual stresses, so it cannot be completely ignored. For this reason, a practical but robust engineering method to model the microstructure field and to calculate the residual welding stresses is proposed in this paper. The current paper aims to establish a weld simulation model, applicable to general purpose finite element (FE) software, able to calculate the welding residual stress (RS) with sufficient preciseness, using only data found in existing literature (for almost all common steel alloys). In this way experimental calibration for each investigated steel grade can be avoided.

2. Theoretical background

2.1. Thermal transient simulation

The heat transfer problem is governed by the following differential Eq. (1), when the heat transfer is carried out through conduction.

$$\rho c\left(\frac{\partial T}{\partial t}+v_x\frac{\partial T}{\partial x}+v_y\frac{\partial T}{\partial y}+v_z\frac{\partial T}{\partial z}\right)=\dot{Q}_G+\frac{\partial}{\partial x}\left(Kx\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(Ky\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(Kz\frac{\partial T}{\partial z}\right) \quad (1)$$

where

ρ = density (Kg/m^3)
c = specific heat (J/(kg K))
T = temperature (K)
t = time (s)
K_{xx}, K_{yy}, K_{zz} = conductivity in the element's x, y, and z directions (W/(m K))
$\dot{Q}$ = heat generation rate per unit volume (W/m^3).
v_x, v_y, v_z = velocity for transport of heat in x, y, and z directions, respectively (m/s).

The boundary conditions of the problem are the surface heat losses through convection (Newton's law of cooling).

$$\frac{q}{A}=h_f(T_s-T_b) \quad (2)$$

where

q/A = is heat flux out of the face (J/s)
h_f = heat transfer coefficient (W/(m^2 K))
T_B = bulk temperature of the adjacent fluid (K)
T_S = temperature at the surface of the model (K).

Heat losses through radiation on the surfaces of the plate away from weld can be ignored, as the radiation influence is negligible at low temperatures. Heat losses through radiation from the weld pool can be taken into consideration, by choosing an appropriate value for the weld heat source efficiency factor [2,6]. The moving weld heat source is modelled using the approach proposed by Goldak in [6]. Goldak proposed a double-ellipsoidal model (see Fig. 2), where two quadrants of