Journal of Constructional Steel Research 145 (2018) 49–57

Contents lists available at ScienceDirect

Journal of Constructional Steel Research

Numerical simulation of a welding process using a prescribed temperature approach

Karlo Seleš [a], Mato Perić [b], Zdenko Tonković [a,*]

[a] *University of Zagreb, Faculty of Mechanical Engineering and Naval Architecture, I. Lučića 5, 10000 Zagreb, Croatia*
[b] *Bestprojekt, Bureau of Energetics and Mechanical Engineering Ltd., Petrovaradinska 7, 10000 Zagreb, Croatia*

ARTICLE INFO

Article history:
Received 21 April 2017
Received in revised form 24 January 2018
Accepted 16 February 2018
Available online xxxx

Keywords:
Abaqus Welding Interface
Prescribed weld temperature
Welding process
Residual stress
Butt-weld
T-joint fillet weld

ABSTRACT

This paper presents an efficient finite element procedure for the prediction of welding-induced residual stresses and distortions in large structures. It is based on a prescribed temperature approach using some features of an Abaqus extension called Abaqus Welding Interface to significantly improve the computational efficiency and speed up the normally time-consuming and cumbersome welding analysis setup performed by the user. To validate the temperature and residual stress solutions obtained by the presented method, two numerical examples are analyzed. Comparison is made with the experimental measurements and the results obtained by the heat generation rate approach using the element birth and death technique. The first example is a butt-welding of two plates, while the second is a T-joint fillet welding of two plates. The results obtained by the proposed procedure demonstrate a good agreement in comparison with the heat generation rate approach as well as the experimental measurements. Furthermore, the computational efficiency is remarkably improved compared to the heat generation rate approach as the CPU time is reduced up to ~70% in both examples.

1. Introduction

Welding is one of the basic procedures of metal joining and is still widely used in the industry due to its reliability, ease of use and low cost. Due to the great local heat input during the welding process and subsequent rapid cooling, permanent plastic deformations appear in the weld and its vicinity, causing the residual stresses and dimensional distortions. Such distortions can be a problem during the assembly of the structure, requiring additional time and financial expenses for the correction of the deformed structure. High tensile residual stresses, in combination with stresses due to workload, have a significant impact on the integrity and lifetime of the structure, as they can accelerate the initiation of stress corrosion cracking, a fatigue crack or a brittle fracture [1–4]. To minimize these consequences, it is very important to know the magnitude and distribution of the residual stresses and strains in the structure as early as possible in the design phase. Therefore, the development of an accurate and efficient technique to estimate and reduce residual stresses and strains has been the subject of numerous studies. With the rapid development of computer technology, numerical welding simulations have become more attractive. Ueda and Yamakawa [5] were among the first to use the finite element method in butt and angular welded plates using temperature-dependent material properties. Since then, enormous progress has been made in the field of Computational Welding Mechanics (CWM). The influence of the selection of the finite element type, the heat input model, the choice of base and weld material model, and the boundary conditions, etc., has been analyzed [6–10]. This effort reflects the trend in both the scientific community and the industry to shift the welding residual stress and distortion assessment from an experimental to a computational-based procedure. A more detailed overview of CWM is presented in [11–13], and recent CWM standards are discussed in [14,15]. The transient heat transfer problem with geometrical and material nonlinearities is solved by employing either fully or sequentially coupled thermo-mechanical analysis. With the assumption that the rate of heat generation due to mechanical dissipation energy is negligible in the heat transfer analysis, a sequentially coupled analysis is commonly applied in the welding simulation process. The temperature field determined in the thermal analysis as a function of time for each integration point is therefore used as an input for the mechanical analysis [16–18]. Furthermore, it is well established that 3D models are required for accurate prediction of post-weld deformation and residual stress distribution. These models are often accompanied with the element birth and death technique to simulate the addition of the weld material. With the industrial need to solve increasingly more-demanding problems in engineering practice, such complex models can be computationally very expensive. This is why current thermal elastic-plastic finite element simulations are usually limited to small and simple structures.

---

* Corresponding author at: Institute of Applied Mechanics, Faculty of Mechanical Engineering and Naval Architecture, University of Zagreb, Ivana Lučića 5, 10000 Zagreb, Croatia.

E-mail addresses: karlo.seles@fsb.hr (K. Seleš), mato.peric@fsb.hr (M. Perić), ztonkov@fsb.hr (Z. Tonković).

https://doi.org/10.1016/j.jcsr.2018.02.012

Due to the limitations of the nonlinear and time-dependent thermal elastic plastic finite element method, alternative solutions, such as the inherent strain method based on the elastic material behavior assumption, are widely used nowadays [19–21]. Herein, the inherent strains are the residual plastic strains caused by the welding process. The main problem in using this method is knowing the distribution of the inherent strains in advance. This particularly applies to the modeling of different directions and speeds in multipass welding simulations, which can be properly simulated by thermal elastic plastic finite element methods.

To partially overcome the abovementioned problem of high computation time in the thermal elastic plastic finite element methods, a novel method is presented in the first phase of the authors' study [22]. The method is based on the 3D to shell elements transition, and it significantly reduces the welding simulation time. In the current study, the welding simulation time is further reduced using a different heat input method. In general, there are two basic approaches for modeling the welding heat input. The first is the heat generation rate approach [23–25] based on the heat flux prescribed on a surface area (surface heat input method) or weld element volume (volumetric heat input method) or a combination of both. The second is the prescribed temperature approach [26,27] where a prescribed temperature load is applied over a weld volume.

In this study, the prescribed temperature approach is introduced. Along with reducing the high computation time, it is also important to reduce the user time needed for numerical model preparation, as it consists of very time-consuming and repetitive tasks. In that sense, an extension to the commercial software package Abaqus [28], the Abaqus Welding Interface (AWI), has recently been developed for 2D and 3D models, and has been tested in [29–31]. In [29,30] the material melting point is used as a targeted prescribed temperature, while in [31] a modified parabolic thermal profile is used. The aim of this study is to determine the equivalent weld temperature which is used as the input parameter in the prescribed temperature approach in order to significantly reduce the welding simulation time while concurrently maintaining the accuracy. Moreover, the aim is to prove the applicability of the proposed method for the numerical analysis of the large welded structures used in today's industry. Certain features of AWI are used in the proposed method, exclusively in thermal analysis, in order to speed up the numerical model preparation. For verification and validation purposes, examples of the butt welding of two plates and T-joint fillet welding are taken from the literature and authors' previous study. All the computations have been performed within the FE software Abaqus/Standard.

The paper is organized as follows. The numerical model is described in Section 2. Section 3 provides a brief review of the used analysis procedure. In Sections 4 and 5, the efficiency and accuracy of the prescribed temperature approach are demonstrated on the examples of the butt welding of two plates and T-joint fillet welding. The results and discussions are given in the Section 6. Finally, some concluding remarks are given in the last section.

2. Numerical model

AWI is an Abaqus/CAE Plug-In based on the model-tree approach. It provides a graphical user interface to set up the welding simulation from within Abaqus/CAE, i.e., to set up the generation of heat transfer properties included in the welding process. In this study, AWI is used in thermal analysis to build the thermal interactions, steps and thermal boundary conditions as it automates these repetitive tasks usually performed by user. A sequentially coupled thermo-mechanical analysis is utilized. The prescribed temperature approach is adopted, representing the heat input coming from the welding torch. The weld beads are deposited in discrete chunks along the welding path. The chunks are represented by the finite element sets and are activated through the Abaqus feature called “model change”. The activation is commonly known as the element birth and death technique and is used to simulate filler metal addition.

The prescribed temperature is assigned as a boundary condition at the interface between the current weld bead chunks and the base material or already-deposited weld bead chunks. The temperature boundary condition is set for the time duration needed for the torch to pass the corresponding chunk. It is linearly ramped to the targeted prescribed temperature. In that time, the chunk is still deactivated and is activated only after the torch has virtually moved to the next finite element set representing the next discrete chunk in its path. Because an activated finite element set has a constant volume temperature, the thermal profile inside the weld is neglected. The main computational advantage of such a method can be seen from the governing finite element equation for transient heat transfer analysis

$$\mathbf{c}\dot{\mathbf{T}} + \mathbf{k}\mathbf{T} = \mathbf{f}_\mathbf{T} \tag{1}$$

where, in the weld region, there is no need to use the heat flux term on the right-hand side to calculate temperatures, as they are already prescribed. In Eq. (1), $\mathbf{c}$ represents the temperature-dependent heat capacity matrix; $\mathbf{k}$ is the temperature-dependent conductivity matrix; $\mathbf{f}_\mathrm{T}$ is the thermal load vector; and $\mathbf{T}$ and $\dot{\mathbf{T}}$ are nodal temperature and its time derivative matrix, respectively. On the other hand, using the standard heat flux input takes a significant amount of computational time for the calculation of temperatures in the weld region. This is due to the high temperature gradients, which result from the temperature difference between high peak weld temperatures and its surroundings, entailing finer temporal incrementation.

When using the prescribed temperature approach, welding parameters such as the welding voltage and welding current are not explicitly included as they are with the heat flux input. Therefore, the magnitude of the prescribed temperature as a main input parameter cannot be easily determined. In this study, the prescribed temperature magnitude determination is proposed by comparison with the heat generation rate approach on a smaller scale reference model where the heat flux magnitude results from the known welding parameters. It can be stated that such a prescribed temperature incorporates not only the welding parameters but also the material model and weld geometry.

3. Analysis methodology

A short overview of the heat generation rate method and the prescribed temperature method used in this study is given in this section. In this study, both methods use the same material properties for both base and weld filler metal, and constant radiation and convective heat transfer coefficients. Welding speed is included with the size of the finite element set in the weld direction and the time duration of the heating step. As mentioned in the Introduction, the major distinction between the two methods is the heat input definition. The first is a simple, commonly used method explained in [22,32] that incorporates the sequential deposition of finite elements (i.e., element birth and death technique), with constant heat flux over the given set of finite elements describing the weld bead chunk. It will be called the CHF (Constant Heat Flux) method. The second method uses the PWT (Prescribed Weld Temperature) approach as previously explained. The mechanical analysis is performed with two options: M1, which is very similar to the mechanical analysis of the CHF method incorporating the element birth and death technique, and M2, which is a simple nonlinear mechanical analysis performed by simultaneous deposition of finite elements, i.e., without the “model change” option. In both mechanical analysis options, the initial temperature condition of the base material is set to room temperature, while the element sets describing the weld beads are set to the prescribed weld temperature used in the thermal analysis. Table 1 presents the list of used methods.

Since the CHF method has already been validated on several examples in engineering practice, it will be taken as a reference solution for