Thin-Walled Structures 120 (2017) 297–306

Contents lists available at ScienceDirect

Thin-Walled Structures

journal homepage: www.elsevier.com/locate/tws

Full length article

Prediction of friction stir welding effects on AA2024-T3 plates and stiffened panels using a shell-based finite element model

R.M.F. Paulo[a], P. Carlone[b], V. Paradiso[b], R.A.F. Valente[a,*], F. Teixeira-Dias[c]

[a] GRIDS Research Group, Department of Mechanical Engineering, University of Aveiro, Portugal
[b] Department of Industrial Engineering, University of Salerno, Italy
[c] Institute for Infrastructure and Environment, School of Engineering, The University of Edinburgh, United Kingdom

ARTICLE INFO

Keywords:
AA2024-T3
Friction stir welding
Shell finite elements
Residual stresses
Distortion
Stiffened panels

ABSTRACT

Manufacturing-induced effects significantly affect in-service behaviour of welded structures, such as integrally stiffened panels for aeronautic applications. Being a complex phenomenon with several variables involved, the assessment of the effects coming from welding usually relies on numerical simulations. Here, a novel shell-based finite element model is proposed to accurately simulate the transient thermal fields and stress-strain distributions resulting from friction stir welding (FSW) processes. The capability of the model to predict (i) residual stresses, (ii) material softening and (iii) geometric distortion of the welded parts is assessed by the modelling and simulation of FSW applied on aluminium integrally stiffened panels.

1. Introduction

Stiffened panels are the common choice for structural elements subjected to bending and, particularly buckling loads, in several demanding applications with high strength/weight ratio, such as the case of airplane wings and fuselages but also ships and off-shore structures. By means of a proper choice of the material as well as essential geometric parameters (particularly the cross sectional dimensions), these structures are supposed to withstand complex load scenarios. Indeed, complex solicitation pattern derive from the combination of longitudinal compressive (buckling) forces, transverse loads, in-plane shear forces, and those perpendicular to the base plate (inducing bending effects) [1]. For this reason, the modelling and prediction of geometric deviations are of crucial importance, particularly those coming from the joining operations of individual panels.

The application of friction stir welding (FSW) processes to join integrally stiffened panels has recently been investigated as an alternative of other joining techniques, such as riveting or fusion welding processes [2–4]. FSW is a well-established solid state welding process that enables to efficiently weld almost all types of aluminium alloys, even those traditionally classified as non-weldable by fusion welding means [5]. Although the effects coming from FSW processes, in terms of residual stresses and geometric distortion during and after joining, are proved to be less invasive compared to other joining processes, the impact of such effects on the performances of the welded structure should be carefully assessed [6–8].

In this study, modelling and numerical analyses of FSW processes were performed using the FEM commercial software package Abaqus [9]. One of the main characteristics of the proposed numerical finite element framework is that the parts to be welded were modelled exclusively using shell finite elements. Regarding the use of Abaqus package, many shell elements have been tested by several authors [10–12]. Nonetheless, some authors adopted solid elements in their models [13–15]. It is agreed that shell elements makes the meshing process simpler and faster, when compared to using solid elements, relying on a reference surface, usually in the mid-thickness of plate, where the nodes are located. In terms of analysis, shell elements provide less computational time in contrast to solid ones, mainly due to the lower number of elements to be used in the model. Moreover shell elements allow for a more straightforward discretization of thin-walled structures, at the same time avoiding to a greater extent over-stiffness effects coming from transverse shear locking. Thin-walled structures like plates and shells are the most common construction elements in nature and technology [16]. Plate and shell structures are often reinforced with slender stiffeners, increasing the load-carrying capacity of thin-walled structures without giving up their lightweight property. To this purpose, a numerical modelling approach is of fundamental importance in the understanding of FSW process effects on the structural behaviour of stiffened panels in order to avoid conservative design choices, often motivated by an attempt to compensate for structural analysis uncertainties.

A 3-stage procedure was created, verified and adopted as a

* Corresponding author.
E-mail address: robertt@ua.pt (R.A.F. Valente).

http://dx.doi.org/10.1016/j.tws.2017.09.009
Received 23 April 2017; Received in revised form 7 August 2017; Accepted 9 September 2017

modelling and numerical framework that can be replicated by other researchers and industry partners, by means of a coupled thermo-mechanical model and a sequence of quasi-static analysis. The summary of the proposed modelling framework is covered in detail in the present paper. In the first stage, a heat source moved longitudinally along the welding line. On the second stage, a cooling step of the joined structure is promoted. During these two initial stages, mechanical boundary conditions were applied to simulate the clamping system. The third stage of the simulation corresponds to the release of the joined structure from the supports, where the boundary conditions are replaced by minimal constrains only to prevent rigid body movements. No re-meshing procedures were needed to reduce the involved computational costs, which is an added value of the proposed modelling/simulation procedure. At the same time, temperature dependence of relevant material parameters was accounted for to ensure a reliable prediction and performance of the proposed numerical procedure. Additionally, thermal softening of the material was considered as being not only temperature dependent but also temperature history dependent, which is a distinctive feature of the presented approach.

The numerical models used in this study were firstly developed (and calibrated) in the simulation of a relatively simple benchmark consisting of single plates joining, being subsequently validated using reference experimental data, obtained by some of the present authors and commented in previous papers [17–19]. Doing so, AA2024-T3 plates were friction stir welded using a HSS unthreaded tool and subsequently microstructurally and mechanically characterized [18]. The residual stress levels coming from the joining process were inferred by the contour method [20], following the hybrid numerical–experimental procedure reported in [19]. The validated model was afterwards applied to simulate FSW process of stiffened panels for aeronautical applications.

2. Materials and methods

The model of the welded plate, with the same dimensions of the one used in the experiments [21], was discretized using shell elements. A sensitivity study was carried out on the proper mesh density needed, the type of shell element to be chosen and the optimum number of integration point across thickness, seeking for a reasonable calculation time without compromising the accuracy of the results. Different types of heat source distribution were tested and the sensitivity of the numerical model to distinct mechanical boundary conditions (simulating the clamping system) was also assessed. Details concerning material modelling, element formulation, discretization, boundary and loading conditions are detailed presented in the following sections.

2.1. Material modelling

In the present work thermal and mechanical properties of AA2024-T3 were defined following previous references in the literature [18,22,23]. Thermal conductivity, thermal expansion coefficient, specific heat capacity, Young's modulus and yield stress were considered as being temperature dependent, whereas density and Poisson's ratio were assumed as temperature independent. An isotropic material model was applied for all the parameters. The material was considered to behave as perfectly plastic and therefore no hardening law was defined in the constitutive model. In fact, Preston et al. [24] described an insignificant effect of work hardening, compared with the perfectly plastic case, on residual stresses numerically predicted for FSW processes with AA2024-T3, since most of the plastic strain occurs at high temperatures when work hardening rates are negligible. Additionally, the assumption of a perfectly plastic behaviour can lead to gains in terms of computational time, since there is no need for updates on the hardening variables.

Thermal softening effects induced by thermal cycles were also considered including a softening model to properly account for the effects of the temperature and temperature history on the yield stress. Among others, Sonne et al. [22] showed that the use of a softening model can lead to important changes in the prediction of residual stresses, compared to the solely use of temperature dependent material properties [22,25–30]. In this regard, a softening model based on the proposal by Myhr and Grong [31] and relying on the overall level of precipitates dissolution and coarsening, was assumed and implemented in the present work. Following the aforementioned contributions by Feng et al. [30] and Sonne et al. [22], the yield stress (σ) can be defined by:

$$\sigma = (\sigma_{\max} - \sigma_{\min})(1 - X_{\mathrm{d}}) + \sigma_{\min}, \tag{1}$$

where $\sigma_{\max}$ is the yield stress of the material in the T3 condition, $\sigma_{\min}$ is the yield stress in the fully softened state and X_{d} is a dissolved precipitates fraction, defined by:

$$X_d = \sqrt{t_{eq}}\,. \tag{2}$$

In this equation, t_{eq} is given by:

$$t_{eq} = \sum_{i=1}^{N_{total}} \frac{\Delta t_i}{t_{ref}\, exp\left[\frac{Q_{ref}}{R}\left(\frac{1}{T_i} - \frac{1}{T_{ref}}\right)\right]}, \tag{3}$$

where Δt_i is the size (time) of the increment, T_i is the current temperature, t_{ref} is the time for total dissolution at the reference temperature (T_{ref}) and defined according to Sonne et al. [22], R is the universal gas constant and Q_{ref} is the effective activation energy for the dissolution of precipitates.

Within a simulation run, for every increment the parameter X_{d} is updated at each integration point starting from a value of 0 (corresponding to the material in the T3 condition) and ranging up to 1 (a fully softened material), according to Eqs. (2) and (3). The calculation of X_{d} was carried out by means of an Abaqus USDFLD user subroutine [9] developed by the authors, being this magnitude defined as a field variable. Actual values of yield stresses are then obtained by an interpolation between upper ($\sigma_{\max}$) and lower ($\sigma_{\min}$) bounds of the yield stress values, according to Eq. (1) and taking into account the current temperature. The curves corresponding to the upper and lower boundaries were based on the literature, although there are some differences in the information provided by different authors [22,24,27,32].

2.2. Model discretization

As previously mentioned, a shell element formulation was used to discretize the plates to be modelled. Two types of elements from the Abaqus library were tested: S4RT and S4T. These are 4-nodes thermo-mechanical coupled elements, where S4RT adopts a reduced integration scheme while S4T a fully integrated one [9]. Regarding the distribution of the integration points across thickness, a Simpson's rule is used by default in Abaqus [9]. Two different numbers of integration points were tested using the S4RT elements: 5 and 9 points across thickness. For the S4T shell element, only 5 integration points across thickness were used, resulting in a total of 20 integration points per element (5 layers of 4 in plane integration points). For this element, the option of 9 integration points across thickness was not tested since preliminary results using the S4RT element did not showed any advantages on using more than 5 integration points along the thickness direction. Three different mesh refinement levels with 0.5, 1, and 2 mm width were tested, all of them composed by equal sized square elements. The reasoning behind the choice for regular meshes at this stage was related to infer if such a simple approach would be effective or, on the contrary, a local re-meshing procedure would be needed. This study showed that the first option (the use of regular meshes) was enough for a good quality of results with a low effort in mesh generation and manipulation.

Furthermore, sensitivity analyses were performed concerning different mesh refinement levels, type of shell elements and number of