Thermal Science and Engineering Progress 5 (2018) 1–14

Contents lists available at ScienceDirect

Thermal Science and Engineering Progress

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

Numerical simulation of thermal and residual stress fields induced by lined pipe welding

Obeid Obeid*, Giulio Alfano, Hamid Bahai, Hussam Jouhara

College of Engineering, Design and Physical Sciences, Brunel University, UB8 3PH Uxbridge, UK

ARTICLE INFO

Keywords:
Lined pipe
Weld overlay
Girth welding
Thermal history
Residual stresses

ABSTRACT

This paper investigates numerical thermal fields and residual stresses induced by single-pass weld overlay (lap-weld) and girth welding (butt-weld) in lined pipe using Tungsten Inert Gas (TIG) welding. A distributed power density of the moving heat source based on Goldak's ellipsoid heat flux distribution is used in a Finite Element (FE) simulation of the lined pipe welding process. In addition, radiation and convection have been incorporated in heat transfer coefficient user-subroutines for the FE code ABAQUS. The 3-D FE model approach has been validated using previous experimental results published for butt-welds of similar sections of carbon-manganese C-Mn steel pipe lined with stainless steel. The FE model has been developed to determine the thermal isotherms and residual stress distributions from weld overlay and girth welding. The use of an inner layer known as a liner has a considerable influence on the thermal history and residual stress distributions. Furthermore, the influence of the weld overlay has been examined thermally and mechanically as it is a key factor that can affect the quality of lined pipe welding.

1. Introduction

For hydrocarbon pipelines where the production fluid is corrosive such as H_2S (sour service) or CO_2 (sweet service), C-Mn steel pipes are usually not suitable. One alternative is lined pipe consisting of an inner layer (the liner) and outer layer (backing steel). The liner is made of corrosion resistant alloy (CRA) such as Alloy625, 304 and 316L stainless steel (SS) whilst the backing steel is made of low-cost carbon steel [1].

Lined pipe welding is a complex process requiring two sequential welds. The liner is typically fixed using weld overlay to seal its end with the outer pipe. Consequently, no gap is left between the liner and the backing steel [2]. A girth weld is then executed to join two adjacent lined pipes. The integrity assessment and lifetime estimation of the lined pipe require consideration of thermal fields and residual stresses induced by such welded structures. A computational procedure based on FEM is an effective alternative to experiments [3]. There are many obstacles which make the experimental investigation of lined pipe welding inflexible, time consuming and prohibitively expensive.

It has been over 25 years since Karlsson and Josefson [4] first proposed a full three-dimensional thermal and mechanical study of circumferential butt-welding operations. Their C-Mn single-pass model using FE-code (ADINAT/ADINA) was validated experimentally by other similar work [5,6]. Developing the finite element code ABAQUS for non-linear analysis enabled Brickstad and Josefson [7] to numerically simulate a series of multi-pass girth-butt-welded stainless steel joints.

Over the last decade or so, a significant development of FE codes in 3-D FE modelling has given a high flexibility in predicting the thermal history and residual stresses in a butt-welded steel pipe. Deng and Murakawa [8] developed 3-D and 2-D FE models to analyse temperature history and residual stresses in multi-pass girth welds with SUS304 stainless steel pipe sections. Their results of 3-D modelling obtained from ABAQUS are in very good agreement with experimental measurements. Akbari and Sattari [9] developed an FE dissimilar cylindrical model in which one joint is made of A240-TP304 stainless steel and the other one is A106-B carbon steel. The thermo-mechanical behaviour and the effect of heat input on the residual stress distribution were discussed in that study. The numerical results obtained by the ANSYS code showed good agreement with the experimental ones using the hole-drilling method.

The thermal and mechanical analyses depend on several principal factors such as material properties, heat input, welding pool geometry, boundary conditions and welding sequence. Subsequently, Deng et al. [10] tried to validate a welding simulation for dissimilar materials consisting of a low alloy steel pipe and an austenitic stainless steel pipe. Some discrepancy between the numerical and the experimental results were found because the weld cladding layer under the low alloy steel joint was not taken into account in the dissimilar materials welding

---

* Corresponding author.
E-mail addresses: obeid.obeid@brunel.ac.uk (O. Obeid), giulio.alfano@brunel.ac.uk (G. Alfano), hamid.bahai@brunel.ac.uk (H. Bahai), hussam.jouhara@brunel.ac.uk (H. Jouhara).

http://dx.doi.org/10.1016/j.tsep.2017.10.005
Received 27 August 2017; Received in revised form 2 October 2017; Accepted 5 October 2017
2451-9049/

Nomenclature

a	half-length of heat source (mm)
b	depth of heat source (mm)
c	half-width of heat source (mm)
C	specific heat capacity (kJ/kg K)
$d\dot{\varepsilon}^p$	plastic strain increment
$d\sigma$	stress increment (Pa)
h_{con}	convective heat transfer coefficient (W/m^{2}K)
H	enthalpy of material (J/kg)
H'	strain-hardening rate (Pa)
I	current (Amperes)
NT11	temperature (°C)
q	power density (Wm^{-3})
Q	total heat input (W)
R	radial distance of the heat torch centre from the pipe axis (mm)
S, S33	axial residual stress (N)
t	current time (s)
t_0	initial time (s)
T	current temperature
T_{amb}	Ambien temperature (°C)
v	welding speed (mm/s)
V	voltage (volts)
WCL	weld centre line
$\dot{\varepsilon}$	total strain
$\dot{\varepsilon}^e$	elastic strain
ε_{em}	effective radiation emissivity
$\dot{\varepsilon}^p$	plastic strain
$\dot{\varepsilon}^t$	Thermal strain
θ	angle of moving torch around the pipe (Rad)
κ	conductivity (W/mK)
μ	welding efficiency (%)
ρ	density (kgm^{-3})
σ_{bol}	Stefan-Boltzmann constant
υ	Poisson's ratio
ω	angular velocity (Rad/s)

simulation.

There are a limited number of studies in the literature which have reported lined pipe welding simulation due to the complex sequence of weld overlay and girth welding. Even so, it is worth noting that the technique used to move the heat source around the weld overlay to fix and seal the liner at the pipe ends and then around the girth weld to join two specimens of lined pipe has not been reported yet. Consequently, Obeid et al. [11] presented a new procedure to simulate a typical lined pipe process including the weld overlay and girth welding. Furthermore, a sensitivity analysis to determine the influence of the cooling time between weld overlay and girth welding and of the welding speed has been conducted thermally and mechanically. In this study, a three-dimensional FE model is developed using ABAQUS [12] to study the thermal and mechanical behaviour induced by the weld overlay and girth welding process, described here as Case A. The numerical approach presented is validated thermally and mechanically by previously published experimental results. Moreover, thermal and mechanical results in the corresponding cases without considering weld overlay, described as Case B, and without liner, described as Case C, were also examined for comparison with Case A.

2. FE modelling

The thermal history fields and the resulting residual stresses are obtained by simulating the weld overlay and the girth welding using ABAQUS 6.13–1 and FORTRAN codes [12]. The procedure consists of two analyses, thermal and mechanical. The first analysis is executed to produce the thermal history of the whole thermal model. In the following analysis, the thermal distribution at each node of the thermal model is employed as a thermal load on the corresponding node in the mechanical analysis in order to determine the stress distribution at this node.

2.1. Finite element mesh

Due to the symmetry of the lined pipe, only one of its two components is modelled. The three-dimensional FE model contains a total of 51840 nodes associated with 10560 elements. Among these, 17400 nodes and 2400 elements represent the liner geometry and the remaining 10560 elements represent the backing pipe geometry.

A fine mesh has been considered in the fusion zone (FZ) and its vicinity, i.e. in the heat affected zone (HAZ), because of the higher gradients of temperature and flux. The element size increases with an increase in the distance from the welding centreline (WCL), in both the C-Mn pipe and the liner. There are 120 sets of divisions in the circumferential direction. Furthermore, the thickness of lined pipe consists of four layers of elements. Three of these layers are for the C-Mn pipe and a one layer is for the liner as shown in Fig. 1.

The FE mesh applied in the structural analysis is the same as that used in the thermal analysis. Nevertheless, both analyses have different element types and boundary conditions. 20-node quadratic hexahedral heat-transfer elements with a single degree of freedom, named DC3D20 in ABAQUS, are used for the thermal analysis. Continuum solid, three-dimensional 20-node reduced integration elements (C3D20R) are employed for mechanical analysis with three translation degrees of freedom in combination with a large displacement (nonlinear geometry).

2.2. Thermal analysis

Transient heat-transfer analysis is executed to evaluate the thermal distributions during welding. Basically, there are two choices for boundary conditions, either specifying the flux or the temperature. In this case, the energy balance for each domain is governed by the classical energy balance equation given as [13]:

$$\rho\frac{\partial H}{\partial t}-\nabla.\,(\kappa\nabla T)=q(x,y,z,t) \tag{1}$$

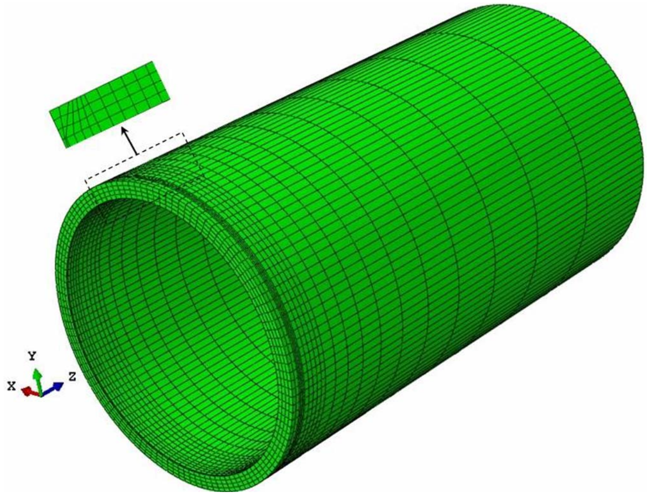

Fig. 1. Three-Dimensional FEM.