Fusion Engineering and Design 85 (2010) 181–189

Contents lists available at ScienceDirect

Fusion Engineering and Design

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

FEM simulation of a small EB welded mock-up and new sequence proposed to improve the final distortions

J. Guirao [a,*], E. Rodríguez [b], A. Bayón [c], J. Pistono [d], L. Jones [e]

[a] Numerical Technologies, S. L., Marqués de San Esteban No. 9, 33206 Gijón, Spain
[b] Department of Mechanical Engineering, University of Oviedo, Campus de Viesques, 33203 Gijón, Spain
[c] Nuclear Generation Division, Iberdrola Engineering, Av. Manoteras 20, 28050 Madrid, Spain
[d] Department of Thermal Machines and Motors, University of Oviedo, Campus de Viesques, 33203 Gijón, Spain
[e] Vacuum Vessel Group, Fusion for Energy, Josep Pla 2, 08019 Barcelona, Spain

ARTICLE INFO

Article history:
Received 30 June 2009
Received in revised form 28 October 2009
Accepted 8 November 2009
Available online 5 December 2009

Keywords:
Weld distortions
Residual stresses
Electron Beam
Finite element
ITER

ABSTRACT

In this paper, an Electron Beam (EB) welded mock-up is used for the calibration of the finite element method welding simulation developed. Once the method is validated, a new simulation welding sequence is proposed with the objective to reduce the mock-up final distortions. The final distortions should be reduced from 7 mm to 1 mm.

Distortion measures and final deformed shapes are compared between two simulation scenarios to establish the effects of the new provisions assumed. The new simulation scenario includes a balanced longitudinal weld sequence in order to minimize the effects of the accumulative angular distortion process. The power of the FE method developed is demonstrated and will allow the optimization of the welding sequences to carry out previous simulation in FEM in future applications of ITER.

1. Introduction

In the last years, mechanic–thermal processes in which there is material contribution or thermal characteristics of the process are modified, using Finite Element Method (FEM), have been widely investigated. The size and the nature (coupled field, highly non linear analysis, etc.) of the problem of welding simulation requires large computational efforts to obtain reliable solutions [1–3].

An algorithm was developed [4] to simulate the thermal cycle during welding in which a space–time FEM is proposed to solve the transient convection–diffusion thermal equation. This method is suitable for the thermal analysis in the weld pool and weld joint region.

A three dimensional transient thermo-mechanical simulation has been performed [5] on a welded T-joint using FEM to predict longitudinal residual stresses. A contour method was used to experimentally validate the residual stresses. A prediction and controlling angular distortion in fillet welded joint and structure were investigated in [6]. In numerical analysis, a new in-house finite element code has been developed based on the idea of iterative substructure method (ISM) to calculate welding distortion in rational time.

Multipurpose ANSYS FE procedure for welding processes simulation of both Laser and TIG welding processes have been developed [7]. The special features are the applicability to a non-uniform gap and the use of a fast iterative procedure that assures the constancy of the fixed maximum temperature along the single pass and between each pass and the following, apart from their shapes and sizes.

Most of these simulations have been carried out using simple geometries to validate the proposed methods. In this paper, the methodology developed [8] will be used for the simulation and optimization of an Electron Beam mock-up welding sequence.

2. Mock-up description

During the preliminary studies made by Pro-Beam (Munich) in order to set the suitability of the Electron Beam welded design of the different parts of the VV, a mock-up with dimensions 1.5 m × 0.9 m × 0.34 m was built [9]. This mock-up includes some of the different types of joins found in PS1 (Poloidal Segment 1) and it is composed of two shells (inner and outer), two ribs, two housings and one key. Before welding the mock-up, every piece was mounted using a set of tack welds done by GTAW (Gas Tungsten Arc Welding). No supporting jigs or structures have been installed during the welding process so that the pieces are free to distort.

* Corresponding author. Tel.: +34 984199692; fax: +34 984199992.
E-mail address: julio@natec-ingenieros.com (J. Guirao).

0920-3796/$ – see front matter © 2009 Elsevier B.V. All rights reserved.
doi:10.1016/j.fusengdes.2009.11.002

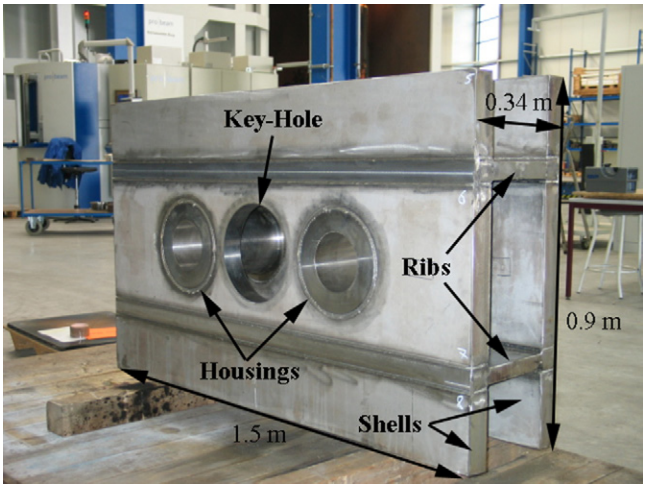

Fig. 1. EB welded mock-up (back view). Dimensions: 1.5 m × 0.9 m × 0.34 m.

All welds in the mock-up were done under similar production conditions as on the of real size models (Fig. 1).

3. Manufacture sequence

The tack welds were all done only in one side (exterior, Fig. 2) by GTAW (5 mm thickness).

Their positions (longitudinal welds) relative to the weld direction (Fig. 3) are summarized in Table 1.

To determine the final distortions of the mock-up, several reference points were defined to take dimensions before and after welding. The measurements were taken using a slide gauge and inside micrometer gauge. Large outer dimensions were taken with a precision of ±0.1 mm. Hole diameters were determined by a micrometer gauge, with an accuracy of ±0.01 mm.

Fig. 4 shows the dimensions checked and Fig. 5 the reference points where the measures have been taken.

The Electron Beam welds between ribs and shell (see Fig. 6) were done in the following order: (i) upper rib joint (front side), (ii) upper rib joint (back side), (iii) lower rib joints, and (iv) central cylinders housings and key.

The parameters of E-beam welding were:

- Acceleration voltage: 80 kV.
- Beam current: 275 mA.
- Welding speed: 5 mm/s.

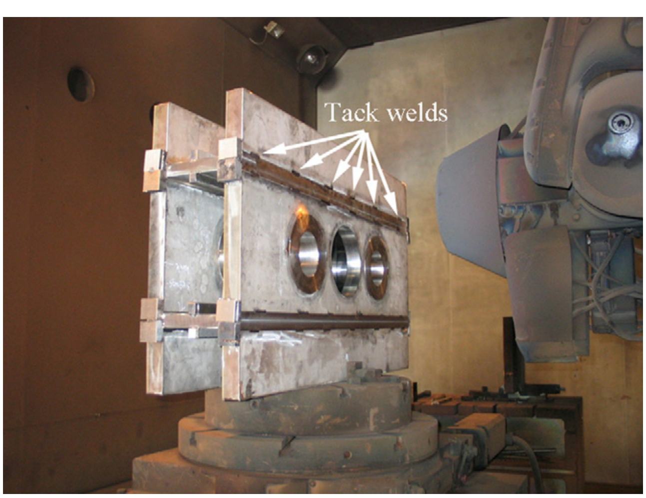

Fig. 2. Tack welds.

Fig. 3. Tack weld position references.

4. Finite element model

A realistic FE model of the mock-up was built, including every weld present in the manufacturing process, using the methodology described in [8] with ANSYS® Mechanical (Release 11.0) simulation software [10]. 3D solid elements were used in both thermal and mechanical models (SOLID70 for thermal analysis and SOLID45 for mechanical analysis). 3D node-to-node contact elements (CONTAC52) were used to simulate realistic boundary conditions (real contact between mock-up and external support, including friction effects). Fig. 7 shows the finite element model, boundary conditions and mechanical loads taken into account in the analysis.

For horizontal plates the following correlations [11] are recommended for upper surface of heated plate or lower surface of cooled

Fig. 4. Dimensions measured.