Title	Comparison on several kinds of T-E-P FEM software for welding
Author(s)	Wang, Yang; Luo, Yu; Zhang, Xueyuan
Citation	Transactions of JWRI. 39(2) P.82-P.84
Issue Date	2010-12
Text Version	publisher
URL	http://hdl.handle.net/11094/5923
DOI	
rights	
Note	

Comparison on several kinds of T-E-P FEM software for welding†

WANG Yang *, LUO Yu **, ZHANG Xueyuan *

KEY WORDS: (Software Comparison) (T-E-P FEM) (Welding Temperature) (Welding Deformation) (Welding Stress and strain)

1. introduction

Numerical simulation has been playing a significant role in manufacturing recently. The general Thermo-elastic-plastic (T-E-P) FEM software is now widely used in mechanical process as during welding. However, the research in comparing with in detail results using different T-E-P FEM software is rare. It is necessary to learn all the differences of calculation results by different software for industrial applications.

In this paper, three kinds of software (ANSYS, ABAQUS and JWRIAN) are employed in T-E-P FEM analysis for bead-on-plate welding, and temperature field and stress field are discussed in detail through a series of calculations. By comparing experimental results which are published, the differences among results which are obtained by different T-E-P FEM software are established.

2. Model for Analysis

Bead-on-plate welding is taken as the research model herein. The length, the width and the thickness of the plate is 400mm, 400mm and 10mm respectively. The material is low-carbon steel. The FE model has 2720 solid elements and 3690 nodes. The smallest size of element is 10mm×2.5mm×2.5mm.

The thermal properties and mechanical properties are dependent on temperature, which reference to paper [1].

DC3D8 and SOLID 70 are used for temperature calculation for Abaqus and Ansys, and to simulate the stress field, C3D8I and SOLID 45 are chosen for Abaqus and Ansys respectively. In order to simplify calculation, the moving welding arc is simplified as a cuboid interior moving heat source.

3. Calculated Results

Comparison of temperature

Figure 1 shows the distribution of temperature at center cross-section on the surface of plate when welding time is 21 sec, and total heat input parameter (Q/h^2) is 4.06 (J/mm^3), welding speed is 10mm/Sec.

From Fig.1, it is clearly seen that temperature results obtained from three kinds of software are consistent, the peak temperature has difference a bit.

Comparison of plastic strain

Figure 2 displays the plastic strain in Y-direction by Abaqus, Ansys and Jwrian at center cross-section in middle of the plate.

Figure 3 is the curve of the temperature history and plastic strain history at center that coordinate is (200, 0, 5).

It is found that plastic strain from Abaqus and Ansys are good in agreement, yet that from Jwrian is a little smaller than the others.

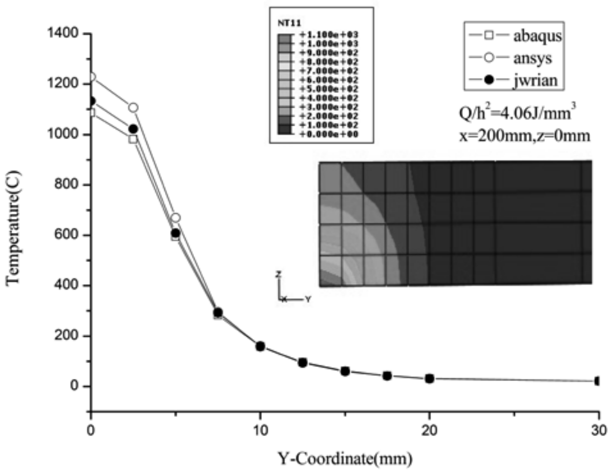

Fig. 1 Distribution of temperature at center cross-section

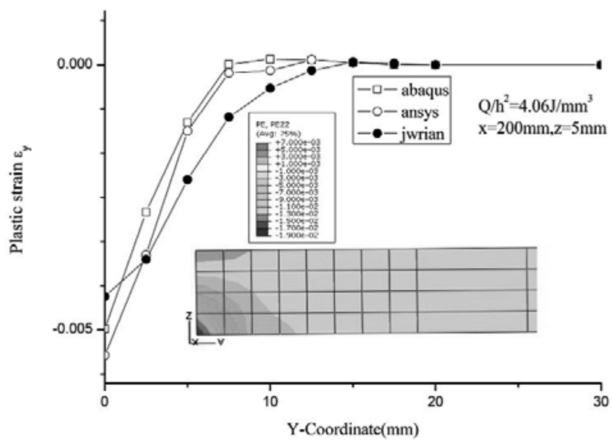

Fig. 2 Distribution of plastic strain at center cross-section

† Received on 30 September 2010
* Graduate School, Shanghai Jiao Tong University, Shanghai, China
** Shanghai Jiao Tong University, Shanghai, China

Transactions of JWRI is published by Joining and Welding Research Institute, Osaka University, Ibaraki, Osaka 567-0047, Japan

Comparison of residual stress

The comparison of residual stresses in welding direction(X) on surface at center cross-section is shown in **Fig. 4**.

According to Fig.4, it is found that results from Abaqus resemble those from Ansys, but the distribution area of residual stress obtained from Jwrian is narrower than Abaqus and Ansys.

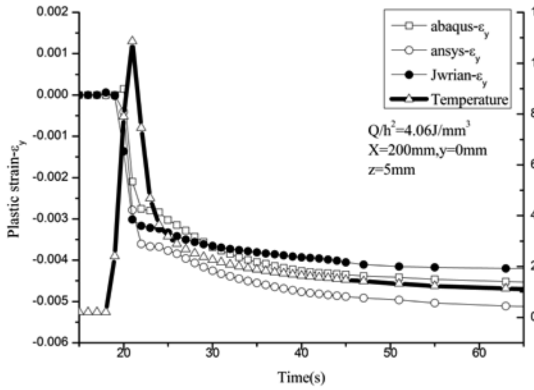

Fig. 3 Temperature and plastic strain histories

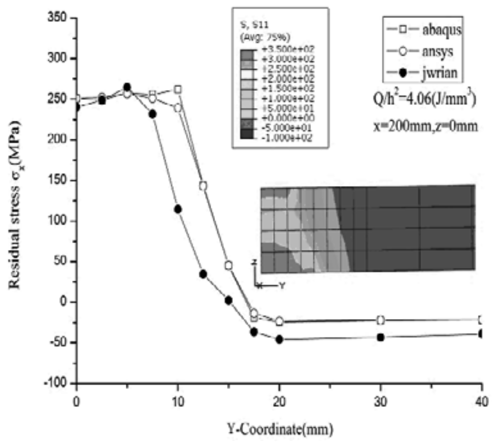

Fig. 4 Distribution of residual stresses at center cross-section

Comparison of deformation

The curve of relationship between transverse shrinkage and heat input parameter Q/h^2 is **Fig. 5**. As seen from Fig.5, the results calculated by the T-E-P FEM software agree with the experimental data [2], and also that from Jwrian is a little smaller than that.

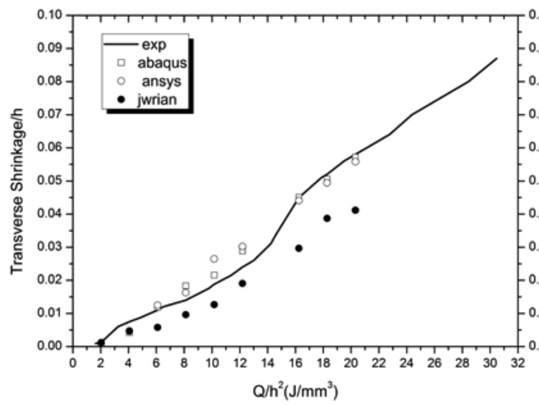

Fig. 5 Relationship between transverse shrinkage and heat input parameter

Figure 6 shows relationship between angular distortion and heat input parameter. Seen as Fig.6, if the heat input parameter Q/h^2 is small, the angular distortion obtained from calculator by the T-E-P FEM softwares are agreement in experimented data. With the increase of heat input parameter Q/h^2, the difference between the numerical results and experimental data increases. Reason may be that metal melting has not been considered in T-E-P FEM analysis. In the case of Jwrian, better results can be obtained in high heat input area compared with the other software.

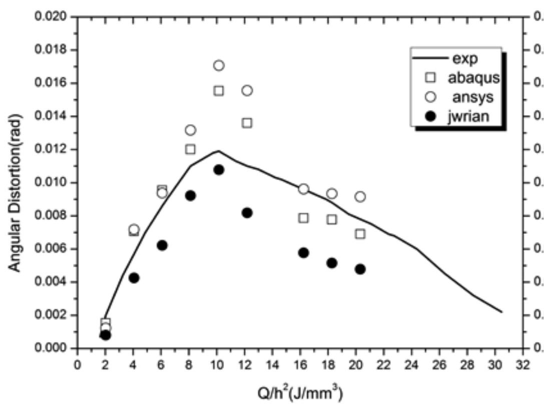

Fig. 6 Relationship between angular distortion and heat input parameter

Comparison of computed time

The computed time of temperature field and stress field are show in **Table 1**. From this table, it can be seen that Abaqus takes the shortest time among the three kinds of software in the same condition.

Table 1 comparison of computed time

Cost time	Abaqus	Ansys	Jwrian
Temperature filed (min)	7	22	8
Stress field (min)	36	90	200

4. Conclusions

The conclusions of this study are summarized as follows:

(1) The temperature results obtained from three kinds of software are consistent; the peak temperature has a small difference.
(2) The calculated results of residual stress accord closely each other, but distribution area of Jwrian is narrower than Abaqus and Ansys.
(3) The calculated transverse shrinkages agree with the experimental data, yet that from Jwrian is a little small than experimental data.
(4) If heat input is small, the calculated angular distortion is agreement with experiment data, with the increasing of heat input, the difference increases.
(5) Abaqus takes the shortest time among the three kinds of software in the same condition.

References

[1] Yu LUO, Hidekazu MURAKAWA and Yukio UEDA: Trans. JWRI, Vol. 26 (1997), No.2.467-475.

[2] Dean Deng: Doctoral Thesis, Osaka University, 2002, pp.21-22.