Title	Efficiency comparison between iterative substructure method and commercial software
Author(s)	Zhang, Linjie; Zhang, Jianxun; Serizawa, Hisashi; Murakawa, Hidekazu
Citation	Transactions of JWRI. 39(2) P.187-P.189
Issue Date	2010-12
Text Version	publisher
URL	http://hdl.handle.net/11094/4473
DOI	
rights	
Note	

Efficiency comparison between iterative substructure method and commercial software†

ZHANG Linjie*, ZHANG Jianxun*, SERIZAWA Hisashi and MURAKAWA Hidekazu****

KEY WORDS: (Finite element method) (Efficiency) (Iterative substructure method) (ANSYS) (Residual stress)

1. Introduction

A Welding process can result in undesirable residual stresses and distortion which is particularly critical for precision components. Because it is impossible to obtain the experimental data of every situation, computer simulations of welding processes are needed in order to predict the impact of different design options on residual stress and deformation. Finite element simulation has become a proven and reliable technique for prediction of welding deformations and residual stresses. However, finite element simulations are currently only used in applications where safety aspects are very important or when a large economic gain can be achieved [1]. The time and cost associated with reliable analysis of complex industrial problems appear to be the main reason contributing to this practice. Investigators have attempted to develop strategies focusing on different dominant factors that influence the efficiency of modeling actual welded structures [2–6]. Noting the fact that the high temperature region which exhibits strong non-linearity is limited to a very small area compared to the size of the model to be analysed and the remaining part is mostly linear, an iterative substructure method (ISM) has been developed to transform welding problem into the combination of a large linear problem and a small but moving strong nonlinear region, to improve the speed of computation.[7,8]

In this work, we carried out a comparison of calculation efficiency between an in house finite element (FE) code developed based on the idea of iterative substructure method (ISM) and commercial FE software ANSYS. To achieve the research purpose, thermal elastic-plastic finite element analysis of laser welding process of TC4 plate was taken as an example and was simulated by ANSYS and an in house developed FE code based on ISM, respectively. In order to make a comparison of residual stress between experiment and simulation, a hole drilling technique was used to measure the residual stresses in the weldments.

2. Experiment Procedures

Welding was done with a continuous wave CO_2 laser which delivers 3KW maximum mean power. The diameter of the minimum focal spot was 0.4mm. The material used in this study was TC4 of which the chemical composition is presented in **Table 1**. The geometry of the welded plate was 200mm×200mm×2mm. Samples were manually cleaned with acetone before welding. Argon was chosen as shielding gas. Welding of beads-on-plate was performed in a welding fixture to ensure that a constant focal length was used.

Then, experimental tests were conducted to collect data of residual stresses on the surface of workpiece. The hole-drilling method was adopted. **Figure 1** shows the positions at which the hole-drilling method was used to measure the residual stress.

Table 1 Chemical compositions of TC4 titanium alloy (wt %)

Al	V	Fe	Si	C	N	H	O	Ti
5.5~6.8	3.5~4.5	0.3	0.15	0.1	0.05	0.015	0.2	Bal

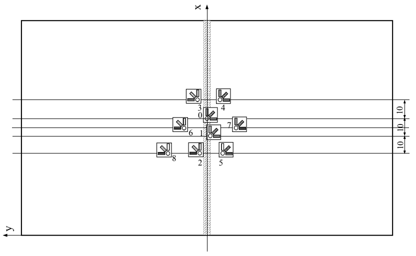

Fig. 1 The positions at which hole-drilling method was used to measure the residual stress.(Unit: mm)

3. Numerical Model

Figure 2 shows the coordinate system and the mesh used to predict the quasi-stationary shape of the molten pool. Only half of the specimen is considered as it is symmetrical about the symmetry plane of the weld. It can also be seen from Fig.2 that a dense mesh is put around the heat source and the vicinity of top surface and bottom surface, while for the other regions, a coarse mesh is employed. Eight node brick element is used in this study and the model contains a total of 20503 nodes and 16800 elements.

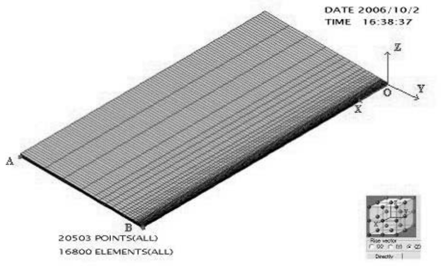

Fig. 2 Mesh of the plate

---

† Received on 30 September 2010

* State Key Laboratory for Mechanical Behavior of Materials, Xi'an Jiaotong University, Xi'an, China

** JWRI, Osaka University, Osaka, Japan

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

Material Properties

The thermal and mechanical properties of TC4 alloy employed in this study are shown in **Fig. 3**.

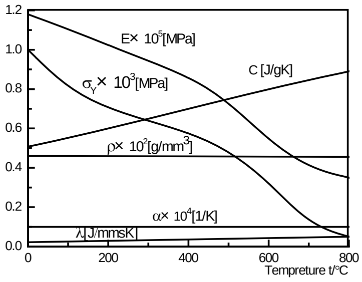

Fig. 3 Thermal and mechanical properties of TC4

Boundary Conditions

In thermal analysis, the thermal load was applied as a heat generation rate corresponding to a volume of internal heat generation power progressing along the weld line during analysis. Convection to the environment from all exterior faces of the workpiece was considered, while radiation heat loss was neglected.

In the structural analysis, the boundary condition is that points A, B and O (Fig. 2) are constrained in the z direction, points O is constrained in the x direction, and the plane of symmetry (i.e. x-o-z) is constrained in the y direction.

Analysis Procedure

Commercial FE software ANSYS and an in house developed FE code based on ISM were used to solve the model, respectively. To ensure a balanced comparison between the analysis results for each case, the same analysis procedure was used for both cases.

The problem was treated as a sequentially coupled analysis. First, a thermal analysis was performed to predict the temperature history of the model. Subsequently, temperature results of the thermal analysis were applied as body loads in a nonlinear transient thermal–mechanical analysis. The same mesh used in the thermal analysis was used for the thermal–mechanical analysis. A volumetric heat source was employed in the thermal analysis. The continuous welding process was approximated by the advancing of a volumetric heat source. During welding, the temperature, and consequently the temperature dependent material properties, changed very rapidly. Thus, material properties and the stiffness matrix were updated at every equilibrium iteration. In addition it is assumed that elements with a temperature higher than 1500 ℃ had little mechanical effect on the surrounding elements.

4. Results and Discussion

Figure 4 shows a comparison of CPU time between ANSYS and ISM. As shown in the figure, the computation speed of ISM is about 13 times faster that that of ANSYS.

Figure 5 depicts the distributions of longitudinal residual stress on the top surface as measured by the hole-drilling method and the finite element method. The experimental result of longitudinal stress at the weld center is about 670MPa. The longitudinal stress at the weld center predicted by ISM and ANSYS are about 800 MPa and 920MPa, respectively. In other words, at the central region of the weld, the results predicted by ISM are closer to the experimental result than that predicted by ANSYS. In addition, it is noted that when ANSYS is used the maximum longitudinal stress appears at the weld center. When ISM is used, the maximum longitudinal residual stress arises in the heat affected zone(HAZ), which is similar to the experimental results. It can also be seen from Fig.6 that ISM predicted a slightly narrower width of the tensile stress zone than the experimental result, while ANSYS predicted a tensile stress zone in which the width is appreciably larger than the experimental result.

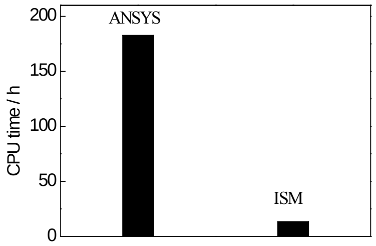

Fig. 4 A comparison of CPU time between ANSYS and ISM.

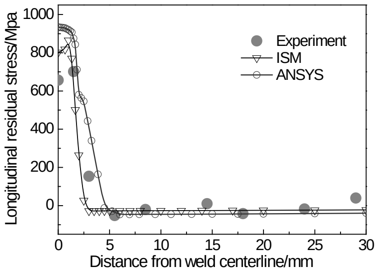

Fig. 5 Results of longitudinal stress

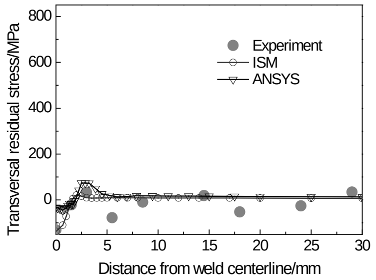

Fig.6 Results of transversal stress

Furthermore, **Fig. 6** represents the distributions of residual stress on the top surface in the Y-direction, as measured by hole-drilling method and the finite element method. As Fig. 7 shows, ISM provided a slightly lower stress than the experimental results at the centerline of the weld, but are close to the experimental results in the HAZ.

5. Conclusions

(1) The computation speed of ISM is about 13 times faster than that of ANSYS;

(2) At the central region of weld, the results predicted by ISM are more accurate than those predicted by ANSYS;

(3) The tension stress zone predicted by ISM is narrower than the experimental result, while that predicted by ANSYS is wider than the experimental result.

References

[1] L.E. Lindgren, J. Thermal Stresses 24 (2001) 305–334.

[2] J.M.J. McDill, A.S. Oddy, Comput. Struct. 54 (1995) 183–189.

[3] J.M.J. McDill, J.A. Goldak, A.S. Oddy, M.J. Bibby, Comm. Appl. Num. Methods 3 (1987) 155–163.

[4] L.E. Lindgren, H.A. Haggblad, J.M.J. McDill, A.S. Oddy, Comput. Methods Appl. Mech. Eng. 147 (1997) 401–409.

[5] Y. Ueda, M.G. Yuan, ASME J. Eng. Mater. Technol. 115 (1994) (1993) 417–423.

[6] A. Bachorski, M.J. Painter, A.J. Smailes, M.A. Wahab, J. Mater. Process. Technol. 92/93 (1999) 405–409.

[7] H. Nishikawa, H. Serizawa and H. Murakawa: Sci. Technol. Weld. Join., 2007, 12, (2), 147–152.

[8] H. Serizawa, S. Itoh, T. Tsuda, H. Nishikawa, M. Shibahara and H. Murakawa: Proc. Int. Conf. on ‘Welding science and engineering’, Xi’an, China, October 2005, Xi’an Jiaotong University, 240–246.