

Numerical simulation of horizontal transversal displacement under different fixed clamping force of welding fixture for superalloy during TIG welding

Zhang Wenyang¹, Guo Shaoqing¹, Yuan Zhongjin², Tang Siyi¹, Liang Jiayu¹ and Zhang Xuejun¹

张文扬, 郭绍庆, 原中晋, 唐思熠, 梁家誉, 张学军

1. Welding and Forge Process Laboratory, AECC Beijing Institute of Aeronautical Materials, Beijing 100095, China;
2. Dassault Systemes SIMULIA Corp. (China), Beijing 100025, China

Receive 19 January 2018; accepted 4 April 2018

Abstract A numerical model is established by Abaqus software for superalloy during tungsten inert gas (TIG) welding with welding fixture to simulate the weld temperature field, stress and strain field, and weld plate's horizontal transversal shift. The clamp force perpendicular to the weld plate varying with welding time was analyzed. The maximum clamp force during welding process was calculated. Under the condition of insufficient of clamping force, a half or one third of the maximum clamp force for example, the weld plate's horizontal transversal displacement could happen and also be computed.

Key words TIG welding, welding fixture, clamping force, transverse displacement, finite element method

0 Introduction

The tungsten inert gas (TIG) welding, manually or automatically, is the most common welding process in aviation industry. Despite many advanced welding technologies are available, such as laser and electron beam welding, TIG welding predominates in welding field due to its simplicity, convenience and utility. Many research works have adopted numerical simulation into studying the complex weld process of TIG welding, such as temperature field, stress and strain field, and final distortion. However, lots of researches^[1-6] only have weld plate in their numerical simulation model. The influence of fixture is not generally included in numerical simulations of welding process, however the fixture constraint plays an important role in accurate modeling of welding process. Yang et al.^[7] took the effect of fixture on the welding stress and distortion of thin aluminum plate joints into consideration. Zhang et al.^[8] proposed numerical models in which the mechanical interaction between welded panel and fixture is considered as contact pair in the finite el-

ement analysis (FEA) of welding process. This could make the FEA of welding process more scientific and accurate. The present work proposes a model concluding a typical fixture in a welding lab and analyzes weld plate's horizontal transversal displacement at different observation points under different clamping force.

1 Model and parameters

GH242 alloy was adopted in the experiment. It is an important high temperature alloy in aviation industry. Weld plate size is 150 mm × 50 mm × 2.4 mm, which is made of GH242 alloy. The GH242 alloy physical property is showed in Table 1. Welding process parameters are listed in Table 2.

The fixture material is made of No. 40 steel with thermal conductivity 45.53 W/m · K and specific heat 480 J/kg · K respectively. The fixture size is 250 mm × 85 mm × 15 mm (lower) and 250 mm × 70 mm × 15 mm (upper) respectively. The schematic of assembly diagram is seen in Fig. 1. Three points on the weld plate are set to be as observation points, seen in Fig. 2.

Foundation item: Project was supported by the National Natural Science Foundation of China (Grant No. 51275496).

Corresponding author: Zhang Wenyang, E-mail: 13681047467@163.com

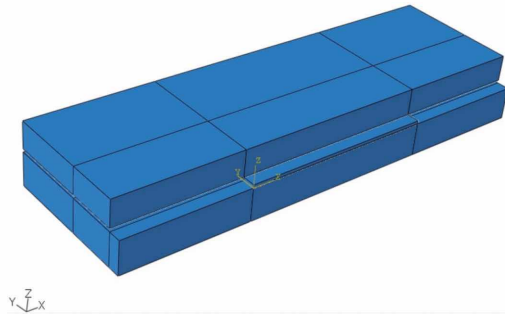
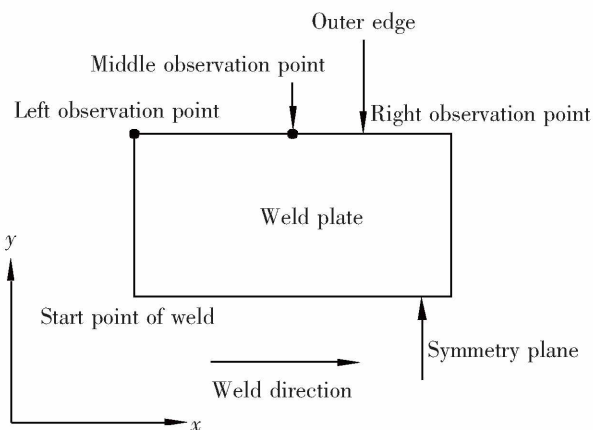
doi:10.12073/j.ew.20180119001

Table 1 GH242 physical property

Specific heat /(J · kg ⁻¹ · K ⁻¹)	Yield strength /MPa	Poisson's ratio	Elastic modulus /(10 ⁸ N · m ⁻²)	Density /(kg · m ⁻³)	Coefficient of thermal conductivity /(W · m ⁻¹ · K ⁻¹)	Temperature /K
386	450	0.35	22.44	9 050	11.3	300
405	450	0.35	22.05	9 050	12.6	373
580	450	0.35	18.91	9 050	22.5	973
630	450	0.35	13.52	9 050	29.2	1 373
720	450	0.35	9.80	9 050	36	1 773

Table 2 Welding process parameters

Heat source power /J	Arc voltage /V	Welding current /A	Power coefficient	Welding speed /(m · s ⁻¹)
700	10	100	0.7	0.005

**Fig. 1 Assembly diagram****Fig. 2 Position of three observation points**

2 Welding temperature field

The present work adopts sequence coupling, com-

putes temperature field firstly and then computes stress and strain field. This research used double ellipsoid heat source mode to do simulation and get weld temperature field, see formula (1) and formula (2). The DFLUX subroutine offered by ABAQUS is needed to simulate the heat source's move. Three radius in double ellipsoid heat source is $a = 0.0014$ m, $b = 0.0017$ m, $c = 0.0016$ m respectively. In the solution of transient temperature field, element type is DC3D8 heat conduction unit, and the convective heat transfer and heat transfer of the corresponding surface are also taken into consideration. Weld temperature field is seen in Fig. 3.

$$q(x, y, z, t) = \frac{6\sqrt{3}f_f P}{\pi^{\frac{3}{2}}a^2c} \times \exp\left\{-\frac{3[x + v(\tau - t)]^2}{a^2}\right\} \times \exp\left(-\frac{3y^2}{a^2}\right) \times \exp\left(-\frac{3z^2}{a^2}\right) \quad (1)$$

$$q(x, y, z, t) = \frac{6\sqrt{3}f_r P}{\pi^{\frac{3}{2}}a^2c} \times \exp\left\{-\frac{3[x + v(\tau - t)]^2}{a^2}\right\} \times \exp\left(-\frac{3y^2}{a^2}\right) \times \exp\left(-\frac{3z^2}{a^2}\right) \quad (2)$$

f_f, f_r heat flux distribution coefficient: $f_f = 2/3$, $f_r = 4/3$.

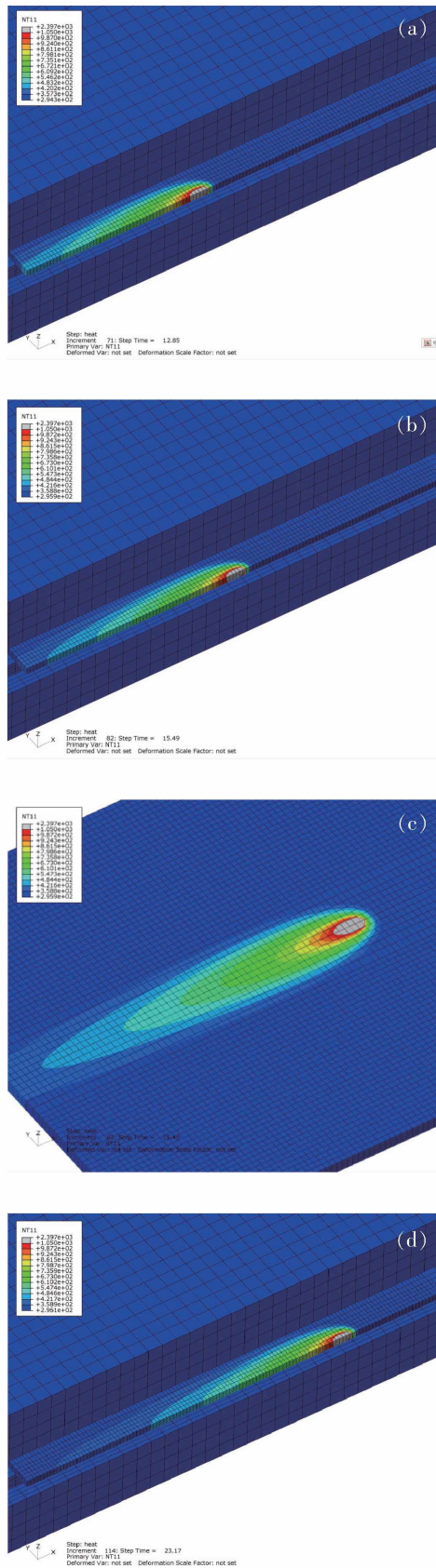


Fig. 3 Weld temperature field at different times (a) – (d) 12.85 s, 15.49 s (half), 15.49 s (full), 23.17 s, respectively

3 Gravity and the support reaction over time

The weld temperature field (Fig. 3) as a load is used to compute the stress and strain field after the temperature field is done. The present work considers the effect of fixture's gravity and the friction between the fixture and the weld plate (the friction coefficient is 0.3). The support reaction over time is seen in Fig. 4.

In the first step with carefully choosing suitable loading time, for the sake of avoiding load shocking, gravity is gradually loaded within the first four seconds, while the upper fixture is in free state. The second step is heat process, at the same time the upper fixture is fixed.

The support reaction force begins and lasts to 25 s until the heat process is over. Then the third step (cooling step) begins and lasts 44.4 s. During the cooling process, the support reaction force achieves its maximum value 7.2×10^4 N.

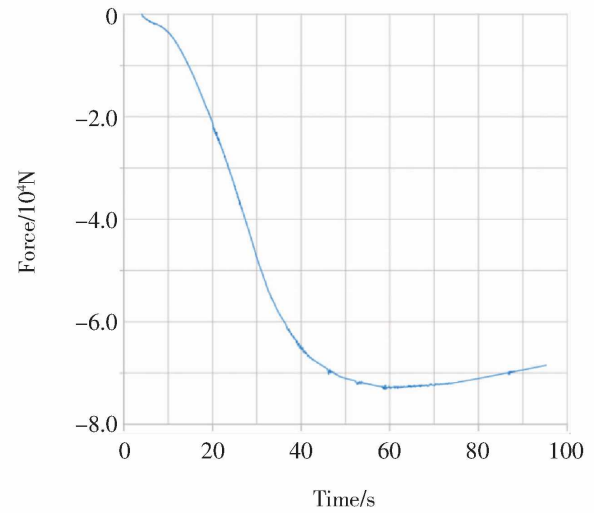


Fig. 4 Support reaction force over time

4 Horizontal displacement

After getting the maximum support reaction, this research respectively used the 1/2 and 1/3 of the maximum support value to apply them as a boundary condition to compute the horizontal transverse displacement, which are similar situations to the insufficient fixture clamping force. The results are seen in Fig. 5.

Fig. 5 shows as a whole that the weld plate will expand firstly and then contract during the welding process under the conditions of different clamping force. In Fig. 5d diagram, two plots (under 1/2 and 1/3

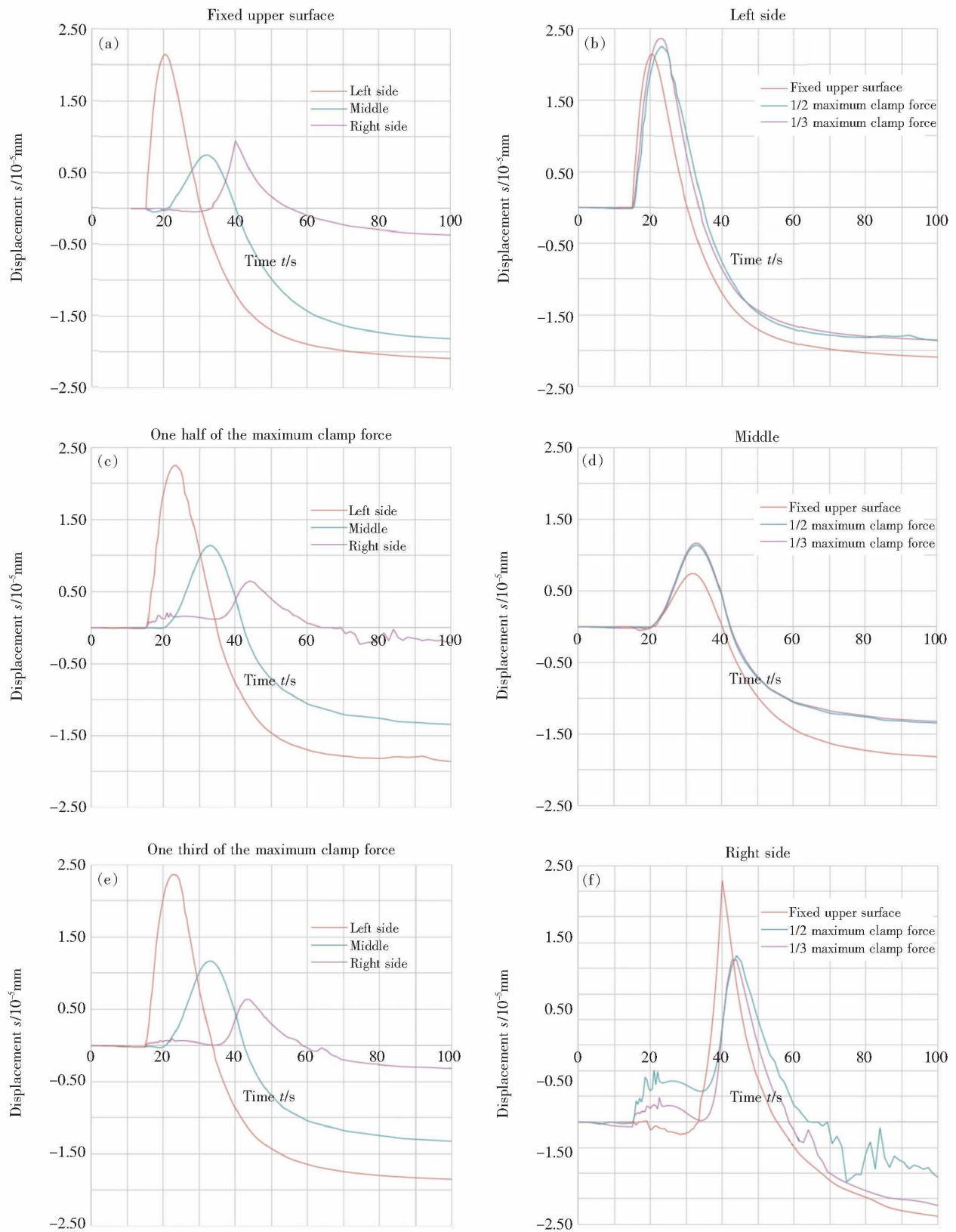


Fig. 5 Horizontal transversal displacement over time (a), (c), (e) shows horizontal transversal displacement at three points respectively (left side, middle, right side) under three different clamp force (fixed upper surface, one half of the maximum clamp force, one third of the maximum clamp force); (b), (d), (f) shows horizontal transversal displacement under three different clamp force (fixed upper surface, one half of the maximum clamp force, one third of the maximum clamp force) respectively at three points

of the maximum clamping force respectively) are coincidence. We can see from Fig. 5a, Fig. 5c, Fig. 5e that the horizontal transversal displacement on left side has the maximum values, and the minimum value on the right side. We also see from Fig. 5b, Fig. 5d, Fig. 5f that the horizontal transversal displacement of the same point at different clamping force is very similar.

5 Conclusions

(1) The present work establishes the physical model which considers the fixture's effect.

(2) The maximum support reaction is computed and is used to simulate the horizontal transversal displacement under three different fixture clamping force.

(3) The horizontal transversal displacement reflects the moving direction of weld heat source; the early being welded parts will expand and contract firstly. Due to the cumulative effect, the early being welded parts have the most horizontal transversal displacement value.

References

- [1] Preston R V, Shercliff H R, Withers P J, et al. Finite element modeling of tungsten inert gas welding of aluminum alloy 2024. *Science and Technology of Welding and Joining*, 2003, 8(1): 10–18.
- [2] Yu S R, Fan D, Xiong J H, et al. Numerical simulation on temperature field in laser welding of thin aluminum alloy plate with different thickness. *Transactions of the China Welding Institution*, 2007, 28(5): 17–20.
- [3] Zhang Z L, Silvanus J, Li H K, et al. Sensitivity of history dependent material mechanical models for numerical simulation of welding process. *Science Technology Welding and Joining*, 2008, 13(5): 422–429.
- [4] Zhang G D, Zhou C Y. Finite Element Simulations of Welding Residual Stress and Creep Damage for Welded Joint. *Acta Metallurgica Sinica*, 2008, 44(7): 848–852.
- [5] Yu S R, Fan D, Xiong J H. Numerical simulation on stress-strain field of laser welding aluminum alloy joints with different thickness. *Transactions of the China Welding Institution*, 2008, 29(10): 25–28.
- [6] Yan D Y, Shi Q Y, Wu A P, et al. Numerical analysis on the residual distortion of Al alloy sheet after friction stir welding. *Acta Metallurgica Sinica*, 2009, 45(2): 183–188.
- [7] Yang J G, Li J, Liu X S, et al. Numerical simulation about the effect of fixture on the welding stress and distortion of thin aluminum plate joints[J]. *China Welding*, 2008, 17(4): 39–42.
- [8] Zhang Z L, Shi Q Y, Yan D Y, et al. Establishment and application of fixture constraint models in finite element analysis of welding process. *Acta Metallurgica Sinica*, 2010, 46(2): 189–194.