Simulation Analysis of Flat Butt Welding Based on ANSYS

Kunpeng He, Kehua Zhan, Yunxiang Wang, Hu Li, Yongzhen Wang

College of Mechanical and Electronic Engineering, Shandong University of Science and Technology, Qingdao 266590, China.

1162626831@qq.com

Abstract

For the welding deformation and residual stress problems in flat welding, the ANSYS finite element software is used to simulate the flat welding process by indirect simulation. The deformation and stress distribution after welding are obtained. To reduce welding deformation and provide a theoretical basis for actual production.

Keywords

Welding deformation; residual stress; finite element.

1. Introduction

Welding splicing is a manufacturing process and technology for joining metal or other thermoplastic materials such as plastics by heating, high temperature or high pressure [1]. Since there are many factors affecting the welding deformation, how to accurately predict the welding deformation is particularly important. If the experiment is conducted, not only a large amount of time is required, but also a large amount of money is required. With the development of finite element software, it provides a better method for welding simulation analysis [2].

In this paper, the finite element software ANSYS is used to simulate the deformation of flat butt welding, and the temperature field and stress field distribution are obtained. Through these calculation data, the welding quality can be accurately tested and evaluated.

2. Establishment of Finite Element Model

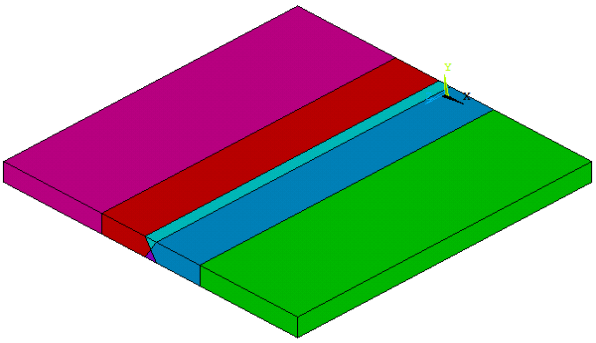

Figure 1. Finite element model

When copper alloy castings are repaired by defects, due to the large linear expansion coefficient of copper alloy, hot cracks are easily generated during welding. The welding process is optimized by simulation calculation to avoid the occurrence of hot cracks in copper alloy castings. Two L300mmW*150mm*H20mm copper alloy plates are butt welded, one weld on the top and bottom, 60° groove, TIG welding.

2.1 Meshing

In the meshing, regular hexahedral meshing is adopted for the weld zone and the remote zone, and free meshing is adopted for the excessive zone, and the mesh of the weld zone is thinner, and the mesh is far away from the zone, which can reduce the calculation. The quantity also guarantees the convergence of the calculation [3].

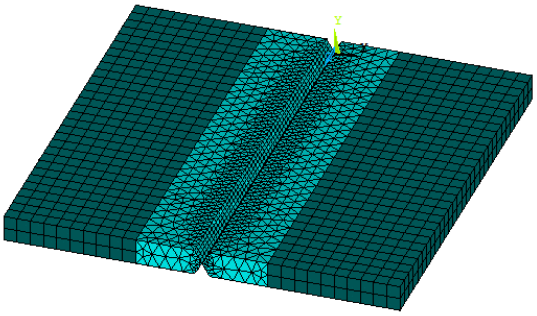

Figure 2. Meshing results

2.2 Life and Death Unit Method

The so-called "unit life and death" is to add or remove materials in the model, so that the corresponding unit in the model is either "born" or "dead". In ANSYS thermal analysis, “killing” a unit in a finite element model does not remove the unit from the model, but multiplies its heat conduction matrix by a small factor (eg, 10-6), the heat of the dead unit. The load, mass and specific heat capacity are set to zero. Similarly, the “birth” of a unit is not to add it to the model, but to reactivate it to restore the unit's specific heat capacity, heat transfer and quality to its original state.

2.3 Welding Heat Source

In order to better simulate the welding process, a Gaussian heat source model was used.

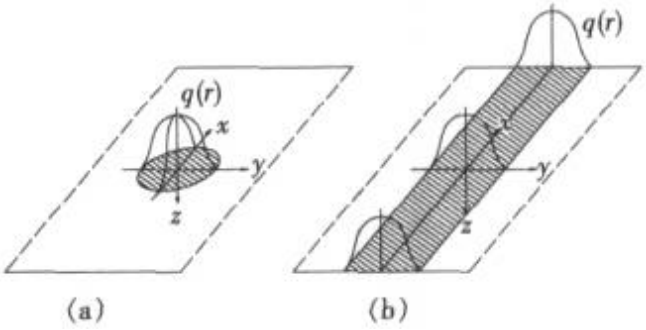

Figure 3. Gaussian heat source model

The heat expression at a distance r from the center of the heat source is:

$$q = \eta UI \tag{1}$$

Where: η is the thermal efficiency during the welding process; U is the voltage of the arc, V; I is the current of the arc, A [4].

Similarly, the heat flux density $q^{'}$ of the weld pool delivered by the welding is similar to the Gaussian normal distribution [5]:

$$q^{'} = \frac{k}{\pi} q e^{-kr^2} \tag{2}$$

Where: K is the coefficient, 1/mm2; r is the distance from the heat source, m.

3. Analysis of Welding Results

3.1 Welding Temperature

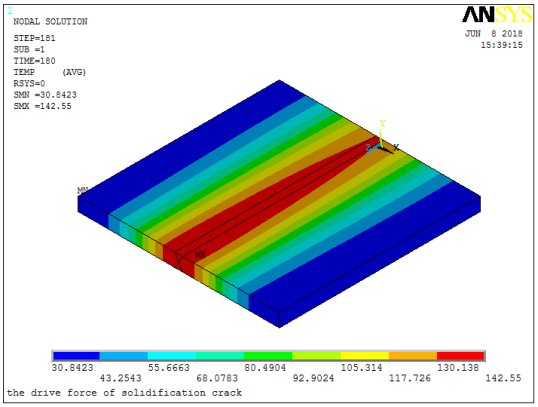

Figure 4. Temperature field distribution

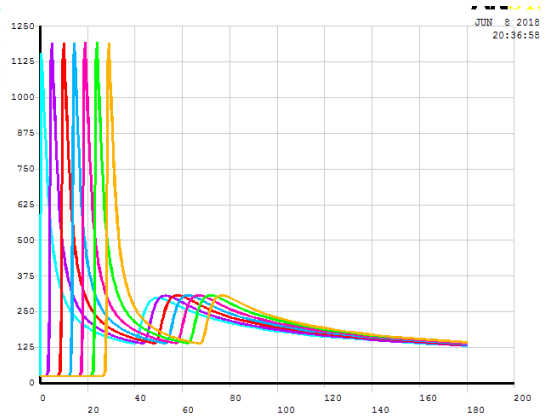

Figure 5. Temperature versus time curve

A number of points are selected on the upper surface of the first weld to obtain a curve of temperature at each point as a function of time. It can be seen from the figure that the temperature has experienced a process of rising from low to high, and after reaching the highest point, the temperature begins to fall back. During the temperature rise process, the curve gradient is larger and the temperature rise is faster; during the temperature drop process, the curve gradient is smaller and the temperature drop is slower.

3.2 Stress Field Analysis

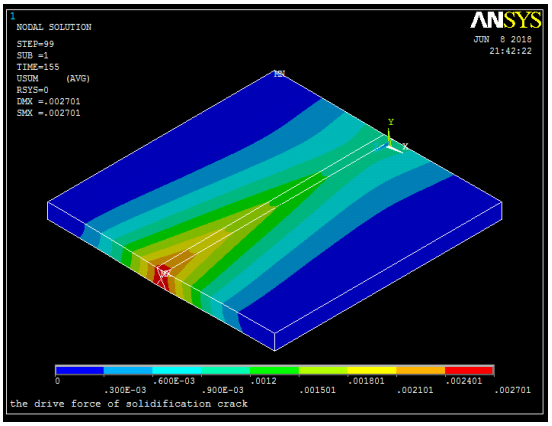

Figure 6. Total deformation after welding

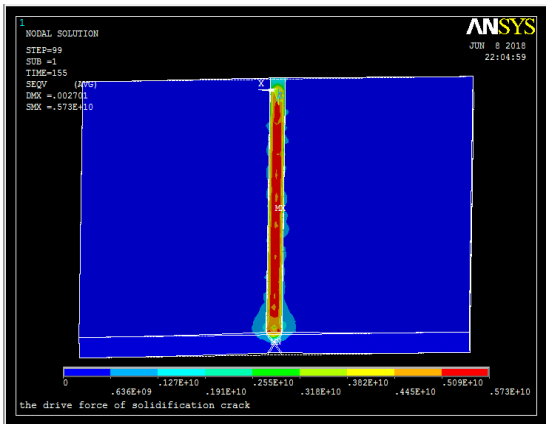

Figure 7. Equivalent stress diagram

It can be seen from the figure that the maximum deformation after welding is 0.271 mm and the maximum equivalent stress is 57 MPa. Figures 6 and 7 show the residual stress and strain of the welded steel plate after double-sided welding, and the stress and strain can be seen. Produced in the vicinity of the weld, when the fixed constraint is fixed, the two sides of the side are fixed, and the stress and strain away from the weld gradually approach zero, which is consistent with the actual situation.

4. Conclusion

The numerical simulation calculation using ANSYS can optimize the process in the whole welding process and avoid the generation of hot cracks. It has great practical significance in production. At the same time, the calculation method of the coupling method also improves the analysis speed.

Acknowledgements

I would like to thank all of my mates in the lab for assisting me in many ways for this thesis.Last but not the least, I would like to express my gratitude to my family for supporting me in many ways throughout my life.

References

[1] Deng Dean, Liang Wei, Luo Yu, Cun Chuan, Ying Yi. Prediction of welding deformation of low carbon steel pipe by thermoelastic finite element method[J].Transactions of the China Welding Society, 2006,27.

[2] Chen Jiaquan, Xiao Shunhu, Wu Gang, Yang Xinyan.Comparison of numerical simulation heat source modes in welding process [J].Welding Technology,2006,35 (1).

[3] Song Wei, Wang Jianli, Wang Hongfeng. Analysis of Deformation of Thin Plate Welding [J]. Thermal Processing Technology, 2013, 42 (15).

[4] Li Yana. Numerical method research and engineering application of welding deformation prediction and control [D]. Liaoning: Dalian Jiaotong University, 2010.

[5] Zhang Ximei, Zhang Shilian, Hu Tieiu. Analysis of deformation and residual stress of T-beam fillet welding [J].Ship Science and Technology, 2008.