Y. Z. ZHONG, J. YUAN, C. HAN, X.M. FANG, G. C. YU, H. C. JI

ISSN 0543-5846
METABK 62(1) 96-98 (2023)
UDC – UDK 669.182:621.791.057:620.16:621-75=111

Q345C WELD DEFORMATION ANALYSIS OF BUTT JOINTS

Received – Primljeno: 2022-06-21
Accepted – Prihvaćeno: 2022-08-20
Original Scientific Paper – Izvorni znanstveni rad

Q345C steel is welded under two different pre-welding heating methods of traditional flame heating and ceramic sheet heating. The finite element analysis of the butt joint weldment by ABAQUS software is used to summarize the law of welding deformation and compare the welding quality under different pre-welding heating methods according to the residual stress and deformation trend of the welding joint weld. The results show that the main stress after welding is concentrated in the weld seam and the surrounding area. The residual stress after welding after the heating and welding of the ceramic sheet is significantly less than the residual stress after the traditional flame heating welding, and the welding after the heating of the ceramic sheet can obtain better welding quality.

Keywords: Q345C; welding simulation; residual stress; the amount of deformation

INTRODUCTION

At present, the welding process has been very mature. How to improve the quality of welding and extend the service life has become the main research problem. During welding, the local temperature rise of welding is fast, and the welding deformation is high, resulting in high residual stress after welding [1]. The welded pipe deformation is too large and the bending is serious. It is not only seriously affects the service life of the weldment structure itself, but also brings difficulties in production and use to the equipment construction and the whole associated with it [2]. Studying the residual stress distribution after welding of welding joints under different pre-weld heating methods is of great significance to improve the welding quality and reliability of welding. It is also the primary prerequisite for ensuring industrial safety production [3]. In this paper, Q345C steel is used as the research object, and the simulation software ABAQUS is used to conduct finite element simulation analysis of the welding process. The distribution of welding residual stresses in different pre-welding heating methods is studied.

FINITE ELEMENT ANALYSIS PRE-PROCESSING

This paper uses ABAQUS software to simulate the welding of the butt joint. When the pre-processing

modeling is carried out, the GUI in the software is usually used to manually enter the parameter information to complete the modeling process. This method can accurately and conveniently define model information, and can also understand the model status in real time.

The test material used is the Q345C steel structure widely used by domestic medium and thick plate enterprises, which has excellent strength, ductility and processability [4]. Its chemical composition is shown in Table 1. The size of the butt weldment is 1000 *1000 *60 mm. Due to the thicker plate and the deeper molten pool, the double ellipsoid heat source model is adopted, and the distribution model is shown in Figure 1. The simulated welding current I = 250A, voltage U = 27 V, welding speed of 0,05 m/s, heat source efficiency of 0,8, heat input of 14 kJ/cm, welding time of 20 s, cooling time of 300 s.

During the welding process, the weld and its surrounding heat affected zone are greatly affected, and the intensity of activity gradually decreases with the area away from the weld. Therefore, in order to ensure the accuracy and computational efficiency of the simulation, the mesh of the weld seam and heat affected zone is dense when meshing. The thermophysical parameters and mechanical properties of welding materials change with temperature, mainly including density, thermal conductivity, elasticity, specific heat, plasticity and coefficient of thermal expansion. When performing welding simulation calculations, the initial ambient temperature is set to 100 °C in the boundary setting, which means that the weldment is heated to 100 °C by conventional flame heating and ceramic sheets. The difference is that the predefined field function is set in the traditional flame heating, indicating that the temperature decreases with the change of thickness. Absolute zero is defined as -273,15 °C and Boltzmann's constant 5,67E-08.

Table 1 **Q345C chemical composition/wt.%**

C	Si	Mn	P	S
0,15	0,23	1,44	0,02	0,005
Cr	**Mo**	**V**	**Ni**	**Cu**
0,025	0,0077	0,031	0,0095	0,016

During the welding process, the weld and its surrounding heat affected zone are greatly affected, and the

Y. Z. Zhong, C. Han, X.M. Fang, G. C. Yu, H. C. Ji, CHINA 22 MCC GROUP CORPORATION LIMITED; J. Yuan, H. C. Ji (E-mail: jihongchao@ncst.edu.cn) College of Mechanical Engineering, North China University of Science and Technology, Hebei, Tangshan, China.

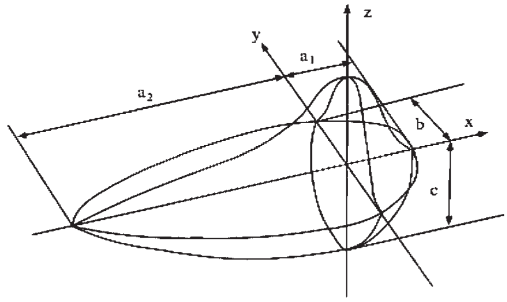

Figure 1 Double ellipsoidal heat source distribution model

intensity of activity gradually decreases with the area away from the weld. Therefore, in order to ensure the accuracy and computational efficiency of the simulation, the mesh of the weld seam and heat affected zone is dense when meshing. The thermophysical parameters and mechanical properties of welding materials change with temperature, mainly including density, thermal conductivity, elasticity, specific heat, plasticity and coefficient of thermal expansion. When performing welding simulation calculations, the initial ambient temperature is set to 100 °C in the boundary setting. It means that the weldment is heated to 100 °C by conventional flame heating and ceramic sheets. The difference is that the predefined field function is set in the traditional flame heating, indicating that the temperature decreases with the change of thickness. Absolute zero is defined as -273,15 °C and Boltzmann's constant 5,67E-08.

SIMULATION RESULTS AND ANALYSIS

After the simulation of the cooling stage after the end of the welding process of the Q345C weldment by ABAQUS, the stress field after the weldment is unevenly heated to 100 °C, and the stress field after the welding is shown in Figure 2. As can be seen from the figure, the maximum stress on the edge of the weld reaches 450 MPa, and the stress is concentrated on the weld and both sides. Under the uniform heating of the weldment, the stress field distribution after the preheating temperature reaches 100 °C after the welding is shown in Figure 3. From the figure, it can be obtained that the stress at the edge of the weld reaches 395 MPa at the maximum, and the overall residual stress value is the smallest.

The weldment selects the same measuring point at a distance of 0, 10, 30 mm from the weld seam, and draws

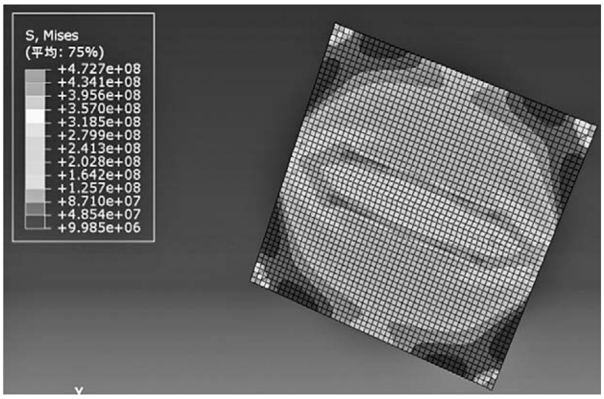

Figure 2 Stress field distribution during the welding cooling phase under conventional flame heating

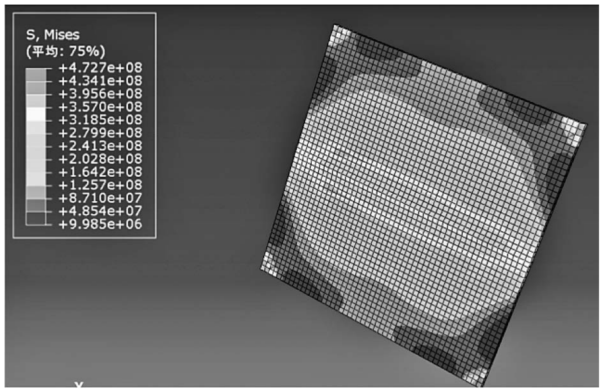

Figure 3 Stress field distribution during the welding cooling phase under conventional flame heating

the stress magnitude during the welding process. The stress change curve of the welding stage under the traditional flame heating condition is shown in Figure 4. The stress change curve of the welding stage under the heating condition of the ceramic sheet is shown in Figure 5. The maximum stress in the spot welding process of 0 mm from the weld seam is 500 MPa and 450 MPa, and the maximum residual stress after welding is 390 MPa and 350 MPa, respectively. The residual stress of the weld seam will increase sharply during welding. After welding, when it is cooled for 600 s in the natural state, the residual stress field at three points will tend to a stable value. Through the comparison of the two heating meth-

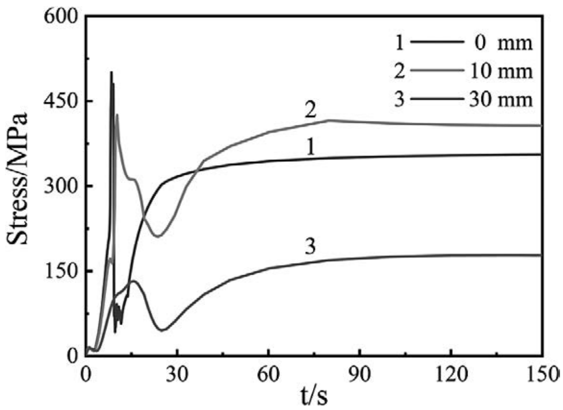

Figure 4 Characteristic point stress change curve during welding under conventional flame heating

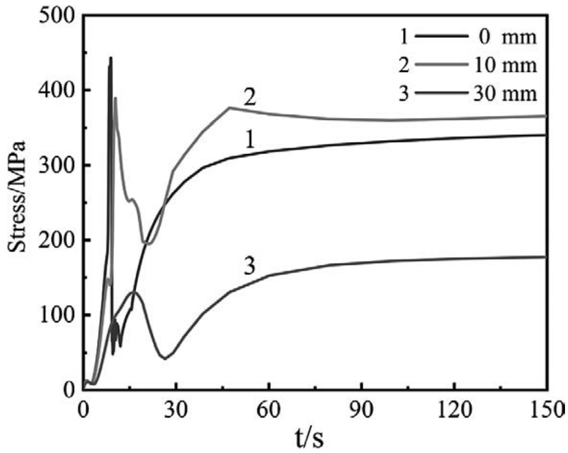

Figure 5 Characteristic point stress change curve when welding under ceramic sheet heating

ods, the residual stress field of welding under traditional flame heating and ceramic sheet heating is quite different. The residual stress after welding under ceramic sheet heating is smaller than that after welding under traditional flame heating, and the welding quality is better.

CONCLUSION

Based on ABAQUS software, the Q345C steel plate deformation analysis after welding is simulated. The pre-processing includes automatic import of mesh model, cross-section creation, assembly creation, material attribute assignment, and description of the loading of welding heat source by using DFLUX sub-programs. Through the comparison of two different heating methods, it can be concluded that the stress concentration phenomenon is more obvious, mainly distributed in the weld and its adjacent area. In the transverse direction, that is perpendicular to the direction of the weld, the trend of stress change is basically the same. The farther away from the weld bead, the smaller the welding heat affected zone and the weaker the thermal cycle effect. Welding under traditional flame heating and ceramic sheet heating has a large difference in the residual stress field, through comparison can be concluded that the residual stress after welding under ceramic sheet heating is smaller than the residual stress after welding under traditional flame heating, and the welding quality is better.

Acknowledgments

This work is supported by the Tangshan Talent Foundation Innovation Team (20130204D) and funded by Tangshan City Major Achievement Transformation Project (Grant No. 19140203F).

REFERENCES

[1] B. Lu, H. Liu, B. Li, et al. Development trend of special aluminum and aluminum alloy profiles and pipes [J], Aluminum Processing, (2017) 5, 40-44.

[2] L. Zhang. Research on application and development trend of aerospace high-strength aluminum alloy materials [J]. New Materials Industry, (2021) 3, 7-11.

[3] H. C. Ji, Z. Ma, X. M. Hung, et al. Damage evolution of 7 075 aluminum alloy basing the Gurson Tvergaard Needleman model under high temperature conditions [J]. Journal of Materials Research and Technology, (2022) 16, 398-415.

[4] C.T. Zhang, M. Q. Gong, L. Zhu, et al. Post-fire mechanical behavior of Q345 structural steel after repeated cooling from elevated temperatures with fire-extinguishing foam [J]. Journal of Constructional Research, (2022) 191, 107201-107206.

Note: The responsible translator for English language is M.M. LI -North China University of Science and Technology, China