Practical Research on Welding Simulation of Q345 Steel

Zhi-ling Wang

Jiangsu Province Xuzhou Technician Institute, Xuzhou 221151, China
379713268@qq.com

Abstract. In this paper, we use TCL/TK integrated language analysis software and semi-ellipsoid heat source model and life-death element method to simulate the butt welding process of Q345 steel plate with thickness of 10 mm. We simulated and analyzed the temperature field, stress field and welding deformation of multi-layer welding. Through welding simulation analysis, we extracted the distribution of welding residual stress and welding deformation of butt joint, and obtained the transverse and longitudinal shrinkage of butt plate and the main reasons. The results show that the multi-layer butt welding seam and its adjacent area are under tension, and the numerical value generally reaches the yield strength, and the two sides are under compression. The tensile stress in the middle of the weld bead is the largest and decreases gradually toward both ends. The cross-sectional area of single-sided multi-layer welding seam is larger and asymmetric along the thickness of the plate. After welding, the transverse shrinkage and angular deformation of the welded parts are larger. At the same time, the accuracy of the simulation results is verified by actual welding and testing.

1. Introduction
As an important method for solving complex engineering problems, finite element method is widely used [1]. With the development of computer technology, more and more scientists and technicians are engaged in the research of process simulation in order to optimize design and process parameters, reduce test cost, improve production efficiency and product quality. In the field of welding, many progresses have been made to optimize the process by numerical simulation [2]. But in the pretreatment process of the model, each weld must correspond to a working condition. At the same time, the welding path, filling and boundary conditions should be set. When welding the whole structure, the efficiency is very low. In engineering applications, various kinds of special finite element software have their own characteristics in geometric modeling, meshing, analysis and calculation. Although in many cases only one software is needed to complete the analysis process of the whole model, the pretreatment of the model is complex, inefficient and error-prone [3]. In this paper, we use TCL/TK software resources to establish the software integration environment, and simulate the welding process and method of Q345. We simulated the temperature field, stress field and welding deformation of multi-layer welding and compared them. After extracting the distribution of welding residual stress and welding deformation of butt joint, we verified the main reason of transverse shrinkage of butt plate.

2. Welding simulation analysis process
The procedure of welding simulation analysis using thermo-elastic-plastic finite element method is shown in Figure 1. Firstly, the geometric model is transformed into mesh model, then the welding heat source parameters and material parameters are input, and the thermal boundary conditions are applied to calculate the welding temperature field. Finally, we take the temperature field and structural
constraints as the boundary conditions to analyze the welding deformation and residual stress, and get the results of welding simulation analysis in the analysis of welding structure [4].

![Diagram of welding simulation process](image1)

**Figure 1.** Analysis process of welding simulation

### 3. Welding Simulation Example

#### 3.1. Physical Model Establishment.

Firstly, we prepare two Q345 steel plates with a dimension of 200 mm x 200 mm x 10 mm and a V-groove. The geometric model of the steel plate is shown in Figure 2. Next, we use MAG method to weld, and the welding parameters are shown in Table 1.

![Geometrical model of butt plate](image2)

**Figure 2.** Geometrical model of butt plate
### Table 1. Welding process parameters

| Number of welding layers | Welding current I/A | Welding voltage U/V | Welding speed V/(mm·s⁻¹) |
|--------------------------|---------------------|---------------------|--------------------------|
| First floor              | 240                 | 28                  | 5                        |
| Second floor             | 295                 | 32                  | 7                        |
| Third floor              | 295                 | 32                  | 7                        |

3.2. Establishment of Finite Element Model.

3.2.1. Finite Element Parameter Setting. When establishing the simulation model file of welding simulation, we divide the solid element mesh into three layers to ensure the accuracy of calculation. At the same time, in order to improve the calculation speed, we control the element mesh of the weld and heat-affected zone to 2 mm, while the element mesh far from the weld zone to 6 mm [6]. According to the characteristics of MAG welding and the restraint mode on site, we choose double ellipsoid heat source as the boundary condition of welding heat source, set the heat transfer coefficient to 0.02N/mm²/sec/K, define the mechanical boundary condition with displacement restraint, and set the heat source type, heat dissipation coefficient and mechanical boundary condition.

3.2.2. Experimental data acquisition. According to the welding process parameters in Table 1, we input the welding speed, welding current, welding voltage and other data in the welding simulation model, and then input the dynamic thermal physical performance parameters of Q345. After setting the above parameters, we select the layer number components, define the welding line and reference line by node method, calculate the cross-sectional area of the weld bead automatically, determine the parameters of double ellipsoid heat source, calculate the length of the weld bead automatically, and calculate the welding time according to the welding speed, so as to complete the setting of the welding boundary and welding conditions, and output the relationship between thermal-mechanical parameters and temperature. The relationship between thermal-mechanical parameters of Q345 and temperature is shown in Figure 3. At the same time, according to the number of meshes, we automatically set the model into blocks, and set the extraction of welding deformation, welding residual stress and other results [5].

![Figure 3. Relationship between thermo-mechanical parameters and temperature of Q345](image)

3.3. Analysis of calculation results.
3.3.1. Welding Deformation Extraction. After the welding simulation calculation is completed, we extract the overall deformation of the butt joint of the experimental panel, as shown in Figure 4. From Figure 4, we can conclude that the main reason for the transverse shrinkage of butt plate is that the base metal expands first in the welding process. When the weld metal solidifies, the expanded base metal will shrink inevitably. This shrinkage is the main component of the transverse shrinkage of butt joint. At the same time, we also find that longitudinal shrinkage occurs along the weld direction, mainly at the head and tail of the weld. The flat plate shrinks in the vertical direction of the weld, and the edge of base metal far from both sides of the weld warps upward, resulting in angular deformation. The deformation trend is consistent with the actual situation [6].

![Figure 4. Contour of welding deformation](image)

3.3.2. Extraction of Welding Residual Stress. After the calculations, the distribution of welding residual stress of butt joint is extracted, as it can be seen from Figure 5 (a) that with the decrease of the distance from the center of the weld, the transverse residual stress in the vertical direction of the weld increases, reaching the maximum stress in the heat affected zone, the stress on the weld falls back, and the stress in the base metal on both sides of the weld presents a symmetrical distribution; Figure 5 (b) is a nephogram of the longitudinal residual stress along the weld direction. The weld seam and its adjacent area are under tension, and the numerical value generally reaches the yield strength, and the two sides are under compression. The tensile stress in the middle of the weld bead is the largest and decreases gradually toward both ends. After the calculations, the distribution of welding residual stress of butt joint is extracted, as it can be seen from Figure 5 (a) that with the decrease of the distance from the center of the weld, the transverse residual stress in the vertical direction of the weld increases, reaching the maximum stress in the heat affected zone, the stress on the weld falls back, and the stress in the base metal on both sides of the weld presents a symmetrical distribution; Figure 5 (b) is a nephogram of the longitudinal residual stress along the weld direction. The weld seam and its adjacent area are under tension, and the numerical value generally reaches the yield strength, and the two sides are under compression. The tensile stress in the middle of the weld bead is the largest and decreases gradually toward both ends.
4. Conclusion
In this paper, we simulated the MAG welding process of Q345 steel flat butt weld by using TCL/TK language to build geometric model, mesh generation and model setting of welding simulation software. Through the above research, we can draw a conclusion that the multi-layer butt welding seam of flat plate and its adjacent area are under tension, the numerical value generally reaches the yield strength, and the two sides are under compression. The tensile stress in the middle of the weld bead is the largest and decreases gradually toward both ends. The cross-sectional area of single-sided multi-layer welding seam is larger and asymmetric along the thickness of the plate. The transverse shrinkage and angular deformation of the welded parts are larger after welding. The accuracy of the simulation results is verified by actual welding and testing.

5. Reference
[1] Jiang Xuewu, Wu Xinyue, Zhu Shijian. Comprehensive application of UG, Hyper Mesh and MSC Marc software for finite element analysis [J]. Computer Aided Engineering, 2007, 16(2):11-14.
[2] Cai Zhipeng, Zhao Haitian, Wu Ge, et al. Series heat source model and its application in welding numerical simulation [J]. Journal of Mechanical Engineering. 2001, 37 (4): 25-28.
[3] Wei Qiye, Huabin. Dynamic simulation of distillation column under software integration environment [J]. Computer simulation, 2004, 21 (9): 67-70.
[4] Dean Deng, Yijun Zhou, Tao Bi, Xiaozhan Liu. Experimental and numerical investigations of welding distortion induced by CO2 gas arc welding in thin-plate bead-on joints [J]. Materials and Design, 2013 (52): 720-729.
[5] Wang Yudong, Jinlei, Hong Qingquan. HyperMesh & HyperView Application Skills and Advanced Examples [M]. Beijing: Machinery Industry Press, 2012.
[6] Zhang Liping, Fang Yuanbin, Wu Bin, etc. The influence of element type on the accuracy of numerical calculation of welding [J]. Welding Machine, 2016, 46 (11): 88-9