Welding Simulation Based on ABAQUS

Xuesong Ding[1, a], Zhaoliang Liu[1]

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

[a]1806210130@qq.com

Abstract

Welded structures are widely used in ships, vehicles, marine engineering, aerospace and other fields due to their good sealing performance and firm structure. Welding is a process of heating the welding part to high temperature and then cooling quickly. The welding parts inevitably produce welding residual stress, which affects the bearing capacity, strength and precision of the welding structure. The study of welding residual stress by numerical simulation not only reduces the blindness of the experiment, but also reduces the research cost effectively. It is of great practical significance to predict welding residual stress and optimize welding process for controlling welding residual stress. Based on the theory of thermal elasto-plastic finite element analysis, based on the ABAQUS software platform, the influence of the energy of different welding lines on the residual stress of the industrial R60702 zirconium plate butt welding head is simulated and analyzed. The results are as follows: For the industrial R60702 zirconium plate, the maximum temperature of the heating heat source will increase when the welding connection energy increases, and the melting point of the material can be reached faster at the beginning of the welding. The line energy increases the size of the equivalent force and the distribution, but the residual stress increases obviously and the transverse stress increases from 72MPa to 92MPa. The direction of the stress is increased from 223MPa to 264Mpa.

Keywords

Zirconium; ABAQUS software; Numerical simulation; Temperature field; Residual stress field.

1. Introduction

Welding is a kind of permanent connection by heating or pressure or both, and by using or not using the welding material, the welded workpiece achieves the combination of atoms on the material. Welded structures have been widely used in aerospace, construction, bridges, ships, automobiles, electronics and other technical fields due to their advantages of good sealing, light weight, free from geometric shape restriction, material saving, simple process, short production cycle and high yield. At present, in industrial production, the output of welded structural steel has reached more than 50% of steel products, which is close to 70% in developed countries. Welding technology has become one of the most important processing methods in industrial production. [1].

2. Theory of Welding Stress and Strain

The highly concentrated heat input in the welding process produces non-uniform temperature field, which results in non-uniform stress-strain field. Therefore, the welding stress-strain process is highly nonlinear. The main methods used by domestic and foreign scholars in analyzing welding stress-strain field are thermal-elastic-plastic finite element method, inherent strain and deformation theory, viscoelastic-plastic analysis method, etc. Thermo-elastic-plastic finite element method can track the generation and dynamic development trend of stress and strain in welding process in detail, so it is more and more used in actual production and research work. Based on the thermal-elastic-plastic finite element method, the stress-strain field of welding is simulated and calculated according to the incremental theory.

2.1 Yield Criterion

Yield criterion [2] is mainly used to determine the stress state of the material when it enters the plastic deformation stage. By calculating an equivalent stress value and comparing the yield strength of the material to determine when the material begins to yield. Yield criterion is often used to analyze the plastic deformation of metals, and the yield criterion is derived from the point of view of energy, so it is also called deformation energy condition. In this paper, the Von Mises yield criterion is adopted. Its physical meaning is that the deformation energy per unit volume needs to accumulate to a certain limit when the material changes from elastic state to plastic state. The limit value is determined by the nature of the material itself and has nothing to do with the stress state of the material. Therefore, this limit stress value can be determined by uniaxial tensile test. When the deformation reaches this critical value, the material yields.

2.2 Strengthening Criteria

Reinforcement criterion is mainly used to describe the change trend of yield surface in the process of yield. There are usually two kinds of strengthening models, namely, isotropic strengthening and follow-up strengthening. Isotropic hardening means that the center of the yield surface remains unchanged during the development of plastic strain, and the yield face expands uniformly around, which is also called isotropic hardening for isotropic materials. Following-up strengthening means that the size of the yield surface does not change, but moves unilaterally with the development of strain. Therefore, the increase of stress value in one direction will lead to the decrease of stress value in the opposite direction.

3. Numerical Simulation of Zirconia Plate Butt Joint

3.1 Geometric Model

The sample is made of industrial pure zirconium sheet with 5mm thickness. The dimensions of the two boards are 100mm * 100 mm. The groove is unilateral 90 °V shape, as shown in Figure 1. TIG welding is used to weld three layers.

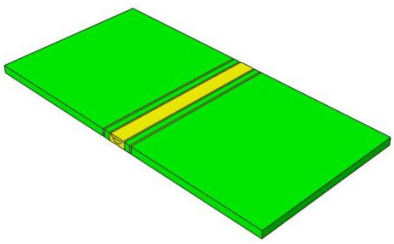

Fig 1. Zirconia plate model

3.2 Model Establishment and Mesh Generation

In the actual welding process, there are intense physical and chemical reactions between welding pool and workpiece, even between welding arc and workpiece, including welding arc, flowing molten metal, the interaction between gas and air on the surface of molten pool. Therefore, in order to ensure the quasi-removal of simulation results in numerical simulation, it is not possible to simplify the model in two dimensions, so it is necessary to establish a three-dimensional model. In this paper, a three-dimensional finite element model is established by means of solid modeling. The size of the model is the same as the actual welding structure. The ZX plane is used as the welding plane and there are three welds. The finite element model after mesh is shown in Figure 2.

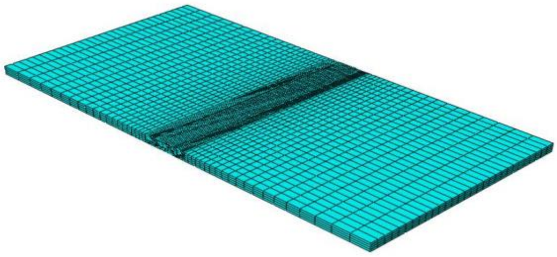

Fig 2. Finite element model

The temperature in the welding process is very high and changes sharply in time and space. Because the temperature in the weld zone changes dramatically, it is easy to be destroyed. Therefore, reasonable mesh generation is the key to the success of numerical calculation. At the same time, mesh generation is closely related to the accuracy of calculation results and the efficiency of computer calculation. In the process of numerical calculation, the rule of mesh generation is that the shape of the element is regular and the mesh density is suitable. If the mesh is too sparse, the calculation will not converge or the result will be wrong. Therefore, the mesh must be divided to the finest possible in the weld and its vicinity. However, in the actual calculation process, the mesh refinement to a certain extent, the calculation accuracy changes little or no longer with the mesh refinement, but the calculation time will be greatly extended, seriously affecting the calculation efficiency. Therefore, for the whole component, generally do not carry out uniform mesh division, but in the more important welded seam division of the fine mesh, far away from the welded seam mesh division is relatively sparse, not only to ensure the accuracy of numerical calculation results and effectively save computing time, but also reduce the configuration of the computer is higher. Please.

3.3 Welding Heat Source Model

Welding heat source is the main source of heat in welding process, and is also the basic condition for welding to be carried out. The characteristics of local concentration, instantaneity and mobility of welding heat source produce extremely uneven temperature field, which forms uneven stress distribution and deformation during and after welding. Welding process is a heat and mass transfer process. With the movement of heat source, temperature changes dramatically in time domain and space domain. Thermophysical properties of materials also change with time. At the same time, there are melting and phase transformation phenomena. Therefore, the selection of welding heat source has a great influence on the calculation accuracy of temperature field and stress-strain field, especially in the weld area near the heat source. In order to accurately study the welding thermal cycle process, scholars have proposed a series of heat source models, including analytical model and finite element model.

4. Calculation and Analysis of Temperature Field

After the calculation is completed, the post-processing module of ABAQUS software is introduced. The post-processing module can view the calculation value of the whole model at a certain load step or at a certain time. It can also dynamically display the distribution of the whole welding temperature field and the change of the node temperature with time by the animation function of ABAQUS. The node temperature values with time are extracted according to needs [2].

In this section, the dynamic evolution of temperature field during welding is selected. Fig 3. shows the temperature field distribution of the first weld at different welding times when the line energy is E1. The dynamic change of temperature field in welding process can be seen from the diagram. The molten pool moves forward with the movement of heat source, which makes the temperature field change continuously.

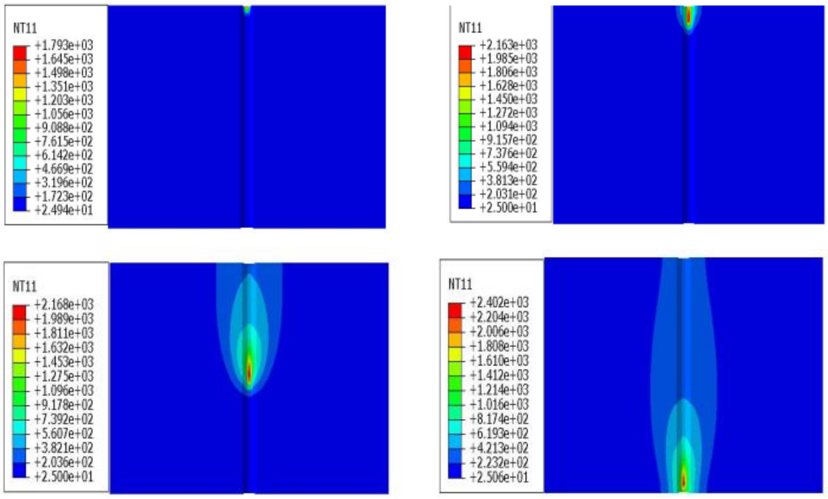

Fig 3. Distribution of temperature field at the first welding seam at different times

5. Conclusion

(1) The results show that the peak value of welding equivalent stress exists in the weld heat affected zone. The maximum value of equivalent stress is close to or equal to the yield strength of the material. When far away from the heat affected zone, the welding equivalent stress decreases rapidly.

(2) When the second weld is heated, heat is transferred to the first weld. Similarly, when the third weld is heated, heat is transferred to the first weld and the second weld. Therefore, the highest temperature of the first weld is higher than that of the second and third welds when the three welds are heated with the same welding line energy.

(3) When the same weld is heated with two different welding line energies, the maximum temperature of the heat source will also increase with the increase of the line energy from E1 to E2.

References

[1] Chang'an Chen, Jinxiang Chen. Analysis of two kinds of ductile fracture morphology of pressure vessel materials [J].Technology and market, 2017, 24 (04): 130~132.

[2] E.L.Wilson, R.E.Nickle. Application of finite element method to the heat conduct analysis. Neuclear Engineering and Design, 4 (1966).

[3] Wang J, Ueda, Murakawa H, et al. Improvement in numerical accuracy and stability of 3-D FEM analysis in welding. Welding Journal, 1996, 7(4): 129~134.