RESEARCH ARTICLE

SIMULATION AND ANALYSIS OF VTYPE WELD JOINT WELDING PROCESS BASED ON 3D MODEL

Lu XiuGuang[1] and Sheng Xuanyu[2]

[1]Shandong Shengli Vocational College, Dongying, Shandong Province, 257097,China
[2]Institute of Nuclear and New Energy Technology, Tsinghua University, Beijing 100084,China

ABSTRACT

ABAQUS software, using 3D models, V-shaped seam welding process was simulated which was commonly used in pressure vessels and tower equipment. Each channel of welding can be calculated and give the temperature distribution and stress distribution after welding. At the same time, the simulation welding process takes into account the cooling after welding. After the computer simulation, the stress distribution and strain energy curve of overall structure also were also given.

INTRODUCTION

With the large-scale development trend of petrochemical units, all kinds of facilities such as towers, reactors, are also becoming heavier and heavier, and bigger and bigger in size. In the past, there are rare500 or 600 tons of tower-type equipment and 700 or 800 tons of reactors, almost no more than thousand tons in mass. The tower apparatus are multi-segment cylinder and welded together. For the spherical tank, stitching weld seam is welded by V-type welds. As for the operation platform and ladder, V type of weld is often used. Due to the higher and higher pressure and temperature in pressure vessels, more and more attention was paid to their security, which makes the weld quality become even more important.

A weld welding seam was not complete in one time, but required multiple welding, including root welding, thermal welding, filler welding, and cover welding. After every welding process is finished, the welded metal was cooled. After cooling down, the second welding process began. For the welded tower or pressure vessel shell, the walls repeatedly withstand heating and cooling, and then the structure must contain residual stress and even larger thermal deformation. So, the analysis on welding process must take into account many factors, which makes manual calculation impossible[1]. Using Weld Molder launched by ABAQUS, this paper simulated the X-weld seam. After the simulation process is completed, we plan to simulate calculation and analysis of welding lug position for tower-type equipment in the actual installation process.

Calculation Model and Parameters

Prerequisites Modeling and Calculation

To complete all operations and calculations in this paper, there are basic operational requirements for software: (a)install CATIA V5-6 R2013, (b) install plug-in that transfer CATIA product to ABAQUS: CATIA WITH ABAQUS TRANSLATOR, (c) install ABAQUS 6. 14-2, (d) install ABAQUS plug-in for simulating weld; (e)install VISUAL STUDIO 2013, (f) install FORTRAN Compiler: INTEL VISUAL FORTRAN XE 2013.

Structure Model

According to the requirement of ABAQUS simulation plug-in, 3 dimensional welding model was built. If 3 dimensional model was established based on the actual size of the parts in the field,

---

Corresponding author: **Lu XiuGuang**
Shandong Shengli Vocational College, Dongying, Shandong Province, 257097,China

we could simulate the real welding process and help evaluating it.

The model was built in software CATIA. The two-dimensional planar model was constructed in [wire frame and surface design] workbench [2]. The detail process was as follows: in [sketch] workbench, the outline zoning was drawn, and the in [wire frame and surface design] workbench, the outline became planar surface after filled with [fill surface] function. Specifically the use of zoning models, establish, and then using a flat fill feature, part section fill a plane. And then using the sketch function, or drawing multiple lines directly in the [wire frame and surface design], also including the outer contour of V-weld. Especially in order to simulate the stress of substrate contact position between base material and substrate, a narrow range was cut out around the weld seam. In the following analysis, grid refinement would be taken in this position. Use multi straight lines and V-shaped curves, split the filled surface into multiple blocks. In dividing and cutting surfaces, we carved the area considering the area to be close to the last welding seam before. In other words, the divided blocks should close to the welding process at every time, such as root welding, thermal welding, fill welding, cover welding. The closer dividing blocks to the real situation, more reliable the results of the simulation [2,3]. The finally built model in CATIA was shown in Fig. 1.
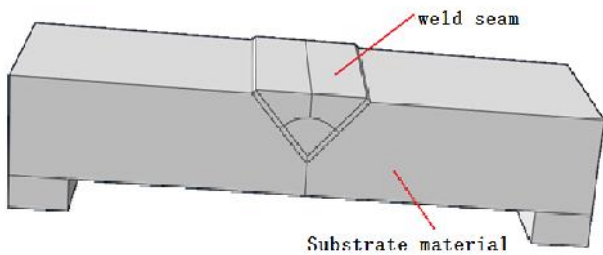

Fig. 1 Models created in CATIA

Material Parameter

In calculation, two kinds of material are defined in the model. One is the model of substrate material; the other is electrode material model. But in the definition of the material, we use the same set of material parameters, namely the substrate material and electrode material performance are the same. The material parameters are shown in table 1[4,5]. Two values are given for yield stress, outside the brackets is the value when plastic strain is 0, that is, material yield stress when plastic strain began to appear; in brackets is the value of yield stress when the plastic strain was 0.25.

Grid division

When the dividing of the model grid, very fine grid should be used for the portion of electrode material, while for parts of the substrate, relatively coarse grid was used. But in this paper, to save the calculation time, we use the same for all elements. The grid size was20mm for all material. Element type is DC3D8, a 8-node linear planar heat transfer element, which has 8 nodes, a Cuboid, in which the first letter D for Diffusion, second letter C represents the Continuum, 3D represents an element in a 3 dimensional, and finally a number 8 represents the number of nodes is 8.The element was a first-order linear element. Grid division within the ABAQUS software was automatically. After setting the grid size, the software automatically generate finite element mesh [6]. The finite element model after dividing grid was shown in Figure 2.
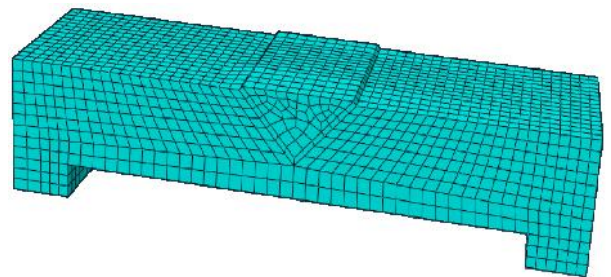

Fig. 2 Finite element mesh

Apart from the defined material parameters and grid division divide the network, the other definition of welded parts was within the Weld Molder plug-in, which can automatically generate the step and boundary conditions. In addition, for the automatically generated structural analysis models, you need to define the boundary conditions, that is, set the point or edge is stationary, and otherwise, the models produce macro-displacement.

In the process, due to the need to call ABAQUS user subroutines, except properly installation of the ABAQUS software, it also needs to install a software compilation tool ITEL VISUAL FORTRAN program. Otherwise, the welding sub-plugin failed to compile user programs, and cannot continue the calculation.

Defined model in the Weld Molder plug-in

In welded model, 3 welding process were defined; all dividing part of welding materials in the geometry of Fig.1 was defined as a welding process.

Table 1 Substrate material parameters

Temperature (℃)	Heat transfer coefficient (×10³W/m²k)	Density (×10³kg/m³)	Modulus of elasticity (MPa)	Poisson's ratio	Coefficient of thermal expansion (×10⁻²m/k)	Yield stress (MPa)	Plastic strain	Specific heat (J/kg·k)
20	55	7.15	93500	0.27	1.17	153 (19278)	0 (0.25)	501
100	51	7.1	89250	0.27	1.17	153 (18746.75)	0 (0.25)	561
200	50	7.05	85000	0.27	1.18	153 (18215.5)	0 (0.25)	590
300	46	7E	78200	0.27	1.22	129.2 (17129.2)	0 (0.25)	619
400	42	6.95	66300	0.27	1.29	119 (9681.5)	0 (0.25)	669
500	39	6.9	51000	0.27	1.32	85 (1997.5)	0 (0.25)	719
600	34	6.85	40800	0.27	1.36	68 (174.25)	0 (0.25)	784
700	30	6.8		0.27	1.38	38 (174.25)	0 (0.25)	855
1500			100	0.27	0	5 (10)	0 (0.25)	

During welding path definition, welding paths should be defined for each weld process based on actual welding process. For simplicity, in the calculations presented in this paper, for the divide blockin electrode material, each defines individual welding path for bottom 3 layers, while for remaining above welding, one welding path is defined. After simplification, the calculation would be reduced, but there was difference with the actual process. Define the initial temperature of the structure was 20℃.

Calculation Result and Analysis

Welding thermal analysis results

When the calculation is complete, for every welding process, we can give temperature distribution and heat transfer on the structure and distribution of temperature in the cooling process. As an example, we give be the first line of welding process, for give detailed calculations for other welding, taking into account the length issue, we no longer give more results.

Fig. 3 shows thermal flow on the base material at the end of the first welding process, the maximum amount value is 1.268×10^7J, at both ends of weld road. Fig. 4 shows temperature distribution of welding materials and substrate after the first welding process. Because we set welding material melting at 1500℃,results show the temperature for welding material part is 1500 ℃.
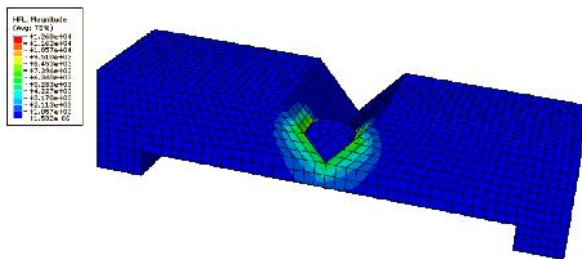

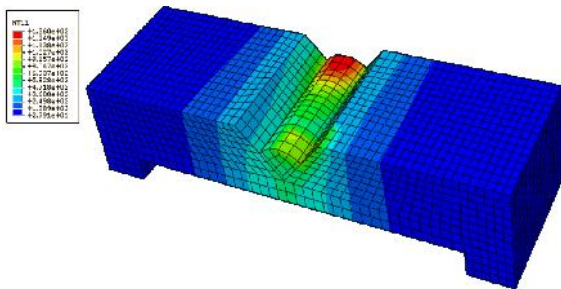**Fig. 3** Heat flow after the first welding process

Fig. 4 Temperature fieldafter the first welding process
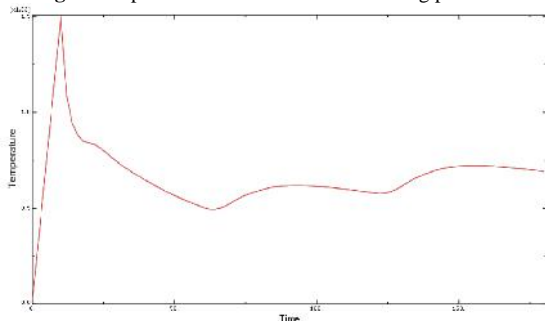

Fig. 5 Temperature curveof node 38

But the displayed result does not affect the calculation of the temperature field, for the actual calculation, only the first welding process affect the welding temperature on the substrate. It can also be seen from the figure, near the first weld process, substrate temperature rises.

Fig. 5 is temperature curve for a node of 38. Node 38 is located in the base V type opening bottom, close to the welding material. As can be seen from the figure, in the 10s, node temperature rise to 1360℃. The following process was cooling the substrate, we did not draw the cooling curves.

Result of Welding Structure Analysis

According to the structure model established within the Weld Molder plug-in, using the results of calculation of welding thermal analysis as an input condition, we calculated the stress distribution in the structure during the welding process. In this way, after welding, residual stress and strain within the structure can be displayed. If you want to analyze the welded structures strength under work load or under other seismic load, the weld result can be evaluated as an input condition. Then we can calculate the welded structures under working loads, and properly evaluate the reliability and integrity of the structure.
Fig. 6 shows the Von Mises stress after the first welding process, and the maximum value is 641.7MPa on a substrate, at the center of the structure near the surface and close to the seam. Fig. 7 shows Structure of plastic strain integral value after first welding process, the node maximum plasticity is 0.03.
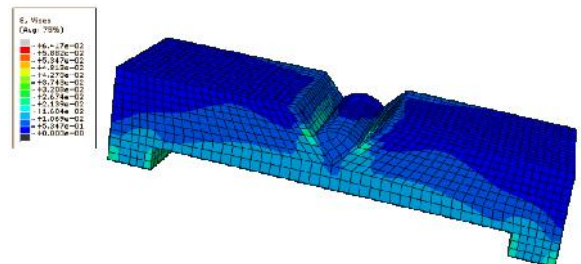

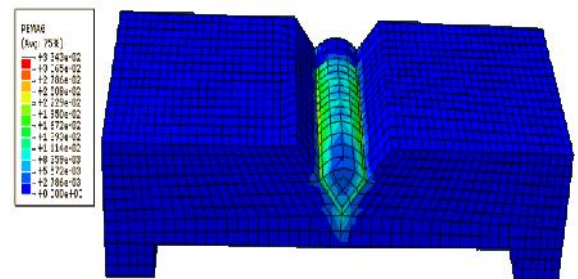**Fig. 6** Von Mises stress after the first welding process

Fig. 7 Structure plastic strain after the first welding process

After finishing all welding processes, Von Mises stress distribution of structure is shown in Fig. 8, the maximum value is 317.1MPa, located below the substrate surface. Fig. 9 shows the change of strain energy as a whole after the welding is completed. During every seam welding process, strain energy increases, during cooling after welding process, strain energy decreases, but the overall response was increased during the welding process.
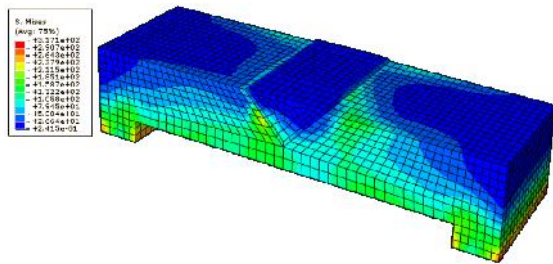
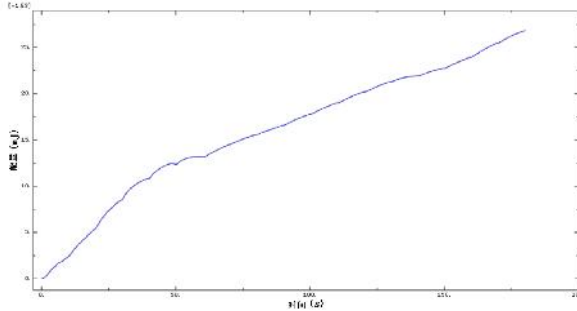**Fig. 8** Von Mises stress distributionwhen the weld is complete

Fig. 9 Strain energy change of the model

By using ABAQUS software plug-in, field and stress distributions were calculated for the V-shaped welded joints welding temperature. Current calculations can only prove that software can finish the simulation of welding process. Whether the simulation results are correct, there are no support experimental results, we now cannot yet conclude.

In the following research, we plan to directly prove the accuracy of calculation results by following method, using overlays strain gages on the structures, measuring the stress distribution of welded structures. At least from the measuring results of welding residual stress and strain, we can indirectly prove correct of ABAQUS software on welding.

Reference

1. Chen Juncheng, Yu Hailiang, Lu Fenggui, Tang Xinhua. Numerical Simulation for Temperature Field in Molten Pool of Deep Penetration Laser Welding of High Strength Steel. Applied Laser, 2008, Vol.28 (3): 177～179.
2. Li Zhenghua, Wu Xiao, Gao Hailong *et.al.* Fracture analysis on weld cracking of box beam based on ANSYS. Hoisting and Conveying Machinery, 2010(7):84~87.
3. Manual Handbook of CATIA.
4. Manual Handbook of Simulia.
5. Sheng Xuanyu edit. Wireframe and Surface Function Detail Explanation of CATIA. Machinery Industry Press: Beijing, 2004.
6. Wang Zhonghui, Wang Yu. Welding numerical simulation method. Electric Welding Machine, 2010, Vol.40(7): 24～27.

How to cite this article:

Lu XiuGuang and Sheng Xuanyu., Simulation And Analysis Of Vtype Weld Joint Welding Process Based On 3d Model. *International Journal of Recent Scientific Research Vol. 6, Issue, 4, pp.3760-3763, April, 2015*

****