Numerical Simulation of Relationship Between Member Dimension and Weld-induced Stress in Plate-fin Type Heat Exchanger*

by Shunta Yamagishi**, Yoshiki Mikami***, Kenji Otani****, Norihiro Hisada****, Shigeru Aoki**** and Masahito Mochizuki***

The core structure of a plate-fin type heat exchanger is composite of brazed plates and fins. Header tanks are welded on the core as inlet and outlet paths. Heat exchangers are essential in the energy system and their importance will continue to grow in the future. In recent years, improvement performance of this heat exchanger is expected; therefore, the number of streams introduced into a core and the operating pressure are expected to increase. This leads to the increase of the number and the thickness of header tanks used. As the result, the effect of welding heat input will become significant. The weld induced deformation and residual stress will affect the structural integrity of the heat exchanger, however, the effect of the welding of header tanks has not been well investigated. The information on the relationship between member dimension and weld induced stress will support the design and fabrication process. In this study, a quasi-three-dimensional simulation model of welding of header tanks on core structure was developed. The effect of member dimensions on weld-induced stress was investigated systematically by the proposed simulation model.

Key Words: Heat Exchanger, Welding, Residual Stress, Distortion, Numerical Simulation

1. Introduction

Heat exchangers are essential components of the energy system and their importance will continue to grow in the future. The plate-fin type heat exchanger is widely used because of its high heat exchange efficiency, multiple fluid heat exchange capability, lightweight structure, and high-reliability as a pressure vessel. The plate-fin type heat exchanger basically consists of a plate-fin core for heat exchange and header tanks as the inlet and outlet of the fluid (Fig. 1). A typical fabrication process for these heat exchangers is as follows: a plate-fin core is vacuum brazed and header tanks are welded onto the core. The thickness of the tube-plates (parting-sheets) and corrugated fins that comprise the core is of the order of millimeters or submillimeters, whereas the thickness of the header tanks is a minimum of ten millimeters. Therefore, the influence of welding will become significant in some case. The combination of these members is determined by design rules established through past fabrication experiences, trial production, and elastic calculations. However, the improvement in heat exchange efficiency is one of the most important issues; therefore, the number of streams introduced into a core and the operating pressure are increasing. This leads to an increase in the number and the thickness of the header tanks welded to the core. There is the possibility of selecting a novel combination of the dimensions for the plate-fin core components and the header tank thickness in the future.

A previous study demonstrated that the welding of the header tank would generate weld-induced stresses in the tube-plate of the core. Therefore, it is important to clarify the relationship between the member dimensions and the weld-induced stress for various combination in order to extend the scope of the design. Numerical simulation is a powerful tool to investigate the relationship between weld-induced deformation and stress, and the member dimensions. A three-dimensional model was used in the previous study; however, it took as long as 40 hours of calculation time for a single pass welding. The welding of a header tank is performed by multi-pass welding; therefore, it is unrealistic to use the three-dimensional model for the calculation of various combination of member dimensions. Therefore, a quasi-three-dimensional model was proposed to perform a

![Fig. 1 Schematics of plate-fin type heat exchanger and extracted part as quasi-three-dimensional model.](image-url)
parametric calculation, and was used for the investigation of the effect of the member dimensions.

2. Numerical simulation model

In order to investigate the influence of the member dimensions on weld-induced stress at a reasonable calculation cost, a quasi-three-dimensional model was proposed.

Figure 1 shows the part modeled in the quasi-three-dimensional model. A cross-sectional part of a heat exchanger with two header tanks was modeled. The model has a tube-plate sandwiched by fins and side bars. Because the plate-fin type heat exchanger has a semi-periodic structure, such a part with a finite thickness was extracted. The plate-fin structure was modeled by a homogenization approach by using an elastic orthotropic stiffness matrix. The quasi-three-dimensional model was proposed. A cross-sectional part of a heat exchanger with two header tanks was modeled. The model has a quasi-three-dimensional model. A cross-sectional part of a heat exchanger with two header tanks was modeled. The model has a quasi-three-dimensional model was proposed. In order to investigate the influence of the member dimensions-parametrically, the boundary conditions were set on the both y-z faces to achieve a plane strain condition by fixing the displacement of both faces in x direction.

Figure 2 shows the finite element model and its dimensions. The core has a size of $20L \times 40L$. The actual sizes of the parts are the proprietary information; therefore, they are denoted in this way. A basic size for components was selected and was varied parametrically. The materials properties of the A5083 aluminum alloy were used for the header tanks and A3003 for the other parts. The material properties and the temperature dependency in the numerical analysis were based on the literature\textsuperscript{1-3}, and show in Fig. 3. Two welding lines, A and B, exist in the model. Each welding line consists of four passes. Thermal and thermal-elastic-plastic analyses have been performed. Welding heat input was simulated by volumetric heat flux. The heat input amount is $598 \text{ J/mm}$, and the heat input time is $2 \text{ s}$.

3. Result of numerical simulation

The residual stress distribution in the cross-section perpendicular to the welding direction is shown in Fig. 4, and the deformation state of the inner surface of the side bar is shown in Fig. 5. The basic tube-plate thickness, header tank thickness are $T$ and $H$, respectively. In addition, the basic dimension of side bar width and distance between header tanks is defined as $W$ and $L$, respectively. The residual stress is compressive at the tube-plate beneath the weld lines, and tensile at the tube-plate located in between the two weld lines. The side bar exhibits an angular distortion by the welding of header tanks, therefore, the side bar located in beneath the weld lines is deforms toward the inside of the core, and the side bar located in between the two weld lines

(a) physical properties for side bar, fin, and tube-plate
(b) mechanical properties for side bar, fin, and tube-plate
(c) physical properties for fin and tube-plate
(d) mechanical properties for header tanks
(e) physical properties for plate-fin structure

Fig. 2 Finite element model for header tanks welded on plate-fin core and mesh division in the welds.

Fig. 3 Material properties of aluminum alloy and plate-fin structure in this numerical analysis.
deforms toward the outside of the core, as shown in Fig. 5. The area of grey in the figure shows the area beneath the weld lines. The angular distortion of the side bar induces the tensile stress in the tube-plate located in between two weld lines. The residual stress distribution and the deformation are similar to those obtained by the previous three-dimensional simulation. Since the quasi-three-dimensional model can evaluate residual stress distribution quantitatively, the influence of member dimension is investigated parametrically in the following sections.

4. Case studies

4.1 Influence of tube-plate thickness

The influence of the tube-plate thickness ($T$) on residual stress was investigated by varying the tube-plate thickness parametrically. The residual stress distribution in the tube-plate along the $y$ direction is shown in Fig. 6. The residual stress of the tube-plate located in between the two welding lines increased with a decrease in tube-plate thickness. In this case, the width of the side bar and the welding condition was constant, therefore the angular distortion due to welding is expected to be almost same. As shown in Fig. 7, the curvature of the side bar displacement in between the weld lines is not dependent on the tube-plate thickness. Consequently, the tube-plate should bear the weld-induced stress resulting from the distortion; therefore, the value of the stress is dependent on the thickness of the tube-plate.
4.2 Influence of header tank thickness

The influence of the header tank thickness \((H)\) on the residual stress was investigated by varying the header tank thickness parametrically. The residual stress distribution in the tube-plate along the \(y\) direction is shown in Fig. 8. The residual stress of the tube-plate located in between the two welding lines increased with the header tank thickness. The increase in header tank thickness increased the influence of the welding. In addition, Fig. 9 shows that as the header tank thickness is increasing, the welding causes the larger deformation of the side bar and the larger local curvature of the side bar in between the two weld lines. The stress in the tube-plate is induced by the angular distortion of the side bar, therefore, it is reasonable that the stress in the tube-plate increases with increasing header tank thickness, i.e., with increasing heat input.

5. Conclusions

In this study, the weld-induced stress in a plate-fin type heat exchanger was investigated using a numerical simulation. A quasi-three-dimensional simulation model of residual stress was proposed considering the cross section along the welding line. The proposed quasi-three-dimensional model shortened the calculation time and enabled parametric calculation by changing the member dimensions. The tube-plate thickness and the header tank thickness were varied parametrically and the variation in the residual stress of tube-plate was calculated. In this paper, the tube-plate thickness and the header tank thickness were selected as parameters for the case study, however, the proposed numerical simulation approach can be applicable for the evaluation of the effect of other parameters.

Reference