Study on Temperature Distribution Characteristics of Condensate in Marine Condensator

Wu Hao¹, Hou Yingzhe¹ and Cui Lin²

¹ Wuhan Secondary Ship Design and Research Institute, Wuhan, Hubei, 430205, China

² Wuhan Polytechnic, Wuhan, Hubei, 430074, China

Abstract. According to the structure and working principles of marine condensator, utilizing FLUNET, a numerical model of the flow field related to the hot well is established and the results were analysed. The flow field and temperature distribution characteristics of hot well with metal mesh and different heat flow distribution schemes are compared and analysed to improve the operation economy and safety of steam turbine units.

1 Introduction

Condensate dissolved oxygen in condenser is an index to evaluate the performance of condenser [1]. The presence of dissolved oxygen in water will lead to serious oxidation corrosion, thus reducing the service life of thermal equipment in power plants to varying degrees.

Different from onshore power stations, which generally adopt base-load operation mode, the condenser, as an important part of the ship's nuclear power plant, is required to be highly maneuverable and has a wide range of load variation during navigation, and its thermodynamic deoxygenation dynamic characteristic affects the safety and reliability of the condenser operation to a large extent [2].

According to the structure and working principle of Marine condenser hot well, the equivalent two-dimensional numerical model of convection heat transfer in the condenser thermal deoxygenation process is established in this paper. The heat transfer process of hot well is simulated numerically, in the case of a metal mesh. Two different heat flow distribution schemes were compared as well. The flow field characteristics and temperature distribution characteristics of the hot well are obtained. Through the analysis and study of the numerical model and the simulation results, the arrangement scheme of the temperature measuring points of the condensate water was put forward.

2 Physical model

2.1 Simplification of model

This paper studies steam bubble of hot well but the influence of heat transfer, for water vapor bubble a numerical simulation of the heat exchange between

involves bubbles generated as well as the water phase change problem, more complex, to simplify the calculation model within the level of heat pipe in heat well established numerical analysis model of convection heat transfer in the numerical analysis model is established for the following assumptions [3]:

- 1) Uniform inlet water temperature and speed.
- 2) Considering that the superheated steam entering from the heat pipe condenses quickly, the inlet condition of the superheated steam bubble is simplified as the wall heat transfer condition.
- 3) The wall temperature of the heat exchange tube is evenly distributed, and the wall temperature of the two groups is equal.
- 4) Without considering the wall heat conduction, that is, the wall is considered to be adiabatic, and the bubble heat transfer is simplified as forced convection heat transfer. Its model is shown in Figure 1.

Figure 1. Hot well cross section.

2.2 Grid Partition

After the calculation area is determined, the calculation area shall be meshed. For the outside of the heat exchange tube, mesh encryption shall be carried out in consideration of the thermal boundary layer effect, and the boundary layer shall be used for meshing, as shown in Figure 2 and Figure 3.

Figure 2. Partial view of boundary layer of heat exchanger

Figure 3. Heat exchanger boundary layer

For the metal mesh area, no direct channel is meshed, but the porous medium model provided by Fluent is used for simulation. Considering the influence of heat transfer in this part, mesh encryption is carried out, as shown in Figure 4 and the grid division of the entire computational domain is shown in Figure 5.

Figure 4. Porous media (metal mesh) mesh encryption

Figure 5. Computational domain grid

2.3 Boundary conditions and initial conditions

The setting of boundary conditions is the key to numerical simulation, as shown in the figure below.

Figure 6. Boundary Condition Setting.

The initial conditions of this model assume that the initial liquid velocity pressure and temperature and other parameters are consistent throughout the calculation area, and the heat flow of the two heat exchangers is distributed according to the area ratio.

3 Numerical Simulation Results and Analysis

3.1 The calculation results of structure with metal mesh

The flow problem in the porous media-pure fluid coupled flow region [4] refers to the existence of two flow regions in the solution region: one is the pure fluid flow region, and the other is the fluid flow region in the porous media. The two parts have a interface, as shown in Figure 7.

Figure 7. Porous media - pure fluid coupled flow region model.

The porous media model method considers the action of solid structure in the flow area as the distributed resistance attached to the fluid, so that the flow can be simulated with a larger grid without losing the flow characteristics, and the distributed resistance can be measured experimentally.

Considering that we are simulating a perforated plate (metal mesh), we can eliminate the permeation term and only use the internal loss term, thus obtaining the following simplified equation of porous media, [5]

$$\frac{\partial p}{\partial x_i} = \sum_{j=1}^3 C_{2ij} \frac{1}{2} \rho v_j |v_j| \quad (1)$$

Assume that 80% of the perforated plates (mesh) are open to flow.

The pressure drop through the plate is given as.

On the basis of the real in-plate fluid velocity, that is, passing through the 80% open area, the loss coefficient is defined by the loss coefficient below K_L is $1-\beta$,

$$\Delta p = K_L \left(\frac{1}{2} \rho v_{80\%open}^2 \right) \quad (2)$$

In the FLUENT model: the velocity of the perforated plate is assumed to be 100% for the whole plate. The open loss coefficient must be converted into the loss of dynamic head per unit length in the porous area. For the first step, the loss factor K_L is calculated and adjusted. It should be based on the speed of 100% open area,

$$\Delta p = K_L \left(\frac{1}{2} \rho v_{100\%open}^2 \right) \quad (3)$$

Or notice that for the same velocity,
 $v_{25\%open} = 4 \times v_{100\%open}$

$$K_L' = K_L \times \frac{v_{25\%open}^2}{v_{100\%open}^2} = 0.5 \times \left(\frac{4}{1} \right)^2 = 8 \quad (4)$$

The adjusted loss factor is 8 for the second one, you have to convert it to the loss factor per unit thickness of the perforated disc assuming a thickness of 1.0mm and the internal loss factor is (SI).

$$C_2 = \frac{K_L'}{thickness} = \frac{8}{10^{-3}} = 8000 m^{-1} \quad (5)$$

Theoretical analysis and experiments show that overheated steam from steam tubes rapidly rises, and because of the blocking of metal mesh, they condense on the upper side of the two metal mesh. So there is a strong heat change between the superheated water vapor and the metal mesh, and the metal network is a good conductor of heat, based on the above analysis and consideration, we can assume that:

1) There is a distribution resistance in the metal network area.

2) Metal net and water vapor heat is strong, the steam parts of the steam are directly added to the metal network, which gives the metal network area an energy source, and the heat and metal net change of the wall is constant.

In order to analyse the effect of heat convection field heat transfer properties, the two kinds of heat flow distribution field were simulated and analysed. The left metal net and the left heat hole were used as a group, and the right metal net and the right heat transfer hole were the second group, and the two groups were divided into two sets of heat change heat.

3.1.1 The first distribution scheme, the metal net and the exchange heat hole for heat transfer by 1:1.

(1) Flow properties

After increasing the porous medium area, the flow field features have some changes, as shown in Figure 8. As you can see from the diagram, the vortex disappears after the barrier 1. At least in two metal mesh, there is no large swirl area in the flow field, which is consistent with the actual situation.

As you can see from the streamline diagram, the flow line is bent by the blocking effect of the metal net, which is reflected in the velocity vector diagram (Figure 9). However, the swirl area of the “blind angle” of the second group of the second group of heat exchanger areas still exists, and there is no significant change in the flow field after the second group.

Figure 8. Motion pattern.

Figure 9. Velocity vector diagram of flow field

(2) Temperature field

Figure 10 and Figure 11 are the temperature distribution characteristics of the rear flow field in the metal network. The increase of the first metal net makes the fluid increase significantly from the first metal plate. At the same time, the temperature effect of the metal net on its upstream is less affected by the flow direction. The heated fluid is heated in the right heat transfer hole. Because of the simplified form of the wall, the heat flow is dragging a long tail. After that, some of the fluid is heated directly through the second metal network, and the other is heated by the left heat hole and then after the second metal network is heated. Because of the low amount of fluid in the left, it makes it less hot, but the "tail" of the heat flow is longer than the right to change the heat hole.

Figure 10. The temperature chart

Figure 11. Temperature nephogram

For the second metal network, the effect is to heat the heated fluid once again, so that the isothermal line is slightly folded after the metal network.

There is also something to be pointed out, similar to the result of section 3.1, which passes two heated hot water from the exit. In other words, the temperature of the exit is higher than the water temperature of the entire flow field area, which is beneficial to the removal of oxygen.

From the temperature distribution, it is also visible that the simplified method of the heat transfer is conservative relative to the actual model of the bubble heat transfer. Because the heat flow of the bubble heat exchanger is distributed in the effect of the bubble, the heat change is more intense and full, and the temperature distribution of the entire flow area is relatively more uniform.

3.1.2 The second distribution scheme, metal net and heat exchange heat transfer heat press 7:3 distribution, highlighting the heat and heat change characteristics of the metal net.

(1) Flow properties

Figure 12 is the flow field flow diagram. There is no obvious difference between this diagram and figure 8. In other words, it is not significant to change the distribution of the convection field.

Figure 13 is the velocity vector distribution of the flow field. Compared with figure 9, the velocity vector distribution of the flow field is different, and there is no obvious difference between the change of heat transfer and the change of the convection field structure.

Figure 12. motion pattern

Figure 13. Velocity vector diagram

(2) Temperature field

Figure 14 and Figure 15 is the distribution of the flow field. Because of this scheme, the ratio of heat flow of metal net distribution increases, making the "tail" of the hot fluid after heat change shorten. Most of the fluid heat transfer is completed in the metal network area, which makes the export temperature more uniform.

Figure 14. The temperature chart

Figure 15. Temperature nephogram

4 Thermal well exit condensation water temperature measuring point arrangement

The temperature simulation results of the metal net are shown in Figure 11 and Figure 15. In either case, the heat well is near the surface of the surface of the surface of the exit, as shown in Figure 16, the temperature of the condensed water is more uniform, so the temperature measuring point is suitable for the position in this position.

Figure 16. Layout diagram of condensate temperature measuring point of hot well.

5 Conclusion

In this paper, the equivalent two-dimensional numerical model of heat transfer heat transfer in the heat of the condenser is established, and the two-dimensional numerical simulation of heat transfer heat in the heat well is simulated, and the different heat flow allocation schemes are compared, and the flow properties and temperature distribution characteristics of the heat well are obtained.

Finally, this paper puts forward the arrangement of the temperature measuring point of the hot well water temperature point, and applies it to the actual engineering problem of the cooling control of the condenser.

References

1. Zhang, J.F., Wang, H., Chen, J. Analysis and Measures of Lower Dissolved Oxygen in Condenser Hot Well. Mechanical Engineer, 2011.Vol 03 (149)
2. Xie, J.Y, Huang, W.J. Mechanism of thermal deaeration and adjustment tactics of deaerator level control system. Guangdong Electric Power, 2006.Vol 12 (19)
3. Zhang, X.J., Wang, X.L. Qiu, J.R. Numerical Simulation of Flow Field in Shellside of Heat Exchanger in Nuclear Power Plant.
4. Yang, K. Research on Simulation Modeling and Vacuum Control of the Marine Condenser [D]. Harbin Engineering University, 2018
5. Chen, K.H, Song, C.Y, etc. Numerical Simulation and Analysis of Porous Medium Model for Wind-Proof and Suppressing Wall. Sintering and Pelletizing, 2008, Vol 33(3)