Physical Model and Numerical Simulation Approach of Steam Flow and Heat Transfer of Pannier-arrangement Condensers

Pingli Hou*†, Maozheng Yu**

ABSTRACT

Through analysing the influence of steam flow direction on the liquid formation and motion behavior in the condenser shell side, the physical model for existing numerical simulation program of condenser is improved by introducing the correlations for flow resistance and condensation heat exchange coefficient in which the influences of steam flow direction are considered according to the available experimental data. Thus a more suitable and general condenser simulation approach is presented and a new condenser calculation program is developed. With the experimental data of a pannier- arrangement experimental condenser, the adaptability of the new condenser simulation approach is verified. General characteristics of this type of condenser are also revealed.

Key Words : steam turbine; condenser; flow; heat transfer; numerical simulation

1. Introduction

Combined cycle power generation is an important development direction for coal power generation in the 21st century. Steam extraction and heat regeneration systems aren't adopted on the steam turbines in combined cycle power plants, and steam turbines are usually designed as side exhaust steam or axial exhaust steam arrangement in order to reduce the height of workshops and be arranged conveniently. Accordingly, condensers are arranged on the ground position as high as steam turbines. In this type of pannier-arrangement condenser, steam flows across the tube bundle at the foreside and the steam flow direction turns rapidly at the tail. The physical model for steam flow and heat transfer, the method for determining heat transfer coefficient and the requirement of tube bundle arrangement are very different from those of common transverse underlung condensers. Therefore, it is necessary to develop the research of steam flow and heat transfer behavior of pannier-arrangement condensers as well as the product design. Numerical analysis method can reveal detailed steam flow and heat transfer processes inside this type of condensers, as well as predict the influence of all kinds of factors on the economy of turbine unit. The current condenser numerical simulation approaches are not suitable for pannier-arrangement condensers, since they all don't allow for the influences of steam flow direction on steam flow resistance and heat transfer coefficient.

In this paper, considering the steam flow features of pannier-arrangement condensers, the correlations for flow resistance and condensation heat exchange coefficient which

* 한국과학기술원 기계기술연구소

** Institute of Turbomachinary, Xi'an Jiaotong University, Xi'an, China

† 연락 저자, 042-869-3070, xjtuhyx@163.com

allow for the influence of flow direction according to the available experimental data[5,6] are constructed. On this base, a more general condenser simulation program is developed, and used to calculate and discuss the performance of a pannier-arrangement test condenser. The adaptability of the developed approach is verified by comparing the simulation results with the experimental data.

2. Physical Model and Numerical Approach

The steam flow space of a condenser is axially divided into many steam rooms by clapboards. In these steam rooms, the cooling water temperature variation is not big and the axial steam motion is limited by clapboards. Then, the flow and condensation heat exchange process of the steam-air mixture in tube bundles can be simplified as two dimensional flow and heat transfer process which changes only on the plane vertical to the tube axis[1].

According to the two dimensional model, the steam-air mixture flow in tube bundle areas of every steam room can be simplified as two dimensional flow of perfect steam-air mixture in a porous medium with distributed resistances and distributed mass sinks.

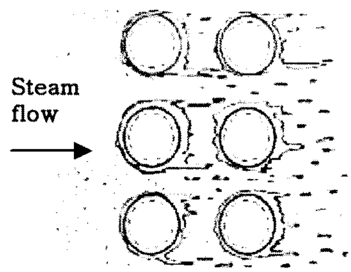
Governing equations group which describes this kind of flow is mainly composed of mass-conservation equation, momentum equations and air component equation. These equations in Cartesian coordinates can be shown as an unified form that

$$\frac{\partial}{\partial x}(\varepsilon\rho u\varphi)+\frac{\partial}{\partial y}(\varepsilon\rho v\varphi)$$
$$=\frac{\partial}{\partial x}(\varepsilon\Gamma_\varphi\frac{\partial\varphi}{\partial x})+\frac{\partial}{\partial y}(\varepsilon\Gamma_\varphi\frac{\partial\varphi}{\partial y})+S_\varphi \qquad (1)$$

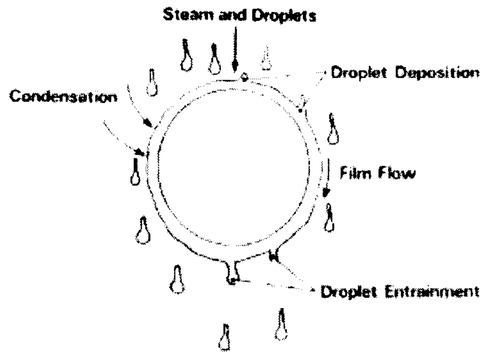
where u, v are components of mixture velocities in the x and y direction; ρ is mixture density; ϵ is the porosity factor of the porous medium; ϕ is an independent variable; Γ_ϕ and S_Φ are respectively the diffusion coefficient and source term, when Φ represents u, v, the air concentrate q and the number 1 respectively, Eq. (1) becomes the momentum equations, the air component equation and the mass-conservation equation respectively; the forms of Γ_ϕ and S_Φ depend on the meaning of Φ, and expressions for them are described in [1].

After dispersing the governing equations group by control-volume integration method, the dispersed equations group is solved by semi-implict method for pressure linked equations. The obtained results include the distributions of the velocity, pressure, temperature and air concentration of the steam flow field on the condenser shell side. In the calculating process, it is necessary to determine the local steam flow resistance coefficient and steam side heat exchange coefficient by existing correlations on the basis of the local steam flow velocities, local air concentration and local tube bank condensate inundation.

At present, correlation for steam flow in the dry tubes bank is generally adopted as the correlation of steam flow resistance coefficient in tube bank area for condenser numerical simulations, but it doesn't allow for the influence of condensate and steam flow direction on the steam flow resistance. Fig. 1 shows the schematic representation of various condensate forms and flow tendencies for various steam flow directions[4]. Davidson and Rowe[2] recognized that single phase resistance correlation can rationally represent the pressure drop of two phase flow for horizontal steam flow, but it would bring considerable error for downward and upward flow.



(a) horizontal flow



(b) downward flow

(c) upward flow

Fig. 1 Schematic representation of various condensate forms and flow tendencies for various steam flow directions

Fig. 2 shows the comparision of single-phase prediction value with experimental value of tube bank resistance for upward flow and downward flow[3]. The data shows that single-phase prediction value is less than experimental result for the upward flow and the former is more than the latter for the downward flow. In this paper, the resistance performance of upward flow and downward flow are revised according to the experimental data in [3]:

for upward flow:

$$\xi_u / \xi_D = 1.732$$

for downward flow

$$\xi_d / \xi_D = 0.85$$

Where ξ is two-phase resistance coefficient, u represents upward flow, d represents downward flow, ξ_D is single-phase resistance coefficient.

In the absence of vapour velocity as condensate flows by gravity on to lower tubes in the bundle, the condensate should thicken around a tube and the condenseate heat transfer coefficient should therefore decrease. Vapour velocity and inundation are important factors that affect the condenstate heat transfer on the shell side. Computer codes for condensers are generally based on the Nusselt equation in which the influences of vapour shear and inundation, namely

$$\frac{\alpha_n}{\alpha_N} = \left(\frac{w}{c}\right)^{-s}\left[1+0.0095\mathrm{Re}_v^{11.8/\sqrt{Nu_N}}\right] \tag{2}$$

where s is correction factor for inundation, detailed meanings of the rest parameters are described in [4].

None of these present day condenser numerical simulation models so far presented accounts of the interactive influence of vapour shear and inundation for the correlation of heat transfer coefficient on the shell side. The literature [3] confirms that the correction factor of inundation for upward flow is higher than that for downward flow. In this paper the influence of vapour flow direction on s is considered according to the experimental data in [3], namely

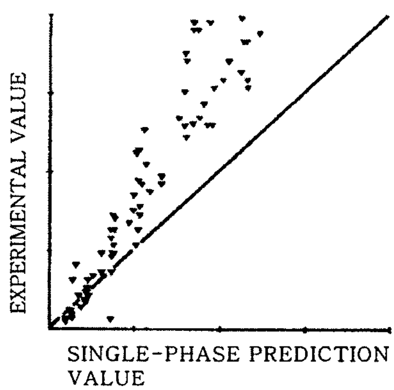
for upward flow:

$$s = \tan^{-1}|v/u|/(\pi/2)\times 0.12+0.223$$

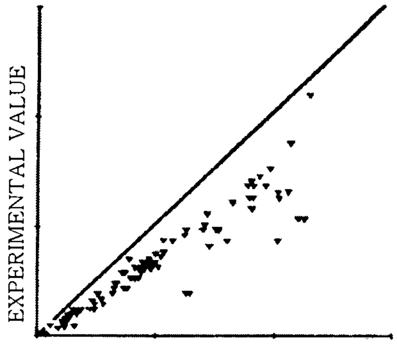
for downward flow:

$$s = 0.223$$

where u, v represent respectively the horizontal component and vertical component of steam flow velocity.



(a) upward flow



(b) downward flow

Fig. 2 Comparision of single-phase prediction value with experimental value of tube bank resistance

3. Configuration and Parameters of the Experimental Condenser

The experimental condenser is a single shell, single pass and pannier-arrangement condenser, whose main design parameters and operation conditions are shown in Table 1.

Table 1. Main Geometric Parameters and Operation Conditions

Condenser pressure (Pa)	27670	Cooling area (m^2)	38.89
Cooling water temperature (K)	290	Mass flux of cooling water (kg/s)	196.04
Water velocity (m/s)	1.19	Mass flux of exhaust steam (kg/s)	2.032
Tube length (m)	1.219	Cooling multiple	96.47
Number of tubes	400	Tube size (mm)	ϕ25.4×1.25

Fig. 3 shows the particular of tube bank. The tube bank is composed of 20×20 tubes in staggered arrangement. From Fig. 3 the main characteristic of this type of condenser is: vapour flows into tube bank from the flank, and air and uncondensed vapour are drawn out at outlet. The computation supposes that the capability of the air pump is great enough to draw out all uncondensed mixture of air and vapour. The parameters of the mixture at outlet are given by computation results. This experimental condenser is not a condenser with reasonable configuration, but it is suitable to examine the rationality and adaptability of a numerical simulation approach.

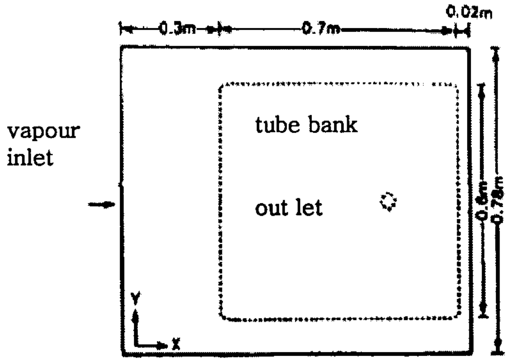


Fig. 3. Experimental Condenser

4. Results and Comparison

Fig. 4 and 5 show respectively the distributions of vapour velocity vectors and vapour streamlines on the shell side of the expetimental condenser. From the figures it can be seen that after steam flows into tube bank area three types of flow are in existence: horizontal flow is dominant in the left part of tube bank area; downward flow is dominant in the top right part; upward flow is dominant in the under right part.

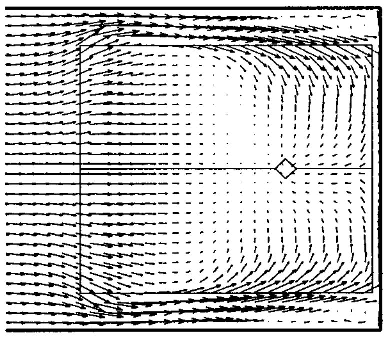


Fig. 4 Diagram of Vapour Velocity Vectors

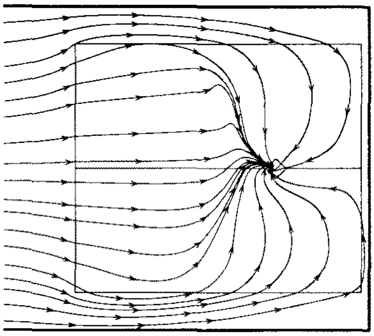
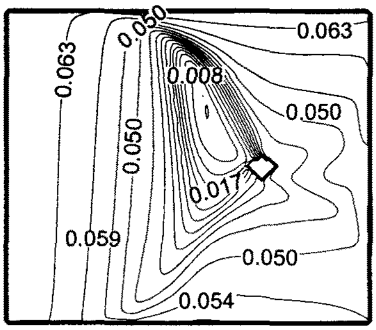
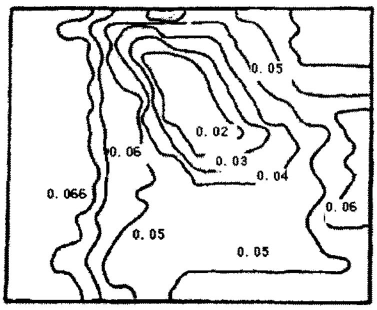


Fig. 5 Diagram of Vapour Streamlines

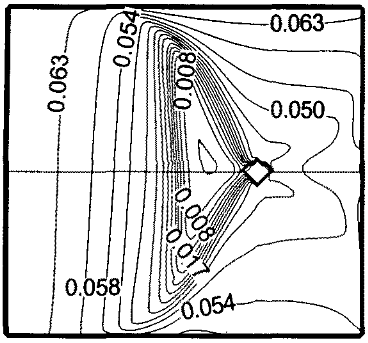
Fig. 6 shows the distributions of condensate per square meter in tube bank area which are respectively obtained by the numerical model considering the influence of steam flow direction on flow resistance(not considering the influence of steam flow direction on inundation), the experiment[5,6] and the single-phase model in which the influence of flow direction on resistance is not considered. From the Fig. it can be seen that in opposition to the general thought that in the lower part of tube bank area the condensate should decrease since it is influenced by inundation, actually the least condensate rate occurs in the top part of tube bank area . The main reason may be explained as: the heat transfer coefficient is lower due to the influence of inundation and therefore the variation of condensate rate in the lower part of tube bank area would be alleviative. At the periphery of the top part tube bank area, the variation gradient is greater due to higher heat transfer coefficient, the steam quantity and vapour volecity decrease rapidly, so lower condensate rate would arise in the top middle part. On the other hand, from comparing Fig. 6a with 6c it can be seen that on the lower part of tube bank area the condensate rate which is obtained by the numerical model considering the influence of steam flow direction on flow resistance is greater than that obtained by the single-phase model. The reason can be explained as: it is difficult for steam to flow into the under right part of tube bank area due to greater steam flow resistance for upward flow in this area, which would increase the horizontal flow from the under left part , and therefore the heat transfer coefficient will be increased. Fig. 7 shows comparision of distributions of horizontal components of vapour volecities which are respectively obtained by the numerical model considering the influence of steam flow direction on flow resistance and the single-phase model in which the influence of flow direction on resistance is not considered. From this figure it can be seen that at the lower middle part of tube bank area horizontal components of vapóur volecities obtained by the former are greater than these obtained by the latter. It can be concluded that the reason that the least condensate rate occurs in the top part of tube bank area is there are great differences between the performances of inundation and flow resistance at the top part of tube bank area and those at the lower part.



(a) Computation Data

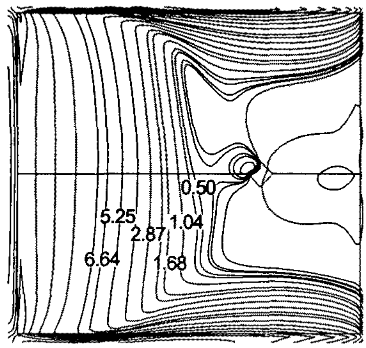


(b) Experimental Data

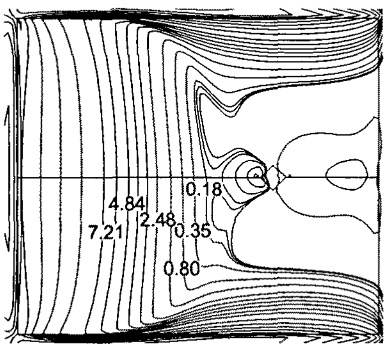


(c)Single-phase Model Result

Fig. 6. Distribution of condensate rate (kg/m^2)

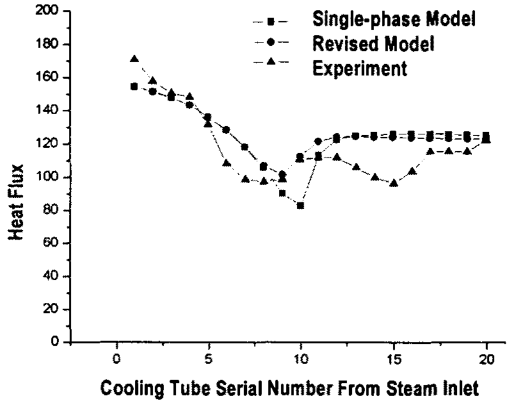


(a) model considering flow direction

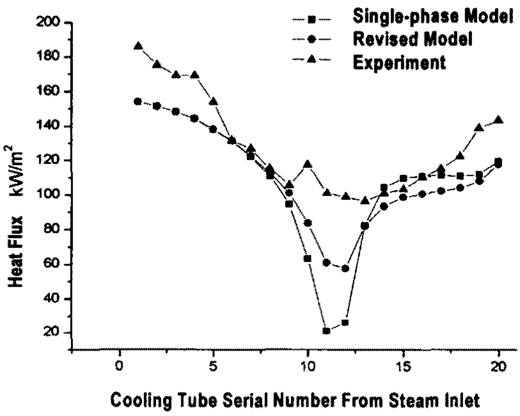


(b)single-phase model

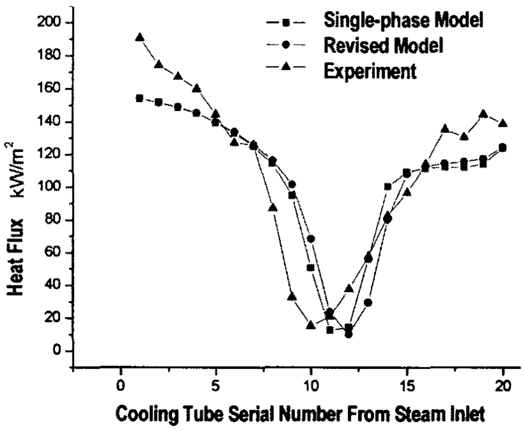
Fig. 7 Comparison of Distributions of Horizontal Components of Vapour Velocities(m/s)



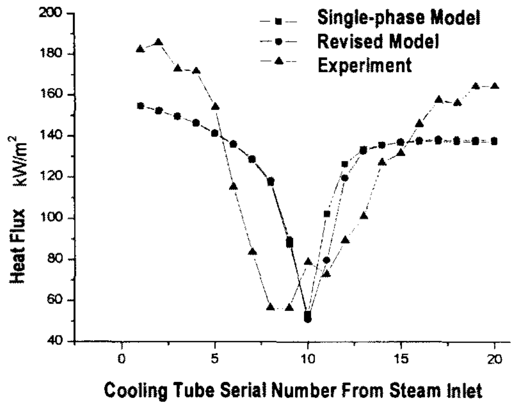
(a) the third row



(b) the eighth row



(c) the thirteen row



(d) the eighteen row

Fig. 8 Heat Flux Distributions on Tube Rows from the Bottom of Tube Bank

Fig. 8 shows comparision of heat flux distributions which are obtained by the revised model considering the influence of steam flow direction on flow resistance , experiment, and the single-phase model on four tube rows. From the Fig. it can be seen that for the thirteen row and the eighteen row the distribulation of heat flux obtained by the revised model accords with that obtained by measured experimental data. For the tubes on the right of the third row, the heat flux obtained by revised model is a little greater than that obtained by measured experimental data. For several tubes in the middle of the eighth row, the heat flux obtained by revised model is less than that obtained by measured experimental data. The results of numerical simulation and experiment show that the area with the lowest heat flux lies in the top middle part of tube bank.

In addition, Fig. 9 shows comparision of heat flux distributions respectively obtained by the revised model and single-phase model. It can be seen that for the top of tube bank the results obtained by two models are all corresponding to that obtained by experimental data. For the middle lower part, the results obtained by single-phase model is less than those obtained by experimental data. The reason is that in this area upward flow dominate and the influence of flow direction on flow resistance and heat transfer is considerable. So the results obtained by revised model are closer to experimental data than those obtained by single-phase model.

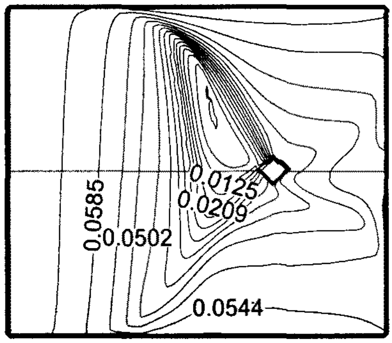


Fig. 9 Distribution of Condensate Rate Resulted from the Model Considering the Influence of Flow Direction (kg/m^2)

Fig. 9 shows distribution of condensate rate resulted from the model considering the influence of flow direction on both flow resistance and inundation coefficient. There is no obvious difference between the distribution of condensate rate obtained by this model and that obtained by the model only considering the influence of flow direction on flow resistance shown in Fig. 6a. So it can be seen that the influence of flow direction on the performance of heat transfer is small.

Noteworthily, the influences of flow direction exist in not only pannier-arrangement condensers but also other types of condensers, so the developed numerical model is suitable for general condensers.

5. Conclusion

The developed numerical simulation approach and program for pannier-arrangement condenser allowing for the influences of steam flow direction can successfully simulate main characteristics of steam flow and heat transfer of condensers.

Flow direction will influence the resistance performance in the lower tube bank area of the pannier-arrangement condenser, furthermore result in a transformation of steam flow and heat load distribution.

Reference

[1] Yu, M.Zh.,Yao X.P., and Wang ,G.Sh., Numerical Analysis of Steam Slow and Heat Transfer Performances of Power Plant Condenser. Power Engineering, 1995, 15(6): 42-48.

[2] Davidson, B.J. and Rowe, M., Simulation of plant condenser performance by computational methods. Condenser Workshop 1980. Naval Postgraduate School, Monterey, CA.

[3] Brickell, G. M., Potential Problem Areas in Simulating Condenser Performance. Power Condenser Heat Transfer Technology, Eds, P.J. Marto & R. H. Nunn, New York, Hemispher Publishing Corp., 1981, pp:51-61.

[4] Moore, M.J. and Sieverding, C.H., Condensers for Large Turbines, Aerothermodynamics of Low Pressure Steam Turbines and Condensers, Washington, Hemisphere, 1987.

[5] Al-Sanea,S., Rhodes,N. and Wilkinson,T.S., Mathematical modelling of two-phase condenser flows. In Proceedings of the International Conference on Multiphase Flow, London, 1985, pp. 19-21.

[6] Bush, A.W., Marshall, G.S. and Wilkinson, T.S., A prediction of steam condensation using a three component solution algorithm. In Proceedings of The Second International Symposium on Condensers and Condensation , University of Bath, U.K., 1990, pp. 223-234.