Development of Numerical Simulation Method for Evaluation of Gas-Liquid Two-Phase Flow Distribution

(Left) Multisystem air-conditioner for buildings
(Right) Centrifugal chiller

MASAYUKI SAKAI[*1] ATSUSHI ENYA[*2]

YASUTAKA AOKI[*2]

Mitsubishi Heavy Industries Group provides heat exchangers used in chemical plants, boilers, air conditioners, etc. In heat exchangers, gas-liquid two-phase flow is distributed to multiple heat transfer tubes by a cone-type or header-type distributor. To predict the flow distribution characteristics, we need to consider not only the behavior of the gas-liquid two-phase flow in the distributor, but also the pressure drop in the heat transfer tubes on the downstream side. Therefore, we developed a flow distribution prediction method by coupling numerical analysis for the flow in a distributor and a one-dimensional model for the pressure drop in heat transfer tubes. The results obtained by the developed prediction method were compared with the experimental results, and it was verified that regardless of the presence or absence of phase change due to heat exchange in heat transfer tubes, the developed method could predict the liquid-phase distribution ratio with practical accuracy.

1. Introduction

In heat exchangers used in chemical plants or air conditioners, gas-liquid two-phase flow is distributed to multiple heat transfer tubes through cone-type or header-type distributors. If the flow distribution becomes extremely uneven, some parts of the heat exchanging area are not used effectively, resulting in the degradation of the heat exchanging performance. In addition, heat transfer tubes with a low flow rate may be excessively heated and damaged. Therefore, it is necessary to accurately evaluate the flow distribution characteristics from the perspective of ensuring the performance and reliability of heat exchangers.

Generally, the flow distribution to multiple heat transfer tubes is affected by the flow state in a distributor and the pressure drop in the downstream heat transfer tubes. Therefore, for prediction of the flow distribution, in addition to the distributor, the downstream heat transfer tubes must also be integrally modeled. The gas-liquid two-phase flow presents various flow regimes depending on the flow rate, quality or heat load, however, and it is still difficult to accurately evaluate the pressure drop in heat transfer tubes by numerical analysis, even with today's advanced numerical analysis technologies for the gas-liquid two-phase flow. Therefore, we developed a new flow distribution prediction method by coupling one-dimensional evaluation equations for which the accuracy has already been verified to determine the pressure drop in heat transfer tubes and unsteady numerical analysis to analyze the behavior of the gas-liquid two-phase flow in a distributor. This paper introduces an overview of the prediction method and describes the verification results for the prediction accuracy through comparison with the experimental results.

2. Prediction method

An overview of the developed prediction method is shown in **Figure 1**. The behavior of gas-liquid two-phase flow in the distributor was simulated by three-dimensional unsteady numerical analysis, while each downstream heat transfer tube was divided into multiple elements in the flow direction, and the heat exchange rate and the pressure drop were calculated by one-dimensional evaluation equations. The details of the numerical analysis method and the one-dimensional model, as well as the coupling method, are explained below.

*1 Heat Transfer Research Department, Research & Innovation Center
*2 Research Manager, Heat Transfer Research Department, Research & Innovation Center

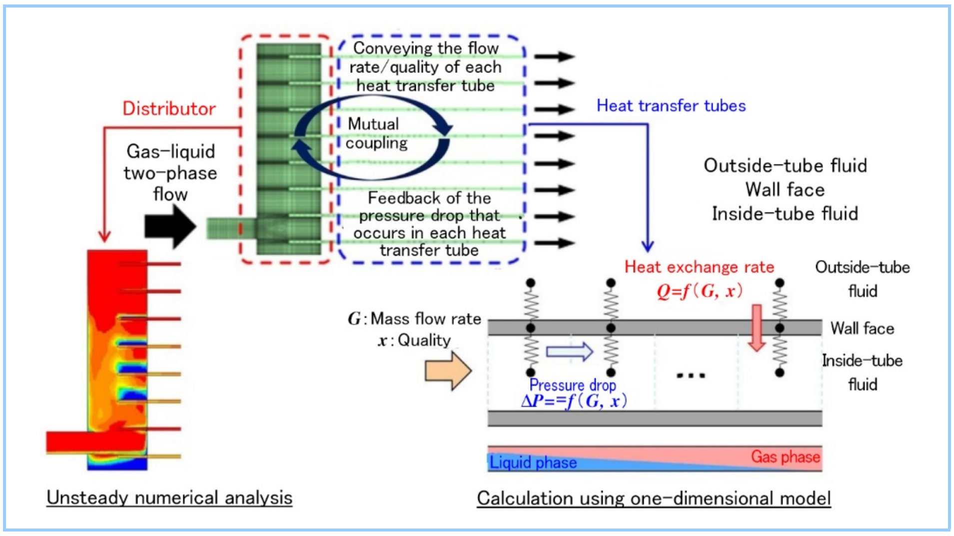

Figure 1 Overview of prediction method

2.1 Numerical analysis method

The two-fluid model, by which the gas phase and the liquid phase are treated by separate equations, was adopted to evaluate the flow regime in the distributor and the gas-liquid separation due to gravity. The generalized thermal-hydraulics analysis software "ANSYS Fluent" was used as the solver. The governing equations for the two-fluid model (mass conservation, momentum conservation and energy conservation) are given as follows:

$$\frac{\partial}{\partial t}(\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \overrightarrow{u_q}) = \dot{m}_{pq} \tag{1}$$

$$\frac{\partial}{\partial t}(\alpha_q \rho_q \overrightarrow{u_q}) + \nabla \cdot (\alpha_q \rho_q \overrightarrow{u_q}\overrightarrow{u_q}) = -\alpha_q \nabla p + \nabla \cdot (\alpha_q \bar{\bar{\tau}}_q) + \alpha_q \rho_q \vec{g} + M_q + \overrightarrow{F_q} \tag{2}$$

$$\frac{\partial}{\partial t}(\alpha_q \rho_q h_q) + \nabla \cdot (\alpha_q \rho_q \overrightarrow{u_q} h_q) = -\alpha_q \frac{dp}{dt} + \bar{\bar{\tau}}_q : \nabla \overrightarrow{u_q} + Q_q + \dot{m}_{pq} h_{pq} \tag{3}$$

where α_q is the volume fraction of the q phase, $\dot{m}_{pq}$ is the phase change rate from the p phase to the q phase, M_q is the momentum generation term of the q phase, $\overrightarrow{F_q}$ is the term of external force that acts on the q phase, and Q_q is the heat input to the q phase. Equations (1) - (3) represent the governing equations of the gas phase when $p = f$ and $q = g$, and the governing equations of the liquid phase when $p = g$ and $q = f$.

The momentum transfer terms caused by velocity difference between the gas phase and liquid phase were calculated by equations (4) and (5), with the gas-liquid interfacial drag M_d and the lift force M_L that acts on bubbles taken into account, respectively. In equations (4) and (5), a_i, C_D and C_L are the gas-liquid interfacial area concentration, drag coefficient and lift coefficient, respectively. The flow regime was judged by physical properties, void fraction, flow rate, etc., and appropriate constitutive equations were applied. For the details, refer to the previously published technical reviews[1],[2].

$$\overrightarrow{M_d} = \frac{1}{8} C_D \cdot a_i \cdot \rho_f (\overrightarrow{u_g} - \overrightarrow{u_f}) \cdot |\overrightarrow{u_g} - \overrightarrow{u_f}| \tag{4}$$

$$\overrightarrow{M_L} = -C_L \rho_f \alpha_g (\overrightarrow{u_f} - \overrightarrow{u_g}) \times (\nabla \times \overrightarrow{u_f}) \tag{5}$$

2.2 One-dimensional model for inside of heat transfer tubes

The heat transfer tubes were divided into multiple elements along the flow direction and the pressure drop in each element was calculated by one-dimensional evaluation equations. Since the pressure drop changes due to not only the flow rate but also the quality, the gas-liquid phase change caused by heat exchange with outside-tube fluid was also considered.

The friction loss in heat transfer tubes was given by the following equations based on the Lockhart-Martinelli correlation[3].

$$\Delta P_{tp} = \Delta P_f \phi_f^2 \tag{6}$$

$$\phi_f^2 = 1 + \frac{C}{\chi} + \frac{1}{\chi^2} \tag{7}$$

where χ is the Lockhart-Martinelli parameter, C is the Chisholm parameter, and ΔP_f is the pressure drop when only the liquid phase flows. For the Chisholm parameter C, different correlations were used according to the size of the heat transfer tubes or channel type, and the Li-Hibiki correlation(4) was used for mini-channels.

2.3 Coupling of numerical analysis and one-dimensional model

The one-dimensional evaluation equations for calculating the heat exchange rate and the pressure drop in heat transfer tubes were incorporated in ANSYS Fluent as user defined functions (UDFs), so that the numerical analysis for the distributor section and the one-dimensional model for the heat transfer tube section were coupled. The information exchange between the one-dimensional model and the solver is shown in **Figure 2**. With reference to the information such as flow rate, void fraction and physical properties that the solver has, the heat exchange rate and the pressure drop were calculated by the one-dimensional model and returned to the solver as the phase change rate $\dot{m}_{pq}$ in equation (1) and the external force term $\vec{F_q}$ in equation (2), respectively.

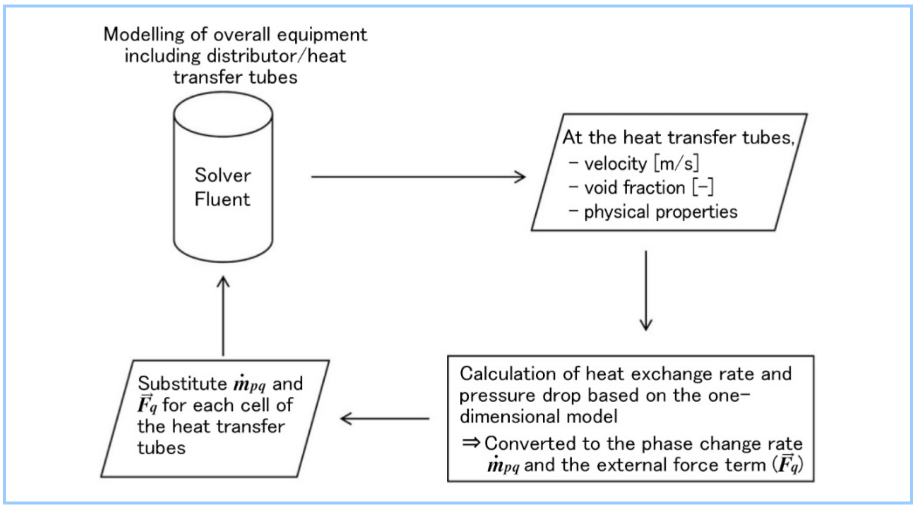

Figure 2 Exchange of information between one-dimensional model and solver

3. Verification calculation

This section describes a calculation example for a header-type distributor of a parallel-flow heat exchanger for air conditioning products such as car air conditioners.

The heat exchanger to be calculated is shown in **Figure 3**. It consists of two headers that are vertically elongated, multiple flat heat transfer tubes and fins, and heat exchange is performed between the fluid in the tubes and air passing between the fins. Both ends of each tube are inserted into the respective headers so that the gas-liquid two-phase flow that enters the lower part of the inflow header is distributed to each tube. In this verification, the calculation was performed under condition (1): non-heating condition using air and water and under condition (2): heating condition involving phase change in heat transfer tubes with the fluorocarbon refrigerant to be used in the actual equipment, and the flow distribution to each heat transfer tube was predicted.

The flow distribution ratio of the liquid phase is shown in **Figure 4**. Concerning condition (2), the calculation result for the case where the developed method was not used (that is, in the case one-dimensional pressure drop evaluation model for heat transfer tubes was not applied) was also described. The tendencies of the flow distributions under both conditions predicted by the developed method reproduced the flow distributions of the experimental results well. When the flow distribution prediction accuracy ε defined by equation (8) was introduced, $\varepsilon = 18\%$ under condition (1) and $\varepsilon = 27\%$ under condition (2) ($\varepsilon = 67\%$ before the developed method is applied). It was verified that by the developed method, the liquid phase flow rate into each heat transfer tube could be predicted within ±30%.

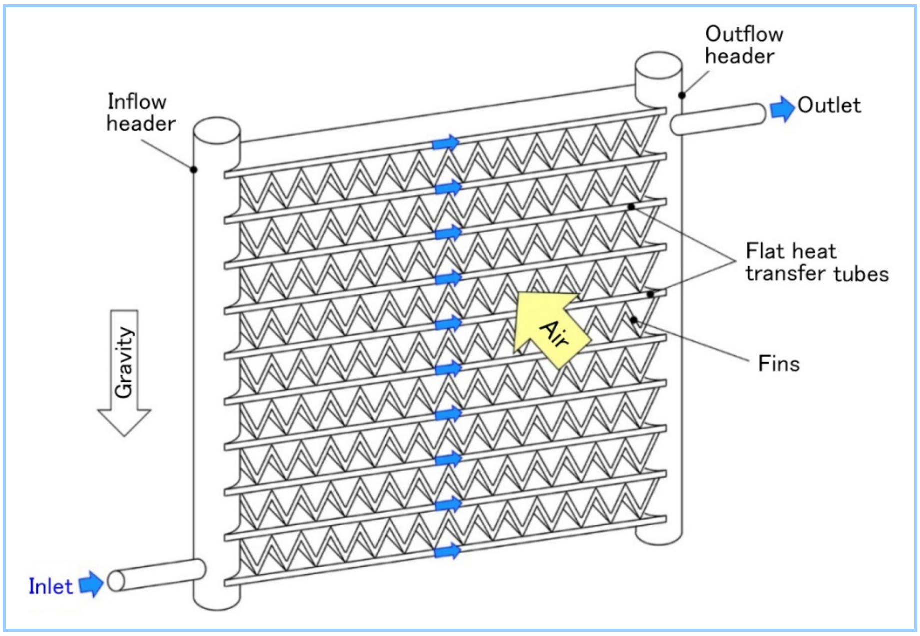

Figure 3　Heat exchanger to be calculated

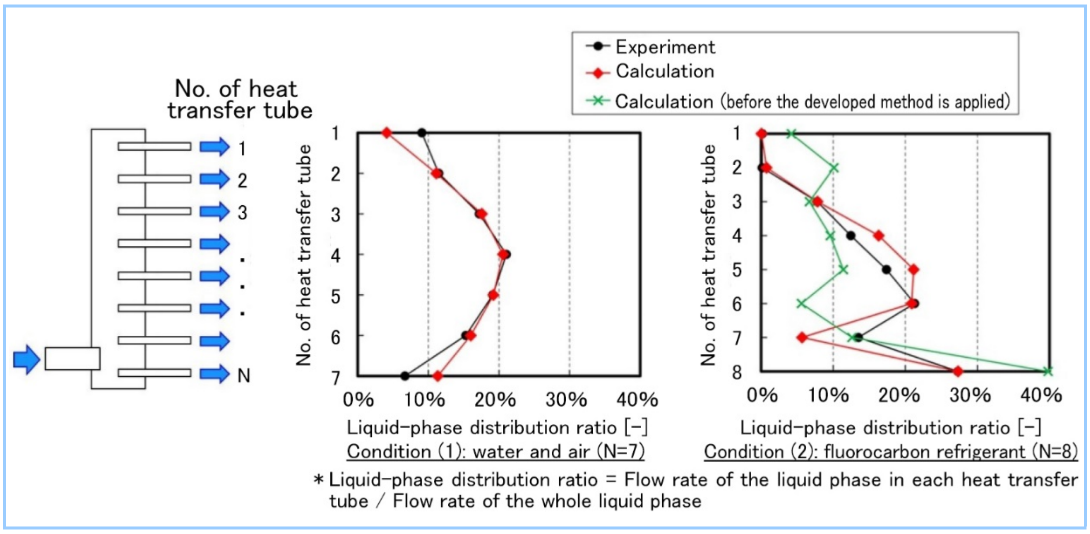

Figure 4　Comparison of liquid-phase distribution ratio

$$\varepsilon = \frac{\sqrt{\frac{1}{N}\sum_i^N\left(G_{exp,i} - G_{cal,i}\right)^2}}{\bar{G}} \tag{8}$$

(N : Number of heat transfer tubes,
$\bar{G}$: Average flow rate of liquid phase in each heat transfer tube,
G_i : Flow rate of liquid phase in heat transfer tube i [exp : Experiment, cal : Calculation])

Figure 5 shows the photo of the visualized inside of the header in condition (2) (experiment) and the contour map of the void fraction (calculation) and **Figure 6** shows the pressure drop between the inlet and the outlet of the heat exchanger in condition (2). Gas-liquid flow state, distribution of void fraction in the header determined by gravity and gas-liquid interfacial drag, and the pressure drop in the heat transfer tubes are in good agreement with those in the experimental results. This shows that the coupling of numerical analysis and the one-dimensional model has achieved the desired effect. The high void fraction region near the inlet tube leads to reduce the liquid phase inflow into the No. 7 tube, while the liquid phase inflow into the No. 4 to 6 tubes increases due to the formation of a low void fraction region in the middle stage of the header. These flow distribution characteristics, which are difficult to explain by gravity effect, could be predicted only by using the developed method.

As described above, we applied the developed prediction method to a header type distributor of a parallel-flow heat exchanger to confirm that the distribution of gas-liquid two-phase flow can be predicted with practical accuracy. Thus, flow distribution characteristics under different flow rate conditions or with different distributor shapes can be quantitatively predicted so that it is possible to design the distributor shape for optimizing the flow distribution.

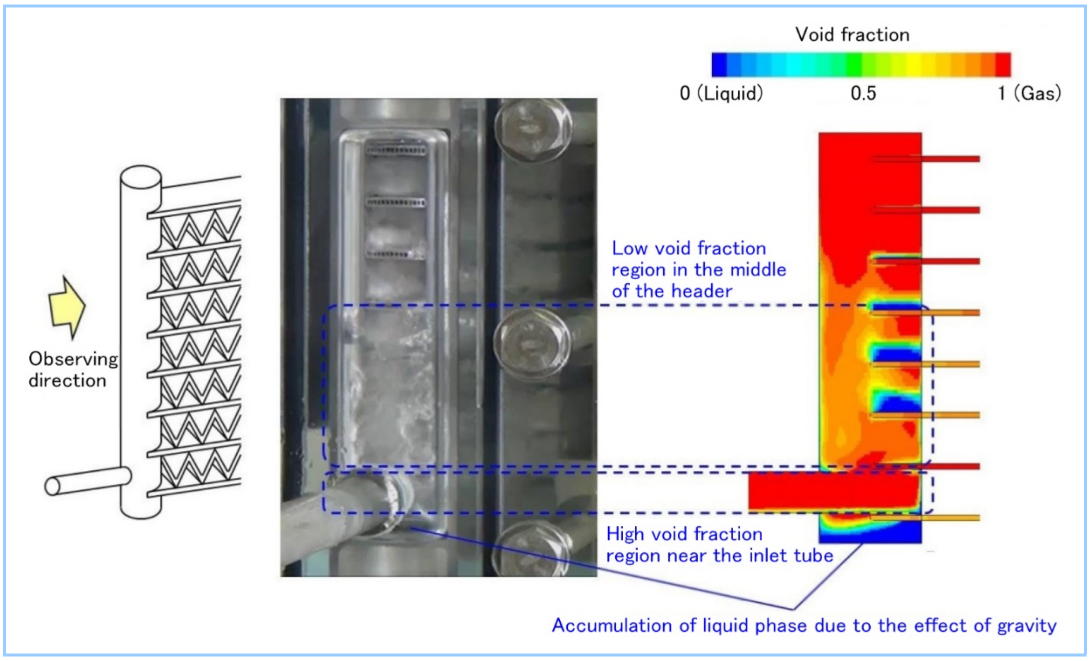

Figure 5 Comparison of flow state in header

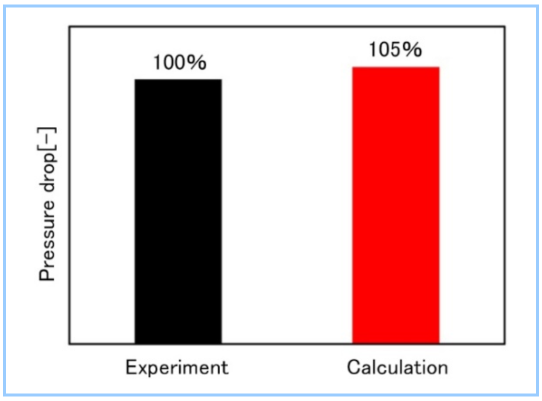

Figure 6 Comparison of pressure drop in heat exchanger

4. Conclusion

We developed a method for predicting the gas-liquid two-phase flow distribution in a heat exchanger by coupling numerical analysis for evaluation of flow behavior in a distributor and one-dimensional model for pressure drop in heat transfer tubes. We performed the verification calculation for a parallel-flow heat exchanger for air conditioners and confirmed that the flow distribution in each heat transfer tube can be accurately predicted. Using this method, we will make efforts to improve the design of heat exchangers and optimize the shapes of distributors toward improvement of the performance and reliability of heat exchangers.

References

(1) Kondo, Y. et al., Development of Two-phase Flow Numerical Simulation Platform for Evaluating Boiling Heat Exchanger Design, Mitsubishi Heavy Industries Technical Review Vol. 55 No. 2 (June 2018), p.1～5

(2) Komuro, Y. et al., Development of Numerical Evaluation Method for Unsteady or Phase-changing Gas-liquid Two-phase Flow, Mitsubishi Heavy Industries Technical Review Vol. 56 No. 1 (March 2019), p.1～5

(3) Lockhart, R. W. et al., Proposed correlation of data for isothermal two-phase, two-component flow in pipes, Trans. ASME, Vol.66No.2 (1945), p.39～48

(4) Li, X. et al., Frictional pressure drop correlation for two-phase flows in mini and micro multi-channels, Applied Thermal Engineering, Vol.116 (2017), p.316～328