Ammonia concentration analysis for the steam condenser by combining two phase flow CFD simulation with condensation and process simulation

K. Karube[1], M. Maekawa[1], S. Lo[2] & K. Mimura[1]
[1]*Chiyoda Advanced Solutions Corporation, Yokohama, Japan*
[2]*CD-adapco, UK*

Abstract

Ammonia corrosion in alumi-brass tubes in steam condensers can be a serious problem. It generally occurs in the high ammonia concentration area. In this case, it is planned to replace the alumi-brass tubes by higher grade material, such as cupronickel. Therefore, it is important to minimize the area to be replaced in order to keep the equipment cost down. It is known that the ammonia concentration is related to the degree of corrosion damage. We propose a hybrid analysis method to obtain ammonia concentration by combining two-phase flow Computational Fluid Dynamics (CFD) analysis and network analysis by a chemical engineering process simulator for the tube bundle. Ammonia concentration obtained by the simulation is therefore used to determine the area to be replaced by the higher grade material tubes.
Keywords: condensation, process simulation, CFD, ammonia concentration distribution, network analysis.

1 Introduction

A steam condenser is installed downstream of the steam turbine to recover the exhaust steam. It is known that ammonia corrosion of the alumi-brass tubes in areas of high ammonia concentration often occurs in steam condensers. Ammonia is used in water treatment agents, such as pH adjusters and boiler compounds. Usually, the condenser has thousands of tubes that consist of both higher grade material (such as cupronickel) and lower grade material (such as

WIT Transactions on Engineering Sciences, Vol 63,
www.witpress.com, ISSN 1743-3533 (on-line)
doi:10.2495/MPF090071

alumi-brass) for reducing equipment cost. The higher grade tube is used in the severe corrosion area. However, ammonia corrosion often occurs in the lower grade tube area when the ammonia concentration is high. It is known that the ammonia concentration is related to the actual corrosion damage. Therefore, it is very important to determine the higher grade tube area quantitatively.

High ammonia concentration is caused by steam condensation. Ammonia concentration in the steam is usually very low. However, high ammonia concentration occurs during condensation governed by ammonia-water vapor-liquid equilibrium. In order to obtain ammonia concentration, coupled analysis is required for two-phase flow dynamics with condensation and ammonia-water vapor-liquid equilibrium. However, direct coupled analysis is very difficult and needs a lot of calculation time. The Computational Fluid Dynamics (CFD) method was used for simulating two-phase flow with condensation [1, 2]. However, it is very difficult to consider phase equilibrium in CFD. A chemical engineering process simulator can simulate rigorous ammonia-water vapor-liquid equilibrium but cannot simulate complicated two-phase flow patterns in the condenser.

In order to solve the above problems, we proposed a hybrid analysis method to simulate ammonia concentration in the steam condenser. In this method, two-phase flow CFD analysis was used to obtain the steam-water flow in the condenser and the flow distribution was passed to the chemical engineering process simulator for network analysis. Some researchers have proposed techniques in combining CFD and process simulation [3, 4]. Although these calculation techniques were applied to the stirred tank etc, only gas and liquid flows were considered. Few simulation examples were proposed for two-phase flow with condensation, such as steam condensers. The proposed method was very useful and was used to determine the area to be replaced by the higher grade material tubes to achieve an optimum condenser design while keeping the cost of manufacturing or modification to a minimum.

2 Numerical method

For analyzing ammonia concentration, we proposed a hybrid analysis method to simulate ammonia concentration in the steam condenser. In this method, two-phase flow CFD analysis was used to obtain the mass flow distribution of steam and water in the tube bundle of the condenser and the computed mass flow distribution was passed to the chemical engineering process simulator for network analysis.

2.1 CFD method

In the CFD analysis, we tried two methods of analysis. One was a single-phase flow analysis and the other was a two-phase flow analysis. In the single-phase flow analysis only the steam flow was calculated and the steam condensation was treated as mass sink in the mass and momentum equations. In the two-phase flow analysis both the steam and water flows were calculated.

www.witpress.com, ISSN 1743-3533 (on-line)

The Navier-Stokes equations were solved for the single-phase flow analysis. The Eulerian multiphase flow model was applied to the two-phase flow analysis [5]. In the Eulerian multiphase flow model, mass and momentum conservation equations were solved for both the gas and liquid phases. Since this analysis assumed isothermal condition, the energy equations were neglected. The gas phase was treated as the continuous phase and the liquid phase as the dispersed phase.

The mass and momentum conservation equations used in present CFD method are given by

$$\nabla \cdot \left(\alpha_k \rho_k \mathbf{u}_k\right) = S_{mass\ k} \tag{1}$$

$$\begin{aligned}\nabla \cdot \left(\alpha_k \rho_k \mathbf{u}_k \mathbf{u}_k\right) = \\ -\alpha_k \nabla p + \alpha_k \rho_k \mathbf{g} + \nabla \cdot \left[\alpha_k \left(\tau_k + \tau_k^t\right)\right] + M_k + S_{pk}\end{aligned} \tag{2}$$

where α, ρ, and $\boldsymbol{u}$ are the phase fraction, the density, and the velocity, respectively. τ and τ^t represent the viscosity stress and the turbulence stress, respectively. In the present study, the standard k - ε model is used as the turbulent model to estimate turbulence stress τ^t. p and $\boldsymbol{g}$ are pressure and gravity acceleration, respectively. The subscript k denotes the phase in the two-phase flow model, where $k = c$ represents the continuous phase, $k = d$ represents the dispersed phase. In the single-phase flow model, subscript .k in the above equation is eliminated. And the phase fractions α satisfies the following equation.

$$\alpha_c + \alpha_d = 1 \tag{3}$$

In the equation (1), S_{mass} represents the interphase mass transfer due to the condensation of steam in the steam condenser, which is considered as a constant (S_{mass} = 1.407 kg m^{-2} s^{-1}) in the present study, simply.

In the equation (2), M, which is eliminated in the single-phase flow model, represents the momentum exchange through the interface or a force per unit volume acting between phases. In the present study, only drag and lift force are considered. Hence,

$$M = M_D + M_L \tag{4}$$

The drag force M_D is estimated using following equation.

$$F_D = \frac{3}{4d} \alpha_d \rho_c C_D \left|\boldsymbol{u}_d - \boldsymbol{u}_c\right| \left(\boldsymbol{u}_d - \boldsymbol{u}_c\right) \tag{5}$$

where d is the diameter of dispersed phase. The drag coefficient C_D is estimated using the modified Schiller and Naumann [6] correlation, shown in following equation.

$$C_D = \begin{cases} \dfrac{24}{\mathrm{Re}_d}\left(1 + 0.15\,\mathrm{Re}_d^{0.687}\right) & ; \quad 0 < \mathrm{Re}_d \le 1000 \\ 0.44 & ; \quad \mathrm{Re}_d > 1000 \end{cases} \tag{6}$$

where the particle Reynolds number is defined as below.

$$\mathrm{Re}_d = \frac{\rho_c |\boldsymbol{u}_d - \boldsymbol{u}_c| d}{\mu_c} \tag{7}$$

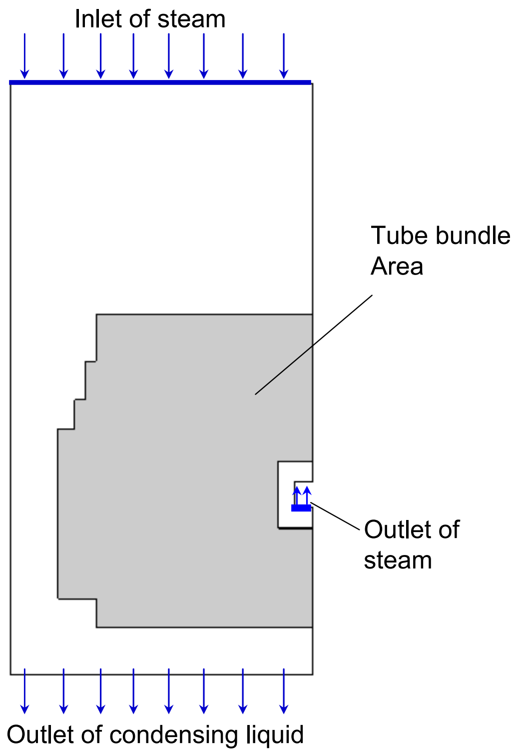


Figure 1: CFD analysis model.

Table 1: Analysis condition summary.

Steam mass flow rate	kg/h	100,000
Density of steam	kg/m^3	0.050
Density of condensing liquid	kg/m^3	1000.0

The lift force M_L is defined as below.

$$M_L = C_L \alpha_d \rho_c (\mathbf{u}_d - \mathbf{u}_c) \times (\nabla \times \mathbf{u}_c) \tag{8}$$

where C_L is the lift coefficient.

In these analyses, the general-purpose CFD software Star-CD from CD-adapco was used, and steady state and isothermal calculation was conducted. Figure 1 shows the CFD analysis model. Around 7,000 regular fine cells were used in the model. A summary of the analysis condition is provided in Table 1. The steam flow in the steam turbine was assumed as the uniform flow distribution for the longitudinal direction of the tubes. Therefore, the two dimensional axi-symmetry model was applied to the sectional plane of the tube bundle. Inlet was located at the top of model and steam flows towards the outlet at the center of the tube bundle. Condensing liquid flows towards the outlet located at the bottom of the model. Steam was condensed as it moves from outside to inside of the tube bundle. The gray part in Figure 1 shows the tube bundle area. Thousands of tubes contained in the tube bundle area of the steam condenser are simply represented by using porous media model. The pressure drop for the tube bundle was calculated by the following empirical correlation, which is counted as the continuous phase momentum source term $S_{p\ c}$ of momentum equation (2) [7].

$$S_{p\,k} = 4\rho_c u_c^2 \,\mathrm{Re}_0^{-0.16}\left[0.25 + \frac{0.1175}{(b_T / d_0 - 1)^{1.08}}\right] N \tag{9}$$

where b_T, d_0, and N are the pitch of tubes, outer-diameter of tube, and tube number per meter, respectively. And Re_0 is the Reynolds number based on tube diameter, which is defined as below.

$$\mathrm{Re}_0 = \frac{\rho_c \boldsymbol{u}_c d_0}{\mu_c} \tag{10}$$

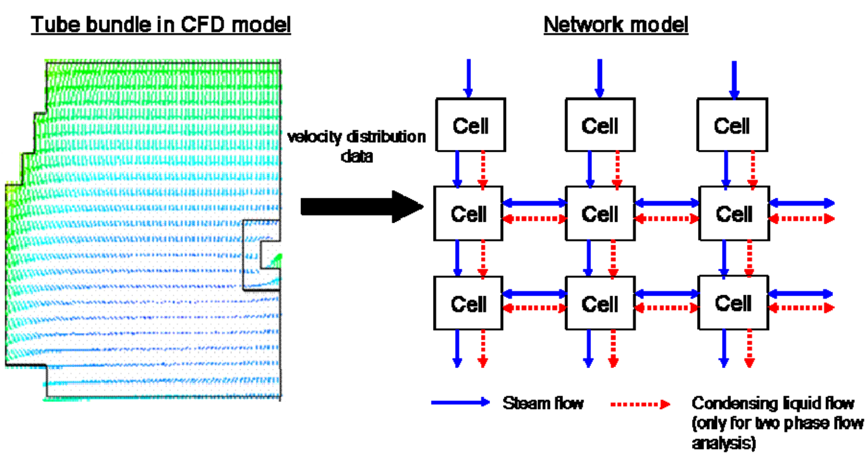


Figure 2: Network model.

2.2 Network analysis method

In the network analysis, the tube bundle area was divided into relatively coarse cells shown as Figure 2. Each cell was modeled by the heat exchanger and flash drum module in the chemical engineering process simulator. Non-uniform cells were used (coarse cells outside the tube bundle and fine cells inside) for simplifying the model.

A commercial process simulator Pro/2 (Invensys SIMSCI) was used for this analysis. In the case of single-phase flow analysis, only the steam mass flow distribution was transferred to the network analysis. Condensed liquid flow was assumed to flow vertically downwards. In the case of two-phase flow analysis, steam and condensing liquid mass flow distributions were passed to the network analysis. The two-phase flow model was considered to have a higher accuracy than the single-phase flow model. Ammonia concentration in the condensation liquid was calculated by the network analysis. The Non Random Two Liquids (NRTL) activity coefficient model was used for the ammonia-water equilibrium relationship.

3 Results

Figure 3(a) and (b) show the steam velocity distribution in the condenser obtained by the single-phase flow analysis and the two-phase flow analysis, respectively. The calculated steam flow pattern was similar in Figure 3(a) and (b), although the highest velocity was a little bit different.

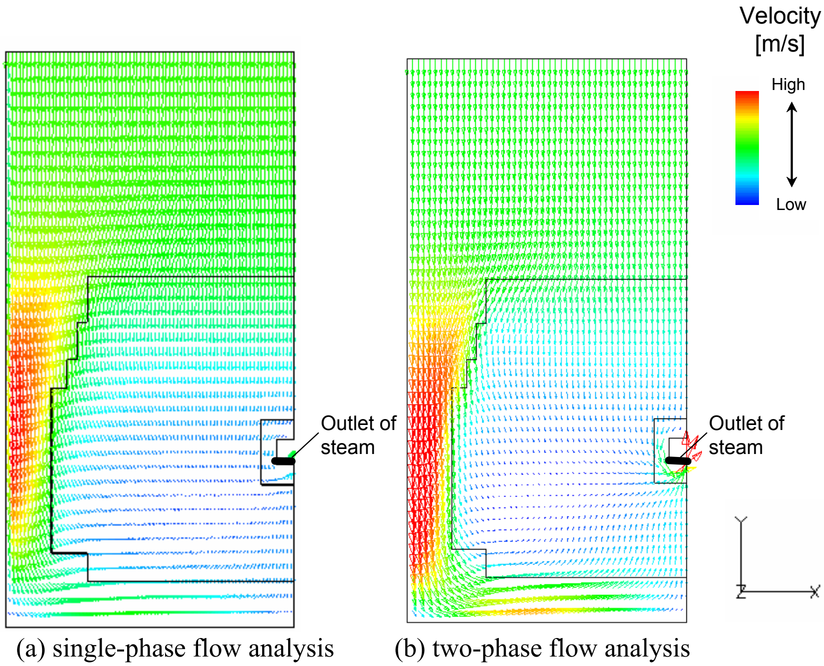
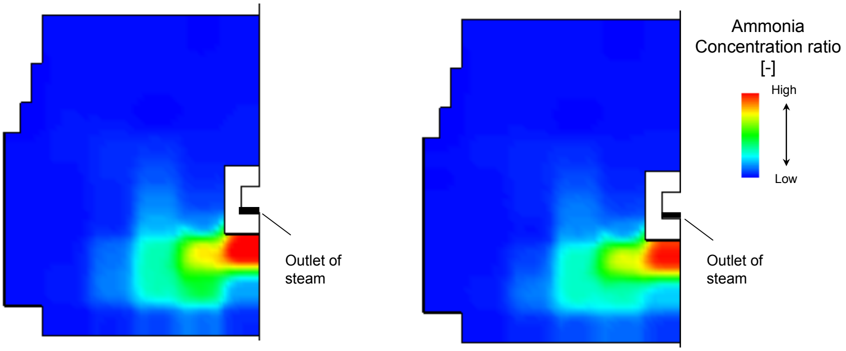


Figure 3: Steam velocity distribution.

Figure 4(a) and (b) show the distribution of the liquid phase ammonia concentration calculated by the network analysis. According to these results, it was confirmed that the ammonia concentration became higher near the steam outlet. It was considered that steam condensation started immediately as the steam entered into the tube bundle. However, ammonia condensed much later because volatility of the ammonia was much higher than steam. Therefore, ammonia was condensed near the outlet where total steam condensation occurred. As we can see in Figure 4(a) and (b), the ammonia concentrations looked very similar, so it was concluded that the difference of steam velocity between both results did not have any significant effect on the ammonia concentration distribution.



(a) single-phase flow analysis (b) two-phase flow analysis

Figure 4: Ammonia concentration distribution in the tube bundle.

4 Conclusions

We proposed a hybrid analysis method to obtain ammonia concentration by combining two-phase flow analysis and network analysis for the tube bundle of a condenser. In this method, two-phase flow CFD analysis was used to obtain mass flow distribution of steam and water in the tube bundle. The calculated mass flow distribution was passed to the chemical engineering process simulator for network analysis.

It was confirmed that there was little differences between single-phase and two-phase flow CFD analysis results. Therefore, single-phase flow analysis was more economical to obtain the flow distribution and made simulation easier without losing accuracy.

The proposed method was very useful in condenser design or modification to determine the area where higher grade material tubes should be used to avoid corrosion problems.

References

[1] S. S. Gulawania, S. K. Dahikara, J. B. Joshia, M. S. Shahb, C. S. amaPrasadb, D. S. Shuklab, CFD simulation of flow pattern and plume dimensions in submerged condensation and reactive gas jets into a liquid bath, *Chemical Engineering Science* ,63, pp. 2420 – 2435, 2008

[2] S. S. Gulawania, J. B. Joshia, M. S. Shahb, C.S. RamaPrasadb, D. S. Shuklab, CFD analysis of flowpattern and heat transfer in direct contact

www.witpress.com, ISSN 1743-3533 (on-line)

steam condensation , *Chemical Engineering Science* ,61, pp. 5204 – 5220, 2006

[3] F. Bezzo, S. Macchietto & C.C.Pantelides, A general methodology for hybrid multizonal/CFD models Part I. Theoretical framework, *Computers & Chemical Engineering*, 28, pp.501-511, 2004

[4] [4] F. Bezzo, S. Macchietto & C.C.Pantelides, A general framework for the integration of computational fluid dynamics and process simulation, *Computers & Chemical Engineering*, 24, pp. 653-658,2000

[5] M. Yasuhara & H. Daiguji, *Numerical Fluid Dynamics-Basic and Applications-*, Tokyo University Inc.: Tokyo, pp. 325-328, 1992

[6] Schiller, L., and Naumann, A., Ŭber die grundlegenden berechungen bei der schwerkraftbereitung. *Z. Vereins deutcher Ing*., 77(12), pp.318-320,1933

[7] N. Suzuki, Chemical Engineering Standard five edition , Maruzen Inc.: Tokyo, pp. 276-277, 1995