HEAT AND MASS TRANSFER IN AN INDIRECT CONTACT COOLING TOWER: CFD SIMULATION AND EXPERIMENT

Jorge Façao and Armando C. Oliveira
Department of Mechanical Engineering, Faculty of Engineering, University of Porto, Portugal

The present work is focused on the computational analysis of heat and mass transfer in an indirect contact cooling tower. The main objectives of the study are to contribute to the understanding of heat and mass transfer mechanisms involved in the problem and to check the possibility of making use of a commercial computational fluid dynamics (CFD) code for simulating mass and heat transfer phenomena occurring in indirect cooling towers. The CFD model uses as boundary conditions the temperatures of the tubes obtained by a correlation model developed by Mizushina. The available mass transfer correlations for indirect cooling towers are presented and compared with a correlation obtained from CFD simulations. The assumption of analogy between heat and mass transfer is also discussed.

1. INTRODUCTION

Computational fluid dynamics (CFD) has been greatly developed over recent years, mostly due to the rapid advance of computer technology. It is now possible to solve scientific problems in complex geometries. Gan and Riffat [1], Gan et al. [2], and Gan and Hasan [3] have published several works about CFD simulation of indirect contact cooling towers, but they have only simulated the sensible heat transfer between the heat exchanger and air. They also have simulated the spray water trajectory. Kloppers and Kröger have presented Poppe, Merkel, and e-NTU methods for direct contact cooling towers operating in counterflow [4] or in crossflow [5]. The mass transfer coefficient was calculated using the analogy between convective heat and mass transfer. Recently, Ren and Yang [6] have presented an analytical model for different flow arrangements in indirect evaporative coolers, considering non-unity of the Lewis number. The mass transfer coefficient was calculated with the help of a Lewis factor. The present study investigates the major mechanisms of coupled heat and mass transfer in an indirect contact cooling tower, namely in the evaporation cooling process. The mass
transfer coefficient was calculated using boundary conditions at the interface between spray water and air supplied by a correlation model. A previous work [7] shows that the mass transfer coefficient has an influence on the rate of change of cooling tower efficiency ($q_g = q_{km}^{0.7}$ to $1.4$).

The indirect contact cooling tower in analysis was designed in order to be used with chilled ceilings with a nominal cooling capacity of 10 kW, [8]. The cooler has a section of 0.6 m × 1.2 m and a height of 1.55 m. Tube bundle has 228 staggered tubes of 10 mm outside diameter, with transverse tube spacing of 60 mm, longitudinal tube spacing of 30 mm, and a total transfer area of 8.6 m². The ratio between heat transfer surface and heat exchanger volume is 25 m²/m³. Figure 1 shows schematically the evaporative cooler and main variables involved. The air flow was forced by a fan located at one tower side.

The mass transfer coefficient, $k_m$, can be obtained using enthalpy difference or using water mass fraction difference [7], assuming that the Lewis number is equal to 1 and neglecting the term ($c_v T$), in calculating the enthalpy of water vapor.

$$m_{air} (i_{air, out} - i_{air, in}) = A k_m \Delta T_{LM}$$

with

$$\Delta T_{LM} = \frac{i_{air, out} - i_{air, in}}{\ln \frac{h_{air, in}}{h_{air, out}}}$$
Outlet air enthalpy could be calculated considering that all the heat goes from water to air.

\[ i_{\text{air, out}} = i_{\text{air, in}} + \frac{m_{\text{water}} c_{\text{water}} (T_{\text{water, in}} - T_{\text{water, out}})}{m_{\text{air}}} \]  

Equation (1) was used to calculate the mass transfer coefficient (average value) from experimental measurements and from CFD simulations. Another alternative is to use the analogy between convective heat and mass transfer [9].

\[ \frac{\text{Sh}}{\text{Nu}} = \frac{k_m k}{\rho D \bar{h}} = \left( \frac{\text{Sc}}{\text{Pr}} \right)^m = \left( \frac{\alpha}{D} \right)^m = \text{Le}^m \]  

or

\[ k_m = \frac{\rho h \alpha}{k} \text{Le}^{m-1} \]  

Here \( m \approx 1/3 \) [9] and the dimensionless quantity \( \text{Le} = \alpha / D \) is called Lewis number, which for water vapor in air is about 0.87.

The mass transfer coefficient for evaporative coolers was correlated by several authors as a function of air mass flow rate. Parker and Treybal [10] were the first to publish a correlation for the mass transfer coefficient in evaporative coolers. They have used a staggered tube bundle with an external diameter of 19 mm and a pitch of \( 2D_0 \). The correlation spans the interval \( 0.68 < G_{\text{air}} < 5 \).

\[ k_m = 0.049 (G_{\text{air}})^{0.9} \]  

Figure 1. Indirect contact cooling tower and main variables.
where $G_{\text{air}}$ is the mass velocity in the narrowest cross section. Parker and Treybal also measured the heat transfer coefficient for the tube bundle (no spray water flow) and found that the mass transfer coefficient predicted by Lewis relationship (Eq. (5)) was 50% smaller than the measured coefficient.

Mizushina et al. [11] developed a correlation for mass transfer coefficient based on their experimental tests of four staggered bundles having a pitch of $2D_0$ with diameters of 12.7, 19.05, and 40 mm. They have found a spray mass flow rate and diameter dependence. Their data spanned $50 < \text{Re}_{\text{spray}} < 240$ and $1.2 < \text{Re}_{\text{air}} \times 10^{-3} < 14$. The correlation may be written in form of

$$k_m \alpha = 5.027 \times 10^{-8} (\text{Re}_{\text{air}})^{0.9} (\text{Re}_{\text{spray}})^{0.15} (D_0)^{-2.6}$$

(7)

where $\alpha$ is a surface area per unity of heat exchanger volume and the Reynolds number is defined by,

$$\text{Re}_{\text{spray}} = \frac{m_{\text{spray}}}{n_{\text{tubes}} \mu_{\text{spray}}} L$$

(8)

If we assume a tube bundle with an outside diameter of 19 mm, $\alpha = 44.8 \text{ m}^{-1}$ and the maximum spray mass flow rate, i.e., $\text{Re}_{\text{spray}}$ of 240, we may simplify Eq. (7) to

$$k_m = 0.036 (G_{\text{air}})^{0.9}$$

(9)

Niitsu et al. [12] have tested bundles with and without fins and they have concluded that for high spray mass flow rates the mass transfer was independent of the spray Reynolds number. The correlation for tubes without fins, with an outside diameter of 16 mm, a longitudinal pitch of $2.38D_0$, and a transverse pitch of $2.34D_0$ was

$$k_m = 0.076 (G_{\text{air}})^{0.9}$$

(10)

Equation (10) is valid for $1.5 < G_{\text{air}} < 5$.

Figure 2 presents the dimensionless correlations presented above as a function of Reynolds number.

2. CORRELATION-BASED MODEL

In order to know the temperature of the tubes in the exchanger a correlation based model was implemented. The model was first presented by Mizushina et al. [13] and applied by several researchers. In this model, the variation of spray water temperature could be evaluated; however, the spray water temperature was considered equal to the temperature at the interface water film/air.

The temperature variation of the water inside the tubes can be calculated through the thermal balance between water and spray.

$$\frac{dT_{\text{water}}}{dA} = \frac{U}{m_{\text{water}} c_{\text{water}}} (T_{\text{water}} - T_{\text{spray}})$$

(11)
The air enthalpy variation was calculated with the definition of enthalpy potential.

\[
\frac{d_i^\text{air}}{dA} = \frac{k_m}{m_\text{air}} (i - i_\text{air}) \tag{12}
\]

The model may use previous correlations (Eqs. (5)–(10)) to calculate \(k_m\), therefore being called a correlation based model.

The evolution of spray water temperature was calculated neglecting the water evaporation.

\[
\frac{dT_\text{spray}}{dA} = \frac{\dot{m}_\text{air}}{\dot{m}_\text{spray}c_p\text{spray}} \frac{d_i^\text{air}}{dA} - \frac{\dot{m}_\text{water}c_p\text{water}}{\dot{m}_\text{spray}c_p\text{spray}} \frac{dT_\text{water}}{dA} \tag{13}
\]

To calculate the outlet air temperature, the evolution of water mass fraction in air could be calculated,

\[
\frac{d\omega_\text{air}}{dA} = \frac{k_m}{m_\text{air}} (\omega_j - \omega_\text{air}) \tag{14}
\]

The temperatures and enthalpy field in the cooling tower were evaluated through the numerical integration of differential Eqs. (11)–(13). The integration was done in the direction of the air flow by the finite differences method. The known boundary conditions were the inlet water temperature, the inlet air enthalpy, and inlet spray water temperature equal to outlet spray water temperature. At the extreme points, the derivatives were approximated by forward or backward differences and in the interior of the grid the derivatives were approximated by central differences. This discretization transforms the differential equations into finite difference equations, whose solutions approximate the solution of the differential equations at discrete points.
3. CFD MODEL

The mathematical formulation of the problem was simplified by approximating the three-dimensional geometry by a two-dimensional one. The flow length is more than double the width. The system was considered in steady-state condition. On a cross-section of the tube bundle for two-dimensional simulation the tubes become separate entities.

The two-dimensional governing equations for steady-state turbulent flow adopted in the FLUENT code [14] were those related to the balance of mass, momentum, energy, species, turbulent kinetic energy, and turbulent energy dissipation rate. The κ-ε model with two equations was adopted to simulate turbulence. The flow field (calculated through the momentum equation) was considered to be independent of the temperature field.

4. BOUNDARY CONDITIONS

The temperature of the tubes was evaluated by the correlation model. The water mass fraction was calculated to a relative humidity of 100% (i.e., saturation condition) with the help of a user defined function programmed in FLUENT. The water mass fraction at air/water interface is a function of local temperature and local pressure.

\[ w_i = 0.62198 \frac{P_{\text{sat}}}{P - P_{\text{sat}}} \]  

(15)

The saturation water pressure was evaluated with a correlation published by Hyland and Wexler [15], for the temperature range of 0 to 200° C.

\[ \ln(P_{\text{sat}}) = C_1/T + C_2 + C_3 T + C_4 T^2 + C_5 T^3 + C_6 \ln(T) \]  

(16)

where

\[ C_1 = -5.8002206E3 \]

\[ C_2 = 1.3914993 \]

\[ C_3 = -4.8640239E - 2 \]

\[ C_4 = 4.1764768E - 5 \]

\[ C_5 = -1.4452093E - 8 \]

\[ C_6 = 6.5459673 \]

The tube bundle was divided into 12 rows, as presented in Figure 3. Table 1 presents tube temperatures evaluated by the correlation model which were used as
boundary conditions for the CFD model. The conditions are similar to experimentally-tested conditions. Note that water film temperatures and air temperatures at air-spray water interface were assumed to be the same. The conditions used in the correlation model were an inlet water temperature of 21°C, a water mass flow rate of 0.8 kg/s, inlet water temperature of 21°C, and spray water mass flow rate of 1.37 kg/s.

<table>
<thead>
<tr>
<th>Tube</th>
<th>$T_i$ [°C]</th>
<th>$T_i$ [°K]</th>
<th>$T_i$ [°K]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>290.53</td>
<td>292.62</td>
<td>293.20</td>
</tr>
<tr>
<td>2</td>
<td>290.85</td>
<td>292.76</td>
<td>293.29</td>
</tr>
<tr>
<td>3</td>
<td>291.02</td>
<td>292.83</td>
<td>293.34</td>
</tr>
<tr>
<td>4</td>
<td>291.09</td>
<td>292.86</td>
<td>293.36</td>
</tr>
<tr>
<td>5</td>
<td>291.09</td>
<td>292.86</td>
<td>293.36</td>
</tr>
<tr>
<td>6</td>
<td>291.05</td>
<td>292.84</td>
<td>293.35</td>
</tr>
<tr>
<td>7</td>
<td>290.97</td>
<td>292.81</td>
<td>293.33</td>
</tr>
<tr>
<td>8</td>
<td>290.88</td>
<td>292.78</td>
<td>293.31</td>
</tr>
<tr>
<td>9</td>
<td>290.77</td>
<td>292.73</td>
<td>293.28</td>
</tr>
<tr>
<td>10</td>
<td>290.66</td>
<td>292.68</td>
<td>293.25</td>
</tr>
<tr>
<td>11</td>
<td>290.53</td>
<td>292.62</td>
<td>293.21</td>
</tr>
<tr>
<td>12</td>
<td>290.40</td>
<td>292.57</td>
<td>293.17</td>
</tr>
</tbody>
</table>

Water mass flow rate of 0.8 kg/s, inlet water temperature of 21°C, and spray water mass flow rate of 1.37 kg/s.
of 0.8 kg/s, and a spray water mass flow rate of 1.37 kg/s. In the CFD model at inlet, the air/vapor mixture has a uniform velocity, temperature (20°C), and water concentration (0.007258). Outflow boundary conditions are used to model flow exit where the details of the flow velocity and pressure are not known prior to solution of the flow problem [14]. The sump temperature was equal to the wet bulb temperature. The density of the mixture was evaluated by considering it as an incompressible ideal gas and $c_p$, $\lambda$, and $\mu$ were calculated for a typical air temperature in the cooling tower, around 20°C. The mass diffusivity and thermal diffusion coefficient were evaluated by the kinetic theory. The effects of radiation heat transfer in the overall heat transfer were neglected.

5. SPACE DISCRETIZATION

A structured two-dimensional grid has been set up with FLUENT’s grid generation package, Gambit. The structured grid has the advantage that convergence occurs more rapidly and, often, more accurately. The grid was refined near the tubes where the largest variations of the thermal and fluid dynamic variables occur. Figure 4 presents the structured grid adopted in numerical computation. The overall geometry has about 347,270 cells. To test the grid independence, a model with an increase in resolution with a factor of 2 in all directions was tested. This represents an increase of the number of cells by a factor of 4 and then the required computation time for the CFD simulation also increases by approximately a factor of 4 [16]. The difference in outlet water mass fraction was about 1%, which means that the solution was independent of the grid size.

![Structured Grid](image)

**Figure 4.** Short representation of the structured grid adopted in the numerical computation.
6. RESULTS

In CFD simulations the solution was considered as converged when residuals were lower than $10^{-7}$ for the energy equation and $10^{-4}$ for the other equations. The evolution of outlet water mass fraction as a function of the number of iterations was also checked.

To check the influence of the inlet relative humidity, two CFD simulations were performed with 90% and 1% as inlet boundary conditions. The CFD simulations showed that the inlet relative humidity doesn’t influence the mass transfer coefficient, as in the theory the Sherwood number was only a function of the air Reynolds number.

The CFD average mass transfer coefficient correlation as a function of air mass flow rate is

$$k_m = 0.0413(G_{\text{air}})^{1.11}$$

The correlation obtained experimentally for the same cooler was [5]

$$k_m = 0.064(G_{\text{air}})^{0.81}$$

In order to compare these correlations and other existing ones, Figure 5 presents a dimensionless representation. For higher air flow rates CFD simulations are very close to the experimental correlation. For smaller air flow rates there was a small deviation. The CFD simulation underestimates the mass transfer coefficient, especially for lower values of Reynolds number.

As mentioned in the introduction, the mass transfer coefficient could be calculated after the heat transfer coefficient using the analogy between heat and mass transfer. Figure 6 presents a comparison of the mass transfer coefficient evaluated

![Figure 5](image_url)

**Figure 5.** Comparison of mass transfer coefficient correlations.
by CFD simulations, by experiment, and by the analogy between heat and mass transfer. Two heat transfer correlations were used: Zhukauskas correlation [17] for a cross flow across tube banks and a heat transfer correlation evaluated by CFD simulations for the cooling tower under analysis. The mass transfer coefficient

Figure 6. Sherwood number evaluated from experimental results through CFD simulations and by analogy between heat and mass transfer.

Figure 7. Contours of air velocity inside the evaporative cooler in analysis.
evaluated through analogy lies below the experimental and CFD correlations. The difference between the Zukauskas correlation and CFD results may be justified by the nonuniform flow distribution in the existing cooling tower (see Figure 7). The location of the fan air inlet (on one tower side) leads to a nonuniform flow pattern. Therefore, to improve flow distribution the fan should be located at tower exit or, with this configuration, baffles could be used to guide the air.

7. CONCLUSIONS

This article presented an analysis performed with the FLUENT code, aimed at improving the understanding of heat and mass transfer mechanisms that occur in indirect contact cooling towers and evaporative coolers.

The conclusions obtained can be summarized as follows.

- The correlation for the mass transfer coefficient obtained through CFD was close to the experimental correlation, especially for higher air flow rates. The minimum deviation for Sherwood number was 4% and the maximum deviation was 32%, which is acceptable for a two-phase turbulent flow.
- The use of analogy between heat and mass transfer to estimate the mass transfer coefficient revealed a deviation to experimental results of 53 and 90% (in Sherwood number), depending on the use of the heat transfer coefficient from the Zukauskas correlation or from CFD simulations. The non-uniform flow distribution inside the cooling tower could be the explanation for the difference between the heat transfer coefficient evaluated from the Zhukauskas correlation and from CFD simulations.

A limitation of the present numerical study regards neglecting the liquid film (spray) in the model. This aspect will be addressed in a future work.

REFERENCES