CFD MODELLING OF EVAPORATION FROM THIN HORIZONTAL LIQUID FILM

Shkurte Kastrati¹, Tomas Hyhlík¹, Radomir Kalinay¹
¹Department of Fluid Dynamics and Thermodynamics, Faculty of Mechanical Engineering, Czech Technical University in Prague, Technicka 4, 166 07 Prague 6, Czech Republic

Abstract
The present study deals with numerical modelling of heat and mass transfer with evaporation from a liquid film in a rectangular channel. The used mathematical model was implemented into the commercial Star-CCM+ code and results were compared with experimental data. From the comparison between numerical and experimental results, it is shown a satisfactory agreement.

Keywords: heat and mass transfer, liquid film, evaporation, CFD

1 Introduction
Evaporation from the wetted surface is of utmost importance in many technological processes. Nonetheless the evaporation of wetted surfaces phenomena lack of a deep understanding because of the complexity of the physics involved, and still needs to be addressed. Despite the fact that various commercial CFD software offer models able to treat the evaporative process, its investigation is still topical in the academic field. Natural convection in presence of water evaporation and condensation plays a crucial role in a number of natural and industrial processes. A realistic and accurate numerical reproduction of such phenomena is a challenging problem from both numerical and theoretical viewpoints.

From a numerical point of view, in most applications, flows cannot be considered laminar and an accurate reproduction of turbulence is required. Moreover, the Boussinesq approximation for incompressible flow, cannot be applied in systems with high temperature and humidity differences.

In the recent years, many researchers have been trying to solve the problems related to evaporation and condensation from thin liquid films. The study of Yan and Tsay [1] is focused on the numerical investigation of laminar steady mixed convection flow between two vertical parallel plates covered with fluid film. In this work the fluid film is treated as a boundary conditions, and it is defined as a Dirichlet boundary condition on the air-fluid interface. Natural convection flow is introduced into the mathematical model, and the Boussinesq approximation is implemented in axial-momentum equation. A similar approach can be seen in the study of Laaroussi [2], where commercial solver FLUENT 6.3 was used to conduct the numerical solution and source terms were introduced in the cells adjacent to the walls for the mass (mixture and species) conservation equations to account for vaporization of the liquid film. This work compares two approaches, how to invoke the buoyancy forces arising from the temperature gradients, hereinafter referred to as thermal buoyancy effects, and buoyancy forces arising from water vapour mass fraction concentration gradients, hereinafter referred to as solutal buoyancy effects, first, using the Boussinesq approximation, and density variation with changing temperature and water vapour mass fraction.

Work of Sosnowski [3] uses a similar approach as in the work Yan and Tsay [1], and Laaroussi [2]; on the air-fluid film interface the Dirichlet boundary condition is defined and the Boussinesq approximation reflecting both, solutal and thermal buoyancy effects is implemented into the incompressible form of Navier-Stokes equations. Numerical analysis is conducted using the Open-Foam code. The change of the thickness of the fluid film in time is evaluated using the velocity of evaporation and condensation.

Concluding foregoing, one approach of simulating convection flow and evaporation process developed by Sosnowski[3] will be investigated and adopted in this contribution. This approach will be

* Corresponding author
E-mail address: shkurte.kastrati@fs.cvut.cz
denoted as *two active scalars* based model, where solutal and thermal buoyancy effects are implemented into momentum equations, the air-fluid film interface is defined as a Dirichlet boundary condition and evaporation rate will be monitored by the *Boundary Species Flux* field function defined in Star-CMM+ code, which includes also the theory of Fick’s law. The subject of this work is very interesting and actual, and yet very challenging. The chosen approach corresponds to the present state of knowledge.

## 2 Problem description

The experiments are performed by Ing. J. Devera [4]. However, for the sake of clarity and a better understanding of these phenomena a brief description of the experimental part is dedicated in this section. The case geometry reported in the Figure 1 of examined phenomena have been measured on a test rig. The test rig is constructed of inlet nozzle directing the incoming air. After the nozzle, there is a rectangular duct of $0.09m^2$ cross-sectional area. On the bottom of the rectangular duct is located a fluid film of width $W$ and length $L$. Rectangular duct ends with a contraction connected to an outlet pipe. The flow rate of entering air is controlled by a small fan located near the end of the outlet pipe. Figure 1 shows design drawing of the test rig. Inlet mass flow rate is kept at a constant value to achieve approximately 0.1 m/s velocity in the mixing area. Inlet air has properties of ambient; i.e. inlet temperature and humidity are given and not regulated. Heating of the fluid film, which is located at the bottom of the mixing area, is provided by the heated aluminium plate and during measurement, the fluid film is maintained at constant temperature. The height of water film might be assumed as constant because of water tank below. A mixture of air and water vapour is then carried through the contraction and outlet pipe outwards in such a way to not influence conditions of the inlet ambient air.

![Figure 1: Schematic of the case investigated [4]](image)

### 2.1 Flow regime

The experiment described in Section 2 can be categorized as a problem of internal flow over a horizontal plate. As reference properties, ambient conditions and heated fluid film are assumed.

To determine whether the driving force is caused by natural or forced convection was used the Richardson number. The Richardson number means the relative intensity between natural and forced convection [5].

In case of ambient conditions given during measurements conducted in the summer, the calculated Richardson number is $Ri = \frac{Gr}{Re^2} > 65$, which implies that the *natural convection is dominant*; however, the role of the *forced convection is still assumed* since the conditions of natural and forced convection equality is exceeded by only a factor of 10.

Judging the flow from the perspective of natural convection, the value of Rayleigh number $Ra = 10^7$ suggests turbulent regime. Although the Reynolds number gives values of laminar flow, still it is
convenient to consider the flow as turbulent – same approach is adopted in [6]. The parameters related to thermal-solute convection are: The total Rayleigh number that characterizes the buoyancy-driven flows

\[ Ra = \frac{g \beta T (T - T_0)L^3}{\nu \alpha_a} + \frac{g \beta_{\omega} (\omega - \omega_0)L^2}{\nu \alpha_w} \]  

composed of the sum of thermal and solute contributions. It can be interpreted as the ratio between the destabilizing effect of buoyancy forces and the stabilizing effect of momentum and thermal diffusion.

The total Grashof number, that also includes a thermal and a solute contribution

\[ Gr = \frac{gL^3}{\nu^2} \beta_T (T - T_0) + \beta_{\omega} (\omega - \omega_0) \]

is the ratio between buoyancy and viscous forces, where \( Ra \) and \( Gr \) are Rayleigh and Grashof number, respectively, \( u \) is velocity, \( T \) is temperature, \( L \) is characteristic length, \( \omega \) is water vapour concentration, \( \beta_{\omega} \) is vapour expansion coefficient, \( \beta_T \) is air thermal expansion coefficient, the subscript zero denotes the reference value of the corresponding variable.

3 Mathematical model

Different physical phenomena have to be modelled by a suitable set of mathematical equations: natural convection in the fluid domain; the heat and mass transfer between the liquid phase and the surrounding gas through change of phase; and the exchange of heat between wetted solid bodies and the thin liquid film or drops laying on their surface. The latter one will not be treated in the present work.

3.1 Governing equations

The gas phase is governed by the incompressible Navier–Stokes equations with the Boussinesq approximation (5) to account for buoyancy effects [3].

\[ \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho_0} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} - \frac{\rho}{\rho_0} g \delta_{i,2} \]

\[ \frac{\rho}{\rho_0} = 1 - \beta_T (T - T_0) - \beta_{\omega} (\omega - \omega_0) \]

where \( u \) is the fluid velocity, \( p \) is pressure, \( T \) is temperature, \( \rho \) is space-time variable density in the fluid flow, \( \omega \) is water vapour concentration. The zero subscript denotes the reference value of the corresponding variable.

The vapour concentration is defined as:

\[ \omega = \frac{m_v}{m_v + m_a} \]  

in which \( m_v \) and \( m_a \) are the vapour and air masses, respectively. Temperature and vapour concentration in the fluid medium are both modelled as active scalars. They are diffused and transported by air according to the advection-diffusion equations as follows:

\[ \frac{\partial T_a}{\partial t} + u_j \frac{\partial T_a}{\partial x_j} = \alpha_a \frac{\partial^2 T_a}{\partial x_i \partial x_i} \]

\[ \frac{\partial \omega}{\partial t} + u_j \frac{\partial \omega}{\partial x_j} = \alpha_\omega \frac{\partial^2 T_a}{\partial x_i \partial x_i} \]

The turbulence modelling is based on the constant value of turbulent Prandtl number and turbulent Schmidt number.
3.2 Air-fluid film interface definition

The air-fluid film interface definition is treated similarly as in the work Yan and Tsay [1] and Sosnowski [3]. From the theory of evaporation, it is assumed a thin saturated layer of moist air above a fluid film, in which moist air and a liquid film are in thermodynamic equilibrium in this layer. The interfacial concentration of water vapour can be evaluated as follows [3]:

$$\omega_v = \frac{M_v}{M_a} \frac{p_v''(T_{ref}) \varphi}{p_{atm} - \left(1 - \frac{M_v}{M_a}\right) p_v''(T_{ref})}$$

(9)

where $M_a = 28.97 \text{ g/mol}$ and $M_v = 18.02 \text{ g/mol}$ are the values of molar mass of air and water vapour respectively; $p_{atm}$ and $p_v''$ are the vapour atmospheric pressure at actual and at saturation condition, respectively; $\varphi$ is relative humidity which equals unity at the water-air interface; $T_{ref}$ is the temperature at the air-water film interface.

The saturated pressure $p_v''$ can be evaluated as in [3]:

$$p_v'' = 611.85 \exp \left(17.502 \frac{(T_{ref} - 273.15)}{(T_{ref} + 273.15)}\right)$$

(10)

Note that expression of (10) is strictly valid for the water-air interface.

4 Numerical approach and simulations settings

Simulations are carried out using the commercial CFD tool of Star-CCM+ and were conducted as three dimensional with gravity effect considered. The continuum is assumed as non-reacting. Such formulation can be described as using a collocated variable arrangement (opposed to staggered) and Rhie-and-Chow type pressure-velocity coupling combined with a SIMPLE type algorithm. This model is more suitable for constant density flows but it can handle mildly compressible flows and low Rayleigh number natural convection [7]. Although, the simulated process is a combination of free and forced convection the solution might be assumed as steady. The flow is modelled as turbulent recalling the identification in section 2.1. Realisable k-$\varepsilon$ Two-Layer turbulence model is selected since it is possible to activate Buoyancy Driven Two-Layer Type model correlating the turbulence parameters (turbulent kinetic energy $k$ and turbulent dissipation rate $\varepsilon$) for flows where buoyancy forces dominate.

Radiation effect is not neglected due to its considerable influence. Since dry air and moist air do not participate in radiation heat transfer the Surface-to-Surface model is selected. Radiation properties of surfaces are considered same for all wavelengths, the Gray Thermal Radiation model is used, and their values are defined according to [8].

4.3 Boundary conditions

Figure 2a describes parts of the computational domain and Table 1 are described how boundary conditions are defined within the computational domain.

<table>
<thead>
<tr>
<th>Part of the computational domain</th>
<th>Boundary condition definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Mass flow inlet</td>
</tr>
<tr>
<td>Walls</td>
<td>Non-adiabatic walls</td>
</tr>
<tr>
<td>Fluid film</td>
<td>Wall of static temperature and species sources</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure outlet</td>
</tr>
</tbody>
</table>

As it is stated in Table 1 the heat transfer through walls is considered. The test rig is manufactured from different materials, therefore, the heat transfer coefficient distribution is not constant. The distribution of heat transfer coefficient along the test rig shows Figure 2b.
The water film area in STAR-CCM+ is defined as a Dirichlet boundary condition and it is considered as an air-water film interface. The water vapour mass fraction on the air-water film interface $\omega_v$ is calculated according to (9). The temperature on the interface is assumed same as the temperature of the plate which is heating up the water film.

To be able to define the fluid film in Star-CCM+, equations (9) and (10) needs to be reproduced by using user-defined field functions.

The water vapour mass fraction determined by (9) is used to define a species source on the area of the fluid film to invoke the evaporative process, via field function of Boundary Species Flux.

Having in consideration that the main interest is in evaporation rate of water vapour relative to the surface of the water film, field function Boundary Species Flux is multiplied by water film surface, after that, it is obtained the water evaporation rate $\dot{m}_{ev}$.

Simulations are initialized in ambient conditions, i.e. properties of a continuum in the domain for zero iteration are same as ambient. Continuum is assumed as a multi-component gas air and water ($H_2O$) and related mass fractions define the species ratio.

5 Results comparison

Simulation results are compared with the experimental measurements from which the boundary conditions are determined. The validity of simulations is confirmed based on a comparison of water evaporation rate, temperature field and the outlet humidity. Table 2 shows boundary conditions and experimental data for three data sets. These datasets were used to compare simulation results of water evaporation rate, outlet temperature and outlet humidity against experimental data.

The second part of Table 2 provides the experimental data. In the Table 2 $\theta_{out}$ represents outlet dimensionless temperature. Outlet water vapour mass fraction $\omega_{vout}$ represents the outlet humidity. Water evaporation rate is quantified as a permil of the inlet mass flow rate of the dry air, this relates the evaporative process with the intensity of the inlet flow.

In table Table 2 are shown the boundary conditions mostly defined using dimensionless numbers (Re, Gr, Ri, Ra, Pr and Sc). For Re and Gr characteristic length $L$ is considered as the length of the fluid film.

Another parameter used to define boundary conditions is ambient pressure $p_{amb}$. Inlet vapour mass fraction $\dot{m}_{in}$ represents the inlet humidity. In Star-CCM+ the species ratio is defined using the vapour mass fraction, therefore it is convenient to employ its usage instead of specific humidity. $\dot{m}_{in}$ is the inlet mass flow of dry air and $\Delta T$ is defined as $\Delta T = T - T_0$ and represents the amount of temperature potential.
Table 2: Three data sets of experimental data determining boundary conditions for simulations and data used for simulation validity assessment based on water evaporation rate, outlet temperature and outlet humidity comparison.

<table>
<thead>
<tr>
<th>Parameter/Dataset</th>
<th>BC1</th>
<th>BC2</th>
<th>BC3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Boundary conditions</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reynolds number</td>
<td>6430.72</td>
<td>6290.36</td>
<td>6412.09</td>
</tr>
<tr>
<td>Grashof number</td>
<td>4.08E+09</td>
<td>3.26E+09</td>
<td>2.79E+09</td>
</tr>
<tr>
<td>Richardson number</td>
<td>98.73</td>
<td>82.49</td>
<td>67.97</td>
</tr>
<tr>
<td>Rayleigh number</td>
<td>5.33E+09</td>
<td>4.26E+09</td>
<td>3.65E+09</td>
</tr>
<tr>
<td>Prandtl number</td>
<td>1.3061</td>
<td>1.3074</td>
<td>1.3076</td>
</tr>
<tr>
<td>Schmidt number</td>
<td>0.4898</td>
<td>0.5086</td>
<td>0.5200</td>
</tr>
<tr>
<td>(P_{\text{amb}}) [Pa]</td>
<td>98490</td>
<td>99200</td>
<td>99200</td>
</tr>
<tr>
<td>(\omega_{\text{vin}})</td>
<td>0.0134</td>
<td>0.0126</td>
<td>0.0125</td>
</tr>
<tr>
<td>(\dot{m}_{\text{eq}}) [kg/s]</td>
<td>0.0106</td>
<td>0.0104</td>
<td>0.0106</td>
</tr>
<tr>
<td>(\Delta T) [°C]</td>
<td>31.95</td>
<td>25.95</td>
<td>21.89</td>
</tr>
<tr>
<td>Experimental data</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(\theta_{\text{out}})</td>
<td>9.43</td>
<td>11.58</td>
<td>13.40</td>
</tr>
<tr>
<td>(\omega_{\text{vout}})</td>
<td>0.0255</td>
<td>0.0219</td>
<td>0.0197</td>
</tr>
<tr>
<td>(\dot{m}<em>{\text{eq}}/\dot{m}</em>{\text{in}}) [%]</td>
<td>12.55</td>
<td>9.45</td>
<td>7.25</td>
</tr>
</tbody>
</table>

6 Results and discussions

Dependencies presented in Figure 3, Figure 4 and Figure 5 are within ranges of \(\Delta T \in <20; 34 >\), \(\dot{m}_{\text{in}} \in < 0.0104; 0.0106 >\), and \(\omega_{\text{vvin}} \in < 0.0125; 0.0134 >\) assumed as linearly dependent.

From the comparison of experimental data and simulation results of water evaporation rate (Figure 3) can be concluded that the implemented model, the two active scalars based model results deviation from the experiment results is within the range of 10%.

![Figure 3: Water evaporation rate dependency on temperature difference \((T - T_0)\)](image)

Comparison of simulation results and experimental data of outlet humidity dependency on inlet humidity (Figure 4) shows good agreement and simulation results deviation is within the range of 10%.
Simulation results of water evaporation rate and outlet humidity are higher for all boundary conditions when compared to the experimental data. This results from the fact that the temperature on air-water film interface is assumed same as the temperature of aluminium plate heating up the water film. In reality, the temperature at air-water film interface is slightly lower due to the process of vaporization. In Figure 5 can be seen the comparisons of outlet temperature dependency on the temperature difference. Outlet temperature is in dimensionless form. For boundary conditions BC1 and BC2, the simulation results and experimental data are within 20% deviation. In the case of boundary condition BC3, the deviation of simulation result from experiment data is higher than 20%.
6 Conclusion

In this work, an attempt has been made to simulate evaporation from a thin horizontal liquid film. The model of interest, the two active scalars based model is firstly described theoretically, in order to highlight its main principles, and then its implementation into commercial Star-CCM+ code is shown. The validity of the developed model is evaluated by comparison with the experimental data. The simulation validity was assessed based on a comparison of water evaporation rate, outlet temperature, and the outlet humidity. It should be pointed out, that the comparison of simulation results and experimental data of water evaporation rate shows a very good agreement; the difference of simulation results from the experimental data is on average less than 10%. Overall, it is achieved of the acceptable match. Simulation results of outlet temperature and outlet humidity are within satisfactory difference from the experiment.

It was said that in this work the heat exchange between the solid substrate and the laying water film is not treated. Evaporation/condensation on solid surfaces absorbs/releases heat; in a wide range of applications, these effects strongly alter the temperature at the solid boundaries. In future work the exchange of heat between the three medias i.e. solid, water film and gaseous phase media respectively, it is recommended to be taken into consideration, and the heat source/sink due to condensation/evaporation in transport equations should not be neglected.

Acknowledgements

We gratefully acknowledge the support from Centre of 3D volumetric velocimetry- (COLA), reg. no. CZ.2.16/3.00/21569, supported by operational programme Prague Competitiveness.

References


