On the development of an OpenFOAM solver to model two-phase horizontal flow in a heated pipe

Josefine M. Wilms and Thomas M. Harms

Mechanical and Mechatronic Engineering, University of Stellenbosch, Private Bag X1, 7602 Matieland, South Africa,

E-mail: fine@sun.ac.za

Abstract

The standard *twoPhaseEulerFoam* solver in OpenFOAM is a C++ based Computational Fluid Dynamics (CFD) code that uses a two-fluid Eulerian approach to model the interaction between two incompressible fluids. It is applicable to isothermal fluids and does not include phase change due to heating or cooling. In this paper *twoPhaseEulerFoam* is adapted to include heat transfer and phase change. An overview of the volume averaged conservation equations for mass, momentum, energy and correlations for evaporation and condensation are given. Simulation results as well as grid and input data for these simulations are presented.

Keywords: OpenFOAM; two-phase flow; heat transfer.

1 Introduction

Convective boiling in horizontal heated channels has many industrial applications including parabolic trough or Fresnel collectors in the solar energy industry: Such collectors consist of mirrors orientated so that the sunlight reflected from them is concentrated on a horizontal receiver tube. This tube contains a heat transfer fluid which is heated to a high temperature by the energy of the sunlight. In the majority of these receivers, the heat transfer fluid is synthetic oil and steam is generated in a secondary heat exchanger.

Since the intention is to generate steam, new technology is aimed at replacing oil with water as a heat transfer fluid. This is known as direct steam generation (DSG). However, unlike synthetic oil, which does not undergo a phase change during heating, water evaporation, associated with DSG, may result in an uneven flow distribution. The detailed engineering of the collector field requires an accurate understanding of the flow rate distribution within these receiver tubes and has led to renewed interest in the numerical modelling of two-phase horizontal fluid motion. Consequently, researchers such as [1] presented a transient model, based on the drift flux model, and [2] used the Modelica library 'DissDyn', which is a homogeneous equilibrium model to investigate the flow field within the DSG receiver tubes.

The underlying objective of our research is to, eventually, develop a model in the open-source environment which can predict the various flow regimes which the heat transfer fluid undergoes during the heating process in direct steam generation plants.

A two-fluid Eulerian methodology is applied to model the transformation of water from liquid to vapour phase. This model comprises of momentum, mass-, and energy conservation equations for each of the two phases and therefore consists of six equations in three spatial dimensions. Momentum transfer between the phases due to drag during relative motion of the phases as well as mass transfer as a result of evaporation is taken into account.

2 Mathematical modelling

In this section the conservation equations for two-phase Eulerian modelling are described and additional closure relations are discussed.

2.1 Governing flow equations

The volume averaged conservation equations for mass, momentum, and energy are given by [3] as

$$\frac{\partial \varepsilon_k}{\partial t} + \nabla \cdot (\varepsilon_k \underline{u}_k) = \frac{\Gamma_{ki} - \Gamma_{ik}}{\rho_k}, \tag{1}$$

$$\frac{\partial \varepsilon_k \underline{u}_k}{\partial t} + \nabla \cdot (\varepsilon_k \underline{u}_k \underline{u}_k) = -\frac{\varepsilon_k \nabla p}{\rho_k} + \frac{1}{\rho_k} \nabla \cdot \left(\varepsilon_k \underline{\underline{\tau}}_k^{eff}\right) + \varepsilon_k \underline{g} + \frac{M_{ki}}{\rho_k} + \frac{\Gamma_{ki} \underline{u}_i - \Gamma_{ik} \underline{u}_k}{\rho_k}, \tag{2}$$

and

$$\frac{\partial \rho_k (h_k + k_k)}{\partial t} + \nabla \cdot (\rho_k \underline{u}_k (h_k + k_k)) = -\varepsilon_k \frac{\partial p}{\partial t} + \nabla \cdot K_k^{eff} \nabla h_k + \Gamma_{ki} h_i - \Gamma_{ik} h_k + E_{ki} + Q_k, \tag{3}$$

where the subscript k denotes the phase which can be liquid, f, or vapour, g. The subscript i denotes the non-k phase. In Eq.'s (1) to (3) ε_k, ρ_k, $\underline{u}_k$, h_k, and k_k are the volume fraction, density, velocity, specific enthalpy and kinetic energy of phase k, respectively. Transfer of mass, momentum, and energy from phase i to phase k are given by Γ_{ki}, M_{ki}, and E_{ki}, respectively and the effective shear stress and thermal diffusivity which governs how each phase interacts with itself is given by $\underline{\underline{\tau}}_k^{eff}$ and K_k^{eff}. Additional terms, which denote the contributions to the governing equations from external sources, are acceleration due to gravity, $\underline{g}$, and the heat source to phase k, Q_k.

The three-dimensional model for unsteady, two-phase flow, given by Eq.'s (1) to (3), requires correlations for the interaction terms in terms of state variables. These correlations are given in the next section.

2.2 Closure

The closure relationships are derived for a continuous liquid phase and a dispersed vapour phase.

2.2.1 Correlations for self-interaction

The interaction of a phase with itself is given in Eq.(2) by the effective stress, $\underline{\underline{\tau}}_k^{eff}$. This term consists of viscous -and turbulent stress contributions which are approximated with the Boussinesq equation:

$$\frac{1}{\rho_k} \nabla \cdot \left(\varepsilon_k \underline{\underline{\tau}}_k^{eff}\right) = \nabla \cdot \varepsilon_k \upsilon_k^{eff} \nabla \underline{u}_k - \nabla \cdot \varepsilon_k \left(\upsilon_k^{eff} \frac{2}{3} \underline{\underline{I}} tr(\nabla \underline{u}_k)^T - \upsilon_k^{eff} (\nabla \underline{u}_k)^T\right). \tag{4}$$

The thermal diffusivity, K_k^{eff}, in Eq. (3) is given by [4] as

$$K_k^{eff} = \frac{\upsilon_k}{\mathrm{Pr}_k} + \frac{\upsilon_k^t}{\mathrm{Pr}^t}, \tag{5}$$

where Pr_k is the dimensionless Prandtl number, Pr^t is the turbulent Prandtl number, assumed to be 0.9 according to [5] , and υ_k is the kinematic viscosity for phase k.

The turbulent kinematic viscosity for the gas phase is given by [4] in terms of turbulent response coefficient, C_t , and the turbulent kinematic viscosity of the liquid phase as

$$\upsilon_g^t = C_t^2 \upsilon_f^t. \tag{6}$$

The turbulent response coefficient is defined as the ratio of the root mean square values of the dispersed phase velocity fluctuations to those of the continuous phase, $C_t = u'_g / u'_f$, and [5] suggests a value of 1.0.

From Eq.(5), it follows that the heat exchange within a phase is primarily dealt with in terms of turbulent parameters. It is, however, currently assumed that the flow remains in the laminar regime and turbulent variables are therefore assumed to be zero. Enthalpy exchange within a phase is therefore very small and will only become significant once a turbulent model is used.

2.2.2 Correlations for phase interaction

The interfacial mass transfer terms, Γ_{fg} and Γ_{gf}, denote the rate of condensation and evaporation, respectively. The rate of evaporation is given by [3] as

$$\Gamma_{gf} = \frac{q_w A_w}{h_{fg}}, \tag{7}$$

where q_w is the wall heat flux, h_{fg} is the enthalpy of formation, and A_w is the contact area with the wall per unit volume. Strictly speaking, this is only applicable to instances where the vapour phase is assumed to be saturated but this correlation is applied here as is. Following [4] it is also assumed that evaporation can only occur at the heated walls of the domain.

The rate of condensation is given by [3] as

$$\Gamma_{fg} = \frac{H(T_{sat} - T_f) a_i}{h_{fg}}, \tag{8}$$

where the saturation temperature and liquid phase temperatures are given by T_{sat} and T_f, respectively and a_i is the interfacial area concentration. Currently, condensation is still neglected.

The interfacial momentum transfer is expressed as a combination of a drag -and lift forces:

$$\underline{M}_{fg} = \underline{M}_{fg}^{lift} + \underline{M}_{fg}^{drag}. \tag{9}$$

The lift force $\underline{M}_{fg}^{lift}$ is given by

$$\underline{M}_{fg}^{lift} = \frac{\varepsilon_f \varepsilon_g}{\rho_g} C_l (\varepsilon_f \rho_f + \varepsilon_g \rho_g)(\underline{u}_{rel} \times \nabla \times \underline{u}), \tag{10}$$

where C_l is the lift coefficient, $\underline{u}_{rel}$ is the relative velocity between phases (i.e. $\underline{u}_{rel} = \underline{u}_g - \underline{u}_f$), and $\underline{u}$ is given by $\underline{u} = \varepsilon_g \underline{u}_g + \varepsilon_f \underline{u}_f$.

The drag force, $\underline{M}_{fg}^{drag}$, is given by

$$\underline{M}_{fg}^{drag} = \frac{0.75 C_{ds} \rho_f \underline{u}_{rel}}{d_g}, \tag{11}$$

where d_g denotes the diameter of the vapour phase and the Schiller-Naumann drag correlation [6] is used for the drag coefficient, C_{ds}:

$$C_{ds} = \begin{cases} \dfrac{24(1 + 0.15Re^{0.678})}{Re} & if\ Re < 1000 \\ 0.44 & if\ \ Re > 1000. \end{cases} \tag{12}$$

The definition of the Reynolds number, Re, used in Eq. (12), is given by

$$\mathrm{Re} = \frac{\underline{u}_{rel} d_g}{\upsilon_f}. \tag{13}$$

Energy interaction between phases, E_{ij}, which forms part of Eq.(3), was neglected in this work.

3 Numerical implementation

OpenFOAM is an object-orientated open-source CFD software library built in the C++ programming language.

The equations, given in Section 2, were implemented by modifying an existing OpenFOAM solver, *twoPhaseEulerFoam*. This modified solver, *twoPhaseEulerFoamPhaseChange*, differs from the standard solver in that it includes two energy equations and provides for mass transfer due to phase change.

3.1 The algorithm

The solution procedure starts by initialising the variables and boundary conditions. Since $\varepsilon_f = 1 - \varepsilon_g$ holds, it is only necessary to solve the mass conservation equation for the vapour phase. The drag and heat-transfer coefficients are computed and the two energy equations are solved. The momentum equations are solved followed by the solution of the pressure correction equation. The velocities and pressure are then updated and if the user defined convergence criterion is satisfied, the time is updated. The algorithm is shown in Figure 1.

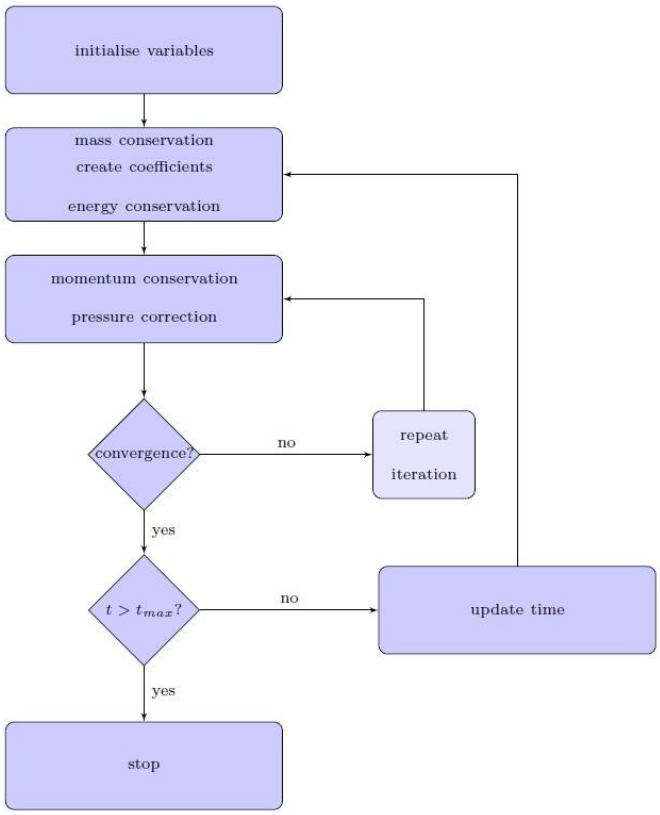

Figure 1: Solution method.

3.2 Simulation setup and initial conditions

The simulations were conducted for a horizontal two-dimensional case, shown in Figure 2. The flow domain, which is 1.0 m in length and 0.03 m wide, is initially filled with water moving at 0.001 m/s from left to right whilst a constant heat flux is applied at the bottom wall. A uniform grid of 15×50 cells was created with OpenFOAM's blockMesh utility.

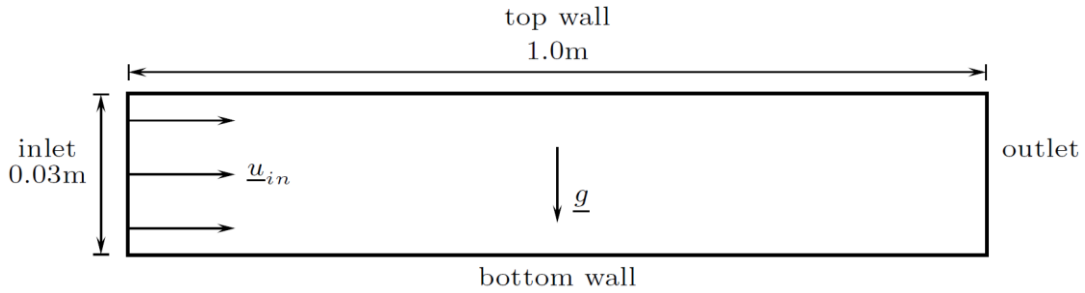

Figure 2: Setup for horizontal fluid motion.

The boundary conditions for the vapour volume fraction, ε_g, pressure, p, velocities, $\underline{u}_g$ and $\underline{u}_f$, and temperatures, T_g and T_f, are summarised in Table 1.

property	**inlet**	**walls**	**outlet**
ε_g [-]	0	zero gradient	zero gradient
$p\left[\frac{N}{m^2}\right]$	zero gradient	zero gradient	zero gradient
$\underline{u}_f$ and $\underline{u}_g\left[\frac{m}{s}\right]$	0.001	no-slip	zero gradient
T_f and T_g [K]	370	zero gradient	zero gradient

Table 1: Boundary conditions for setup shown in Figure 2.

The properties, used in the simulations, for the water -and vapour phases are summarised in Table 2.

property	**water**	**vapour**
kinematic viscosity $\nu\left[\frac{m^2}{s}\right]$	1.0e-06	1.6e-05
density $\rho\left[\frac{kg}{m^3}\right]$	1000	1
Prandtl number Pr $[-]$	2.24	0.7
heat capacity $c_p\left[\frac{m^2}{s^2K}\right]$	4200	1800
heat capacity $c_v\left[\frac{m^2}{s^2K}\right]$	4200	1400

Table 2: Water and vapour properties.

In Table 2 c_p and c_v are the specific heat capacities at constant pressure and volume, respectively. The enthalpy of formation (or latent heat), h_{fg}, is taken as 1,675,853 J/kg, a constant wall heat flux, Q_{wall}, is specified as 300,000 J/(kgs), and the saturation temperature is given as 373 K.

3.2.1 Results and discussion

The vapour volume fraction and temperature results at times, t = 10 s, 20 s, and 30 s, which demonstrate evaporation due to a heat flux at the bottom wall, are shown in Figure 3 and Figure 4, respectively.

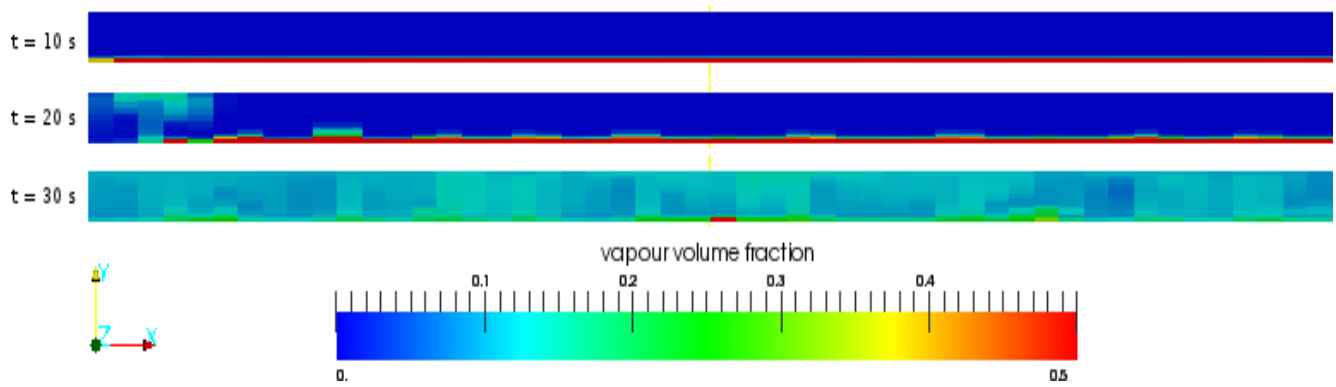

Figure 3: Vapour volume fraction at t = 10 s, 20 s, 30 s.

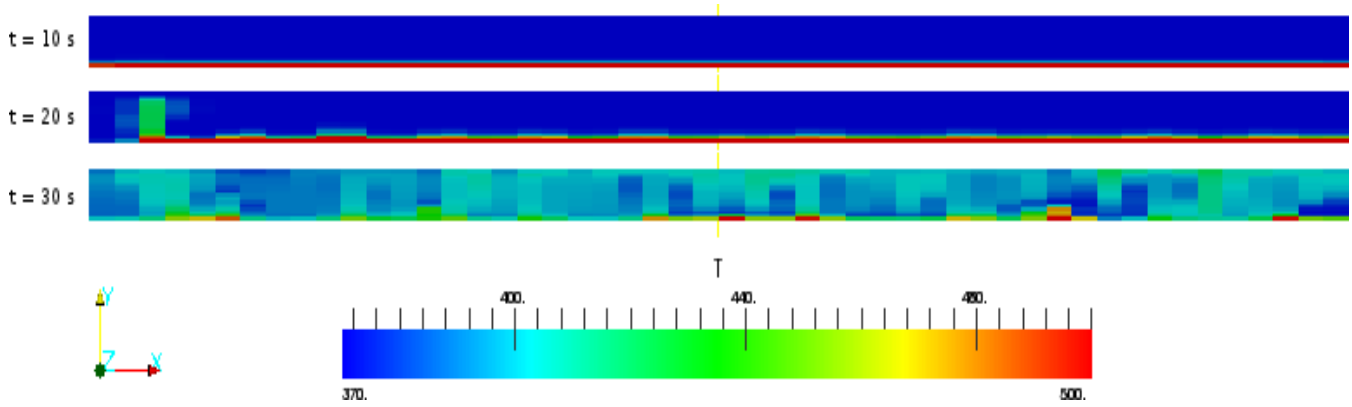

Figure 4: Temperature [K] at t = 10 s, 20 s, 30 s.

The temperature result, shown in Figure 4, was computed by assuming that the energy amount going out of the hot vapour is equal to the energy amount entering the liquid phase. The mixture temperature, *T*, is therefore given by

$$T = \frac{\rho_g \varepsilon_g c_{vg} T_g + \rho_f \varepsilon_f c_{vf} T_f}{\rho_g \varepsilon_g c_{vg} + \rho_f \varepsilon_f c_{vf}} . \tag{14}$$

The domain shown in Figure 2 is initially filled with water at a temperature of 370 K. From Figure 3 it is seen that vapour initially forms at the heated bottom wall and rises up due to buoyancy effects. For illustrative purposes, Figure 1 is plotted for a vapour volume fraction ranging from 0 to 0.5 and not from 0 to 1.0. Figure 4 shows that the temperature increases as the vapour volume fraction increases. Note that condensation has not been taken into consideration and that a laminar model was used for the simulation.

4 Conclusion

This work is still in its initial phases. The observed phase change and behaviour of the vapour phase are in line with expected behaviour. Future work, entailing the inclusion of an additional transport equation in order to determine the bubble diameter and resulting inter-facial forces, is essential for accurate modelling of the condensation process. In addition, a turbulence model needs to be implemented, energy interaction between phases needs to be accounted for, and the effect of vapour compressibility on the phase change needs to be investigated.

References

[1] Y. Taitel and D. Barnea, "Transient solution for flow of evaporating fluid in parallel pipes using analysis based on flow patterns", *International Journal of Multiphase Flow*, vol. 37, pp. 469-474, 2011.

[2] T. Hirsch, W. Steinmann and M. Eck, "Simulation of transient two-phase flow in parabolic trough collectors using Modelica", in *4th International Modelica Conference*, Hamburg, 2005.

[3] H. Anglart and O. Nylund, "CFD application to prediction of void distribution in two-phase bubbly flows in rod bundles", *Nuclear Engineering and Design*, pp. 81-98, 1995.

[4] E. Michta, "Modeling of Subcooled Nucleate Boiling with OpenFOAM", Royal institute of technology, Stockholm, 2011.

[5] A. Ghione, "Development and validation of a two-phase CFD model using OpenFOAM", Master's thesis. Royal Institute of Technology, Stockholm, 2012.

[6] L. Schiller and Z. Naumann, "A drag coefficient correlation", *Zeitschrift des Vereins Deutscher Ingenieure*, 1935.