Numerical Simulation of Phase Change Modeling Including Conjugate Heat Transfer

Nima Samkhaniani¹, MohamadReza Ansari²

¹ Ph.d Post-Graduated, Tarbiat Modares University; nima.samkhaniani@modares.ac.ir
² Associate Professor, Tarbiat Modares University; mra_1330@modares.ac.ir

Abstract
In the present study, volume of fluid method in OpenFOAM CFD package is extended to consider phase change phenomena due to heat transfer from solid region. Both fluid phases are considered immiscible and incompressible and mass transfer is accounted for Lee model. Newtonian flows are solved using a finite volume scheme based on PISO algorithm. This code is validated with three simple test cases.

Keywords: Film Boiling, OpenFOAM, VOF, Heat Transfer

Introduction
In last two decades, various computational schemes for modeling of interfacial and two phase flow have been extended. Interface resolving method combined with single fluid formulism has been applied extensively for modeling of isothermal interfacial flow [1, 2]. Single-fluid formalism is based on solving a single set of transport equations for the whole computational domain and treating the two phases as a single fluid with variable material properties. Changes in these properties are accounted for by advection a phase indicator or color function. Single-fluid model is called as a direct numerical simulation of interface motion (not of turbulence) [3]. The key to these methods is the use of a single-phase set of conservation equations. Surface tension, mass transfer and etc are added as source term in equations. Interface resolving method can be divided in two main groups: sharp interface method and diffuse interface method [1]. Three most popular interface resolving methods are volume of fluid (VOF) [4], level-set (LS) [5] and front-tracking (FT) [6]. Each method has its own advantages and disadvantages. VOF method keep the mass conserve better than LS or FT, however it represents interface curvature or normal vector less accurate than LS or FT [1]. VOF methods can be classified based on interface reconstruction to: 1) VOF with interface reconstruction (IR-VOF) such as SLIC-VOF [7], PLIC-VOF [8], PROST [9] 2) VOF without interface reconstruction which called color function VOF (CF-VOF) such as CICSAM [10]. (CF-VOF) for ease of implantation and omit of geometrical reconstruction of interface are getting popular. In past decade interface resolving method is extended to include mass transfer in evaporation process. Extension of phase change in interface tracking methods is reported for VOF [11], level-set [12], front-tracking [13], CLSVOF [14]. Analysis of conjugate heat transfer generally requires the solution of the Navier-Stokes and energy equations for the fluid side coupled with the energy equation for the solid side. Numerical methods for the solution of the conjugate problem with single-phase Fluids are well established. However, analysis of conjugate heat transfer involving boiling fluids represents a much more challenging problem. The problem is a moving-boundary problem mathematically. Two fluid phases are separated by a moving and deformable interface with both phases possibly impinging on a solid wall. The location of the phase interface is not known a priori and must be found as part of the solution procedure.

In present article, OpenFOAM code is developed to simulate conjugate heat transfer in boiling. The solver is validated with two simple test cases, and then it will apply on much more complex phenomena of film boiling.

Numerical Method
The equations that need to be solved in the boiling model are the conservations for mass, momentum, energy and volume fraction:

\[
\frac{\partial}{\partial t} (\rho U) + \nabla \cdot (\rho U U) = 0
\]

\[
\frac{\partial}{\partial t} (\rho U U) + \nabla \cdot (\rho U U U) = -\nabla p + \nabla \cdot (\rho U U) + \alpha \nabla \nabla \nabla \nabla
\]

\[
\frac{\partial}{\partial t} (\rho C T) + \nabla \cdot (\rho C T U) = \nabla \cdot (k VT) - m^* h_{fg}
\]

\[
\frac{\partial}{\partial t} (\rho L G) + U \cdot L G + \alpha L \nabla L G = - \frac{\rho L \nabla L G}{(\rho L - \rho G)}
\]

The volume fraction filed (αL) determines the liquid volume fraction in each cell.

\[
\alpha_L (x,t) = \frac{V_{vol} \rho \text{water}}{V_{vol} \rho \text{water} + V_{vol} \rho \text{gas}} = \begin{cases} 
1 & x \text{ liquid} \\
0 & x \text{ gas}
\end{cases}
\]

Therefore the physical properties of vapor and liquid such as: viscosity (μ), density (ρ), thermal conductivity (k), specific capacity (CP) are defined as:

\[ y = a_L y_L + (1 - a_L) y_G \]

where y = (μ, ρ, k, CP)
To close equations, an appropriate mass transfer model should be added. Here, Lee model for mass transfer is considered

\[ \dot{m} = r_e (1 - \alpha_r) \rho_e \frac{T - T_{sat}}{T_{sat}} \]  

(7)

Heat transfer equation in solid region is defined as:

\[ \frac{\partial}{\partial t} (\rho C_p T) = \nabla \cdot (k \nabla T) \]  

(8)

Boundary condition between solid region and fluid region is defines as:

\[ T_F = T_S \]  

(9)

\[ \frac{\partial T_S}{\partial n} = \frac{\partial T_F}{\partial n} \]  

(10)

which show the continuity of temperature field and heat flux on boundary.

In each time step, the following tasks are performed by the solver:

1- advection of VOF field
2- initial solution of velocity field
3- calculation of energy equation in fluid
4- solution of pressure-velocity coupling (PISO algorithm)
5- calculation of energy equation in solid

**Results and Discussion**

**Stephan Problem**

The one dimensional Stephan problem has become a benchmark for validation of phase change model [11], [15], [16], [14]. In this problem, a vapor film is generated beside super-heated wall and pushes away liquid from it. Interface is at the saturation temperature. “Figure 1” shows boundary conditions for this problem.

\[ \eta = \frac{\exp(\eta^2) \sqrt{\rho_e C_p L \eta}}{\sqrt{\pi h_{fg}}} \]  

(12)

The physical properties of Stephan problem is shown in “Table 1”. “Figure 2” compares interface position from simulation with analytical. In Stephan problem, interface is assumed that it is at saturation temperature, so a high diffuse liquid property has been chosen to damp any interface temperature deviation from saturation temperature.

**Conjugate Heat Transfer**

A simple conduction problem is presented to validate conjugate heat transfer. In Figure 3, a quiescent two phase flow placed next to the hot wall. This problem can be considered as serial heat resistances.
Film Boiling
Boiling of quiescent liquid near hot solid surface is called pool boiling. Some major regimes in pool boiling are convective heat transfer, nucleation boiling and film boiling. In film boiling, vapor layer separates saturated or sub-cooled liquid from the hot solid plate. Due to liquid higher density than vapor, Rayleigh-Taylor instability happens which amplifies small perturbation at interface and lead to bubble growth. The critical or the most unstable wavelength is given by:

$$\lambda_c = \frac{2\pi}{\sqrt{\frac{3\sigma}{g(\rho_L - \rho_v)}}}$$  \hspace{1cm} (14)

where $g$ is gravity. For simulation of this problem, computational domain with width of $\lambda_c$ and height of $3\lambda_c$ for fluid and $\lambda_c/2$ for solid are considered.

Thermo physical properties of film boiling problem are listed in Table 2. Solid bottom face has a fixed value temperature of 510 k. Fluid saturation temperature is 500 k. side faces are considered as symmetry plane. Top face has fixed value pressure and fluid-solid face has no slip condition.

### Table 2: film boiling simulation details

<table>
<thead>
<tr>
<th></th>
<th>Liquid</th>
<th>Gas</th>
<th>Solid</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (kg/m³)</td>
<td>200</td>
<td>5</td>
<td>1000</td>
</tr>
<tr>
<td>Viscosity (kg/ms)</td>
<td>5×10⁻⁴</td>
<td>1×10⁻³</td>
<td>-</td>
</tr>
<tr>
<td>Specific Heat (J/kgk)</td>
<td>400</td>
<td>200</td>
<td>490</td>
</tr>
<tr>
<td>Heat Conductivity (J/kgkm)</td>
<td>40</td>
<td>1</td>
<td>55</td>
</tr>
<tr>
<td>G (m/s²)</td>
<td>9.81</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Latent Heat (J/kg)</td>
<td>1×10⁴</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Surface tension (N/m)</td>
<td>0.1</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Solid temperature and volume of fluid distributions are shown in Figure 5. A mushroom shape of bubble in node and anti-node is reported here which is similar to other numerical simulations [11, 13, 16] in this field.

### Conclusions
In this study, a brief report of OpenFOAM code development was presented. Volume of fluid method in this code was extended to simulate boiling phenomena. Governing equations and algorithm was presented. New solver was validated with three problems. Stefan problem and conduction problem was presented to validated phase change phenomena and conjugate heat transfer, respectively. Film boiling was finally simulated and shape of bubble was compared with previous numerical studies. Future study must focus on the film boiling and the effect of heat transfer in solid on film boiling.

### References


