International Journal of Current Engineering and Technology
E-ISSN 2277 – 4106, P-ISSN 2347 - 5161

Available at http://inpressco.com/category/ijcet

Research Article

CFD Simulation of Subcooled Flow Boiling using OpenFOAM

Harikrishnan G[Å*], Ebin P S[Å] and Jayakumar J S[Å]

[Å]Department of Mechanical Engineering, Amrita Vishwa Vidyapeetham University, Amritapuri-690525, Kerala, India

Accepted 10 January 2014, Available online 01 February 2014, **Special Issue-2, (February 2014)**

Abstract

Subcooled flow boiling in vertical channels have many industrial applications, including heat removal from core of nuclears. In the present work, subcooled flow boiling is simulated using open source simulation software OpenFOAM. A new solver is developed based on the OpenFOAM framework to predict subcooled flow boiling by incorporating various flow boiling models. Vapor generation in the heated wall is modeled with a wall heat flux partitioning model. Additional source terms are added to the governing equations to account for the phase change process. Energy equation is implemented for the liquid phase. The solver is validated against benchmark calculation of the DEBORA subcooled boiling data.

Keywords: *subcooled boiling, CFD, VOF, OpenFOAM, bubbleFoam, Numerical.*

1. Introduction

Boiling is the phenomenon of phase change from liquid to vapor, occurs when a liquid is brought into contact with a surface maintained at a temperature sufficiently above the saturation temperature of the liquid. It is characterized by rapid formation of vapor bubbles at the solid liquid interface that detach from the surface when they attain a certain size and attempt to rise to the free surface of the liquid. As compared with single phase convection, boiling phenomena offers an additional heat transfer in the form of latent heat, and hence a higher heat transfer coefficient also.

The development in modern cooling systems aims at efficient, space and weight saving designs. The consequential demand for highest possible heat transfer rates has lead to very promising concept of providing for a controlled transition from pure single-phase convection to subcooled boiling flow in highly thermally loaded regions. The flow boiling process has wide industrial applications as it offers a higher heat transfer capacity at low wall temperature. Sub cooled flow boiling can be found in many practical applications, such as heat exchangers, steam generators, refrigeration systems and it is especially important in water- cooled nuclear power reactor. The capability to predict two-phase flow behaviors in forced convective sub cooled boiling flow is of considerable interest to nuclear reactor safety.

Due to its widespread applications, boiling heat transfer has been intensively studied in the past and still is subject of ongoing research activities in many groups all over the world. A better physical understanding of the boiling process can be achieved by either highly resolved generic boiling experiments or by numerical modeling and simulation. Recently, many researchers have done efforts to model flow boiling process in various conditions using different computational tools. Researchers are still going to improve the accuracy of prediction by modifying the numerical models.

Simulation of flow boiling in vertical pipes using CFX-5 CFD codes shows good agreement with experimental results (Koncar, *et al*, 2005). Boiling at walls is modeled with a wall heat flux partitioning model. The turbulence induced by the bubbles also taken into account in turbulence modeling. Many of the models suggested for wall boiling assume the total wall heat flux to be superimposed of additive components – single phase convective flux, quenching flux, evaporative flux. An alternate wall boiling is implemented (Steiner, *et al*, 2004), called Bubble Departure Lift-off (BDL) model. In this model, correction parameters are implemented for each heat flux additive term. The bubble diameter at departure is obtained from a force balancing on the lift off bubble. The overall agreement between the predictions of BDL model and the experimental data is very good.

Population balance equation combined with a 3-dimensional two-fluid model has employed to predict the boiling flows (Tu and Yeoh, 2003). The modeling has done in CFX-4.4, in which a MUSIG solver implemented is further modified to account for wall boiling and condensation in bulk fluid. A general population balance equation also employed to account for the bubble coalescence and break up.

To enhance the prediction capability of sub cooled flow boiling a new mechanistic bubble size mode is implemented (Yun, et al, 2012). In addition to this, a new

*Corresponding author: **Harikrishnan G**

wall boiling model also suggested. These models are implemented in STAR-CD 4.12 software. The bubble departure size is modeled with a force balance model. A new velocity wall function also included. To implement new S_Y model, a new transport equation is included. To validate the advanced solver, DEBORA experimental conditions are simulated using STAR-CD 4.1.

In the present work, a new solver is developed to predict the subcooled flow boiling within the OpenFOAM 2.1.1 framework. OpenFOAM is an open-source CFD software package built on the C++ programming language. The OpenFOAM package is built with many executables called solvers but none of them exactly predict the flow boiling phenomena. A base solver, *bubbleFoam* was selected and modified by incorporating various flow boiling models and energy equation. The new solver so developed is called *boilingFoam* and is capable to predict flow boiling as required. The governing equations in the *bubbleFoam* need to modify by implementing the wall boiling and bubble condensation models as well. The boiling at walls can be implemented according to wall heat flux partitioning model. The solver is based on several assumptions, notably of incompressible phases, saturated gas phase, spherical bubbles and no bubble-sliding.

2. Code validation

The algorithms implemented in OpenFOAM to solve Navier-Stokes equations are validated against the benchmark solution of Lid driven flows.

2.1 Natural Convection in a differentially heated square cavity

Solution of differentially heated lid driven cavity is taken up as the validation problem. This consists of two-dimensional flow in a square cavity of side L. Vertical walls of the cavity are different temperatures while horizontal walls are insulated. The results are compared against the benchmark results (Davis, 1983).

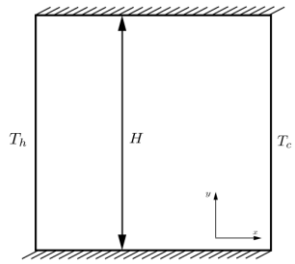

Fig.1 Square cavity with differentially heated walls

In this problem the vertical walls are at temperature T_h and T_c respectively and other two walls are insulated. Heat transfer from vertical walls results in density changes to the fluid in the cavity, and leads to buoyancy-driven recirculation. Simulations are carried out for Reyleigh numbers ranging from 10^3, 10^4 and 10^5.

2.1.1 Governing equations: In natural convection problems the driven force is given changes in fluid density due to temperature evolution. Even though the thermodynamic properties of the fluid are assumed to be constant, buoyancy body force term in momentum equation are added allowing relating density changes to temperature. This is achieved through the Boussinesq approximation that couples the energy and momentum equations. The governing equations for Newtonian incompressible fluid under laminar steady state flow can be written as in Equations

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \quad (1)$$

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\vartheta\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \quad (2)$$

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\vartheta\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right)+F_b \quad (3)$$

$$u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}=\vartheta\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right) \quad (4)$$

where u, v are velocities and T is temperature. F_b is the buoyancy force.

2.1.2 Solver : buoyantBoussinesqSimpleFoam
Steady-state solver for buoyant, turbulent flow of incompressible fluids

2.1.3 Results: The results are compared with benchmark solutions (Davis, 1983) for Rayleigh number from 10^3, 10^4 and 10^5 and is presented in Figs 2 to 4.

.

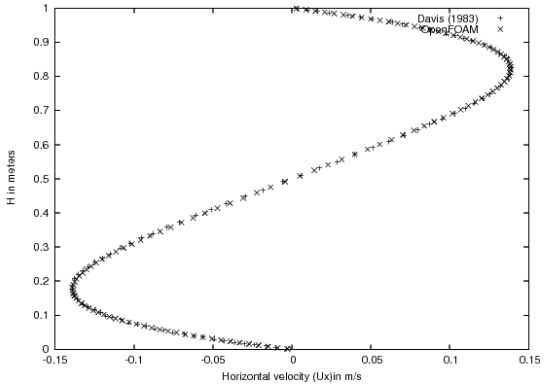

Fig.2 Horizontal velocity along vertical centerline for Re=10^3

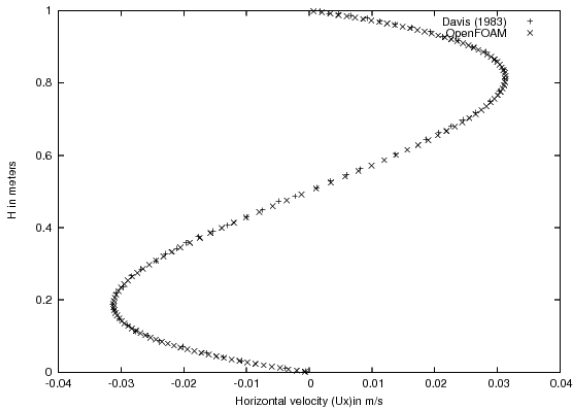

Fig.3 Horizontal velocity along vertical centerline for Re=10^4.

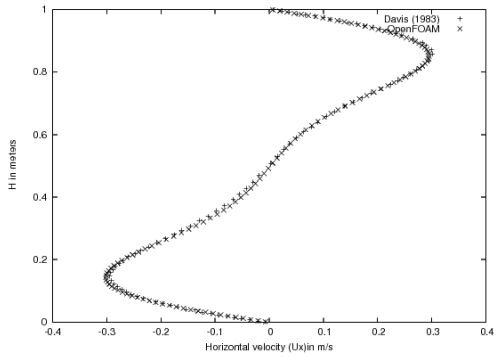

Fig.4 Horizontal velocity along vertical centerline for Re=10^5

Result shows good agreement with experimental values hence validate the algorithms.

2.2 Lid Driven flow in a 3-D Square Cavity

To validate algorithms for Navier-Stokes equations in 3D geometries, a lid driven 3D cavity flow is simulated. The top wall moves in the x-direction at a speed of 1 m/s while the other three are stationary. Initially, the flow is assumed incompressible laminar, isothermal flow. Figure 5 shows 3D cavity with a moving lid.

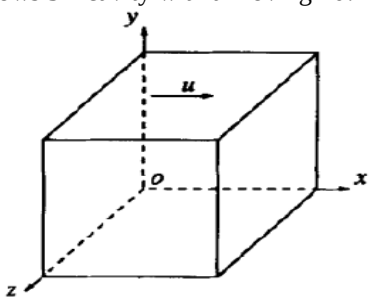

Fig.5 3D-square cavity with moving upper lid

2.2.1 Governing equations: The flow inside the cavity is governed by incompressible Navier-Stokes equation

$$\frac{\partial \vec{U}}{\partial t} + \nabla . U\vec{U} = -\frac{1}{\rho}\nabla p + \vartheta \nabla^2 \vec{U} \qquad (5)$$

2.2.2 Solver: icoFoam: Transient solver for incompressible, laminar flow of Newtonian fluids.

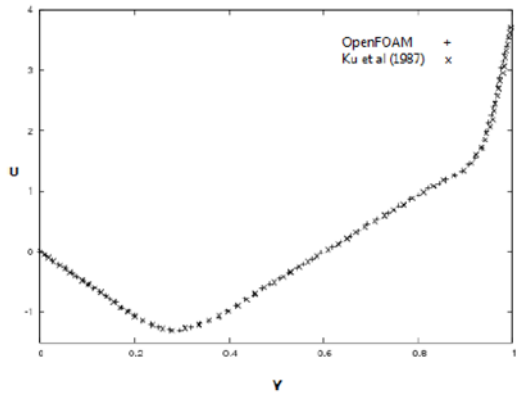

Fig.6 Variation of u along horizontal centerline

2.2.3 Results: The horizontal and vertical component of velocity is extracted along a vertical and horizontal center line to compare with benchmark solutions (Ku, *et al*, 1987)

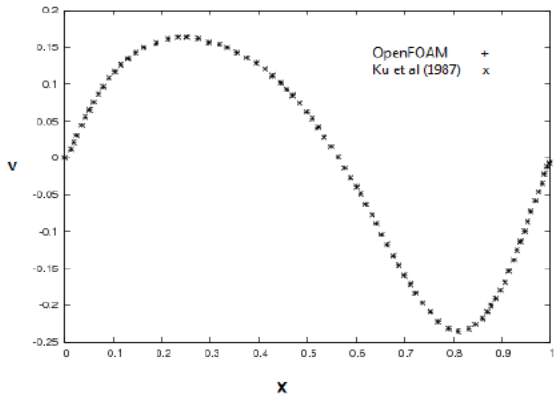

Fig.7 Variation of v along vertical centerline

Result shows good agreement with experimental values.

3. Simulation of subcooled flow boiling

In subcooled flow boiling, the bulk fluid temperature is below the saturation temperature of the liquid, but liquid close to the heated wall attains the saturation temperature and result in bubble formation in the wall regions. The bubble formed then detaches from the wall and move to the bulk liquid where it condense. To model wall boiling a heat flux partitioning is adopted in the near wall regions.

3.1 Governing equations

The codes are written in OpenFOAM framework to successively solve different conservation equations – mass, momentum, and energy equations – for an incompressible liquid-gas system.

Mass conservation equation

The continuous phase is liquid and dispersed phase is gaseous. The solver first solves the phase continuity equation given as:

$$\frac{\partial \alpha_k}{\partial t} + \nabla . \alpha_k U_k = \frac{\Gamma_{ki} - \Gamma_{ik}}{\rho_k} \qquad (6)$$

where, α and U represents the phase volume fraction, density and velocity respectively, k denotes the phase and can be either liquid (l) or gas (g), and i is the non-k phase. Hence Γ_{ki} denotes the evaporation rate especially due to bubble nucleation at the walls and Γ_{ik} is the condensation rate per unit volume.

Momentum conservation equation

The momentum equation for each phase is solved separately. The momentum equation for phase k is given as:

$$\frac{\partial(\alpha_k\vec{U}_k)}{\partial t} + \nabla.\alpha_k U_k\vec{U}_k = -\nabla.\left(\alpha_k(\bar{\tau}_k + \bar{\tau}_k^t)\right) - \frac{\alpha_k}{\rho_k}\vec{\nabla}p + \alpha_k\vec{g} + \frac{\vec{M}_k}{\rho_k} + \frac{\Gamma_{ki}U_i - \Gamma_{ik}U_k}{\rho_k} \quad (7)$$

The subscript *k* and *i* represent the two phases. The first term on the right hand side is the combined viscous and Reynolds stress. The second term accounts the pressure drop in the channel. Third term is gravity force, and $\overrightarrow{M_k}$ is the interfacial force per unit volume.

Energy conservation equation

Finally energy equation is solved for the liquid phase. The vapor phase is always assumed to be at saturated state, hence no need to solve energy equation for vapor phase. The energy conservation equation in terms of specific enthalpy can be written as:

$$\frac{\partial\left((1-\alpha)i_l\right)}{\partial t} + \nabla.(1-\alpha)i_l\vec{U}_l = -\frac{1}{\rho_l}\nabla.[(1-\alpha)(q_l + q_l^t)] + \frac{(1-\alpha)}{\rho_l}\frac{Dp}{Dt} + \frac{\Gamma_{lg}i_{g,sat} - \Gamma_{gl}i_l}{\rho_l} + \frac{q_w^{"}A_w^{"}}{\rho_l} \quad (8)$$

where q_l and q_l^t represent molecular and turbulent heat fluxes in the liquid phase. The third term denotes the phase change processes and last term implies the heat transferred from wall. $q_w^{"}$ is the wall heat flux density and $A_w^{"}$ is the wall contact area per unit volume.

3.2 Modeling of wall boiling

Boiling models are used to compute the rate of bubble formation and evaporation at the heated walls.

3.2.1 Wall heat flux partitioning: The wall boiling phenomena is modeled with the heat flux partitioning model (Krepper and Rzehak, 2011). Accordingly, the heat flux applied to the external wall is written as a sum of three heat fluxes as:

$$Q_{total} = Q_{conv} + Q_{quen} + Q_{evap} \quad (9)$$

where $Q_{conv}, Q_{quen}, Q_{evap}$ represents the convective, quenching and evaporative heat fluxes respectively. Each of these components are then modeled as a function of the wall temperature and local flow parameters. Then Equation 9 is solved iteratively for the local wall temperature, T_w, which satisfies wall heat flux partitioning. Each of the heat flux component can be modeled as given below.

The turbulent convection is modeled exactly as that of turbulent single phase convection as:

$$Q_{conv} = h_c(1 - A_w)(T_w - T_l) \quad (10)$$

where h_c is the convection heat transfer coefficient is calculated using:

$$h_c = \frac{\rho_l C_{pl} U_\tau}{T^+} \quad (11)$$

where U_τ is friction velocity and $T^+$is non dimensional temperature. The quenching heat flux is modeled by:

$$Q_{quen} = h_Q A_w(T_w - T_l) \quad (12)$$

where quenching heat transfer co efficient $h_Q$is given by:

$$h_Q = \frac{2}{\sqrt{\pi}} f\sqrt{t_{wait} k_l \rho_l C_{pl}} \quad (13)$$

f is bubble detachment frequency and t_{wait} is the waiting time between successive bubble detachment.

The evaporative heat flux is modeled as:

$$Q_{evap} = \frac{\pi}{6} d_w^3 \rho_g f N^{"} h_{fg} \quad (14)$$

$N^{"}$is the nucleation site density, $d_w$is the bubble detachment diameter and h_{fg}s the latent heat of vaporization.

Bubble detachment frequency

Bubble detachment frequency is modeled as:

$$f = \sqrt{\frac{4g(\rho_l - \rho_g)}{3C_d d_w \rho_l}} \quad (15)$$

Nucleation site density

Nucleation site density I modeled as:

$$N^{"} = N_{reff}\left(\frac{T_w - T_l}{\Delta T_{reff}}\right)^p \quad (16)$$

$N_{reff}$is taken as 0.8x10^6 m^{-2} and ΔT_{reff} as 10K.

Bubble size at detachment

Bubble detachment diameter is depends upon many factors heat flux, system pressure, liquid properties etc. The diameter can be modeled as:

$$d_w = d_{ref} e^{\frac{T_{sat} - T_l}{\Delta T_{reff}}} \quad (17)$$

d_{ref} is taken 0.6mm and ΔT_{reff} = 45K.

Local bubble diameter

To calculate the evaporation and condensation rate at bulk fluid and heat transfer rate, local bubble diameter is required. A linear model to predict local bubble diameter as a function of liquid sub cooling given as:

$$d_B = \frac{d_{B.1}(T_{sub} - T_{sub,2}) + d_{B,2}(T_{sub} - T_{sub,1})}{T_{sub,2} - T_{sub,1}} \quad (18)$$

where $d_{B,1} = 0.1mm$ at $T_{sub,1} = -13.5K$ and $d_{B,2} = 2mm$ at $T_{sub,2} = 5K$.

Bubble influence area

The wall area fraction influenced by vapor bubble is calculated by

$$A_w = \pi(a\frac{d_w}{2})^2 N^{"} \quad (19)$$

where a is the bubble influence factor, for which value 2 is assumed here. $A_w = 1$ indicate that the whole surface is covered with bubbles.

3. 3 Modeling of momentum transfer

Interfacial momentum transfer is modeled for interfacial forces which includes drag force, lift force, turbulent dispersion force and wall lubrication force. The interfacial drag force is calculated as:

$$\vec{F}_D = \frac{3}{4}\frac{C_D}{d_b}\alpha_g\rho_l|\vec{u}_g - \vec{u}_l|(\vec{u}_g - \vec{u}_l) \quad (20)$$

where C_d is the drag co efficient for bubble. Drag coefficient c_D is calculated as:

$$c_d = \frac{24}{Re}(1 + 0.1Re^{0.75}) \quad (21)$$

The lift force on liquid phase can be calculated by:

$$\vec{F}_L = \alpha C_l\rho_l(\vec{u}_g - \vec{u}_l) \times \nabla \times (\vec{u}_l) \quad (22)$$

where C_L is lift force coefficient acts in the direction of decreasing velocity, is given by Tomiyama (1998).

The effect of dispersion of the vapor bubbles due the turbulent eddies in the liquid phase is taken into account by the turbulent dispersion force.

$$\vec{F}_{TD} = -\frac{3}{4}\frac{C_D\mu_l}{d_B\sigma_t}(\vec{u}_g - \vec{u}_l)\frac{\nabla\alpha}{1-\alpha} \quad (23)$$

3. 4 Turbulence Modeling

In flow boiling, the dispersed phase moves along with the fluctuations in the continuous phase. Hence the turbulence stresses are modeled only for liquid phase. The liquid turbulence is calculated with $k-\varepsilon$ turbulence model, which belongs to the eddy viscosity turbulence model. In bubbly two-phase flows, an additional production of liquid turbulence generated by fluctuating wakes behind the large bubbles may occur. The so-called bubble-induced turbulence is taken into account by additional viscosity term, which is added to the molecular viscosity of the liquid phase. The effective liquid viscosity is given as:

$$\mu_l^{eff} = \mu_l + \mu_l^t + \mu_l^b \quad (24)$$

where μ_l is the liquid molecular viscosity, μ_l^t is turbulent viscosity and μ_l^b is the bubble induced turbulent viscosity. It can be modeled as:

$$\mu_l^b = C_{\mu b}\rho_l\alpha d_B|\vec{u}_g - \vec{u}_l| \quad (25)$$

Parameter $C_{\mu b}$usually takes a value 0.6.

3.5 Implementation of wall boiling model

The first stage of current work is to implement a new solver called *boilingFoam*, by modifying *bubbleFoam*. Major concern is given to the modeling of wall boiling, based on a wall heat flux partitioning. Equation (4) shows the heat flux balance in wall partitioning model.

During subcooled boiling flow, heat and mass exchange between the phases takes place on the heated wall and in the subcooled liquid flow. Here vapour bubbles are generated in the micro cavities commonly designated as nucleation sites, which are randomly distributed over the heated surface. The vapour bubble is generated on a nucleation site when the surface temperature sufficiently exceeds the liquid saturation temperature at the local pressure. These bubbles may slide along the walls, eventually depart from it, and migrate further into the liquid where it condenses.

At first it is needed to predict the wall temperature based on the heat flux applied. The equation (9) is solved iteratively to predict the wall temperature. Different heat fluxes in equation (9) can be replaced with corresponding correlations, and equation (9) becomes:

$$Q_{total} = h_c(1 - A_w)(T_w - T_l) + h_Q A_w(T_w - T_l) + \frac{\pi}{6}d_w^3\rho_g f N^{"} h_{fg} \quad (26)$$

where Q_{total} is the heat flux applied. Other parameters A_w, d_w, N'', f are calculated using the corresponding correlation. $T_l$is liquid temperature and T_w is unknown wall temperature.

Equation (26) is solved iteratively using bi-section method to obtain the unknown wall temperatureT_w. With these wall temperature various parameter such as nucleation site density, bubble departure diameter etc are recalculated and finally evaporative heat flux as given by equation (27). Then the mass of vapor generated at the wall can obtained as:

$$\Gamma_{lg} = \frac{Q_{evap}}{h_{fg}} \quad (27)$$

The wall boiling and bubble formation occurs in the near wall cells, Hence the above specified boiling model is applied to the near wall cells only. In the bulk these bubbles condenses as the liquid temperature is below saturation temperature. The condensation is modeled as:

$$\Gamma_{gl} = \frac{h_{LG}A_i(T_{sat} - T_l)}{h_{fg}} \quad (28)$$

where $h_{LG}$the interfacial heat transfer co is efficient calculated as:

$$h_{LG} = \frac{N_u K_b}{d_B} \quad (29)$$

$N_u$is the Nusselt number and $K_b$is the liquid thermal conductivity.

All these calculations are implemented in two header files, massCondence.H and wallBoiling.H and these are then added to the source file of the main solver.

massCondense.H: Models for bubble condensation in the bulk are implemented in this header file. Equation (23) is changed to OpenFOAM framework and implemented.

wallEvap.H: The wall boiling models including the wall heat flux partitioning are implemented in this header file. Equation (26) is solved iteratively to predict the local wall temperature and then equation (27) is solved to get rate of evaporation.

alphaEqn.H: The phase continuity equation (Equation (6)) is solved to obtain vapor phase fraction.

UEqn.H: Momentum equation is solved for both liquid and vapor phases. PIMPLE algorithm is selected for calculation of velocities and pressure.

HbEqn.H: Vapor phase is always assumed to be at saturated condition and hence energy equation is solved for liquid phase only. Equation (3) is implemented for liquid phase.

The geometry is a rectangular cavity of 0.01m x 0.01m x 1.5m. The geometry and meshes are generated with a utility called *blockMesh*.

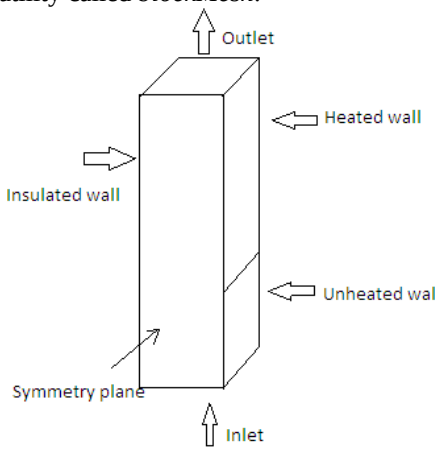

Fig.8 Outline of rectangular pipe

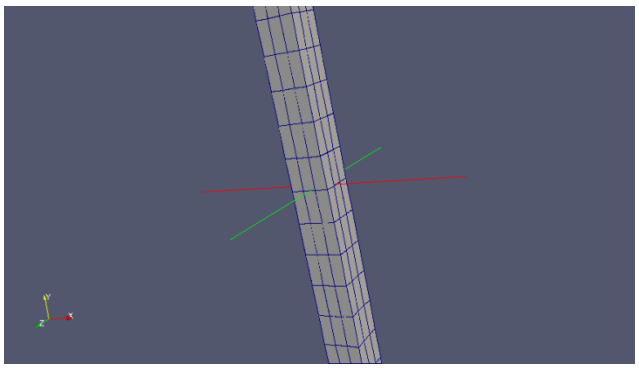

Fig.9 Rectangular pipe with grids

The case directory consist of three sub directories, *0*, *constant* and *system* sub directory. The *0* - sub directory includes the initial and boundary conditions for various fields. The constant sub directory includes a sub directory called *polyMesh* and three dictionaries – *transportProperties,g* and *RASproperties. polyMesh* sub directory hold all the geometrical information including the *blockMeshDict.* All of the property values are specified in *transportDictionary*, as the solver read these values as the code runs. The gravity constant is specified in *g*-dictionary. The *system* subdirectory consists of three dictionaries – *control*, *fvSoltions* and *fvSchemes.* Start time, end time and time interval are specified in *controlDict*. *fvScheme* dictionary holds the different schemes for the equation descretisation. Finally solver settings are included in *fvSolution* dictionary.

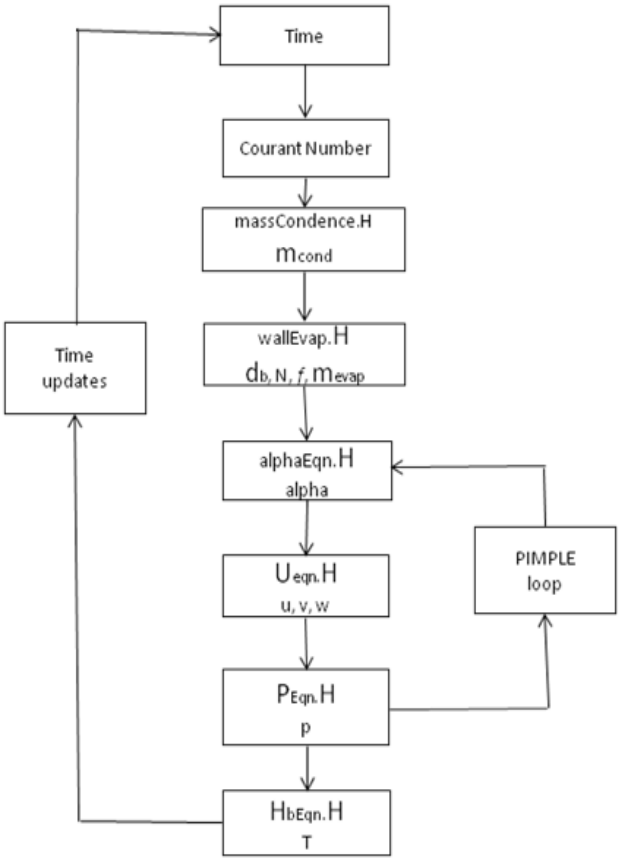

Fig.10 Flow chart of *boilingFoam*

4.0 Results

A new solver is developed in OpenFOAM framework to predict the subcooled flow boiling in vertical pipes. The new solver *boilingFoam* is a modified version of base solver *bubbleFoam*. The solver *bubbleFoam* is capable to solve for dispersed two-phase flows with strong density ratio, but it is not possible to model the heat transfer and mass transfer between the phases. Hence *bubbleFoam* is modified by incorporating various flow boiling models.
A wall boiling models is implemented to predict the boiling at the heated walls. Additional viscosity terms are added to the effective viscosity term, to account for bubble induced turbulence. Energy equation is implemented for the liquid phase. The vapor phase is assumed to be at saturated state.

After setting the initial and boundary conditions, iteration begins by entering command *boilingFoam* in the command prompt. Results are stored as time directories at specified times in the *controlDict*. Finally the results obtained can be visualized with *paraView*, a post processing tool for OpenFOAM. The distribution of various flow parameters are shown in figures 11 and 12.

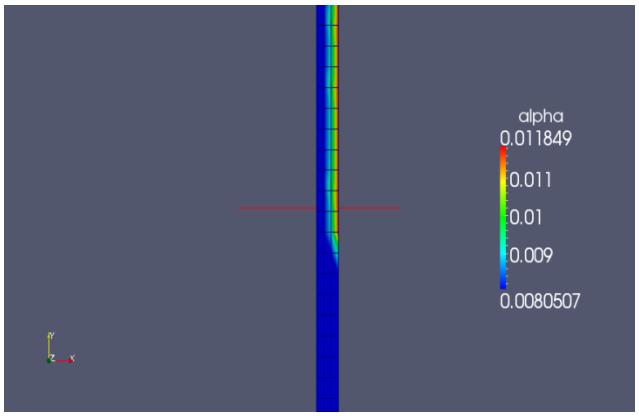

Fig.11 Distribution of field alpha (void fraction) at t=0.014sec

Vapor void fraction is high in regions close to the wall as compared with bulk liquid, because of the wall boiling phenomena.

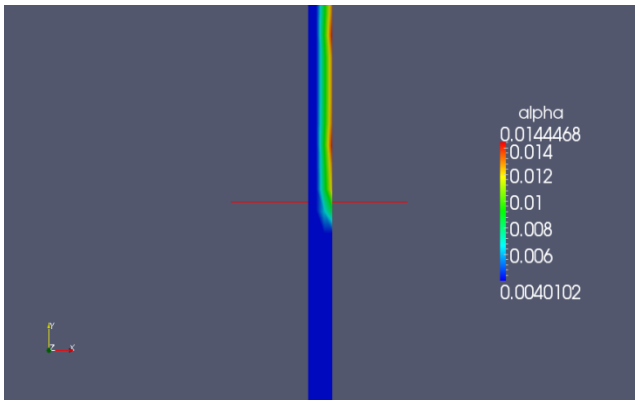

Fig 12 Distribution of field alpha (void fraction) at t=0.02sec

5.0 Conclusions

1) Wall boiling models are implemented to bubbleFoam to predict the evaporation at the heated walls
2) Additional viscosity terms are added to the effective viscosity, to account for bubble induced turbulence.
3) The base solver *bubbleFoam* is modified with new wall boiling and condensation models and a new solver *boilingFoam* which is capable to model heat and mass transfer between phases is developed.
4) Energy equation is implemented for the liquid phase.
5) The solver predicts the parameters correctly in regions close to the heated wall.

References

Davis G, (1983), Natural convection of air in a square cavity: a bench mark numerical solution, *International Journal of Numerical Methods in Fluids*, Vol. 3, pp. 249-264.

Ku.H.C, Hirsh.R.S, Taylor.T.D, (1987), A pseudospectral method for solution of the three dimensional unsteady Navier-stokes equations, *Journal of computational physics*. Vol.70, pp. 439-462.

Tomiyama, (1998), Struggle with computational bubble dynamics, *Third International Conference on Multi-phase Flow*.

Tu J Y, Yeoh G.H, (2001), On numerical modeling of low-pressure subcooled boiling flows, *International Journal of Heat and Mass Transfer* 45 (2002) 1197–1209

Steiner H, Kobor A and Gebhard L, (2004), A wall heat transfer model for subcooled boiling flow, *Proceedings of ASME-ZSIS International Thermal Science Seminar II*, Solvenia, June 13-16.

Konker B, krepper E and .Egorov, (2005), CFD modeling of subcooled flow boiling for nuclear engineering applications, *International Conference Nuclear Energy for New Europe*, Slovenia, September 5-8.

Krepper E, Rzehak R (2011), CFD for subcooled flow boiling: Simulation of DEBORA experiments, *Nuclear Engineering and Design*, 241, 3851– 3866.

Yuna B, Splawskib A, Simon Lob and Chul-Hwa Song (2012), *Prediction of a subcooled boiling flow with advanced two-phase flow models*, Nuclear Engineering and Design 253, 351–359.