Multi-Phase Flow Modeling to Simulate Boiling Phenomenon using CFD

Biradar Anilkumar M Assistant Professor Department of Mechanical Engineering Hirasugar Institute of Technology, Nidasoshi Belagavi Dist, Karnataka State Email- *biradaranilkumar@rediffmail.com*

Shivayogi. Salutagi Technology Specialist (CFD) Email- shiva.salutagi@gmail.com Niranjan M Ukkali Assistant Professor Department of Mechanical Engineering Hirasugar Institute of Technology, Nidasoshi Belagavi Dist, Karnataka State Email- niranjan.ukkali05@gmail.com

B.M.Dodamani Assistant Professor Department of Mechanical Engineering Hirasugar Institute of Technology, Nidasoshi Belagavi Dist, Karnataka State Email- bhimanagouddodamani@gmail.com

ABSTRACT

Thermal systems design plays an important role in design of Heat exchanger, Thermal simulation models, Heat transfer equipments, Boiling and Condensation devices, Hence modeling of thermal systems can be done by using various Analystic and software packages. In that context Computational Fluid Dynamics (CFD) is one of the key role. In this paper discusses about Multi-Phase flow modeling to simulate the phenomenon of Boiling using CFD tool and an experimental study of subcooled flow boiling is costly and time consuming. Computational fluid dynamics (CFD) is a cost effective and accurate alternative to model testing, with variations on the simulation being performed quickly. The objective of this project is to simulate and validate boiling and two phase flow in CFX-5.7.1 for high pressure with implementing wall boiling model for two experimental cases in the literature. This analysis includes phase change mechanisms and interphase momentum, heat and mass transfer processes. The RPI model is used for distributing the heat flux for heating the subcooled liquid and for vapor generation. The effect of various drag and non-drag forces is considered, which gives different modeling approaches. This paper also highlights the effects of various parameters on vapor volume fraction and different bubble diameters are also investigated. The flow parameters like velocity, temperature and vapor volume fractions are predicted in CFX-5.7.1. These results are compared and validated with numerical and experimental data from earlier research carried out by various researchers. The successful validation of this model, the flow modeling of inbed evaporator tubes of fluidized bed combustion is performed. The CFX results are compared and validated with "Babcock and Wilcox" circulation program. With the distribution of vapor volume fraction and velocity profiles, the flow separation and localized high vapor volume fraction can be identified with various geometries. This analysis will be helpful for further improvement.

Keywords—CFD, Mutli-phase flow, Modelling of Thermal systems, Flow visualization

I. INTRODUCTION

Computational Fluid Dynamics (CFD) is a computer-based tool for simulating the behavior of systems involving fluid flow, heat transfer, and other related physical processes. It works by solving the equations of fluid flow (in a special form) over a region of interest, with specified (known) conditions on the boundary of that region. Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design timescales and improve processes throughout the engineering world. CFD provides a cost-effective and accurate alternative to scale model testing, with variations on the simulation being performed quickly, offering obvious advantages.

The set of equations which describe the processes of momentum, heat and mass transfer are known as the Navier-Stokes equations. These partial differential equations can be discretized and solved numerically. The following paragraph explains the procedure to analyze the problem by CFD in brief.

The geometry of the region of interest is defined. If the geometry already exists in CAD, it can be imported directly. The mesh is then created. After importing the mesh into the pre-processor, other elements of the simulation including the boundary conditions (inlets, outlets etc.) and fluid properties are defined. The flow solver is run to produce a file of results, which contain the variation of velocity, pressure and any other variables throughout the region of interest. The results can be visualized and provide the engineer an understanding of the behavior of the fluid throughout the region of interest. This can lead to design modifications, which can be tested by changing the geometry of the CFD model and seeing the effect. The process of performing a single CFD simulation is split into four components

II. LITERATURE REVIEW

An important distinction in single-phase flow is whether the flow is laminar or turbulent, or whether separation flow or secondary flows exist. This information helps in modeling specific phenomena because one has an indication of the flow character for a particular geometry. Analogously in multiphase flow probably the key toward understanding the phenomena is the ability to identify the internal geometry of the flow; i.e., the relative location of interfaces between the phases, how they are affected by pressure, flow, heat flux and channel geometry, and how transitions between the flow patterns occur.



Fig 2.1 Vertical annual flow

Fig 2.2 Co-current flow

There are two fundamental types of flow patterns, one can identify, stratified and dispersed. A stratified flow pattern is one in which the two phases are separated by a continuous interface at a length scale comparable to the external scale of the flow; e.g., a liquid film on a wall with a gas or another immiscible liquid in the center of the channel. The complete separation of the two phases usually occurs due to density differences (horizontal flow) combined with a relatively low mass flow rate of the phase near the wall compared to the other phase in the center of the channel (e.g., vertical annular flow in Fig. 2.1, 2.2). These separated flow patterns can occur when the phases flow in the same direction (co-current flow) or in opposite directions (counter-current flow). The transition between these two types of stratified flow is governed by the balance between buoyancy and inertial forces

A dispersed flow pattern is one in which one or more phases are uniformly dispersed within a continuum of another phase with a length scale much smaller than the external scale; e.g., gas bubbles or solid particles in a liquid or liquid droplet in a gas or another immiscible liquid in Fig. 2.1, 2.2. In this case the dispersed phase forms into nearly regular shaped particles with their stable size-governed again by a balance of buoyancy, inertial and surface tension forces. The transitional flow regimes between these two fundamental types can take on much geometry. Some of the more common transitional flow patterns are churn-turbulent and slug flow; i.e., dispersed-stratified flows where the discontinuous phase begins to form a continuum near the wall (bubbly-film) or in the center of the channel (wispy-annular). These flow patterns will be discussed in more detail of flow boiling. [1]

Boiling is the process in which a liquid evaporates and forms vapor pockets or regions within the continuous liquid phase. Boiling can take many forms. Consider the common everyday occurrence of a pot of boiling water on top of the stove. In this case a stagnant pool of liquid is heated and boiling occurs in the liquid at the bottom of the bulk liquid pool. This overall process is called pool boiling. To form the vapor phase within

the continuous liquid phase one must heat the liquid to a temperature above its saturation temperature, T_{sat} . If one were to look at the bottom of the pot of water on the stove one would notice the vapor bubbles nucleating at this heater surface. This nucleation occurs within preferred-sites or crevices within the heater surface aided by trapped vapor and gas. These vapor bubbles will move up from heated surface due to buoyancy effects and fresh liquid comes in contact again with heated surface. Likewise liquid temperature increases and more vapor will be generated. In pool boiling, the phases will not in bulk flow. [2]

III. MATHEMATICAL FORMULATION OF THERMAL SYSTEMS

3.1 Flow Equations

The two-fluid model of sub cooled nucleate boiling flow consists of a dispersed phase (vapor bubbles) and a continuous phase (liquid) and is based on two sets of averaged transport equations. At averaging, the so-called "interpenetrating continua" approach is used, where each phase is treated as a continuum that fills up the entire control volume and is described by its own system of averaged equations for mass, momentum and energy. Averaged equations for both the phases are coupled with additional closure relations describing the exchange of mass, momentum and energy at the interface as well as the turbulence within each phase.

These equations which govern each phase are as follows. [3] *3.1.1 Continuity equation of phase k*

$$\frac{\partial(\rho_k \alpha_k)}{\partial t} + \nabla \cdot (\rho_k \alpha_k U_k) = \sum_{j=1, j \neq k}^{N_P} (\Gamma_{kj} - \Gamma_{jk})$$
(3.1)

3.1.2 Momentum equation of phase k

$$\frac{\partial(\rho_k \alpha_k U_k)}{\partial t} + \nabla \cdot (\rho_k \alpha_k U_k U_k) = \nabla \cdot [\alpha_k \mu_k^e (\nabla U_k + (\nabla U_k)^T)] - \alpha_k \nabla P_k + \alpha_k \rho_k g + \sum_{j=1, j \neq k}^{N_p} (\Gamma_{kj} U_j - \Gamma_{jk} U_k) + \sum_{j=1, j \neq k}^{N_p} F_{kj}$$
(3.2)

3.1.3 Energy equation of phase k

$$\frac{\partial(\rho_k \alpha_k H_k)}{\partial t} + \nabla \cdot (\rho_k \alpha_k U_k H_k - \alpha_k \lambda_k^e \nabla T_k) = Q_k + \sum_{j=1, j \neq k}^{N_p} (\Gamma_{kj} H_j - \Gamma_{jk} H_k) + \sum_{j=1, j \neq k}^{N_p} E_{kj}$$
(3.3)

Inter-phase transfer terms in the momentum and energy equations Γ_{kj} and F_{kj} denote the mass and momentum transfer terms from phase j to k.

3.2 Inter-Phase Momentum Transfer:

The momentum exchange between an ideal bubble and the surrounding fluid is induced by interfacial forces acting on the bubble, as well as mass transfer during evaporation and condensation and the non-drag forces. Whereas the additional momentum exchange during phase change always depends on mass transfer. The momentum exchange through mass transfer in a flow field is taken into account through additional sources and sinks.

The inter-phase transfer of momentum [14] is modeled with the interfacial forces, which include drag force (F_{lg}^{d}), virtual mass force (F_{lg}^{vm}), lift force (F_{lg}^{L}), turbulent dispersion force (F_{lg}^{TD}) and wall lubrication force (F_{lg}^{LW}). Generally the total interfacial force is usually expressed in terms of several effects as follows.

$$F_{\rm lg} = F_{\rm lg}^{d} + F_{\rm lg}^{vm} + F_{\rm lg}^{L} + F_{\rm lg}^{TD} + F_{\rm lg}^{LW}$$
(3.4)

3.2.1 Drag Force

The **drag force** (Ishii and Zuber, 1979) [5] exerted on the gas phase bubbles is a vector with a direction of the relative velocity of the gas phase can be expressed as:

$$F_{\rm lg}^{d} = -F_{gl}^{d} = \frac{1}{8} \rho_l A_{\rm lg} C_D | U_g - U_l | (U_g - U_l)$$
(3.5)

For a bubble of a given shape, undergoing motion in a Newtonian incompressible fluid, C_D depends only on bubble Reynold no as follows.

$$\operatorname{Re}_{b} = \frac{\rho_{l} |U_{g} - U_{l}| d_{b}}{\mu_{l}}$$
(3.6)

The drag force coefficient (C_D) is dependent on the flow regime (i. e Re_b) is correlated from Ishii and Zuber [4] as follows.

$$C_{D} = 24 \frac{1 + 0.1 \operatorname{Re}_{b}^{0.75}}{\operatorname{Re}_{b}}, \quad \text{for } 0 < \alpha_{g} \le 0.10$$

$$C_{D} = \frac{2}{3} d_{b} \sqrt{\frac{g \Delta \rho}{\sigma}} \left[\frac{1 + 17.67(1 - \alpha_{g})^{1.238}}{18.67(1 - \alpha_{g})^{1.5}} \right]^{2}, \text{ for } 0.10 < \alpha_{g} \le 0.25$$

$$C_{D} = 9.8(1 - \alpha_{g}), \quad \text{for } 0.25 < \alpha_{g}$$
(3.7)

The interfacial area density can be determined for equal-sized spherical bubbles as

$$A_{\rm lg} = \frac{6\alpha_g}{d_b}$$
(3.8)

The eq. (3.8) is valid only for a low volume fraction of gas phase, when distortion free, spherical bubbles are encountered.

IV. MODEL PREPARATION USING CFX PLATFORM

4.1 Geometry details and mesh

Simulation of subcooled flow boiling is performed in six-heated rod bundle using CFX-5.7.1. The figure 4.1 shows the computational domain with test section. The testing has been done for one tenth of the cross section, keeping the periodic symmetry of the bundle. Since the present model is only applicable to two-phase bubbly flow, only 2 m of the section is modeled. The wall region has been modeled to capture turbulence adjacent to the wall with $y^+ = 100$. The figure 4.2 shows the hexahedral mesh used in the simulation. The first prism height is calculated using mesh calculator as follows.

Inputs: density (ρ) = 778.39 kg/m³, velocity (V) = 1.4285 m/s,

Running length (L) = 2 m, viscosity (μ) = 1.0 e-4 Pa.s, required y⁺ = 100

Output: Re =
$$\frac{\rho VL}{\mu}$$
 = 2.22 e7,
First prism height = $\frac{\sqrt{80} \cdot y^+ \cdot L}{\text{Re}^{\frac{13}{14}}}$ = 0.269 mm

The input and operating conditions are given in table 4.1.

Table 4.1: Data of input and operating conditions (Nylund case)

\dot{q} (KW/m ²)	P (bar)	\dot{m} (kg/m ² s)	$T_{s}(K)$	ΔT subcooling(K)
540	49.7	1112	536.54	3.5

The above table shows the various properties of sub cooling parameters of the sample with respective to pressure mass flow rate and Temperature



Fig. 4.1: Hexahedral mesh used in simulation



Fig. 4.3: Liquid temperature plot



Fig. 4.2: Test section with six heated rods



Fig. 4.4: Vapor volume fraction plots on the c/s

V. CONCLUSIONS

- This exhaustive study determines three-dimensional two-fluid model for each vapor phase and liquid phase is considered separately. The inter-phase heat and mass transfer processes are coupled with two-fluid model. The RPI model is used to distribute the heat flux for heating the sub cooled liquid and for vapor generation.
- The void fraction, velocity, and temperature profiles for sub cooled flow boiling of water are estimated for two experimental cases in the literature. These results are compared with experimental and numerical data in the literature and it is found that present RPI model of CFX shows good agreement.
- The effect of various drag and non-drag forces is tested along with the effect of various parameters on vapor volume fraction. Different bubble diameter correlations are tested and suitable correlation is used for simulation.
- It is observed that sub cooled water enters the test section and subsequently boils due to the uniform heating from the heated wall. Vapor bubbles will form adjacent to the heated wall and then move towards the centre.
- The liquid temperature increases in the domain due to liquid heating and remains constant after saturation is reached. It is found that the void fraction increases along the length of channel. The liquid velocity increases as the vapor generation increases along the flow direction. The vapor phase moves faster than liquid phase due to low density and low viscosity than continuous liquid.

- The calculation including non-drag forces gives better results. The present study shows good agreement with experimental data than Anglart-Nylund's study.
- It is seen that the lift force pushes the bubbles towards the low velocity region and wall lubrication force pushes the bubbles away from the wall. Due to these non-drag forces, radial void fraction distribution is closer to experimental data. The void fraction peak occurs nearer to the heated wall and then decreases from heated wall to outer wall.
- After validation of this model, the flow modeling of in bed evaporator tubes is performed. The CFX results are compared and validated with B & W circulation program.
- With the distribution of vapor volume fraction and velocity profiles, it is found that there are high velocity and high vapor formation or flow separation regions near internal section of the bends of the tubes, which may be DNB effect due to local overheating. To reduce these effects, new tube profile is proposed.

REFERENCES

- [1] John G. Collier, "Convective Boiling and Condensation", 3rd edition, Atomic Energy Research Establishment, Harwell.
- [2] "Steam, its Generation and Use", 40th edition, The Babcock and Wilcox Company, Barberton, Ohio, USA, 1992.
- [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, 163(1996), pp. 81-98.
- [4] M. Ishii and N. Zuber, "Drag Coefficient and Relative Velocity in Bubbly, Droplet or Particulate flows", AIChE Journal, Vol. 25, pp.843-853 (1979).