*International Journal For Technological Research In Engineering*
*Volume 10, Issue 3, November-2022* *ISSN (Online): 2347 - 4718*

# OPTIMIZATION OF BOILING PHENOMENON OF WATER USING FLUENT BY CFD SIMULATION

[1]Pragati Parmar, [2]Dr. Rahul Kumar Singh
[1]M.tech scholar, [2]HOD & Assistant Professor
Rabindranath Tagore University, Bhopal

***Abstract: The nucleate boiling is the current promising way for heat transfer and for calculating the critical heat flux. The understanding of the thermal and hydrodynamic features of the flow plays a fundamental role in this study. The current experimental techniques are still inadequate to capture the small scales involved in the flow, while the recent advances in the multiphase CFD techniques provide innovative tools to investigate the two-phase flow. However, the scientific literature concerning with numerical modeling of flow boiling patterns is still poor, such that several aspects of the flow are not clarified yet. In order to save lots of procedure time, the flow is sculptured with an axis y metrical formulation in fluent Ansys 14.5.The purpose of this study is to demonstrate the modeling of forced convection sub-cooled nucleate boiling using the in-built boiling model available under Eulerian multiphase model. The exact position of bubbles volume fraction is obtained by considering all these parameters. This will make easier the estimation of volume fraction at x=0.1,0.2,0.3......n points. This method reduces the calculating time for critical flux which makes very industrial work easier.***

***Keywords: Computer Fluid Dynamics, Nucleate Boiling, Simulation***

## 1. INTRODUCTION

When boiling happens on a solid surface at low superheat bubbles will be seen to make repeatedly at most popular positions known as nucleation sites. Nucleate boiling will occur in Pool Boiling and in Forced-Convective Boiling. The heat transfer coefficients area unit is very high however despite a few years of analysis empirical correlations for the coefficients have massive error bands. A lot of problem arises from the sensitivity of nucleate boiling to the micro geometry of the surface on a micro meter length scale and to its wettability it is difficult to find out applicable ways in which of quantifying these characteristics. There's still disagreement regarding the physical mechanisms by which the heat is transferred thus phenomenological models for nucleate boiling at the present do no good and sometimes even worse than the empirical correlations. The empirical correlation of wide application has been given by Gorenflo (1991) supported the final scaling of fluid thermal and transport properties with reduced pressure p/pc and reduced temperature T/Tc. Recent reviews of the voluminous analysis literature on mechanisms in boiling given by Dhir (1990) and Fujita (1992). This further describes the options of nucleate boiling on that there's broad agreement and indicates the areas of disagreement and more development.[1]

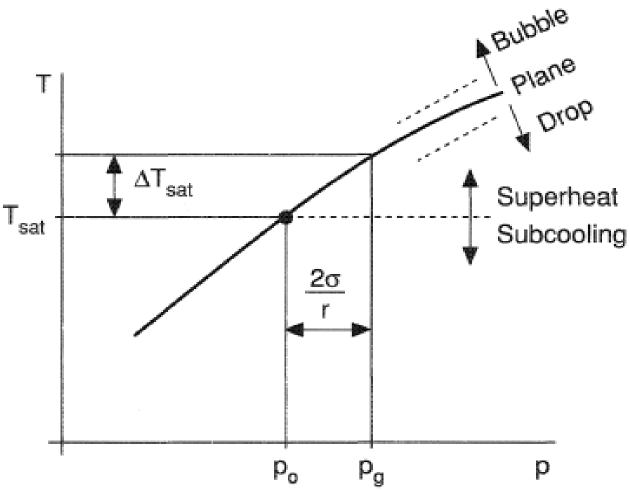

**Fig - Equilibrium at plane and curved interfaces.**

A spherical bubble of finite radius r has an interface of curvature 2/r and this has two effects:

(1) For mechanical equilibrium of the bubble interface there should be an excess internal pressure of $2\sigma/r_e$ to resist the collapsing membrane stress caused by the physical phenomenon σ

(2) The pressure for a given surface temperature is (Kelvin equation)

$$\frac{P}{Psat} = exp(\frac{-2\sigma vM}{RT}) \tag{1.1}$$

There is an identical result with exponent of opposite sign for the vapor pressure in equilibrium with a drop of liquid.

**MECHANISM OF BUBBLE GROWTH:-** The mechanism of growth of a bubble in uniformly superheated liquid represented antecedently is changed once nucleation happens at a solid wall. Growth as an ideal hemisphere (Fig1.7a) is prevented by the issue of displacing liquid from the the solid boundary so a micro layer of liquid is left beneath the bottom of the bubble (Fig1.7b). The curvature at the edge of the bubble depends on the native viscous and mechanical phenomenon stresses. It's generally sharp enough to offer the looks of a contact angle between the bubble and also the wall however there's no triple contact line therefore the properties of the wall will exert no influence. The density of the micro layer at the bubble boundary will be calculable from viscous physical phenomenon theory while not elaborate thought of the bubble form. Because it grows the bubble displaces liquid thus by the time it reaches some extent at distance R from the nucleation web site in time t the liquid at R has been in motion for time t and also the physical phenomenon of slow liquid that's overtaken by the bubble is of thickness $\delta_{Ro}$, where[10]

$$\delta_{Ro} - \sqrt{\nu_1 t} \qquad (1.5)$$

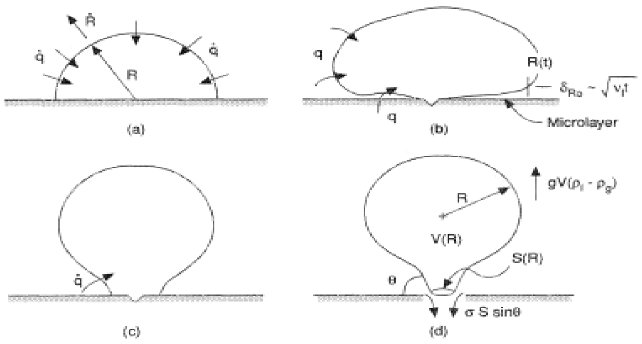

Fig : Bubble growth and detachment.

**LATERAL CONDUCTION:-** Above discussed mechanism are focused around the nucleation sites and fluctuate as bubbles grow and depart thus there should be some unsteady lateral physical phenomenon of warmth within the wall, (Fig1.9). A wall manufactured from a fabric with infinite thermal diffusivity will have a homogeneous steady superheat. In pool boiling of water at low heat fluxes such measurements make sure that there's sturdy cooling by micro layer evaporation ( Equilibrium at plane and curved interfaces) they show that mechanisms (Unstable equilibrium and growth of a bubble nucleus) and (contact angle) are less effective than the transient physical phenomenon 'quenching' model suggests which they care for a wall space of influence that's no larger than the utmost projected areas of the bubbles (Fig1.10) the final level of convective cooling is many times the extent expected for single section convection.

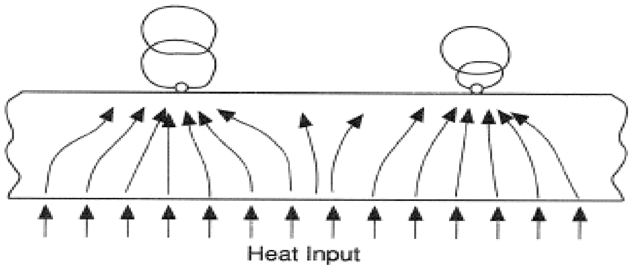

Fig1. 9. Lateral conduction in the wall.[18]

## 2. LITERATURE REVIEW

**1.** Zhihao Chen, Yoshio Utaka.**[52]** We have antecedently discovered and through an experiment measured a micro layer shaped to a lower place a growing bubble throughout nucleate pool boiling. The initial micro layer thickness was of micrometer order and augmented linearly with distance from the bubble beginning website. The quantitative degree of contribution of the micro layer evaporation to bubble growth was still not elucidated though an out sized variety of experimental studies are conducted on the distribution and evaporation characteristics of the micro layer.

**2.** E.Sattari,M.A.Delavar,E.Fattahi,K.Sedighi**[53]**In the given paper the mixture of three-dimensional equal and two-dimensional non-isothermal Lattice Boltzman methodology (LBM) unit wont to simulate the nucleate pool boiling development. So on validate the projected model rising bubble development is simulated later the boiling technique is investigated by using a operate for heat transfer. The investigation is compared with altogether totally different numerical works and is found to be in superb agreement. The implications of the parameters nonetheless as contact angle heat flux and the length on departure dimensionless time and the diameter of bubble unit considered.

**3.** Kazuo Ikeda**[54]** Pressurized water reactor fuels are developed to fulfill the requirements of the market. A spacer grid may be a key element to boost thermal hydraulic performance of a water-cooled reactor fuel assembly. Combining structures (vanes) of a spacer grid promote fluid combining and enhance heat removal from fuel rods. A bigger combining vane would improve combining result which might increase the departure from nucleate boiling profit for fuel. However the raised pressure loss at massive combining vanes would scale back the fluid flow at the mixed fuel core which might cut back the margin.

**4.** D.Bestion**[55]** System thermal hydraulic codes model all two-phase flow regimes but they are restricted to an out sized description. Two-phase CFD tools predict two-phase flow with the way finer space resolution but the current modelling capabilities unit of measurement restricted to unfold bubbly or drop flow and separate-phase flow. Plenty of less experience exists on plenty of difficult flow regimes that blend the existence of unfold fields with the presence of big interfaces sort of a free surface or a moving-picture show surface. A list of come-at-able reactor issues that might show pride in associate in nursing all flow regime CFD model is given. The first downside is to identify the numerous styles of native flow configuration. It's shown that a 4-field model has much better capabilities than a two-fluid approach to identify most complex regimes.

**5.** Sina Nabati Shoghl, Masoud Bahrami, Mostafa Keshavarz **[56]** This work investigated the dynamics of bubbles in pool boiling of nano fluid with coated and number eleven dodecyl salt (SDS) resolution with utterly totally different nano particles. Also CFD module was used for mathematical modeling of bubbles in pure water boiling. Different macroscale parameters such as shapes, numbers and build contact with angle of bubbles to boot were investigated through an experiment and verified by CFD modeling results. Porous layers of nano-particles on stainless steel substrate in conjunction with SDS additions were shown to change formation of bubbles compared to reference condition.

## 3. PROBLEM DESCRIPTION

In this study we will consider upward, vertical flow in a pipe with a heated wall. The flow domain is shown schematically in Figure 3. 1. The pipe is 20 mm in diameter (inner) and 2.0 m in length. The wall provides heat to the fluid at the rate of 345.6 kW/m². As the wall temperature rises above the fluid saturation temperature, steam bubbles are formed and they migrate away from the wall. Since the bulk flow is sub cooled, the bubbles condense near the center of the pipe. Outlet profiles of velocity magnitude and turbulence quantities generated for a simulated flow field without boiling will be used as inlet information to the boiling simulation. This is done to ensure a fully-developed profile of these quantities at the inlet.

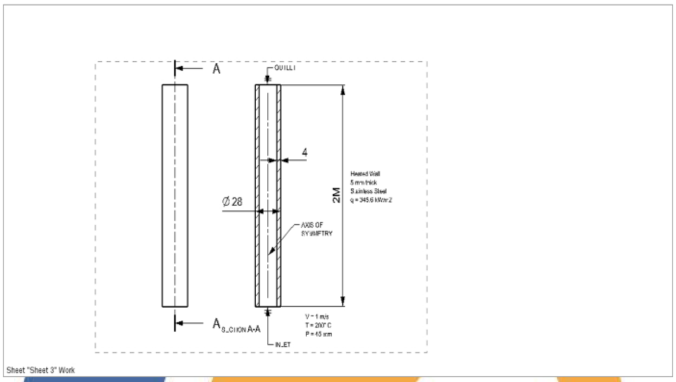

Figure 3.1: Schematic of the Problem

## 4. METHODOLOGY

Computational fluid dynamics is a computer-based simulation method for analysing fluid flow, heat transfer and related phenomena such as chemical reactions. This project uses CFD for analysis of flow and heat transfer. Some examples of application areas are: aerodynamic lift and drag,power plant combustion, chemical processes, heating/ventilation and even biomedical engineering (simulating blood flow through arteries and veins). CFD analyses carried out in the various industries are used in R&D and manufacture of aircraft, combustion engines as well as many other industrial products.

It can be advantageous to use CFD over traditional experimental-based analyses, since experiments have a cost directly proportional to the number of configurations desired for testing unlike with CFD, where large amounts of results can be produced at practically no added expense. In this way, parametric studies to optimise equipment are very inexpensive with CFD when compared to experiments.

This section briefly describes the general concepts and theory related to using CFD to analyse fluid flow and heat transfer as relevant to this project. It begins with a review of the tools needed for carrying out the CFD analyses and the processes required followed by a summary of the governing equations and turbulence models and finally a discussion of the discretisation schemes and solution algorithms is presented.

### CFD COMPUTATIONAL TOOLS

This section describes the CFD tools required for carrying out a simulation and the process one follows in order to solve a problem using CFD. The hardware required and the three main elements of processing CFD simulations: pre-processor, processor, and post-processor.

There is a variety of commercial CFD software available such as Fluent, Ansys CFX, ACE, as well as a wide range of suitable hardware and associated costs depending on the complexity of the mesh and size of the calculations. Commercial CFD packages can cost up to about $20000 (US Dollars) per year for licenses, maintenance, and support. Complicated transient cases with fine meshes will require more powerful computer processors and RAM than simpler cases with rough meshes. A typical engineering workstation (i.e. 32 GB processing RAM with quad processors) at a cost of approximately $3000-$5000 (US Dollars) or a combination of several processors running in parallel is probably the minimum investment needed to get started.

One of the purposes of this project is to use all open-source CFD software instead of commercial software for the simulations. This type of software is advantageous for smaller companies to use, as the cost of commercial CFD package licenses can be prohibitive.

### PROBLEM SOLVING WITH CFD

There are many decisions to be made before setting up the problem in the CFD code. Some of the decisions to be made can include: whether the problem should be 2D or 3D, which type of boundary conditions to use, whether or not to calculate pressure/temperature variations based on the air flow density which turbulence model to use, etc. The assumptions made should be reduced to a level as simple as possible yet still retaining the most important features of the problem to be solved in order to reach an accurate solution.

## 5. RESULT AND DISCUSSION

After doing CFD Analysis of Nucleate Boiling process using Ansys Fluent,then we are getting the following result. For obtaining the required result we have to follow some following steps:

1.Set up and solution for 1st single phase flow:phase is liquid water(L)

2. We use double precision Ansys fluent 14.5.

3.Mesh generartion:We use hexa mesh.

4. Models:(i)Energy equation is on.

(ii)**k-epslion** where k-turbulence kinetic energy epslion-turb disspitation rate

5.Material:We use water liquid $H_2O$( Liquid) in this model density as piecewise linear profile of temperature.

6. Set point **1-Temp**-473.15K
**Density**-864.7kg/m³
Set point **2.-Temp** -543.15K
**Density**-770.6kg/m³

7.Cell zone condition: Fluid(water liquid).

**Boundary conditions:**

8.1.**Inlet Type** :Normal to boundary as the velocity specification method

(a)Magnitude-1m/s (b)Turbulent intensity-4%,(c)Hydraulic dia-.0154 (d)Thermal condition is 473.15K at the inlet.

8.2.**OuletType:** Gauge Pressure-0atm **Type**-Normal to boundary

(a) Turbulent intensity-4% (b) hydraulic diameter -.0154 3 (c) Thermal condition is 530.55 to back flow temperature.

9.Operating condition-Pressure-4.5e^6 and Gravity-9.81m/s2
10.Convergence criteria:For continuity 1e^-8 for all the remaining residual we use 1e^5
11.Now intialise the solution .
12.Run the calculation for 500 iteration.

We see in the process the solution convergences in 320 iteration approx.

| **Physical property of liquid** | **Temp(T)** | **Value** | **Temp(T)** | **Value** |
|---|---|---|---|---|
| Cp (j/kg k) | 473.15K | 4493.6 | 543.15K | 5067 |
| $\lambda_1$ (w/mk) | 473.15K | 0.5928 | 543.15K | 0.664 |
| Viscosity (kg/ms) | 473.15K | 1.339e^-4 | 543.15K | 9.995e^-5 |

Some more results for single phase flow:

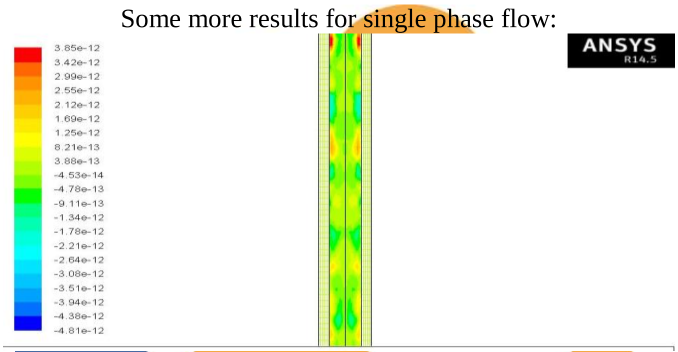

Fig- Contours of Mass Imbalance

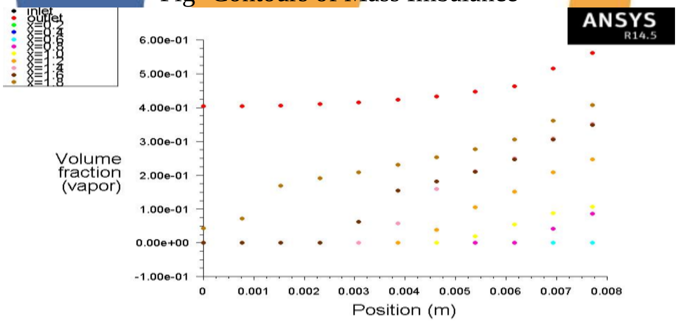

Fig- Profile of Volume fraction

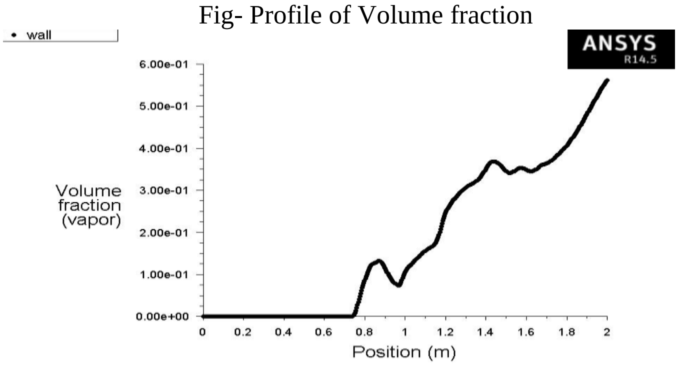

Fig- Profile of volume fraction along the wall

## CONCLUSION

In this study we have consider upward, vertical flow in a pipe with a heated wall. As the temperature increase the heat rate also increase and the wall temperature rises above the fluid saturation temperature, steam bubbles are formed and they migrate away from the wall. Since the bulk flow is sub cooled the bubbles condense near the center of the pipe .Outlet profiles of velocity magnitude and turbulence quantities generated for a simulated flow field without boiling will be used as inlet information to the boiling simulation. This is done to ensure a fully-developed profile of these quantities at the inlet.

Fully developed profile are get by Ansys fluent software which is a commercial software also.The analysis of the thermal and hydrodynamics features of the flow and the wall heat transfer led to the conclusions that follow the Velocity boundary condition at the channel inlet forces the bubble to move downstream but the thinning of the liquid lm as effect of the evaporation is not captured. Differently when axed pressure difference is imposed among the terminal sections of the channel the bubble slows down as evaporation starts thus decreasing the lm thickness. The bubble may decelerate generating a backflow even though the nose continues to travel downstream to the channel.

The heat transfer performance is improved by the two-phase ow with respect to the single phase case throughout the heated length of the channel. The maximum enhancement is measured in the wake of the bubble, with the two phase flow heat transfer coefficient exceeding the single phase one by more than 20% along the diameters behind the bubble. There are two main reasons for such an improvement. Firstly the bubble transit has squeezed the thermal boundary layer against the wall and the local lm evaporation has cooled down the superheated liquid.

Finally we can conclude that in this study we are showing the volume fraction of fluid is highest at outlet in multiphase while it decreases after reaching at outlet in single phase flow and its shows different positions. As by studying this we can conclude that different rotating devices can attach where we are getting the highest vapour volume fraction.

### LIST OF REFERENCES

1. Bar-Cohen, A.: Hysteresis phenomena at the onset of nucleate boiling, Proc. Engineering Foundation Conf. on Pool and External Flow Boiling, Santa Barbara, 1–14 (1992).
2. Chen, J. C.: Correlation for boiling heat transfer to saturated fluids in convective flow, Ind Eng. Chem .Process Design and Development, 322–329 (1966).
3. Cooper, M. G.and Chandratilleke, T. :Growth of diffusion-controlled vapor bubbles at a wall in a known temperature gradient.Int.J.Heat Mass Transfer24, 1475–1492 DOI: 10.1016/0017-9310(81)90215-5 (1981).
4. Cornwell, K.:The influence of bubbly flow on boiling from a tube in a bundle, Int.J.Heat Mass Transfer 33,

2579–2584 DOI: 10.1016/0017-9310(90)90193-X(1990) .

5. Del Valle, V. H. and Kenning, D. B. R. : Subcooled boiling at high heat flux. Int. J. Heat Mass Transfer 28, 1907-1920 DOI: 10.1016/0017-9310(85)90213-3 (1985).

6. Dhir V. K.: Nucleate and transition boiling under pool and external flow conditions,Process.9th,Int.HeatTransferConf.,Jerusalem,1,129–156DOI: 10.1016/0142-727X(91)90018-Q (1990).

7. Fujita, Y. : The state-of-the-art nucleate boiling mechanism, Proc. Engineering Foundation Conf. on Pool and External Flow Boiling, Santa Barbara, 83–98 (1992).

8. Gorenflo, D.: Behaltersieden,VDI Warmeatlas 6th edn., VD1-Verlag, Dusseldorf (1991).