CANDIDATE'S DECLARATION

I hereby declare that the work presented in this dissertation report entitled, **"CFD MODELLING OF NUCLEATE POOL BOILING"** is submitted in partial fulfilment of the requirement for award of the degree of **Master of Technology in Chemical Engineering** with the specialization in **Computer Aided Process Plant Design (CAPPD)** at the Indian Institute of Technology Roorkee, is an authentic record of my original work carried out under the guidance of **Dr. V.K Agarwal, Prof. & Head**, Department of Chemical Engineering, Indian Institute of Technology Roorkee..

Date-

Place - Roorkee (**PRERANA SIKARWAR)**

CERTIFICATE

This is to certify that the above statement made by the candidate is correct to the best of my knowledge.

Dr. V.K Agarwal

Professor and Head,

Department of Chemical Engineering

Indian Institute of Technology, Roorkee

Roorkee, Uttarakhand-247667

INDIA

ACKNOWLEDGEMENT

It is with a deep sense of gratitude that I profoundly acknowledge my indebtness to my supervisor, **Dr. V.K Agarwal, Prof. & Head,** Department of Chemical Engineering for his perceptive suggestions and comments that helped me to remain motivated and enthused in compiling this work. His infallible supervision and guidance has made this work a more rewarding experience for me. My special thanks are due to **Dr.Vimal Kumar** Assistant Professor, Department of Chemical Engineering for his invaluable guidance.

Last but not the least, it is all owed to the blessings of my parents which helped me to complete the work in due time.

(PRERANA SIKARWAR)

ABSTRACT

Nucleate boiling is a very significant two phase process and finds wide application in many chemical industries. It is an extremely advantageous mechanism, due to its ability to transfer enormous amount of heat at low temperature gradients. Modelling of nucleate pool boiling is very is desirable in order to avoid overheating or damage of heating material in many industrial applications of it.

The main objective of the project work is to develop an effective heat transfer model for calculation of heat transfer coefficient. The set up used for study is taken from BHAUMIK et.al. (2004) literature. The set up consist of a vertical cylinder in which a horizontal heating rod is immersed. Different constant heat flux is given to heating rod made up of stainless steel. For every constant heat flux provided to heating surface heat transfer coefficient is measured at both atmospheric and sub-atmospheric conditions.CFD code ANSYS 13 is used for simulation. Simulations have been carried out in 2D heating system using a transient Eulerian-Eulerian multiphase model. RPI boiling model of Kurul & Podowski is used for simulation process. Simulations have been carried out for benzene & water at atmospheric & sub-atmospheric pressure. Simulated results are obtained for heat transfer coefficients, wall super-heat, quenching heat flux to total flux, convective flux to total flux, evaporative flux to total flux ; heat transfer coefficient obtained for different constant heat flux for both atmospheric and sub-atmospheric conditions are compared. Obtained results are validated with previously reported experimental data.

CONTENTS

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

h	Heat transfer coefficient (w/m^2k)
x	Liquid mole fraction
y	Vapour mole fraction
k	Thermal conductivity
d	Diameter
q	Heat flux(w/m^2)
R_a	Arithmetic average roughness(μm)
g	Gravitational coefficient(m/sec^2)
P	Pressure(N/m^2)
T	Temperature(K)
A_0	empirical constant
B_0	empirical constant
h_{fg}	latent heat of evaporation (J/kg)
K_0	empirical constant
T_s	saturation or bubble point temperature (K)
α	heat transfer coefficient (W/m^2K^{-1})
β	mass transfer coefficient in liquids (m/ s^{-1})
X_1	mole fraction of HFC-32 in liquid
Y_1	mole fraction of HCFC-32 in vapour
σ	surface tension (N/m)
μ	kinematic viscosity (m^2/s)

D_b	Departure diameter
m_e	Evaporative mass transfer
N_w	Active nucleation site density
V_d	Volume of bubble based on bubble departure diameter
h_{sl}	Heat transfer coefficient based on Ranz-Marshall correleation
ρ_v	Vapour density
ρ_l	Liquid density
h_{fv}	Latent heat of evaporation
t_w	Waiting time
a_i	Interfacial area
S_q	Source term
ε	Thermal performance
V_q	Volume of phase q
ρ_q	Density of phase q
$\breve{\rho_{\breve{q}}}$	Effective density
α_q	Fraction of phase q
m_{pq}	Characterizes the mass transfer from p^{th} phase to q^{th} phase.
m_{qp}	Characterizes the mass transfer
$\overrightarrow{v_q}$	Velocity of q phase.
h_q	Enthalpy of q phase
v_{qp}	Inter-phase velocity q phase to p phase
v_{pq}	Inter-Phase velocity p phase to q phase
$F_{lift,q}$	Lift force
R_{pq}	Interaction force

q_q	Heat flux of q^{th} phase
q_{wall}	Wall heat flux
q_q	Evaporative heat flux.
q_q	Quenching heat flux
h_{qp}	Inter-phase enthalpy from q^{th} phase to p^{th} phase.
h_{pq}	Inter-phase enthalpy from p^{th} phase to q^{th} phase.
m_q	Mass of q^{th} phase
m_p	Mass of p^{th} phase
q_f	Convective heat flux.
α_{quench}	Minimum fluidisation velocity
$\alpha_{wallfunction}$	Single phase heat transfer coefficient
m_e	Evaporative mass transfer
m_{lt}	Vapour to liquid mass transfer
m_{vt}	Superheated liquid to vapor mass transfer
q_{lt}	Vapour to liquid heat transfer
q_{vt}	Superheated liquid to vapour heat transfer
T_v	Vapour temperature
Q_{vi}	Inter-phase to vapour heat transfer
$C_{P,v}$	Specific enthalpy of vapour at constant pressure
Pe	Peclet number
Re	Reynolds number
Ja	Jacob number
Pr	Prandtl number
Nu	Nuseelt number

Subscripts and superscripts

max	Maximum
min	Minimum
id	Ideal
l	Liquid
v	Vapour
n	No. Of components
s	Saturation
v	Vapour
w	Wall
a	Additional
eff	Effective
b	Bubble
mix	Mixture
bp	Boiling range

CHAPTER-1

INTRODUCTION

Boiling is basically that mode of heat transfer in which phase change occurs; liquid phase changes in to vapor. Boiling aids in transferring lager amount of heat with small area and aids in improving the thermal performance of the components for the process industries. There are basically two types of boiling which occur in industries. Boiling can be divided in to basically two categories:

- Pool Boiling
- Flow boiling.

In the present work nucleate pool boiling is taken in to consideration. In pool boiling, boiling takes place on heating surface submerged in initially stagnant fluid.

Nucleate boiling is very significant part of heat exchange process; it has wide applications in many industries like petrochemical, air-separation, air-conditioning, power generation etc. it also finds it application in electronics industries in high heat flux cooling.

Heat transfer process in pool boiling process-:

- Natural convection.
- Partial nucleate boiling.
- Fully developed nucleate boiling.
- Transition boiling.
- Film boiling.

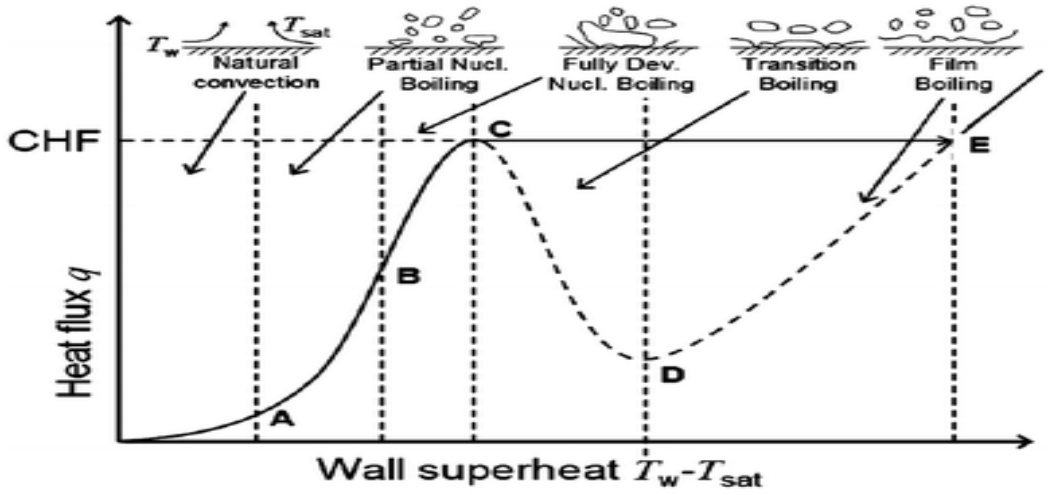

Fig 1.1 Value of wall superheat with respect to heat flux

1.1 NUCLEATE BOILING MECHANISM- Nucleate boiling takes place when the temperature of the heating surface becomes higher than the saturated temperature by a certain value provided heat flux value should be lower than the critical heat flux. At onset of nucleate boiling the bubbles covered only a small portion of heating surface and most of the liquid is in direct contact with the liquid. Bubbles formed at local nucleation sites, more sites become active as temperature of liquid is enhanced. As wall superheat increases heat transfer coefficient of fluid also increases until CHF value is reached. Nucleate boiling cycle is divided is divided in to following stages-:

- Formation of superheated thermal boundary layer adjacent to the heating wall.
- Set up of the conditions for bubble generation in active nucleation sites in thermal and mechanical equilibrium.
- Growth of bubble and departure of bubble followed by removal of thermal layer this whole process is repeated till the cold liquid comes in contact with the heating wall.

Heat Transfer Mechanism:

- **Bubble agitation mechanism -** Due to the process of growing and detached bubble on heating surface, intense convection in the liquid present near wall takes place which in turn causes the natural convection near the wall in local forced convection process.

- **Vapor liquid exchange mechanism -** A superheated layer forms on the heating surface due to the transient conduction from the heating wall to the liquid, this heat is take up by the bubble departed from the heating surface which introduces the cyclic thermal boundary layer stripping phenomena.

- **Evaporation mechanism -** Growth of vapor bubbles takes place on the superheated layer formed on the heating surface. Two heat transfer process occurs in a growing bubble: Macro evaporation and micro evaporation; macro evaporation takes place over the top of the bubble surrounded by thermal boundary layer and micro evaporation takes place underneath the bubble.

1.2 COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics is that branch of computational technology which helps us to analyze the dynamics of flowing fluid. With the help of CFD we develop the model which represents the system or device in which we are interested in studying. After this flow physics and chemistry is applied to the model and CFD software will give us the idea of fluid dynamics. Therefore with the help of CFD we can design the model of any system and predict its outcome in advance.

Navier-Stokes equations are the basis for solution of any CFD problem. Navier-Stokes equations represent the single phase fluid flow; one can get the equation by implementation of Newton's Law of Motion to a fluid element and it also known as momentum equation. It is accompanied by continuity and energy equation. The other most significant consideration in solving CFD problem is discretization techniques, in discretization spatial volume is discretized in to small cells and then appropriate algorithm is implemented in order to solve the equations.

Steps In CFD Analysis Process:

The required steps for performing a CFD analysis are given below:

- Formulate the Flow Problem.
- Model the Geometry and Flow Domain.
- The volume occupied by fluid is divided in to discrete cells known as mesh. This mesh can be uniform or non-uniform.
- Physical modeling is defined e.g equations of motion + enthalpy + radiation + species conservation.
- Establish the Boundary and Initial Conditions. It includes the condition of flowing fluid at the boundaries of the considered problem. If considered problem is transient, then initial conditions of flowing fluid are also defined.
- Organize the Simulation Strategy.
- Set up the Input Parameters and Files.
- Perform the Simulation.
- Invigilate the Simulation for Completion.
- Obtain the results by post processing.
- Make Comparisons of the Results.
- Repeat the Process in order to analyze the Sensitivities.

CFD codes- There are various CFD codes available for the prediction of dynamics fluid flow. In our case we need to simulate nucleate pool of liquid, CFD codes are very strong tool we can observe the heat transfer pattern with the aid of suitable boundary conditions and multiphase model. Following are some of the CFD code available which can be used for the simulation of nucleate pool boiling-:

- ANSYS FLUENT
- CFX
- NEPTUNE

In the present case CFD code ANSYS FLUENT 13 is used for the simulation of nucleate pool boiling.

1.3 MOTIVATION

It has been cleared from the literature review that lot of experimental as well as numerical studies have been conducted by researchers in order to investigate nucleate pool boiling. Many researchers have done experiment to determine the effect of roughness on heat transfer coefficient and heat flux in nucleate pool boiling. Researchers have also done experiment to investigate the heat transfer coefficients by using different boiling media; some of them have also used nanofluid as boiling media. Few researchers have also conducted experiment in microchannels. They have also used coated and non-coated heating surface in order to determine its effect on nucleate pool boiling. They have also developed and modified correlations for heat transfer. Most of the studies available on nucleate pool boiling whether computational or experimental have been conducted at atmospheric pressure. Only few have used sub-atmospheric pressure. Mostly researchers used refrigerants but in chemical engineering we need to use organic chemicals.

1.4 OBJECTIVES

Keeping the above in the mind the present investigation has been planned with the following objectives:-

- To develop CFD model for nucleate pool boiling of liquids on heating surface.
- To generate grid and grid sensitivity analysis.
- To solve the mathematical model in ANSYS FLUENT 13 and validated the simulated results with the previously reported experimental work.
- Calculation of heat transfer coefficients and other heat transfer parameters at atmospheric and sub-atmospheric pressure.

CHAPTER- 2

LITERATURE REVIEW

Nucleate boiling is significant phenomena in chemical engineering. Nucleate boiling falls under multiphase problem. As it is a multiphase problem, mechanism of mass, momentum and energy exchange through the interface is quite complicated. Numerical models proposed for modeling of nucleate boiling mainly depends on the correlations for various range of operating conditions. In this chapter, we briefly discuss the experimental and numerical studies available on nucleate pool boiling.

2.1 EXPERIMENTAL STUDIES ON NUCLEATE POOL BOILING

Rao et al., (2004)- They have experimentally evaluated pool boiling heat transfer coefficient for mixtures. They have taken acetone–isopropanol–water and acetone–MEK (methyl-ethyl ketone)–water ternary systems for the experimental purpose. They have also obtained correlation for pure components; surface–liquid interaction parameter and the surface roughness group is taken in to consideration while obtaining correlation. They have shown the following in the experiment

- Evaluated heat transfer coefficients of mixture are having lower values than pure components constituting the mixture.
- When they have compared the experimental data with the available literature; either over prediction or under prediction have been find out. Therefore two correlations have been used by them for obtaining ideal heat transfer coefficient.
- Following are the correlations for heat transfer coefficients of mixture.

$$h_{id}=.23\cdot\left(\frac{k_1}{d_b}\right).\left(\frac{q d_b}{k_1 T_s}\right)^{.674}.\left(\frac{\rho_v}{\rho_1}\right)^{.297}.\left(\frac{d_b^2\lambda}{\alpha_1^2}\right)^{.371}.\left(\frac{\rho_1\alpha_1^2}{d_b\sigma}\right)^{.350}.\left(\frac{\rho_1-\rho_v}{\rho_1}\right)^{-1.73} \quad \text{..... (2.1)}$$

$$\frac{1}{h_{id}}=\sum_{i=1}^{n}\frac{x_i}{h_i} \quad \text{..... (2.2)}$$

$$\frac{h_i}{h_0} = \left(\frac{q}{q_0}\right)^{n(p^*)} F(p^*)\left(\frac{R_a}{R_{ao}}\right)^{.133} \quad \text{.....(2.3)}$$

- For organic liquids,

$$F(p^*) = 1.2p^{*.27} + \left(2.5 + \frac{1}{1-p^*}\right)p^* \quad \text{..... (2.4)}$$

$$n(p^*) = .9 - .3p^{*.3} \quad \text{.....(2.5)}$$

- For water,

$$F(p^*) = 1.73p^{*.27} + \left(6.1 + \frac{.68}{1-p^*}\right)p^{*2} \quad \text{.....(2.6)}$$

$$n(p^*) = 0.9 - 0.3p^{*.15} \quad \text{.....(2.7)}$$

- Due to the diffusion of light components to the interface of bubble, reduction in the value of temperature gradient has been observed. The trend of diffusion in multi-component is very different from binary mixtures; therefore they have obtained effective temperature gradient by taking in to consideration multi-component diffusion coefficient.
- They have observed that heat transfer coefficient depends on equilibrium vapor and liquid concentration of light component.

Kim et al., (2005)- They have performed nucleate boiling experiment for refrigerant. Pure R113 and constant temperature have been used for experimental purpose. Experiment has been performed for sub-cooled, saturated and superheated conditions. They have analyzed the behavior of bubble growth by changing the condition of pool, by the help of dimensionless parameters for initial and thermal growth regions. They have also examined the bubble growth behavior during sub-cooled, saturated, superheated pool boiling. They have done it with the help of jakob number.

In the experiment quantitative analysis of single bubble growth is done during all conditions. Microscale heater array and wheatstone bridges have taken in to use for maintaining the constant wall temperature. They have concluded following from the experiment

- For initial growth region, they have found out growth rate is proportional to $(t^*)^{2/3}$ for all pool temperature except $Ja^* < 20$ ($T_b < 40^0$C).
- For thermal growth region, it is proportional to $(t^*)^{1/5}$ for all pool temperatures.

Das et al., (2007)- They have done experimental investigation on heat transfer for smooth surfaces and for the surfaces with different nucleation sites under conditions of saturated pool boiling. They have formed the surfaces by micro-drilling with distinct nucleation sites. They have found out that Rohsenow correlation [W.M. Rohsenow, A method of correlating heat transfer data for surface boiling of liquids, Trans. ASME 74 (1952) 969–976] among various correlations available gives most accurate results when compared with experimental data. They have observed that with the increase in density of nucleation site, heat transfer coefficients also increases with diminishing pattern.

Zhao et al., (2008)- They have performed experiment for determining in nucleate pool boiling for refrigerant on a horizontal surface of copper. Refrigerants used for the experiments are HFC-134a, HFC-32, HFC-125 and binary mixtures of two types: non-azeotrope mixture HFC-32/134a, azoetrope mixture HFC-32/125. Experiment is performed at saturated conditions (.9Mpa)

Following Correlations used in estimating heat transfer coefficient in nucleate pool boiling -:

- Stephan Corner Equation

$$\frac{\alpha}{\alpha_{id}} = \frac{1}{1+A_0|Y_1-X_1|(.88+.12P)} \quad \text{..... (2.8)}$$

- Schlunder Equation

$$\frac{\alpha}{\alpha_{id}} = \frac{1}{1+\frac{(T_{s2}-T_{s1})|Y_1-X_1|}{\Delta T_{id}}\left[1-exp\left(\frac{-\beta_0 q}{\beta_l h_{fg} \rho_l}\right)\right]} \quad \text{..... (2.9)}$$

- Jungincal Equation

$$\frac{\alpha}{\alpha_{id}} = \frac{1}{1+K_0|Y_1-X_1|\left(\frac{\rho_v}{\rho_l}\right)q^{.48+.1X_1}} \quad \text{(2.10)}$$

They have concluded following from the experiment:

- Reduction in the rate of heat transfer in binary mixture mainly depends on concentration of mixture and value of heat flux.
- Out of the available correlations present none can determine the accurate heat transfer coefficient well. Especially in the case of binary mixtureHFC-32/125, every correlation available unable to determine the general trend of the experiment results at different heat fluxes.

Gerardi et al., (2010) - They have examined the growth of bubble in nucleate pool of water with the help of high speed video and synchronized infrared thermometry. They have used high speed video and synchronized infrared thermometry in order to get required information on generation of bubble and heat transfer. They have determined departure diameter of bubble, wait and growth time of bubble and nucleation site density for a indium–tin oxide surface kept on a sapphire substrate. The data obtained from it is highly significant in validating models of heat transfer, two phase flow including CFD with methods of interface tracking. The whole data collected from individual nucleation sites on pattern of bubble growth has been used to obtain commonly used but validated poorly heat transfer model of nucleate pool boiling. The compatibility between the model and data has been find out quite good. The heat flux portioning model which has been used showed that transient conduction after departure of bubble has played an important role in nucleate pool boiling.

Hosseini et al., (2011) - They have performed the experiment for determining the effect of roughness of surface on heat transfer coefficient of nucleate pool boiling. They have conducted experiment on horizontal circular copper surfaces by using refrigerant R113. Different sand paper grit size has been used by them in order to get different surface roughness. A profilometer has been used to obtain value of average surface roughness. Heat flux in the range of 8

200kW/m^2 is used. Heat transfer coefficients have been calculated by determining input heat flux wall and superheat of the samples.

Following correlations have been used for determining heat transfer coefficient:

$$h = 55\, q^{.67}\, p_r^{\,n}\, (-\log_{10} P_r)^{-.55}\, M^{-.5} \quad \text{..... (2.11)}$$

Where exponent n is calculated as

$$n = .12 - .2(\log_{10} Rp) \quad \text{.....(2.12)}$$

The parameter Rp is maximum peak height of surface profile in µm.

$$Rp = \frac{Ra}{.4} \quad \text{..... (2.13)}$$

From the experiment they have found that surface roughening enhances the heat transfer coefficient of boiling. It is found that the heat transfer coefficient of the sample can be increased up to 38.5%, they also found that for rough surfaces an increase in value of Ra does not increase the heat transfer coefficient as much as it does in polished surfaces.

2.2 MODELLING OF NUCLEATE POOL BOILING

Liao et al., (2004)- Physical model for growth of vapor bubble in saturated nucleate boiling has been developed by them. The model takes in to account both transfer of heat through the liquid surface and from the bulk superheated liquid which is surrounding the bubble. While developing model, thermal interaction of the temperature around the growing bubble and vapor bubble growth with micro layer is taken in to account. An asymptotic and numerical solution describes the structure of thin liquid thermal boundary layer around the bubble. Thin unsteady thermal boundary layer which is present around the fast growing bubble allows transfer of heat from the bulk liquid to vapor. Sometimes the amount of heat transfer through this layer is much more than the heat transfer from microlayer. They have thrown light on the experimental observations on inappropriate amount of heat transfer through the microlayer carried out by Yaddanapudi and

Kim (2001). It has been found that in case of thick super-heated thermal boundary layer, sometimes heat transfer through the vapor bubble dome is significant in growth of vapor bubble.

Christopher et al., (2010) - They have presented a theoretical model to guess the microlayer thickness. They have done investigation on heat transfer across microlayer during heterogeneous nucleate boiling. They have assumed single directional flow in heat transfer with the effects of disjoining pressure. Modified Clausius-Clapyron equation is used to determine the variation of temperature and pressure across the evaporating interface. It has shown the variation of the evaporation rate, interfacial temperature, and the microlayer thickness with respect to radial position. The theoretical model gives the following information

- The microlayer thickness is found to be a function of the heating conditions and fluid properties.
- As the bubble size increase, heat transfer across the microlayer also increases. The increment in thickness is not as fast as increment in interfacial surface area of microlayer.
- When the coefficient of ideal evaporation is 1, high thermal resistance is offered by microlayer; when coefficient of ideal evaporation is .03, then the thermal resistance offered by evaporation at the interface also plays important role.

Steiner et al., (2005) - Chen-type superposition model has been proposed by them for evaluation of effective wall heat flux in subcooled boiling flow. It alters the contribution of nucleate boiling by taking in to account the effect of flow forces of the subcooling of the thermal boundary layer. It shows good agreement with available data especially in the PDB region. In the PDB region, where bulk fluid plays an important role in nucleate pool boiling which shows that the model captures the flow induced suppression very well. Major disadvantage of this model is that significant deviations occur in vicinity of the Fully Developed Boiling (FDB) regime.

2.3 NUMERICAL STUDIES ON NUCLEATE POOL BOILING

Yoon et al., (2001) - A mesh free method has been developed by them for determination of gas-liquid two phase flow. For the analysis of two phase flow, they have integrated particle method (MPS) and a gridless method (MAFL) for determination of any Lagrangian-Eulerian calculation. Numerical simulation is conducted for a generation of gas bubble in viscous liquids for isothermal flow. Results obtained from the numerical simulations were compared with the available correlations. Coupling of energy equation with the equation of motion is done in order to calculate nucleate pool boiling. They have obtained results for heat transfer rate, bubble growth rate, bubble departure radius. obtained results matched well with the experimental data .Simulations were conducted for the shape of bubble generating in still viscous liquid in two dimension and usual shapes of bubble were regenerated that were obtained in previously conducted experiment by researcher for different range of liquid properties. The mesh free numerical method is used for the calculation of bubble growth and departure process. Bubble departure diameter calculated from the conducted simulation is found to be proportional to the square root of the surface tension and contact angle. liquid agitation due to bubble motion has significant contribution in high heat transfer rate in nucleate pool boiling.

Son et al., (2002) - They have performed numerical simulation for heat transfer during growth of a bubble in nucleate pool boiling. Model used for simulation purpose is divided in to three parts: the bubble, micro- region (ring shaped zone between bubble and heating wall) and macro region (surrounding liquid around the bubble. Detailed modeling of macro region is done. Navier-Stokes equations both for liquid and vapor phases have solved with the help of finite element method. It came out from the simulation flow pattern of liquid around the bubble depends on both vapor flow inside the bubble and the movement of the bubble surface. It has been observed that due to transfer of cooler liquid towards the wall by the fluid enhances the heat transfer; buoyancy has small effect in comparison to forced convection. Heat fluxes, temperature fields, Velocity, bubble contours, and departure diameter were calculated.

Mukherjee et al., (2007) - They have done numerical simulation for the growth of single bubble with dynamic contact angle during nucleate pool boiling. Simulation has been carried out for the single nucleating bubble having different contact angle with the base of heating wall. Dynamic contact angle models based on the contact line velocity and Static contact angle model is used for the simulation purpose. Navier-Stokes equations using SIMPLE(semi implicit method for pressure linked equations) method have solved. Level-set technique is used for the capture of liquid-vapor interface. The effect of static contact angle on bubble dynamics and vapor volume growth rate compared with the results obtained with the dynamic contact angle.They have concluded following from the simulations.

- The vapor volume growth rate is proportional to the contact angle.
- Surface wettability is inversely proportional to the bubble departure diameter.
- When dynamic contact angle model is used bubble acquires a stick/slip behavior at the base of the wall while it acquires a smooth behavior when static contact angle is used.

Narumanchi et al., (2008) -They have done numerical simulation for turbulent jet impingement. CFD code Fluent has used for the computational purpose. Eulerian multiphase model has enabled for the simulation purpose for nucleate pool boiling.CFD model and code has validated against available experimental studies. results obtained from the simulation shows good agreement with the available experimental results.

Vyskocil et al., (2008) - They have conducted simulations for the convective boiling flow in vertical tube with the CFD codes. Codes used for the simulations purpose were NEPTUNE and FLUENT. The mechanistic boiling model of Kurul and Podowski (1990) is implemented in NEPTUNE CFD code in order to model nucleate pool boiling; similar model is applied in the FLUENT 6 by the help of user defined function (UDF) in FLUENT 6. They have used R-12 for the simulation purpose both for NEPTUNE and FLUENT 6. Eulerian multiphase model is enabled for the simulation purpose in FLUENT 6. Realizable k-∈ epsilon model is used for modeling turbulence with first order implicit unsteady formulation. results obtained from both FLUENT 6 and NEPTUNE CFD code shows good agreement with each other.

Hazi et al., (2009) - Numerical simulation is performed for the heterogeneous boiling for slowly flowing and still fluid on horizontal plate. lattice Boltzmann technique is used for the simulation purpose. They have calculated departure diameter of bubble and frequency. They have shown that there is no relation between the static contact angle and bubble departure diameter. They have also shown that frequency of bubble detachments varies exponentially with the static contact angle. From the numerical simulation they have found that in still fluid the bubble departure diameter varies with $g^{1/2}$.

Zhuan et al., (2010) -They have performed numerical simulation for investigation of nucleate boiling of water in micro-channels. VOF multiphase model is used for the simulation purpose. They have analysed Marangoni heat transfer through the bubble surface and later on it is compared with heat flux at beginning at onset of nucleate boiling. They have divided the growth of bubble in channel in two sections: in first section surface tension plays significant role in bubble growth while in second section heat transfer at beginning controlled the boiling process. Results obtained from the simulations have shown generation of bubble, merger of bubbles, shrinking, and departure.From the simulation results they have concluded following

- In common channels nucleate boiling exhibits free bubble flow characteristics while in microchannel it exhibits confined bubble flow.
- From the investigation it has been observed that Marangoni heat transfer on bubble surface plays significant role at onset of boiling. They have divided whole boiling process in two sections- in first section the growth of bubble is inhibited by surface tension and Marangoni convection governs at onset of nucleate boiling; in second section bubble grows very fast and departs off from the wall.

Aminfar et al.,(2012) - They have performed numerical simulation for nucleate boiling of nano-fluids. Two phase and three phase model mixtures and control volume strategies have been used to anticipate boiling curve. Same method is used for the investigation of nanofluids with alumina particles. They have concluded from the results obtained from numerical simulation, two phase simulations have provided more accurate results than three phase simulations.

Ryu et al.,(2012) - They have conducted numerical simulation for nucleate boiling of water with the aid of two dimensional Lattice Boltzmann method. Method used for simulation is based on free energy. Lattice Boltzmann method included the techniques of interface capturing and energy equations with phase change model. They have done simulation for the bubble growth in superheated liquid and results obtained were compared with analytical solutions. Numerical simulations were performed by using different values for wall superheat, surface tension, and gravity force and contact angle. They have performed simulations for single and multiple nucleation sites. From the results of numerical simulation they have concluded that the Lattice Boltzmann method with a phase change model is suitable for the direct numerical simulation of nucleate pool boiling.

CHAPTER-3

MATHEMATICAL MODELING

In industry and nature we encounter so many flows which consist of different phases. Gas, liquid & solid are physical phases of matter. In multiphase flow, we can define phase as a material which has a separately defined volume fraction & has its own way of response to the potential field in which it is immersed. Our problem of nucleate pool boiling falls in category of multiphase flow. In this chapter, we described different approaches to multiphase modeling, models & model equations for nucleate pool boiling.

3.1 MULTIPHASE MODELING APPROACH: There are two approaches which is used in the numerical calculation in modeling of multiphase flows.

- Eulerian –Lagrangian approach.
- Eulerian-Eulerian approach.

3.1.1 Euler-Lagrange Approach

In this approach fluid phase is handled as a continuum by solving the time- averaged Navier-Stokes equation. The other phase which is dispersed one is solved by observing a large number of particles, bubbles through the calculated field. The dispersed phase can interchange mass, momentum and energy with the other phase (fluid). The basic assumption which is made in this model is that the second phase (dispersed), even though high mass loading. The path of the dispersed phase i.e. particle, droplet, bubble are computed individually at different intervals. Due to this feature this model is highly appropriate for modeling of spray dryers, coal and liquid fuel combustion. It cannot be used for the modeling in those flows where the volume fraction of dispersed phase is not negligible like fluidized beds.

3.1.2 Eulerian-Eulerian Approach

In this approach the each & every phase present in flow is treated mathematically as mutual penetrating continua. As we know that each phase has its own volume and it can't be occupied by other therefore the concept of volume fraction is introduced. It is assumed that volume

fraction of each & every phase is continuous function of space & time and the sum of all volume fraction is equal to one. Conservation equations are applied on each phase & we obtain a set of equations having look-alike structure for all phases. The derived equations are closed by giving essential relations. These relations can be obtained from empirical information. Three models are available with this approach: The volume of fluid model, the mixture model, the eulerian model.

The VOF model

In this model a surface-tracking method is used. This surface-tracking method is implemented to a fixed Eulerian mesh. This model can be used for only those mixtures which consist of two or more fluids which are not miscible with each other; position of the interface between the fluids is very significant. A single set of momentum equations is used by all the fluids present in mixture. Volume fractions of each of the phases present in each computational mesh is followed through out. This model is appropriate in those applications where we are interested in position of interface of fluids e.g. flow of large bubbles in liquid, free surface flows.

Mixture model

This model is applicable for mixtures containing two or more phases (fluid or particulate). As we know that in Eulerian model each phase is handled as mutual penetrating continua. Momentum equation for mixture is solved and it recommends relative velocities to report the phases. This model can be implemented for the mixtures having particle-laden flows with minor charging, sedimentation, cyclone separators and bubbly flows. It can also be implemented for modeling of homogenous flow without relative velocities of dispersed phase.

Eulerian model

This is most complicated model available in ANSYS FLUENT. In this model a set of n momentum and continuity is solved for each & every phase, after that coupling is done through the pressure & inter-phase exchange coefficients. The way in which coupling is achieved depends upon the phases present e.g. in granular flow coupling is done in different way than non-granular flow. In granular flows properties are determined by implementation of kinetic theory. Interchange of momentum between the phases also depends on the type of phases present in

mixture. Eulerian multiphase is widely used for modeling of fluidized beds, risers, bubble column etc.

Eulerian model is based on the following assumptions:

- All phases shared single pressure.
- Conservation equations (momentum & equations) are solved for each phases.

3.2 Selection Of Model

As we have already discussed that VOF model is suited for free surface flow which is not focus of our research. Therefore only two models are left, now we have to make a choice between the Mixture model & the Eulerian model. As we are using RPI boiling model for modeling of nucleate pool boiling which is available with eulerian model only; so for our problem we have used Eulerian model.

The Eulerian multiphase model is used for the modeling of discrete yet interacting phases present in flow. The phases present in flow can be of any combination it can be liquid-gas, solid-liquid, and liquid-gas. In Eulerian model each phase is treated separately in contrary to the Eulerian-Lagrange approach. As we know that when we switch from single phase to multiphase so many changes occurs like in single phase only a single set of conservations are solved while in multiphase some more equations also come in picture. So now we have some more equations in addition to the original but some modifications must be done in original equations in order to apply in multiphase. The changes include introduction of volume fraction for phases & in mechanism of interchange of mass, momentum, heat & mass between the phases.

3.3 Eulerian Model Theory

It is used for the modeling of multiphase flows. By the help of it modeling of different yet interacting phases.

ANSYS FLUENT solution of this model based on following assumption:

- All phases shared single pressure.
- Conservation equations are also applied on each phase.

Limitations

- It can be used only for viscous flows.
- It can't be implemented for modeling of streamwise periodic flow.
- It can't be implemented when melting & solidification occurs in system.
- When tracking of particles is to be done in parallel way, we use discrete phase model with eulerian model only when option of shared memory is disabled.

3.3.1 Following section discuss the equations used to calculate the multiphase solution:

- **Conservation of Mass :**
  - **Continuity Equation:**

$$\frac{1}{\rho_{rg}}\left(\frac{\partial}{\partial t}\left(\alpha_q\rho_q\right)+\nabla.\left(\alpha_q\rho_q\overrightarrow{v_q}\right)\right)=\sum_{p=1}^{m}\left(m_{pq}-\dot{m}_{qp}\right)+s_q \quad \text{(3.1)}$$

- **Conservation of momentum:**

$$\frac{\partial}{\partial t}\left(\alpha_q\rho_q\overrightarrow{v_q}\right)+\nabla.\left(\alpha_q\rho_q\overrightarrow{v_q}\overrightarrow{v_q}\right)=-\alpha_q\nabla p+\nabla.\overline{\overline{\tau_q}}+\alpha_q\rho_q\bar{g}+\sum_{p=1}^{n}\left(k_{pq}\left(\overline{v_p}-\overline{v_q}\right)m_{pq}\overline{v_{pq}}-\overline{v_q}\dot{m}_{qp}+\right)+\left(\bar{F}_q+\overline{F_{lift,q}}+\bar{F}_{vm,q}\right) \quad \text{(3.2)}$$

If m_{pq} > 0, it means mass transfer takes place from phase p to q(v_{pq} =v_{qp}); if m_{pq} < 0 v_{pq}=v_q. Similarly if $m_{qp} > 0$ then $v_{qp} = v_q$. Above equations must be closed with the correct expressions for the interphase force.

- **Conservation of energy:** For conservation of energy in Eulerian multiphase problems, enthalpy equation for each phase is needed.

$$\frac{\partial}{\partial t}\left(\alpha_q\rho_q h_q\right)+\nabla.\left(\alpha_q\rho_q\overrightarrow{v_q}h_q\right)=-\alpha_q\frac{\partial p_q}{\partial t}+\overline{\overline{\tau_q}}+\nabla\overline{u_g}+S_q+\sum_{p=1}^{n}\left(Q_{pq}+m_{pq}h_{pq}-\dot{m}_{pq}h_{qp}\right) \quad \text{(3.3)}$$

- **Volume Fractions:**

$$V_q = \int_V \alpha_q dV \quad \text{(3.4)}$$

$$\sum_{q=1}^{n} \alpha_q = 1 \quad \text{(3.5)}$$

$$\breve{\rho}_{\breve{q}} = \alpha_q \quad \text{(3.6)}$$

Equations solved by ANSYS FLUENT:

- **Continuity Equation:**

Volume fraction of each is obtained from the following equation-:

$$\frac{1}{\rho_{rg}}\left(\frac{\partial(\alpha_q \rho_q)}{\partial t} + \nabla.\left(\alpha_q \rho_q \overrightarrow{v_q}\right)\right) = \sum_{p=1}^{n}\left(m_{pq} - \dot{m}_{qp}\right) \quad \text{(3.7)}$$

- **Fluid-Fluid Momentum Equations:**

$$\frac{\partial}{\partial t}\left(\alpha_q \rho_q h_q\right) + \nabla.\left(\alpha_q \rho_q \overrightarrow{v_q} h_q\right) = -\alpha_q \nabla p + \nabla.\overline{\overline{\tau_q}} + \alpha_q \rho_q \bar{g} + \sum_{p=1}^{n}\left(K_{pq}\left(\overline{v_p} - \bar{v}_q\right) + m_{pq}\overline{v_{pq}} - \dot{m}_{qp}\bar{v}_{qp}\right) \quad \text{(3.8)}$$

- **Conservation of Energy:**

$$\frac{\partial}{\partial t}\left(\alpha_q \rho_q h_q\right) + \nabla.\left(\alpha_q \rho_q \overrightarrow{v_q} h_q\right) = -\alpha_q \frac{\partial p_q}{\partial t} + \overline{\overline{\tau_q}} + \nabla\overline{u_g} - \nabla\bar{q}_q + S_q + \sum_{p=1}^{n}\left(Q_{pq} + m_{pq} h_{pq} - \dot{m}_{qp} h_{qp}\right) \quad \text{(3.9)}$$

3.4 NUCLEATE BOILING MODEL

As we know that term “subcooled boiling" is used to describe the physical condition where the temperature of the wall is sufficient to cause boiling to occur at the wall even though the temperature of the average liquid of most of the volume is less than the saturated temperature. In subcooled boiling, the energy is transferred directly from wall to the liquid. Some part of this energy is utilized in enhancing the temperature of the liquid & part of it is used to generate vapor. Average liquid temperature is also enhanced by interphase heat transfer. Some energy is also transferred directly from wall to vapor. These fundamental mechanisms are the base of Renessealer Polytechnic Institute models also known as RPI models.

In ANSYS FLUENT, nucleate boiling model is developed in the context of eulerian multiphase model. Modeling of two phases are done, primary phase (liquid) and secondary phase (vapor bubbles). Multiphase flows are governed by conservation equations of continuity, momentum and energy. These equations are solved for each phase. This phenomenon is modeled by mechanistic boiling model of Kurul & Podowski (1990). These models are compatible with three different kinds of wall boundaries: when temperature of wall is constant, constant heat flux, constant heat transfer coefficient.

3.4.1 RPI Model

The basic assumption of this model is that total heat flux from the wall is transferred in three parts, namely the convective heat flux, the quenching heat flux, the evaporative heat flux.

$$q_{wall} = q_c + q_e + q_q \quad \text{.....(3.10)}$$

The wall surface to which heat is given is further divided in to area A_b , area which is in influence of nucleating bubbles and remaining portion (1- A_b), which is covered by fluid.

Convective Heat Flux: This heat flux is used to calculate the heat transfer in single phase.

$$q_c = h_c (T_w - T_1)(1-A_b) \quad \text{.....(3.11)}$$

Where h_c is the heat transfer coefficient of single phase and T_w and T_1 are the wall and liquid temperatures respectively.

Quenching Heat Flux: This heat flux is used to calculate that part of wall heat flux which is transferred during bubble departure and generation of bubble at the same nucleation site.

$$q_q = \frac{2k_1}{\sqrt{\pi T \lambda_1}}(T_w - T_1) \quad \text{(3.12)}$$

Where k_1 is conductivity, T is the periodic time and

$$\lambda_1 = \frac{k_1}{\rho_1 c_{p1}}$$

Evaporative Heat Flux: This heat flux represents that portion of heat flux which is utilized in evaporation of liquid.

$$q_E = V_d N_w \rho_v h_{fv} f \quad \text{(3.13)}$$

Where $V_d$the volume of bubble based on bubble diameter is, N_w is active nucleation site density ρ_v, h_{fv} is vapor density and latent heat of evaporation and f is the bubble departure frequency.

The above equation needs closure for the following parameters:

- **Area of influence**

  It depends on departure diameter and nucleation site density

$$A_b = K \frac{N_w d_w^2 \pi}{4} \quad \text{(3.14)}$$

the area of influence has to be restricted, in order to ignore numerical instabilities due to unbound empirical correlations for the nucleate site density. It is limited by the following expression

$$A_b = \min\left(1, K\frac{N_w d_w^2 \pi}{4}\right) \quad \text{.....(3.15)}$$

Value of K used here is generally 4, but it's not universal it can vary between 1.8 and 5.

- **Frequency of bubble departure**

Application of RPI model takes that frequency of bubble diameter which is based on inertia controlled growth.

$$f = \frac{1}{t} = \sqrt{\frac{4g(\rho_l - \rho_v)}{3\rho_l D_w}} \quad \text{.....(3.16)}$$

- **Nucleate Site Density**

It is generally given by correlation which depends upon the wall superheat. The general expression is of the following form:

$$N_w = c^n (T_w - T_{sat})^n \quad \text{..... (3.17)}$$

Parameters used in above equation are having following values:
n= 1.805 and c = 210.

- **Bubble departure diameter**

The default bubble departure diameter (mm) used for the RPI model is given on following empirical correlations and is reported as:

$$D_w = \min(.0014, .0006e^{\frac{-\Delta T_w}{45}}) \quad \text{.....(3.18)}$$

While kocamustafaogullari and Ishii use

$$D_w = .0012(\rho *)^{.9}\ .028\sqrt{\frac{\sigma}{g\ (\rho_l - \rho_v)}}\varphi \quad \text{.....(3.19)}$$

3.4.2 Interfacial Momentum Transfer

The interfacial momentum transfer consists of four parts: drag, lift, virtual mass and turbulent drift forces. Modeling of virtual mass is done by using standard correlation available in the Eulerian multiphase model within ANSYS FLUENT. Specific sub-models also available for drag, lift, turbulent drift forces. User can also use its own correlation via help of UDF available within ANSYS FLUENT.

Interfacial area

It is very significant parameters for the calculation of drag and heat transfer. If dispersed boiling takes place then area depends on diameter of bubble would be sufficient enough. If during boiling bubble coalescence occurs then it should be modified. The following values can be used:

$$A_i = \frac{6a_v}{D_b} \quad \text{(3.20)}$$

$$A_i = \frac{6a_v(1-a_v)}{D_b} \quad \text{(3.31)}$$

3.4.4 Interfacial heat transfer

It is divided in to following parts

- Vapor to liquid heat transfer.
- Heat transfer from superheated liquid to vapor.

Vapor to liquid heat transfer

When bubble detaches from the heating wall and travel towards the sub-cooled region, transfer of heat transfer takes place from the bubble to liquid and it is given by following expression:

$$q_{1t} = A_i h_{s1}(T_{sat} - T_1) \quad \text{(3.32)}$$

Where A_i the interfacial area already described above & $h_{s1}$is heat transfer coeffiecient based on Ranz-Marshall Correlation.

$$h_{s1} = \frac{k_1}{D_b}(2+.6Re^{.5}Pr^{.33}) \quad \text{(3.33)}$$

Superheated liquid to vapor heat transfer

The interface to vapor heat transfer is obtained by the help of "constant time scale return to saturation method". The fundamental assumption of this method is that vapor remains at temperature of saturation by fast evaporation/condensation. It is given by following expression:

$$q_{vt} = \frac{a_v \rho_v C_p}{\delta t}(T_{sat} - T_v) \quad \text{(3.34)}$$

3.4.3 Mass transfer:

- **Transfer of mass from heating wall to vapor**

  The evaporation mass flow is implemented at the cell adjacent to wall and it is obtained from evaporative heat flux, given by following expression:

$$m_E = \frac{q_E}{h_{fv} + C_{p,l}\Delta T_{sub}} \quad \text{(3.35)}$$

- **Interfacial Mass Transfer**

  It is directly proportional to the interfacial heat transfer. It is assumed that all the heat that is giving to the heat transfer is used in mass transfer. It can be given by the following expression

$$m = m_{lt} + m_{vt} = \frac{q_{lt}+q_{vt}}{h_{fv}} \quad \text{(3.36)}$$

CHAPTER-4

NUMERICAL SOLUTION AND SIMULATION SETUP

4.1 DISCRETIZATION TECHNIQUES:

Discretization is used to convert the continuous models & equations in to discrete counter parts. Discretization is done in order to make it appropriate for numerical evaluation and for easy implementation on digital computers. By the help of this technique we can either limit the length of simulation or the number of particles involved in simulation. The stability of any discretization method is determined numerically than analytically via simple linear problems. Special attention must be given to ensure that it should handle discontinuous functions properly. Some discretization methods are given following:

- Finite volume method
- Finite element method.
- Finite difference method.

In the present case finite volume method is used for simulation process.

4.1.1 Finite volume method

Finite volume method is standard technique, highly used in commercial software and research codes. The equations that governed the flow are solved on small control volumes. it converts the partial differential equations of the Navier- Stokes equations in conventional form and then discretize it. Finite volume basically means a small volume surrounding around each node point on a mesh. In this technique, partial differential equation having a divergence term in volume integral, this is converted in to surface intergal by the help of divergence theorem. Later on these terms are calculated as fluxes at the surface of each finite volume. As flux entering and leaving the volume is constant, therefore the method is conservative in nature. Another benefit of this method is that it ensures the conservation of fluxes through a certain volume. Though the overall solution that obtained will be conservation but there is no assurance it would be the correct solution.

4.2 GEOMETRY DESCRIPTION

The Present Study consists of 3D heating system which has been used as the computational domain. The selected apparatus has a diameter of .21m, a height of .40m, and heating rod having .032m outer & .018 inner diameters. In order to investigate the heat transfer coefficient & other parameters same size has been selected for this work.

Following figures represents the experimental setup taken from literature *BHAUMIK et al. (2004)* and structure for CFD analysis

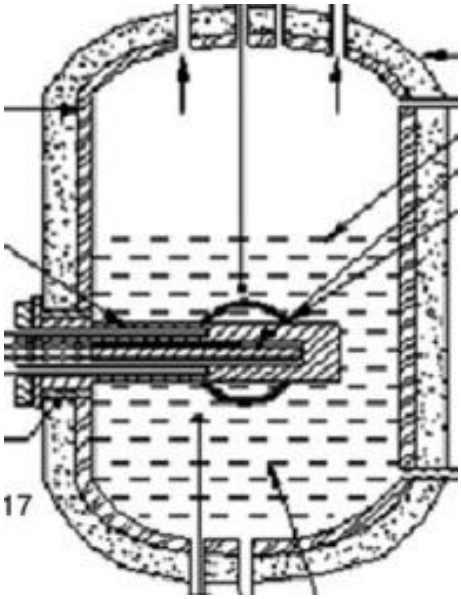

Fig 4.1

Experimental setup from *BHAUMIK et al. (2004)*(2008)

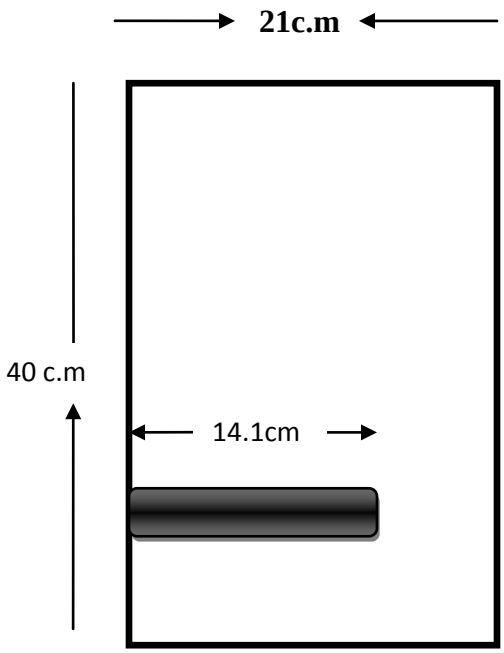

Fig 4.2

Structure for CFD analysis

Following figures shows the computational domain and as well as grid generation in the domain and zoomed view of grid which have been created in gambit. 2D geometry has been meshed of structured rectangular cells.

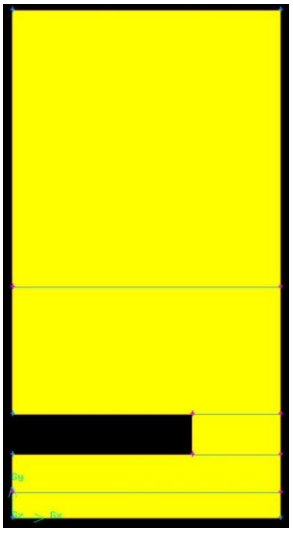

Fig 4.3
Generated grid in Gambit

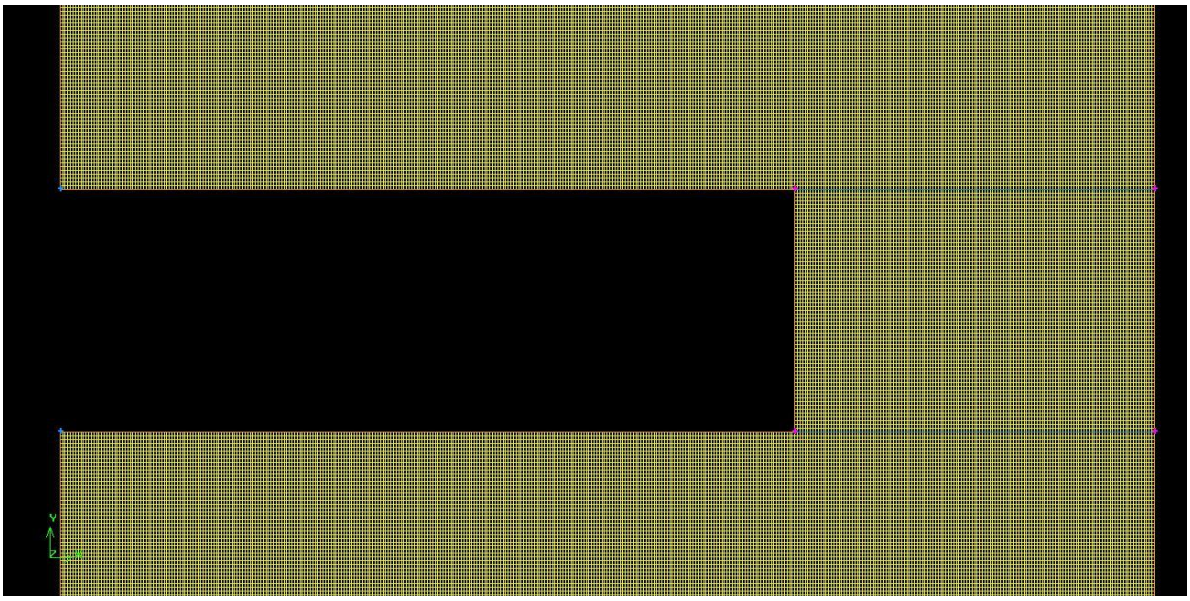

Fig 4.4
Zoomed view of mesh

Table 4.1
Detail of geometry used for simulation purpose.

Mesh size a) Near heating surface b) In interior	 .0005m .001m
Heating surface length	14.5 cm
Cylinder length	40 cm
Cylinder diameter	21cm
Heating surface outer diameter	32mm
Heating surface inner diameter	18mm

4.3 Boundary Conditions

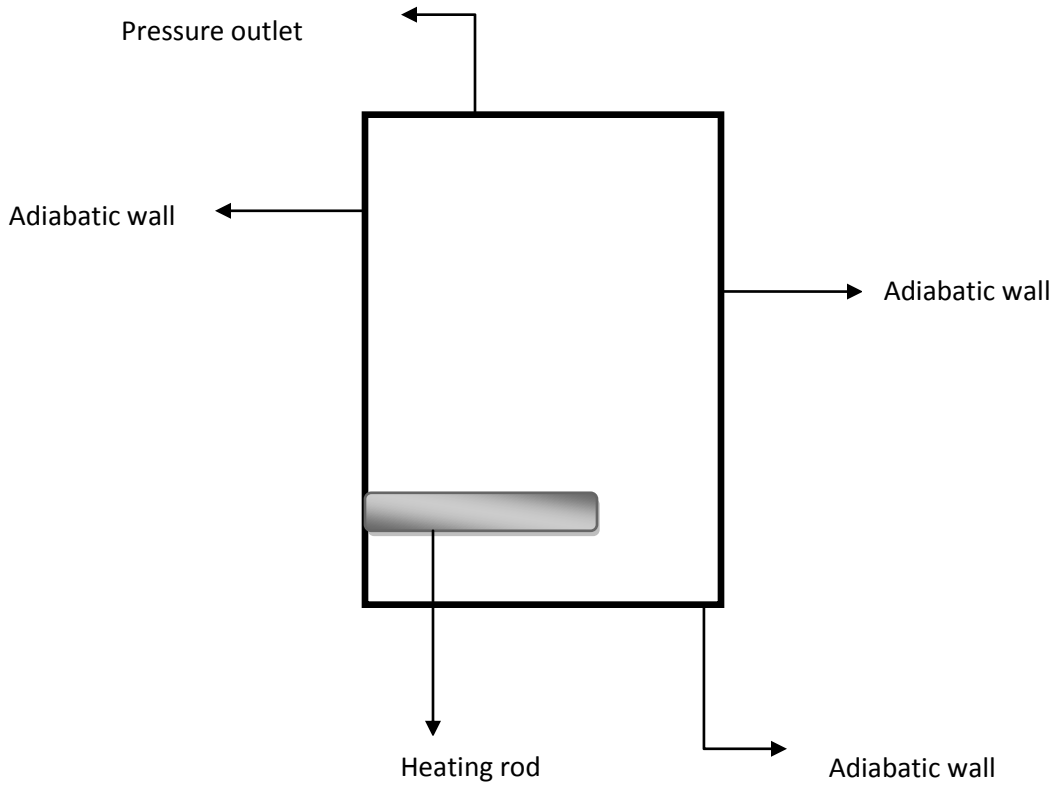

Fig 4.5 Boundary conditions

4.4 Model Parameters and Solver Settings

Table 4.2

Model parameters and solver settings used for simulation

Solver	Pressure based
Multiphase	Eulerian
Boiling Model	R.P.I boiling model
Energy	Enabled
Viscous	RNG k- epsilon model
Materials	Heating surface- stainless steel Heating media- liquid benzene / water
Properties (Benzene)	Benzene liquid- piecewise linear Benzene vapor- At boiling point
Properties (Water)	Water liquid- Piecewise linear Water Vapor-At boiling point
Phase-1	Benzene liquid/liquid water
Phase-2	Benzene vapor/water vapor
Phase interaction	Drag- boiling- ishii Lift- boiling-moraga Heat Transfer - Ranz Marshall Mass Transfer–boiling(transfer from liquid to vapor phase)
Operating Condition(Benzene)	Pressure- .1 atm to 1 atm Temprature- 340K Density-2.784 kg/m^3 Gravity – (-9.91m/sec^2)
Operating Condition(Water)	Temperature-370 k Pressure- 0.1 atm to 1 atm Density- 0.6 kg/m^3
Boundary condition	Heating surface- wall Outlet – pressure outlet

	Container surface- wall(adiabatic)
Solver Settings	
Pressure-Velocity Coupling	Full Multiphase Coupled scheme
Courant number	8
Explicit Relaxation Factors	Pressure -1 & Momentum-1
Under Relaxation Factors	Density -1 Body Force -0.5 Vaporization Mass -0.5 Volume fraction- 0.3 Turbulent kinetic energy- 0.3 Turbulent dissipation rate- 0.3 Turbulent viscosity- 0.5 Energy-0.6
Residual Monitors	Switch on compute local scale and choose local scale.
Initialization (Benzene)	Phase-1 temperature- 325 K Phase -2 temperature- 353.3 K
Initialization (Water)	Phase-1 temperature- 360 K Phase -2 temperature- 373.3 K
Iteration	Time step-.001 sec Max. iteration/ time interval-20

Following flowchart describes the required number of steps to be performed during simulation in ANSYS FLUENT 13.

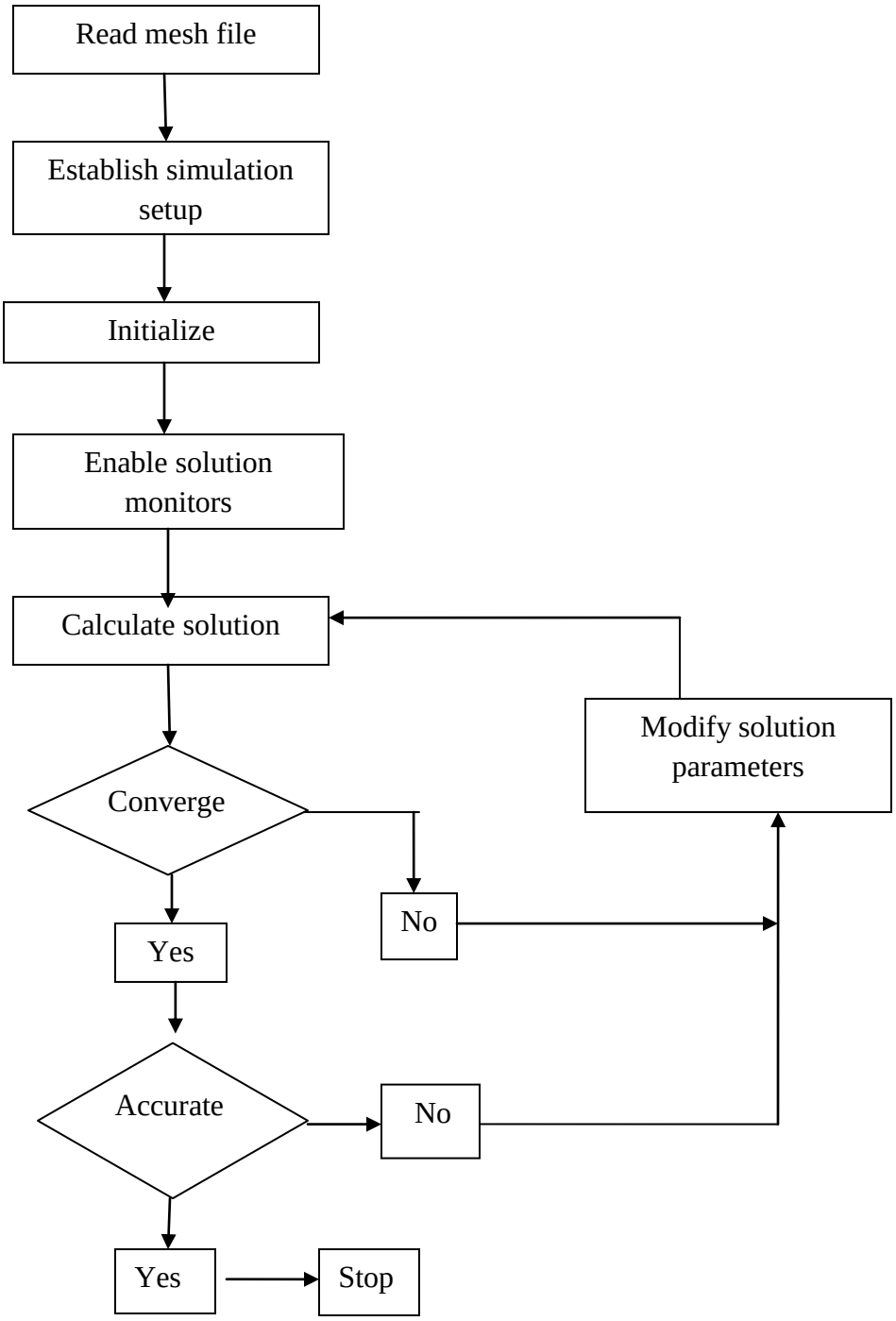

Fig. 4.6 *Represents general procedure of simulation in ANSYS FLUENT 13.*

CHAPTER-5

RESULTS AND DISCUSSIONS

CFD modeling of nucleate pool boiling has been performed using ANSYS FLUENT 13. Simulations have been carried out using in a 2D heating system with stainless steel as heating rod and water and benzene as heating media using a transient Eulerian-Eulerian model. Results obtained from simulations are validated with previously experimental data. Heat transfer coefficients for both water and benzene are calculated and the effect of pressure on heat transfer coefficients is also calculated

5.1 **For Benzene**

5.1.1 Grid Sensitivity And Validation Of Model

The first important step in analyzing the results obtained from simulation is Grid sensitivity analysis. Grid sensitivity analysis is done in order to study the effect of grid size resolution on the results obtained by simulation. In order to carry out this, geometry of the container has been meshed with three different mesh size 0.005, .002 and .0003 for simulation purpose. Results obtained for grid size .0003 agrees well with the experimental values.

5.1.1.1 Grid sensitivity analysis

Figure 5.1 shows the variation of heat transfer coefficient with respect to heat flux for different grid size in order to determine the effect of grid on results obtained.

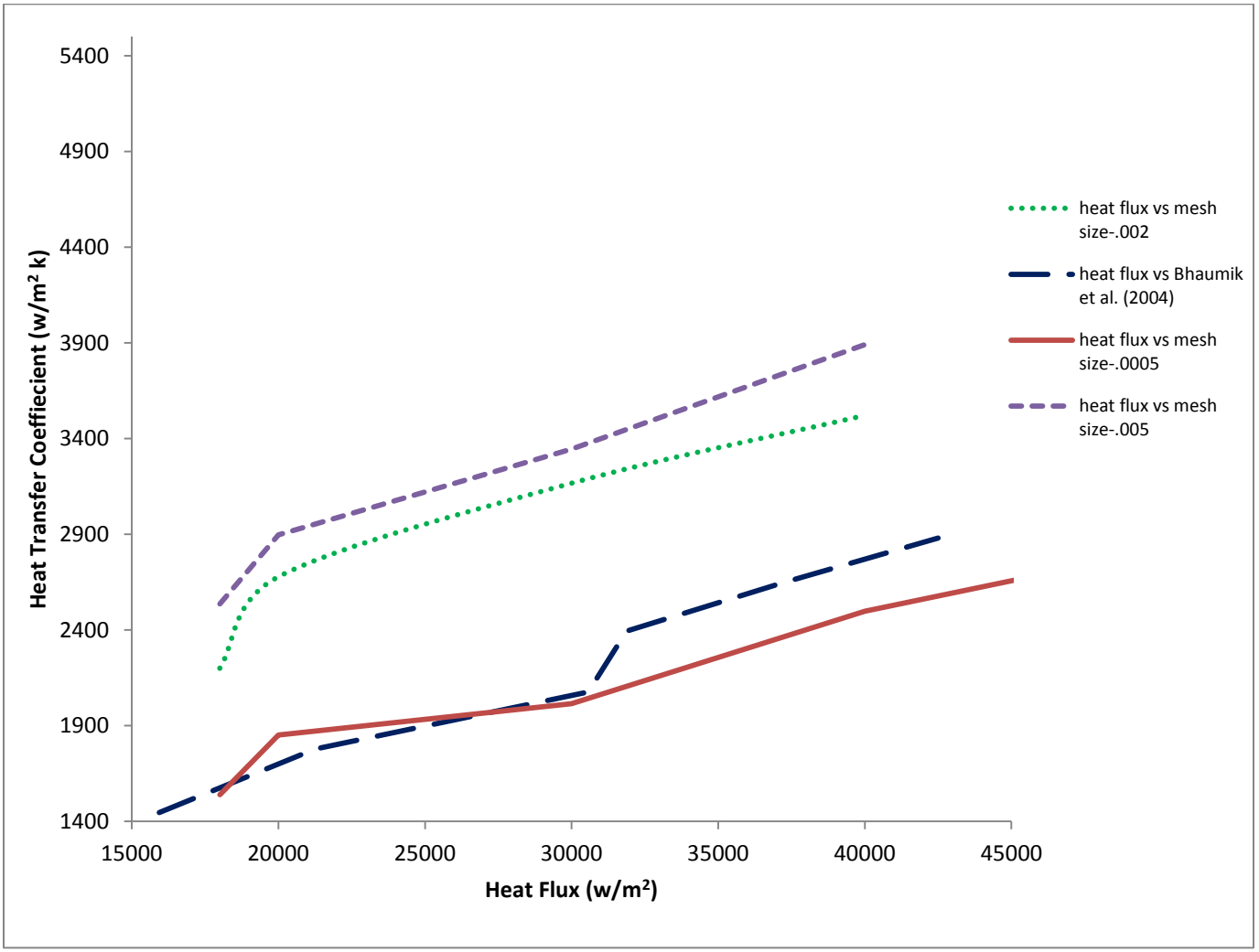

Figure 5.1 Value of heat transfer coefficient with respect to heat flux for different grid sizes

5.1.1.2 Validation

Figure 5.2 shows the variation of heat transfer coefficient with the respect to heat flux with the help of values taken from literature BHAUMIK et al. (2004) and present work in order to validate the present work.

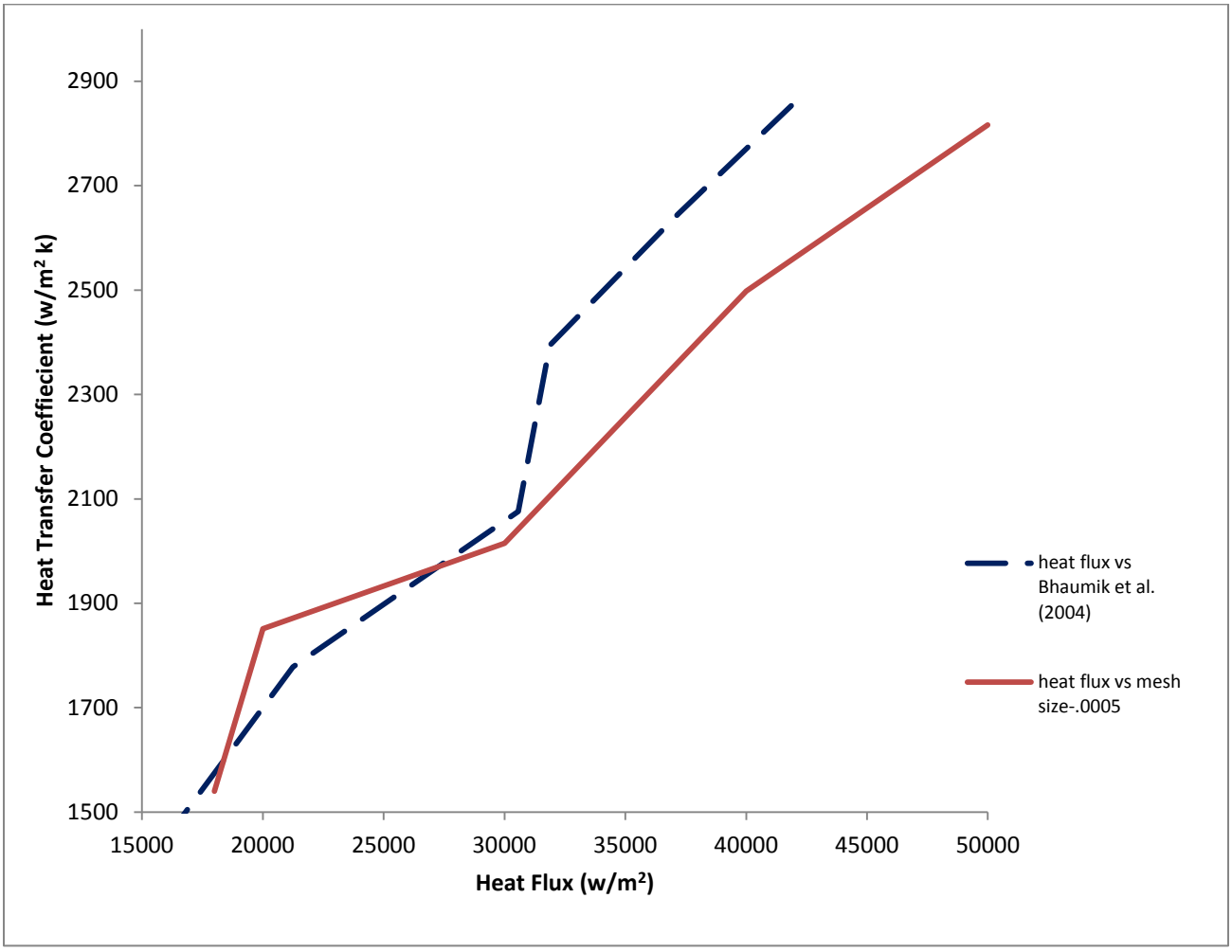

Figure 5.2 Value of heat transfer coefficients with respect to heat flux from literature and present work.

5.1.2 Results for benzene

Simulations have been carried out for different constant heat flux value assigned to stainless steel heating rod. Simulations have been carried out at atmospheric and sub-atmospheric pressure.

- **At atmospheric pressure for benzene**

Figure 5.3 shows variation of heat transfer coefficient with respect to heat flux. It is clear from the figure that with the increase in heat flux supplied to heating rod heat transfer coefficient also increasing. As we know with the increase in heat flux wall superheat also increase which increase the boiling rate which in turn increases the heat transfer coefficient.

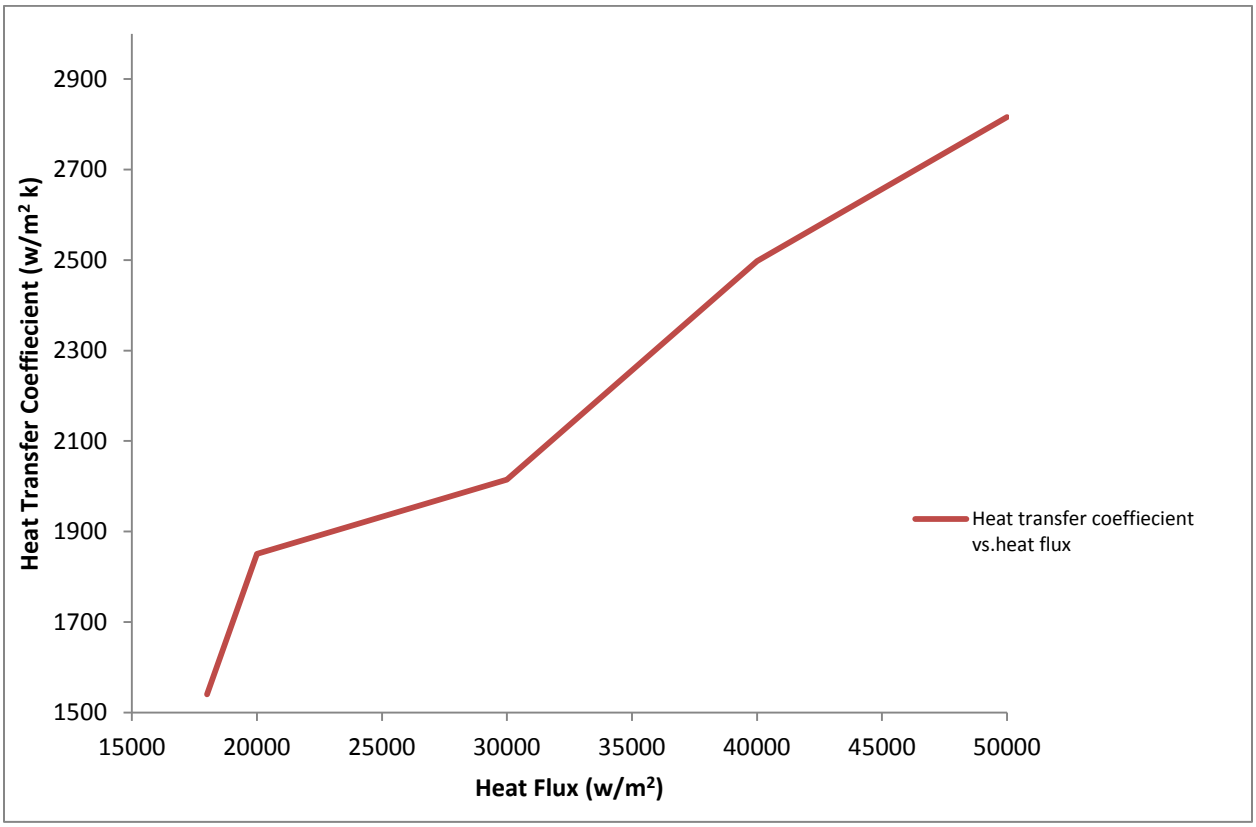

Figure 5.3 Value of heat transfer coefficient with respect to heat flux.

Figure 5.4 show the variation of wall superheat with respect to heat flux. It is clear from the figure that wall superheat increases with the increase in value of heat flux supplied to heating rod. As we know that the wall superheat is difference between wall temperature of heating surface and saturation temperature of liquid being heated, so with the increase in heat flux value supplied to heating rod its temperature would also increase which in turn increases the wall superheat.

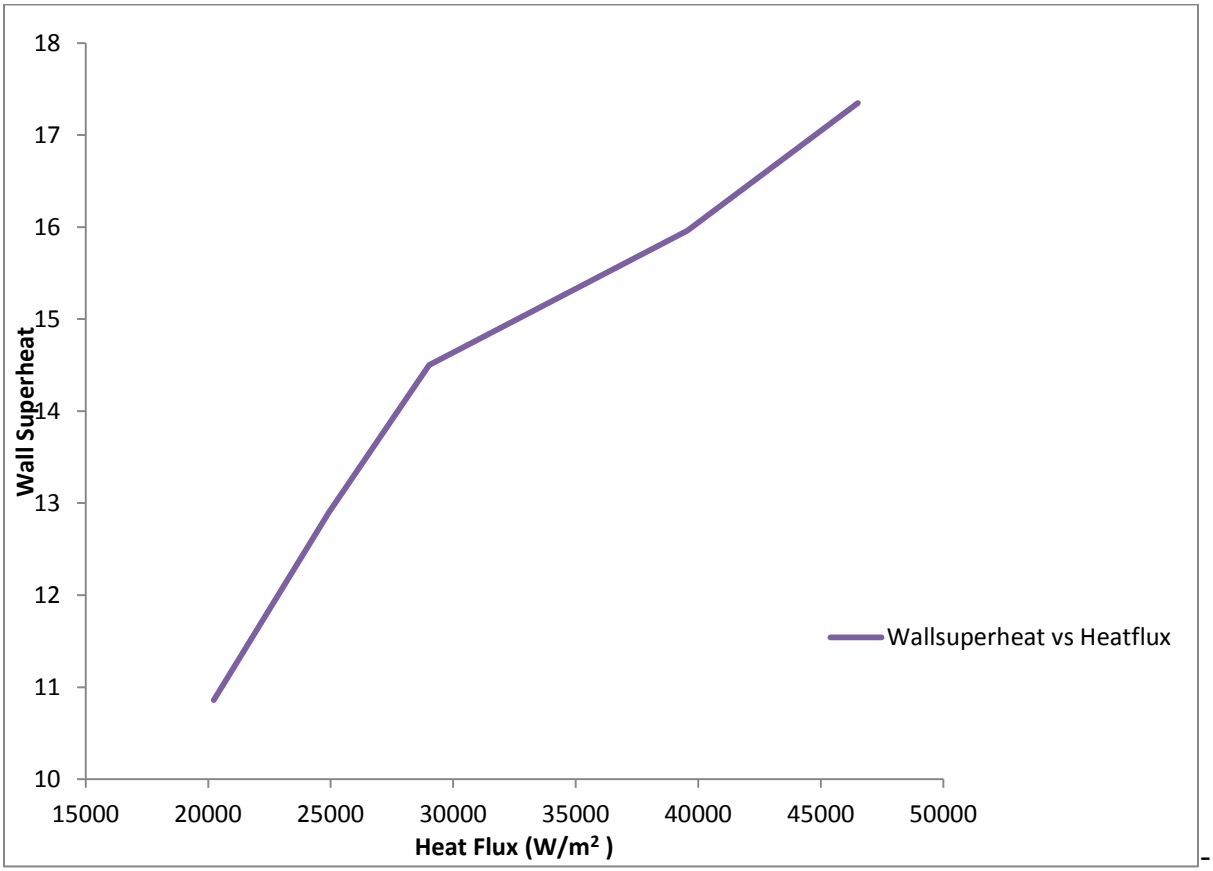

Figure 5.4 Value of Wall superheat with respect to heat flux

Figure 5.5 shows the variation of quenching heat flux to total heat flux with respect to total heat flux. It is clear from the figure that most part of the wall heat flux is transferred by quenching heat transfer. As generation of bubble increases with increase of heat flux, more bubbles will depart from the heating surface and more cooler liquid will come into contact with heating surface thus increasing the rate of heat transfer by quenching.

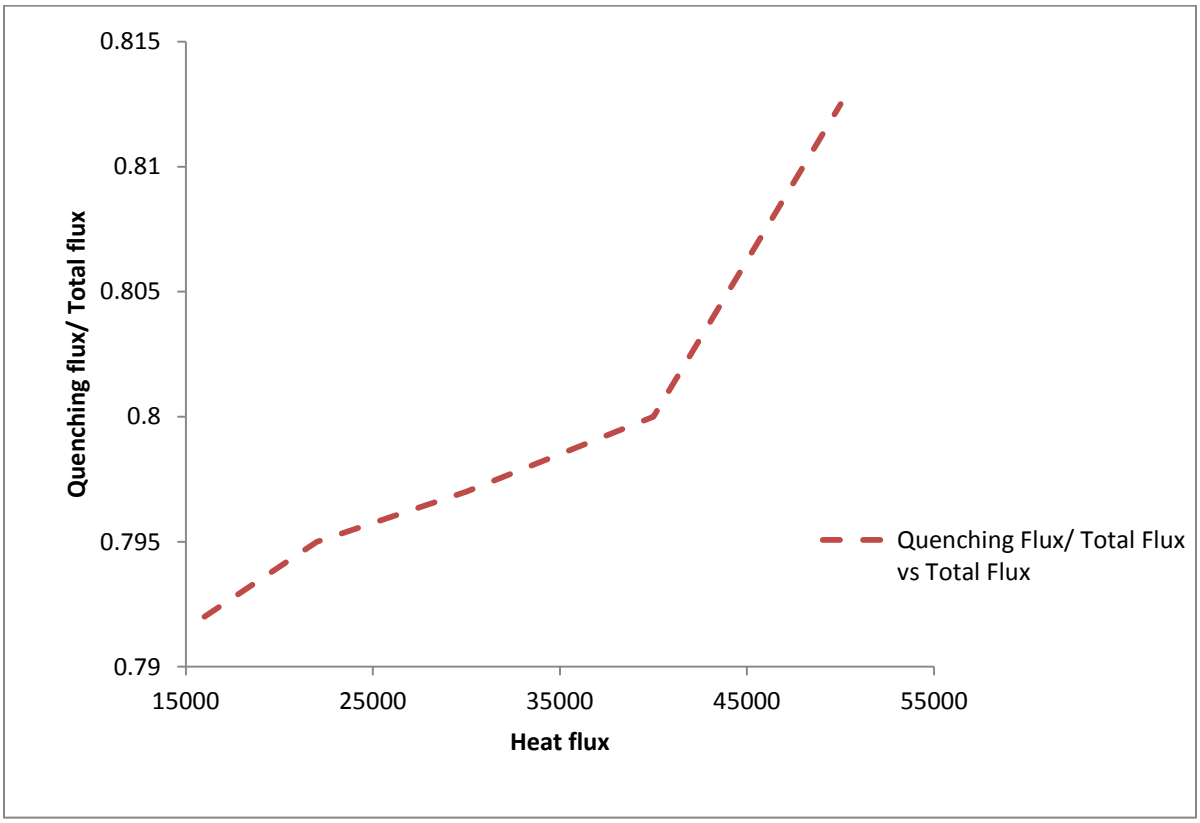

Figure 5.5 Values of quenching flux/ total flux vs. total flux

Figure 5.6 shows the variation of evaporative heat flux to total heat flux with respect to total heat flux. It is clear from the figure that evaporative heat flux increases as total flux increases. As generation of bubble increases with wall heat flux due to which rate of heat transfer by evaporation also increases.

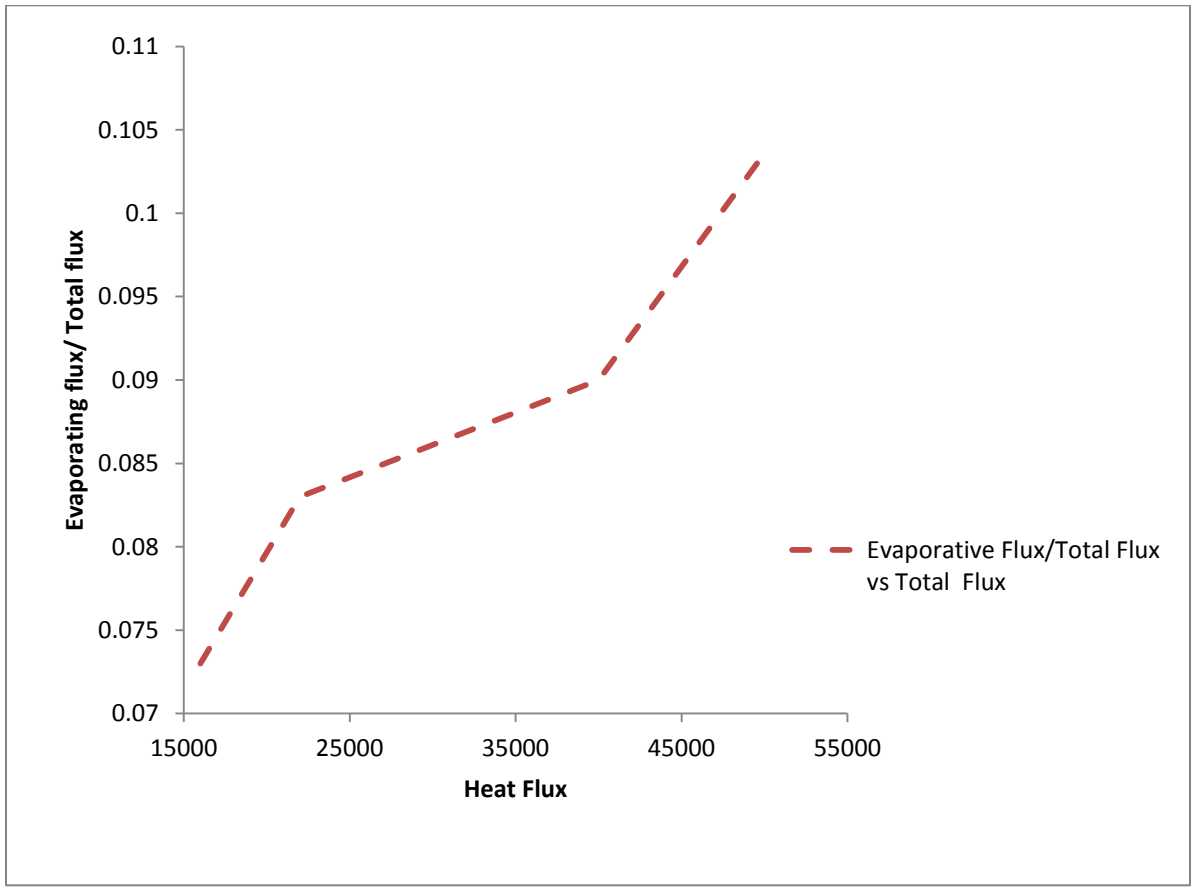

Figure 5.6 Values of evaporative flux/ total flux vs. total flux

Figure 5.7 shows the variation of convective heat flux to total heat flux with respect to total heat flux. It is clear from the figure that convective heat flux decreases as total flux increases. At low heat fluxes less heating surface is occupied by bubbles in comparison to high heat fluxes so heat transfer by convection is more at low value of heat flux.

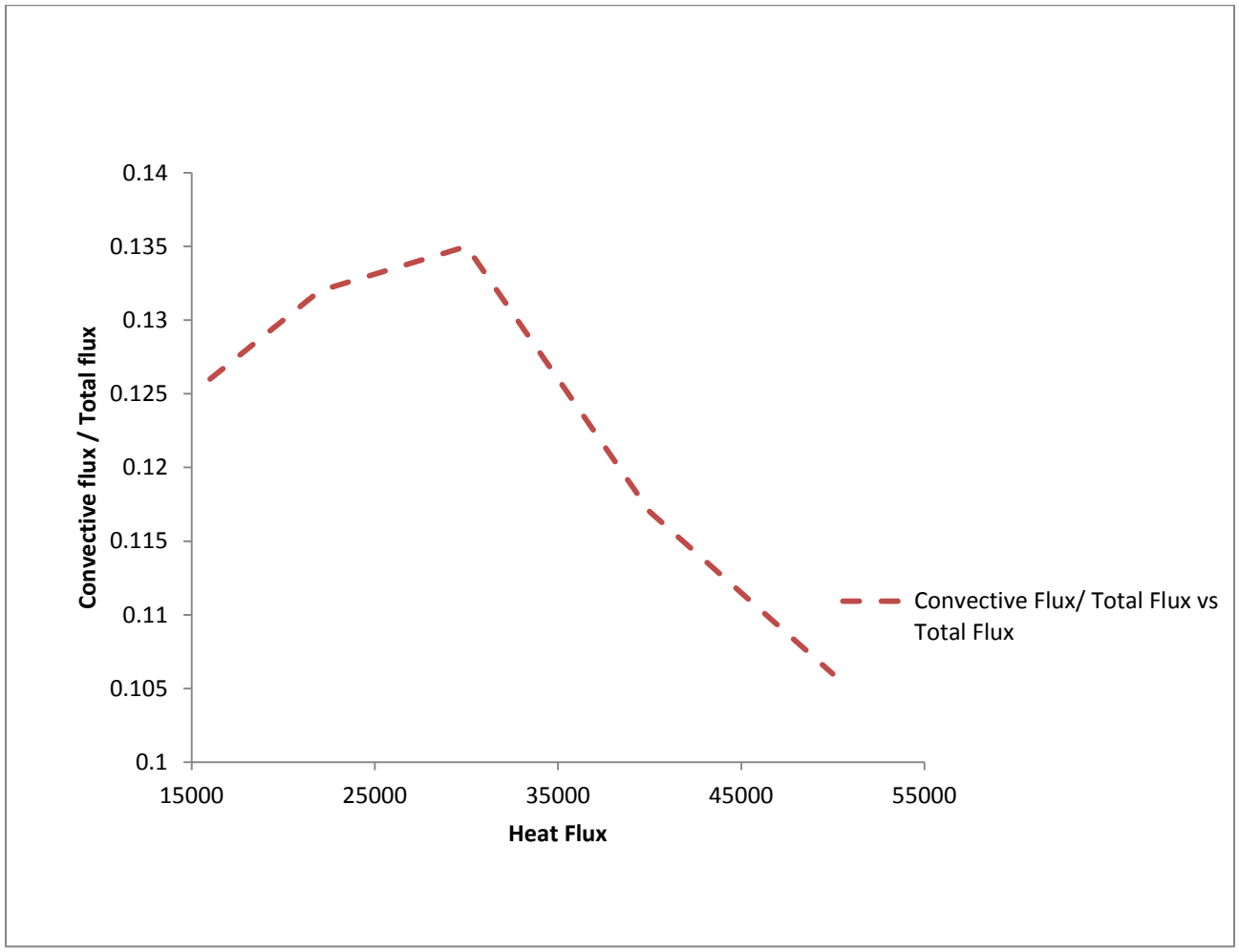

Figure 5.7 Values of evaporative flux/ total flux vs. total flux

- **At pressure equal to .78 atm**

Figure 5.9 shows the comparison of heat transfer coefficient with respect to heat flux at atmospheric and sub-atmospheric conditions. It is clear from the figure that with the increase in pressure, value of heat transfer coefficient also increasing. As we know that with the increase in pressure, surface tension also decreases which increases the formation of more bubbles which in turn increases the heat transfer coefficient.

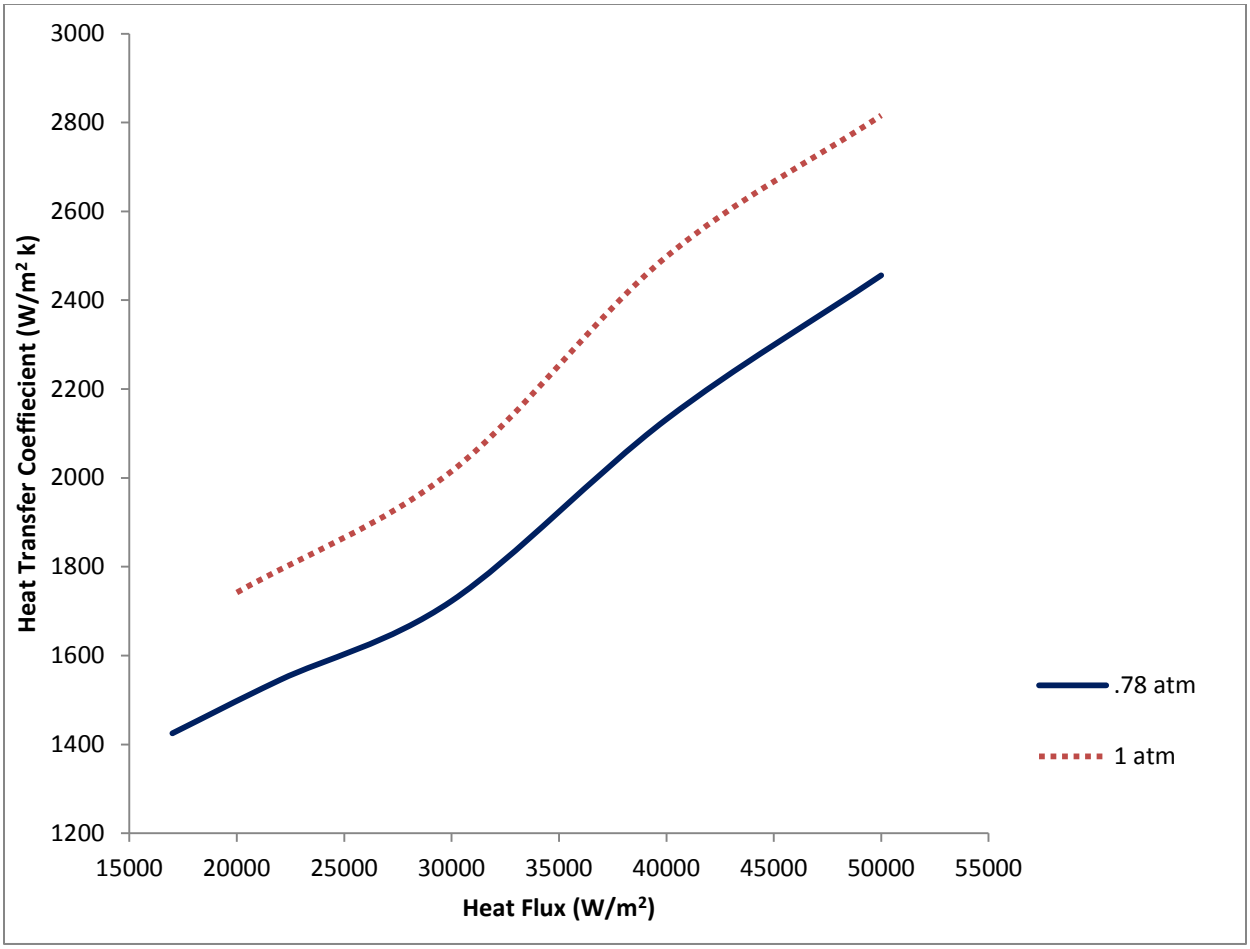

Figure 5.8 Values of heat transfer coefficients with respect to heat flux at different pressure.

5.2 FOR WATER

5.2.1 Grid Sensitivity and Validation of model

Grid sensitivity: Results obtained from simulations with grid size .0005 cm agree well with previously obtained experimental data.

Validation: Figure 5.6 shows the variation of heat transfer coefficient with respect to heat flux with the help of values taken from different literature and present work in order to validate the present work.

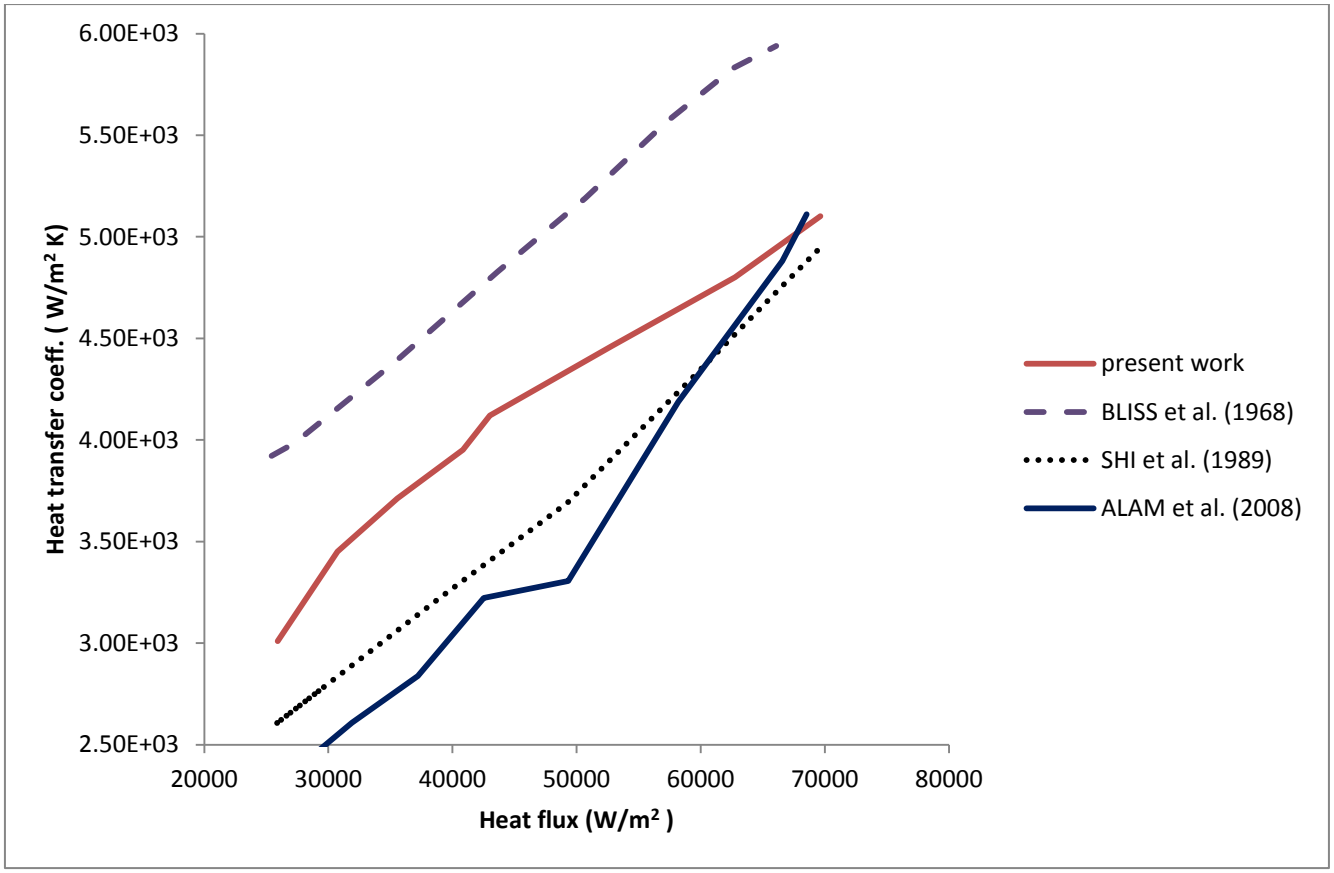

Figure 5.9 Value of heat transfer coefficient with respect to heat flux from literature and present work

5.2.2 **Results for water -** Simulations have been carried out for different constant heat flux value assigned to stainless steel heating rod. Simulations have been carried out at atmospheric and sub-atmospheric pressure.

- **At atmospheric pressure for water**

Figure 5.7 shows variation of heat transfer coefficient with respect to heat flux. It is clear from the figure that with the increase in heat flux supplied to heating rod heat transfer coefficient also increasing. As we know with the increase in heat flux wall superheat also increase which increase the boiling rate which in turn increases the heat transfer coefficient.

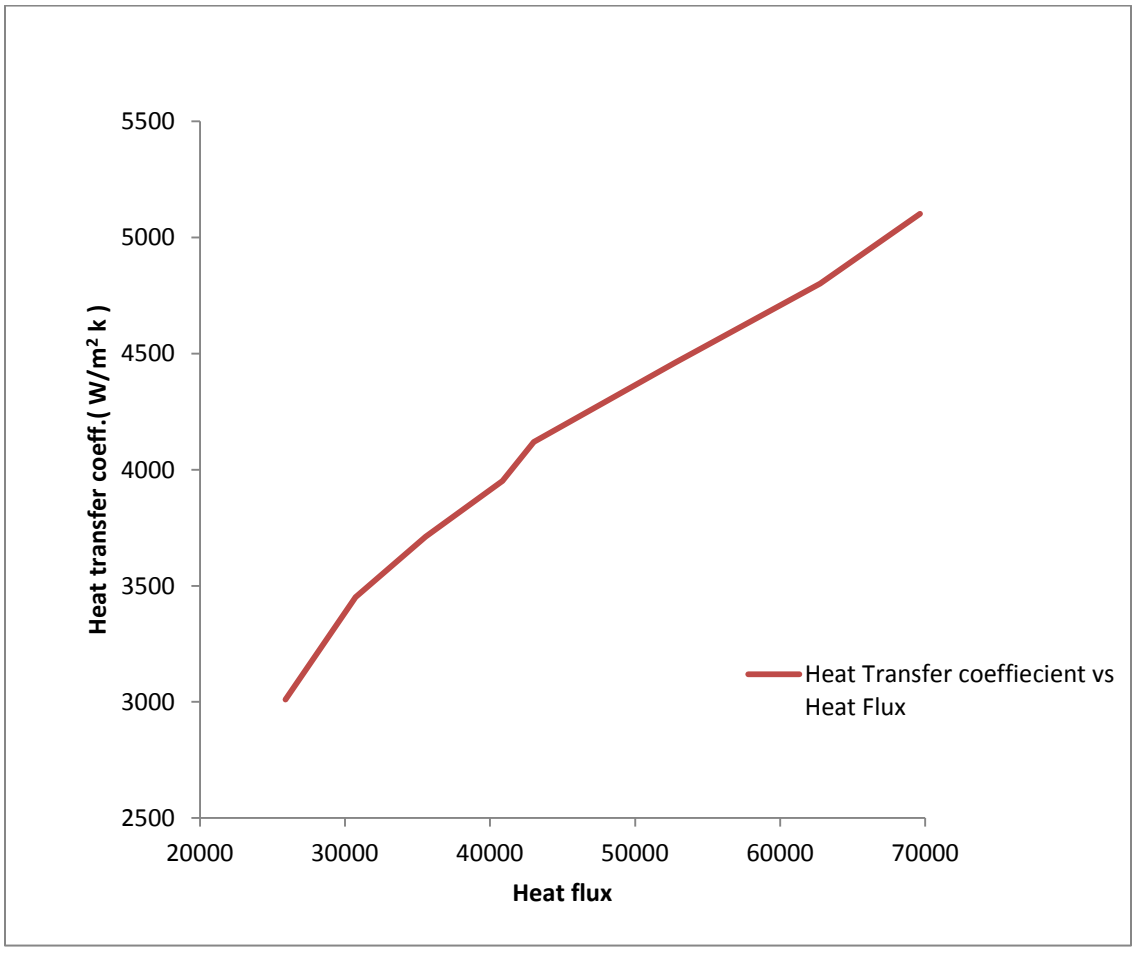

Figure 5.10 Value of heat transfer coefficient with respect to heat flux.

Figure 5.8 shows the variation of wall superheat with respect to heat flux. It is clear from the figure that wall superheat increases with the increase in value of heat flux supplied to heating rod. As we know that the wall superheat is difference between wall temperature of heating surface and saturation temperature of liquid being heated, so with the increase in heat flux value supplied to heating rod its temperature would also increase which in turn increases the wall superheat.

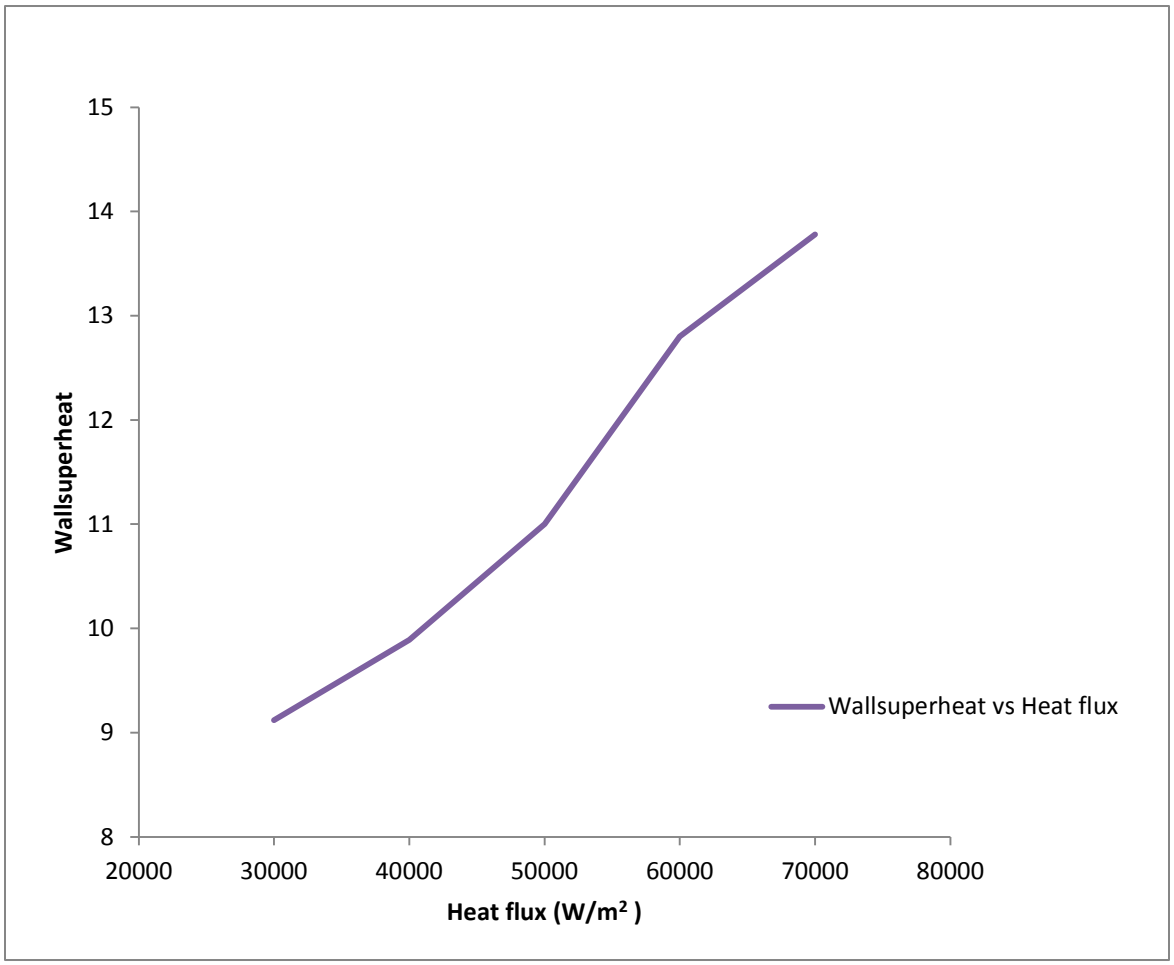

Figure 5.11 Values of wall superheat with respect to heat flux.

- **At pressure equal to .78 atm**

Figure 5.9 shows the comparison of heat transfer coefficient with respect to heat flux at atmospheric and sub-atmospheric conditions. It is clear from the figure that with the increase in pressure, value of heat transfer coefficient also increasing. As we know that with the increase in pressure, surface tension also decreases which increases the formation of more bubbles which in turn increases the heat transfer coefficient.

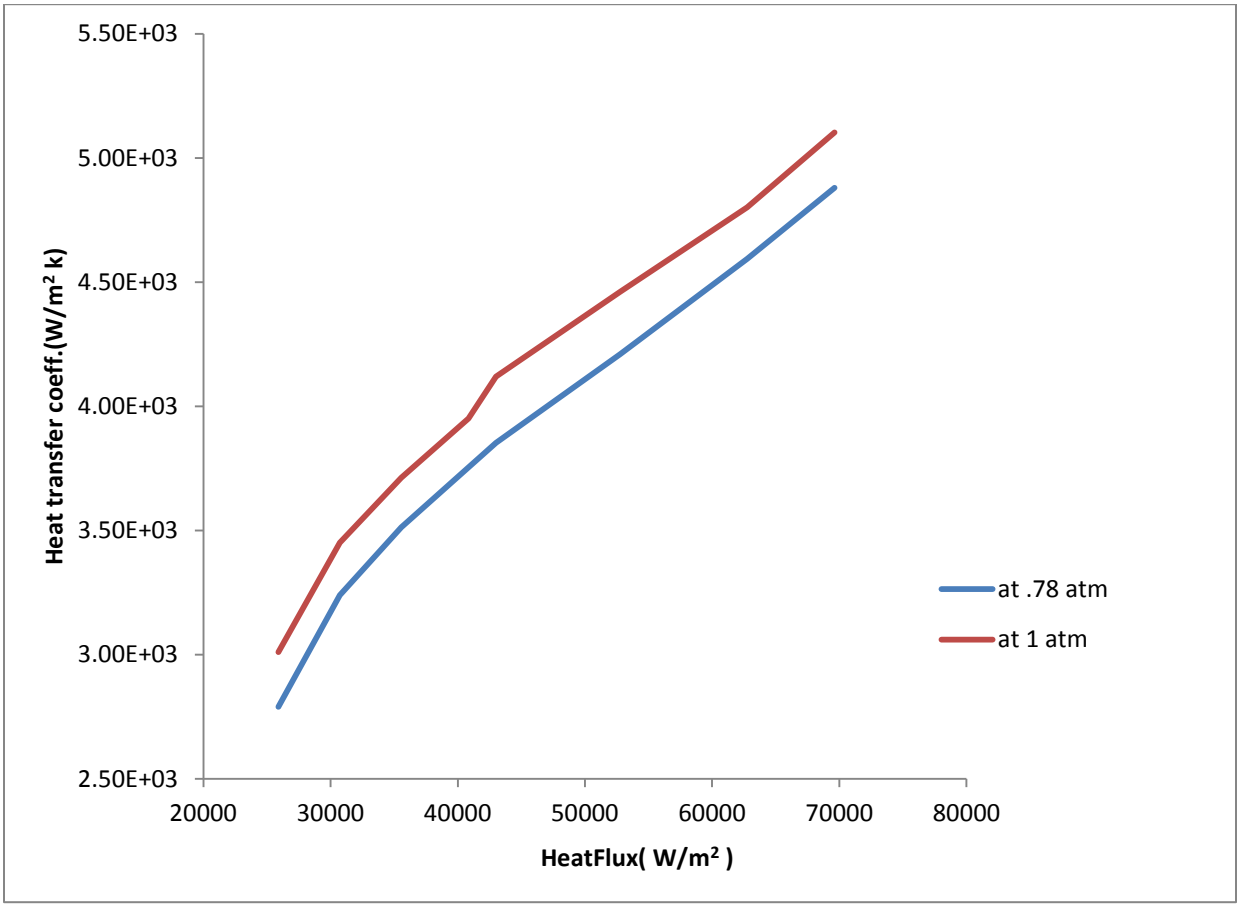

Figure 5.12 Values of heat transfer coefficient with respect to heat flux at different pressure.

CHAPTER-6

CONCLUSION

6.1 CONCLUSION

In the present work numerical simulation has been performed for investigation of nucleate pool boiling in 2D. Geometry for the simulation purpose is taken from the literature **BHAUMIK et al. (2004).**Heating media is benzene and heating rod is of stainless steel .Eulerian –Eulerian approach is used to perform simulations.

- CFD code ANSYS FLUENT 13 is used for the simulation purpose. Eulerian model is enabled for multiphase modeling and for modeling of nucleate pool boiling RPI boiling model is used.
- Fine grid is used for simulation purpose and grid sensitivity analysis is carried out in order to understand the effect of grid size on solution obtained. The results obtained by using mesh size .0003 m are well agreed with the experimental data obtained from literature.
- Results obtained from CFD simulations are validated against classical calculated data of heat transfer coefficient. Data used for validation purpose has taken from the literature ***BHAUMIK et al. (2004).*** Results obtained shows agreement with the data obtained literature.
- From the simulated results it has been found that the heat transfer coefficient increases with the increase in the value of constant heat flux assigned to heating rod. Wall superheat also increases with increase in heat flux value.
- It is also cleared from the result most part of wall heat flux is transferred by quenching heat transfer then as convective and quenching.
- On decreasing the operating pressure the value of heat transfer coefficient also decreases.

Recommendations:

- In the present work numerical simulations are carried out for 2D geometry over stainless steel (uncoated surface). Further simulations can be performed with different heating surface or coated surface.
- Simulation can also be carried out in 3D and results obtained can be compared with the previously obtained 2D results.
- Present work is done for benzene (pure fluid). It can be extended for other fluids and mixtures like binary and ternary systems.

CHAPTER-6

REFERENCES

1. ***A.K. Das, P.K. Das *, P. Saha***, "Nucleate boiling of water from plain and structured Surfaces", Experimental Thermal and Fluid Science 31 (2007) 967–977.

2. ***Abhijit Mukherjee, Satish G. Kandlikar,*** "Numerical study of single bubbles with dynamic Contact angle during nucleate pool boiling", International Journal of Heat and Mass Transfer 50 (2007) 127–138.

3. ***Craig Gerardi, Jacopo Buongiorno *, Lin-wen Hu, Thomas McKrell*** , "Study of bubble Growth in water pool boiling through synchronized, infrared thermometry and high-speed Video", International Journal of Heat and Mass Transfer 53 (2010) 4185–4192.

4. ***David M. Christopher, Zhang Lu,*** "Heat Transfer in the Microlayer under a Bubble During Nucleate Boiling", Tsinghua Science & Technology 15(2010) 404-413

5. ***G.Son, N. Ramanujapu, V. K. Dhir,*** "Numerical Simulation of Bubble Merger Process on Single Nucleation Site During Pool Nucleate Boiling" ,J. heat transfer 124(2002) 51-62.

6. ***G.Vinayak Rao, A.R. Balkrishnan,*** "Heat transfer in nucleate pool boiling of Multicomponent Mixtures",Experimental Thermal and Fluid science 29 (2004) 87-103.

7. ***Gabor Hazi , Attila Markus,*** "On the bubble departure diameter and release frequency based On numerical simulation results", International Journal of Heat and Mass Transfer 52 (2009) 1472–1480.

8. ***Gihun Son a, Vijay K. Dhir***, "Numerical simulation of nucleate boiling on a horizontal Surface at high heat fluxes", International Journal of Heat and Mass Transfer 51 (2008) 2566 -2582.

9. ***H. Aminfar , M. Mohammadpourfard , M. Sahraro, "***Numerical simulation of nucleate Pool boiling on the horizontal surface for nano-fluid using wall heat flux partitioning method" , Computers and Fluids 66(2012) 29-38.

10. ***Han Young Yoon, Seiichi Koshizuka, Yoshiaki Oka,*** "Direct calculation of bubble growth, departure, and rise in nucleate pool boiling" , International Journal of Multiphase Flow 27 (2001) 277-298

11. ***Helfried Steine, Alexander Kobor, Ludwig Gebhard,*** "A wall heat transfer model for Subcooled boiling flow", International Journal of Heat and Mass Transfer 48(2005) 4161– 4173.

12. ***Jeongbae Kim, Byung Do Oh, Moo Hwan Kim, "***Experimental Study of Pool Temperature Effects on Nucleate Boiling" ,International Journal of Multiphase Flow 32 (2006) 208-231.

13. ***L.Vyskocil, J.Macek,*** "Boiling flow simulation in NEPTUNE_CFD and FLUENT CODES", Nuclear Research Institute (NRI), Dept. of Thermal hydraulic Analyses (2008).

14. ***Petra Genske, Karl Stephan,*** "Numerical simulation of heat transfer during growth of Single vapor bubbles in nucleate boiling", International Journal of Thermal Sciences 45 (2006) 299-309.

15. ***R. Hosseini, A. Gholaminejad, H. Jahandar*** , "Roughness Effects on Nucleate Pool boiling of R-113 on Horizontal Circular Copper Surface" , World Academy of Science, Engineering and Technology 79 (2011) 679-684.

16. ***Rui Zhuan, Wen Wang***, "Simulation on nucleate boiling in micro-channel", International Journal of Heat and Mass Transfer 53 (2010) 502–512.

17. ***Seungyeob Ryu, Sungho Ko,*** "Direct numerical simulation of nucleate pool boiling using a two-dimensional lattice Boltzmann method", Nuclear Engineering and Design 248 (2012) 248–262.

18. ***Sreekant Narumanchia, Andrey Troshk , Desikan Bharathan, Vahab Hassani,*** "Numerical Simulations of nucleate boiling in impinging jets: Applications in power electronics cooling" International Journal of Heat and Mass Transfer 51 (2008) 1–12.

19. ***Y.H.Zhao, Y.H.Diao , T. Takaharu,*** "Experimental investigation in nucleate pool Boiling of refrigerant mixtures", Applied Thermal Engineering 28 (2008) 110-115.

20. ***S.Bhaumik, V.K Agarwal, S.C Gupta,*** "A generalized correlation of nucleate pool boiling of liquids.", Indian Journal of Chemical Technology vol.11 September 2004 pp 719-725.