IJERST

International Journal of Engineering Research and Science & Technology

ISSN : 2319-5991
Vol. 4, No. 1
February 2015

www.ijerst.com

Email: editorijerst@gmail.com or editor@ijerst.com

International Journal of Engineering Research and Science & Technology

ISSN 2319-5991 www.ijerst.com
Vol. 4, No. 1, February 2015

Research Paper

NUMERICAL SIMULATION OF NUCLEATE BOILING CASE USING FLUENT

Sphurti Sweta[1]*, Suresh Kumar Badholiya[2] and Santosh Dixit[3]

**Corresponding Author:Sphurti Sweta* ✉ *sphu.swe@gmail.com*

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 axisymmetrical 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

INTRODUCTION

When boiling happens on a solid surface at low superheat, bubbles will be seen to make repeatedly at most popular positions referred to 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 microgeometry of the surface on a micrometer 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 that the heat is transferred thus phenomenological models for nucleate boiling at the present do no higher, and sometimes worse, than the empirical correlations associate empirical correlation of

[1] M.Tech Scholar, Bhopal Institute of Technology, Bangrasia, Bhopal, Madhya Pradesh 462045.
[2] Assistant Professor, Bhopal Institute of Technology, Bangrasia, Bhopal, Madhya Pradesh 462045.
[3] Ansys Incharge (MYSL), Bhopal Institute of Technology, Bangrasia, Bhopal, Madhya Pradesh 462045.

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 (see Reduced Properties) (Bar-Cohen, 1992).

Computer Fluid Dynamics

Computational fluid dynamics, sometimes abbreviated as CFD, may be a branch of hydraulics that uses numerical ways and algorithms to resolve and analyze issues that involve fluid flows. Computers are accustomed perform the calculations needed to simulate the interaction of liquids and gases with surfaces outlined by boundary conditions. With high-speed supercomputers, higher solutions are often achieved in progress analysis yields software package that improves the accuracy and speed of advanced simulation eventualities like sonic or turbulent flows (Blaner, 1975; Chen, 1966).

Computational fluid dynamics, sometimes abbreviated as CFD, may be a branch of hydraulics that uses numerical ways and algorithms to resolve and analyze issues that involve fluid flows. Computers ar accustomed perform the calculations needed to simulate the interaction of liquids and gases with surfaces outlined by boundary conditions. With high-speed supercomputers, higher solutions are often achieved in progress analysis yields software package that improves the accuracy and speed of advanced simulation eventualities like sonic or turbulent flows. Initial experimental validation of such software package is performed employing a construction with the ultimate validation returning in all-out testing (Cooper, 1981)

FINITE VOLUME METHOD

In the finite volume technique, the governing partial differential equations (typically the Navier-Stokes equations, the mass and energy conservation equations, and also the turbulence equations) area unit recast in an exceedingly conservative type, so solved over distinct management volumes (Cornwell, 1990).

Abbreviations and Acronyms

CFD-Computer Fluid Dynamics, N-S Equation-Navier Strokes Equation, Cp-SpecificHeat, FVM-Finite Volume Method, FEM-Finite Element Method, FDM-Finite Distinction Method, RANS-Reynolds-Sveraged Navier-Stokes, RSM-Reynold Stress Model

EQUATIONS

The finite volume equation yields governing equations within the type, Equation (1) (del Valle, 1985)

$$\frac{\partial}{\partial t}\iiint QdV + \iint FdA = 0 \qquad ...(1)$$

where Q, is the vector of conserved variables, F is the vector of fluxes (see Euler equations or Navier–Stokes equations), V is the volume of the control volume element, and A is the surface area of the control volume element.

In FVM technique, a weighted residual equation is formed (Dhir, 1990; Fujita, 1992)

$$R_i = \iiint W_i Q dV^e \qquad ...(2)$$

where R_i is the equation residual at an element vertex *i*, *Q* is the conservation equation expressed on an element basis, W_i is the weight factor, and V^e is the volume of the element (Gorenflo, 1992).

Finite distinction technique (FDM) (Judd and Chopra, 1993)

$$\frac{\partial Q}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = 0$$

where Q is the vector of conserved variables, and F, G, and H are the fluxes in the x, y, and z directions respectively.

REYNOLD-AVERAGED NAVIER-STOKES (CFD, 2011)

Reynolds-averaged Navier-Stokes (RANS) equations square measure the oldest approach to turbulence modeling. AN ensemble version of the governing equations is resolved (Pope, 2000) that introduces new apparent stresses called painter stresses. This adds a second order tensor of unknowns that varied models will give totally different levels of closure. It's a typical idea that the RANS equations don't apply to flows with a time-varying mean flow as a result of these equations square measure 'time-averaged'. In fact, statistically unsteady (or non-stationary) flows will equally be treated this is often typically brought up as URANS. There's nothing inherent in painter averaging to preclude this, however the turbulence models accustomed shut the equations square measure valid solely as long because the time over that these changes within the mean occur is giant compared to the time scales of the turbulent motion containing most of the energy (Farge, 2001).

RANS models are often divided into 2 broad approaches: (Goldstein, 1995) Boussinesq Hypothesis This methodology involves mistreatment AN pure mathematics equation for the painter stresses that embrace deciding the turbulent body, and looking on the extent of sophistication of the model, resolution transport equations for deciding the turbulent mechanical energy and dissipation. Models embrace k-ε (Launder and Spalding), (Lundgren, 1969) mixture Length Model (Prandtl), (Coulucci, 1998) and 0 Equation Model (Cebeci and Smith) (Fox, 2003). The models on the market during this approach square measure usually brought up by the quantity of transport equations related to the tactic for instance, the blending Length model may be a "Zero Equation" model as a result of no transport equations square measure resolved; the may be a "Two Equation" model as a result of 2 transport equations (one for and one for) square measure solved (Pope, 1985).

REYNOLD STRESS MODEL (RSM)

This approach tries to really solve transport equations for the painter stresses. This suggests introduction of many transport equations for all the painter stresses and thence this approach is far a lot of pricey in CPU effort (Zhihao, 2015)

LITERATURE REVIEW

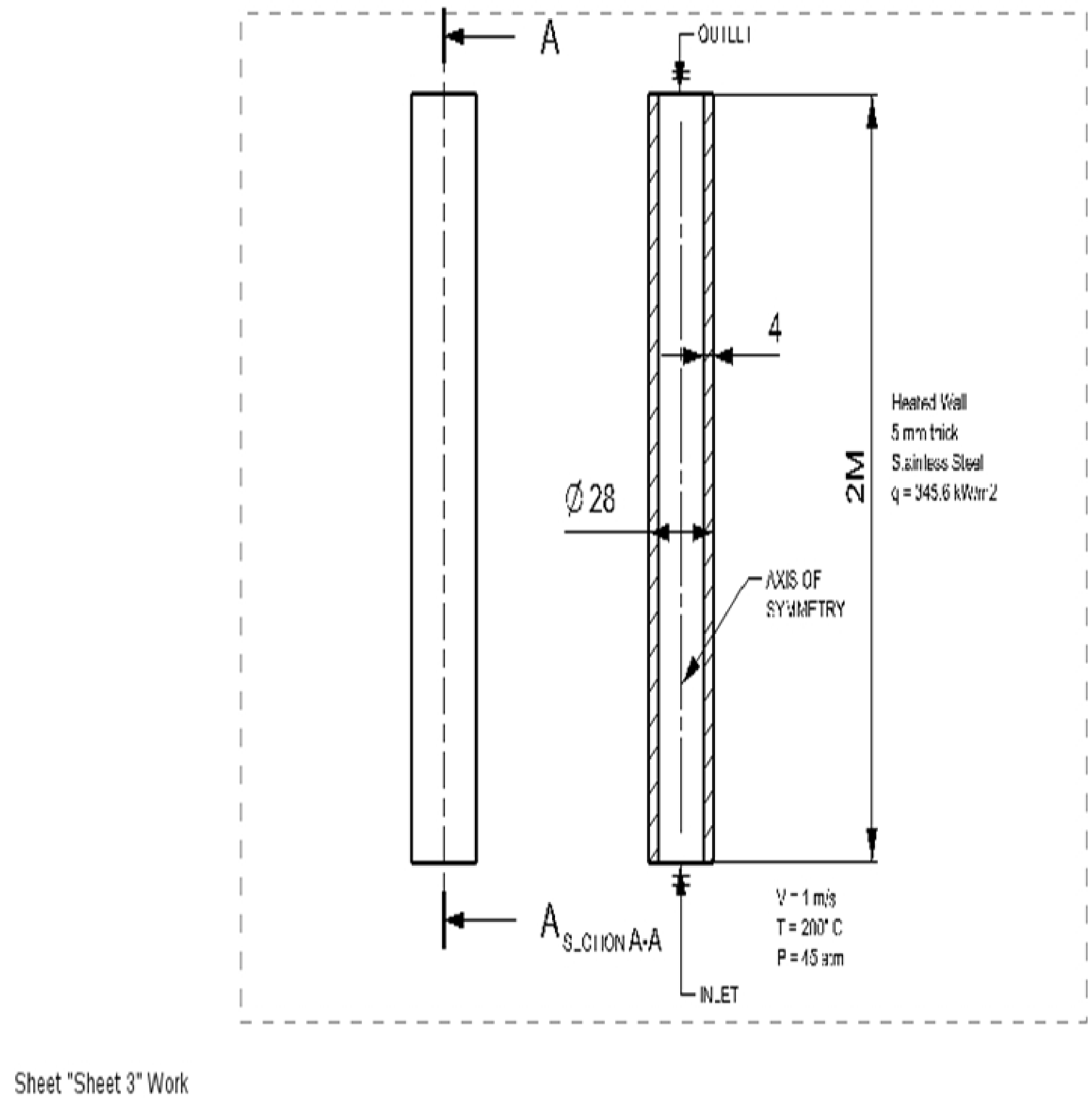
In this work we have a tendency to investigate the current capabilities of CFD for wall boiling. The procedure model used combines the Euler/ Euler two-phase flow description with heat flux partitioning terribly similar modelling was antecedently applied to (Sattari, 2014) boiling water beneath air mass conditions relevant to atomic energy systems. Similar conditions in terms of the relevant non-dimensional numbers are realized within the DEBORA tests exploitation dichlorodifluoromethane (R12) because the operating fluid. This expedited measurements of radial profiles for gas volume fraction, gas rate, liquid temperature and bubble size (Kazuo, 2014).

After reviewing the theoretical and experimental basis of correlations employed in the model, provides a careful assessment of the mandatory recalibrations to explain the DEBORA tests (Bestion, 2014). It's then shown that at intervals an explicit vary of conditions completely different tests will be simulated with one set of model parameters, because the subcooling is remittent and also the quantity of generated vapor will increase the gas fraction profile changes from wall to core peaking. This can be a significant impact not captured by the current modelling (Mostafa, 2014).

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 1. The pipe is 20 mm in diameter 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 centre of the pipe Outlet profiles of velocity magnitude and turbulence quantities generated for a simulated flow field without boiling

Figure 1: Schematic Diagram



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.

METHODOLOGY

CFD Governing Equations (Mostafa, 2014)

This section is a summary of the governing equations used in CFD to mathematically solve for fluid flow and heat transfer, based on the principles of conservation of mass, momentum, and energy. Details of how they are actually used in the CFD Computations (Farge and Schneider, 2001)

Conservation Equations: The conservation laws of physics form the basis for fluid flow governing equations (previously listed as Equations 1-3 in Section 2.1: Governing Equations and Numerical Schemes). The laws are:

a. Law of Conservation of Mass: Fluid mass is always conserved. (Equation 1) (Sreeyuth *et al.,* 2014).

$$\frac{\partial(\rho u_i)}{\partial x_i} = 0$$

b. Newton's 2nd Law: The sum of the forces on a fluid particle is equal to the rate of change of momentum. (Equation 2) (Lamas *et al.,* 2012).

$$\frac{\partial}{\partial x_i}(\rho u_i u_j) = \frac{\partial}{\partial x_i}\left(\mu \frac{\partial u_j}{\partial x_i}\right) - \frac{\partial p}{\partial x_j}$$

c. First Law of Thermodynamics: The rate of head added to a system plus the rate of work done on a fluid particle equals the total rate of change in energy. (Equation 3) (Zhang *et al.,* 2014).

$$\frac{\partial}{\partial x_i}(\rho u_i T) = \frac{\partial}{\partial x_i}\left(\frac{k}{C_p}\frac{\partial u_j}{\partial x_i}\right)$$

The fluid behavior can be characterized in terms of the fluid properties velocity vector u (with components *u*, *v*, and *w* in the *x*, *y*, and *z* directions), pressure *p*, density ρ, viscosity ì, heat conductivity *k*, and temperature *T*. The changes in these fluid properties can occur over space and time (Xiumin *et al.,* 2008). Using CFD, these changes are calculated for small elements of the fluid, following the conservation laws of physics listed above. The changes are due to fluid flowing across the boundaries of the fluid element and can also be due to sources within the element producing changes in fluid properties. This is called the Euler method (tracking changes in a stationary mass while particles travel through it) in contrast with the Lagrangian method (which follows the movement of a single particle as it flows through a series of elements) (Xiumin, 2008).

RESULT

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 dissipation rate.
5. Material: We use water liq H_2O(L) in this model density as piecewise linear profile of temperature.
6. Set point 1-Temp-473.15 K Density-864.7 kg/m³

   Set point 2.-Temp -543.15 K Density-770.6 kg/m³
7. Cell zone condition:Fluid (water liq).
8. Boundary conditions:

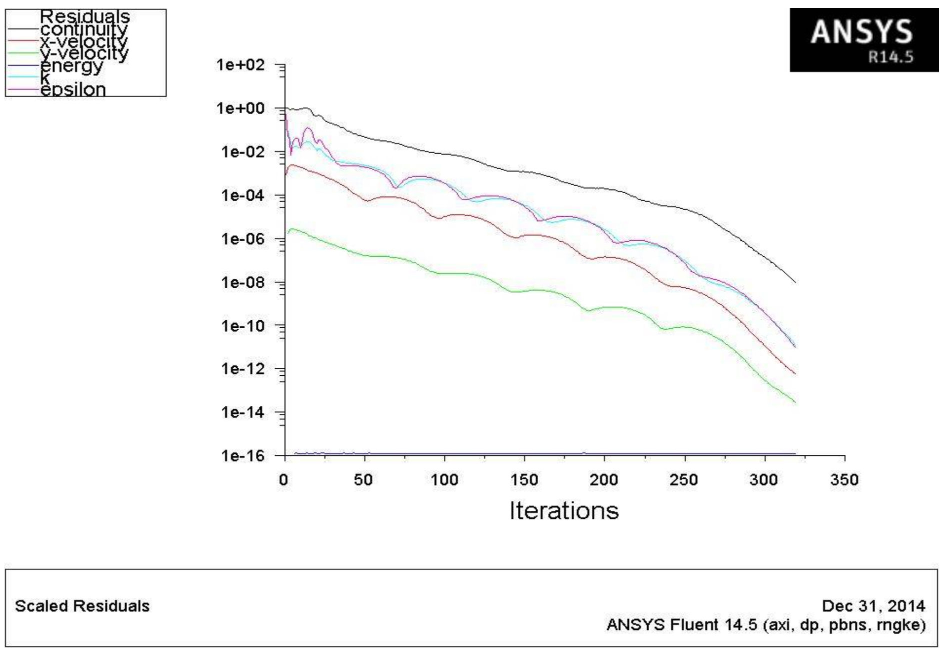
8.1. Inlet Type: Normal to boundary as the velocity specification method (a) Magnitude-1 m/s; (b) Turbulent intensity-4%; (c) Hydraulic dia-.0154; (d) Thermal condition is 473.15K at the inlet.

8.2. Oulet Type: Gauge pressure-0 atm Type-Normal to boundary (a) Turbulent intensity-4%; (b) hydraulic dia-.0154 3; (c) Thermal condition is 530.55 to back flow temperature.

9. Operating condition-Pressure-4.5e^6 and Gravity-9.81 m/s^2

10. Convergence criteria:For continuity 1e^-8 for all the remaining residual we use 1e^5

Figure 2: Residual Plot of Single Phase



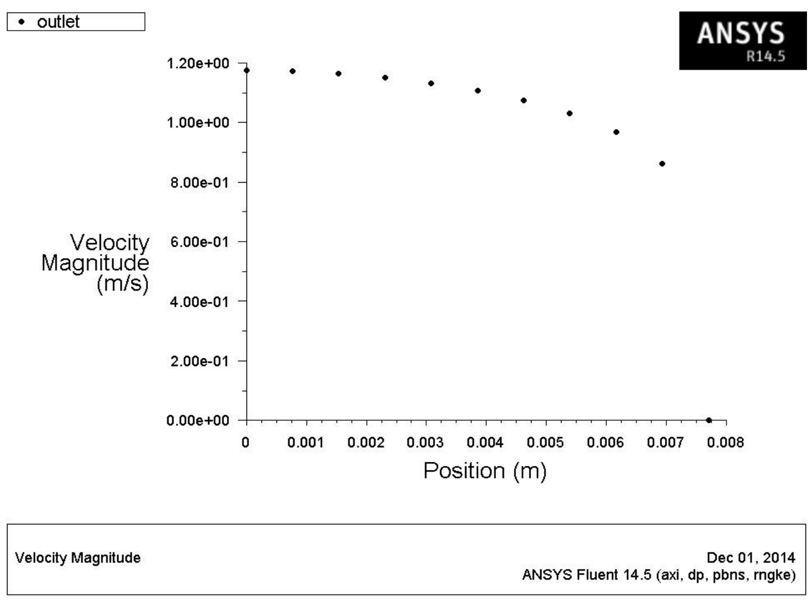


Figure 3: Contours of Mass Imbalance

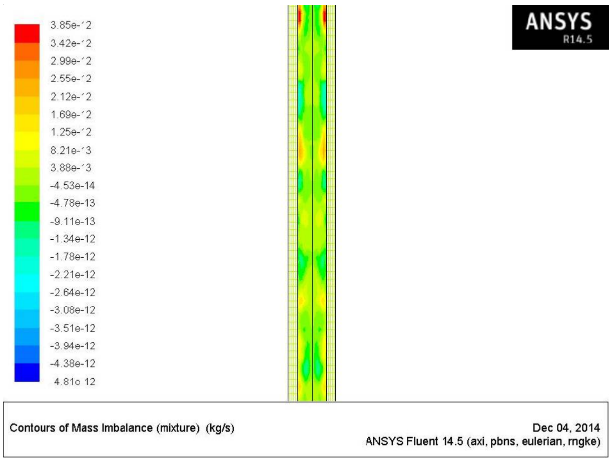
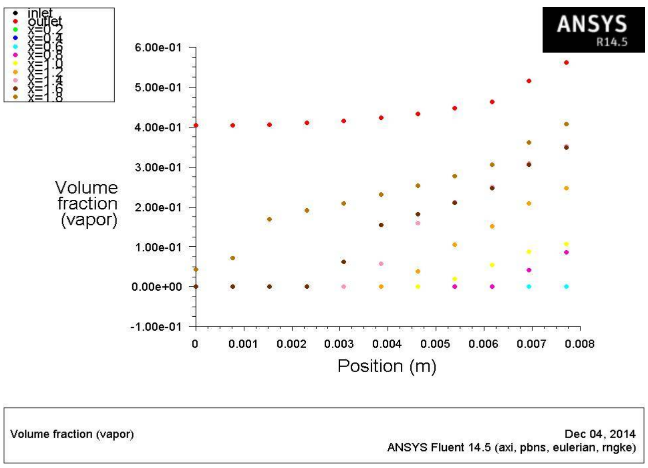


Figure 4: Convergence History of Mass Flow Rate

Figure 5: Profile of Volume Fraction



11. Now initialize the solution .
12. Run the cal for 500 iteration.
13. We see in the process the solution convergences in 320 iteration approx.

Fully developed velocity profile for single phase flow.

Finally, we get the fully developed velocity profile by the CFD Fluent Ansys.

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 .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.

Fully developed profile are get by Ansys 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. Velocity boundary condition at the channel inlet forces the bubble to move downstream but the thinning of the liquid 1m 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 liquid velocity in the proximity of the channel wall is temporarily increased, therefore the development of the thermal layer to the steady situation is delayed. The measured liquid 1m thickness gives D = 20. For such thickness the lm can not be assumed stagnant and the assumption of pure heat conduction cross the lm leads to overestimation of the heat transfer coefficient. When multiple bubbles evaporate in sequence, the bubble ahead cools down the thermal layer such that the bubble behind grows less.

ACKNOWLEDGMENT

The first and biggest acknowledgment is for my supervisor Mr. Suresh Badholiya (Assistant Professor, BITS, Bhopal) who gave me the opportunity to begin this post graduation course. Mr. Suresh Badholiya (Assistant Professor, BITS, Bhopal) has been my supervisor since my synopsis work and I am grateful for her constant guidance.

I am thankful to the Head, Department of Mechanical Engineering BITS, Bhopal Mr. Rohit Chaudhary, for making all the facilities available and for providing me the unstinting support and guidance in the absence of which the thesis would have been the shadow of its present form.

I also humbly acknowledge the guidance and support of my esteemed Faculty, who have also been the guiding light through out the Project. My sincere gratitude to Respected Mr. Ajit Rastogi, Mr. Harisharn Sahu, and all the staff members of Mechanical Department of BITS Bhopal. I own my deep sense of gratitude and thanks for their constructive criticism, valuable suggestions and constant encouragement at all stages of the development of the Project.

I am also thankful to Mr. Santosh Dixit CFD, Ansys incharge RISE Bhopal for given his kind support for my dissertation work. I remember with gratitude the suggestions and good wishes of Friends in other Universities and Institutions name as Mr. Rahul Kumar Singh (AISECT, Bhopal), Mr. Chandra Raj Singh Baghel (GEC, Bhopal). And no words to thank the Almighty for the profuse blessings. Last but not least, I thank my family for their constant support especially my father Mr. R K Pandey, who always supports me and motivate to do better in my work as well as life.

REFERENCES

1. Bar-Cohen A (1992), "Hysteresis phenomena at the onset of nucleate boiling", Proc. Engineering Foundation Conf. on Pool and External Flow Boiling, Santa Barbara, pp. 1-14.

2. Bestion D (2014), "The difficult challenge of a two-phase CFD modelling for all flow regimes", *Nuclear Engineering and Design*, Vol. 279, November 2014, pp. 116-125.

3. Blander M and Katz J L (1975), "Bubble nucleation in liquids", *AICHE Journal,* Vol. 21, pp. 833–848.

4. Bloch B, Muselmann W, Saier M, Bloch G, Muselmann W and Saier M (2013), "A phenomenological study on effects leading to the departure from nucleate boiling in subcooled flow boiling", *International Journal of Heat and Mass Transfer*,Vol. 67, December, pp. 61-69

5. Chen J C (1966), "Correlation for boiling heat transfer to saturated fluids in convective flow", *Ind Eng. Chem . Process Design and Development,* pp. 322–329.

6. Cooper M G and Chandratilleke T (1981), "Growth of diffusion-controlled vapor bubbles at a wall in a known temperature gradient", *Int.J.Heat Mass Transfer,* Vol. 24, pp. 1475–1492. DOI: 10.1016/0017-9310(81)90215-5.

7. Colucci P J, Jaberi F A, Givi P and Pope S B (1998), "Filtered density function for large eddy simulation of turbulent reacting flows", *Physics of Fluids A,* Vol. 10, No. 2, pp. 499–515. Bibcode:1998PhFl...10..499C. doi:10.1063/1.869537.

8. Cornwell K (1990), "The influence of bubbly flow on boiling from a tube in a bundle", *Int.J.Heat Mass Transfer,* Vol. 33, pp. 2579–2584. DOI: 10.1016/0017-9310(90)90193-X.

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

10. Dhir V K (1990), "Nucleate and transition boiling under pool and external flow conditions", Proc. 9th Int. Heat Transfer Conf., Jerusalem, Vol. 1, pp. 129–156, DOI: 10.1016/0142-727X(91)90018-Q.

11. Eckhard Krepper and Roland Rzehak (2011), "CFD for subcooled flow boiling: Simulation of DEBORA experiments Nuclear Engineering and Design", Vol. 241, September 2011, pp. 3851-3866.

12. Farge Marie and Schneider Kai (2001), "Coherent Vortex Simulation (CVS), A Semi-Deterministic Turbulence Model Using

Wavelets", *Flow, Turbulence and combustion,* Vol. 66, No. 4, pp. 393–426. doi:10.1023/A:1013512726409.

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

14. Fox Rodney (2003), *Computational models for turbulent reacting flows,* Cambridge University Press.

15. Goldstein Daniel and Vasilyev Oleg (1995), "Stochastic coherent adaptive large eddy simulation method", *Physics of Fluids* A, Vol. 24, No. 7, pp. 2497. Bibcode:2004PhFl...16.2497G. doi:10.1063/1.1736671.

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

17. Judd R L and Chopra A (1993), "Interaction of the nucleation processes occurring at adjacent nucleation sites", *J. Heat Transfer,* Vol. 115, pp. 955–962 DOI: 10.1115/1.2911392.

18. Kazuo Ikeda (2014), "CFD application to advanced design for high efficiency spacer grid", *Nuclear Engineering and Design*, Vol. 279, November, pp. 73-82.

19. Lamas M I, Sáiz Jabardo J M, Arce A and Fariñas P (2012), "Numerical analysis of the bubble detachment diameter in nucleate boiling", University of A Coruña, Mendizábal s/n, 15403 Ferrol, A Coruña, Spain

20. Lundgren T S (1969), "Model equation for nonhomogeneous turbulence", *Physics of Fluids A,* Vol. 12, No. 3, pp. 485–497. Bibcode: 1969PhFl...12..485L. doi:10.1063/1.1692511.

21. Pope S B (1985), "PDF methods for turbulent reactive flows", *Progress in Energy and Combustion Science*, Vol. 11, No. 2, pp. 119–192. Bibcode:1985PrECS..11..119P. doi:10.1016/0360-1285(85)90002-4.

22. Pope S B (2000), *Turulent Flows*, Cambridge University Press. ISBN 978-0-521-59886-6.

23. Paz C, Suárez E, Concheiro M, Copo A and Tobio V (2013), "CFD Implementation And Experimental Validation Of The Chen Model For Heat Transfer In Nucleate Boiling", Author(s), Published.

24. Sattari E attari E, Delavar M A, Fattah E and Sedighi K (2014), "Numerical investigation the effects of working parameters on nucleate pool boiling", *International Communication in Heat and Mass Transfer*, Vol. 59, December, pp. 106-113.

25. Sina Nabati Shoghl, Masoud Bahrami and Mostafa Keshavarz (2014), "Experimental investigation and CFD modeling of the dynamics of bubbles in nanofluid pool boiling", *International Communications in Heat and Mass Transfer*, Vol. 58, November, pp. 12-24.

26. Mostafa Keshavarz Moraveji (2014), "Experimental investigation and CFD modeling of the dynamics of bubbles in nanofluid pool boiling", *International Communications in Heat and Maa Transfer*, Vol. 58, November, pp. 12-24

27. "Numerical predictions of flow boiling characteristics: Current status, model setup and CFD modeling for different non-uniform heating profiles", *Applied Thermal*

Engineering, In Press, Corrected Proof, Available Online 20 September 2014.

28. Sreeyuth Lal, Yohei Sato and Bojan Niceno (2014), "Direct numerical simulation of bubble dynamics in subcooled and near-saturated convective nucleate boiling", *International Journal of Heat and Fluid Flow*, In Press, Corrected Proof, Available online 27 November 2014.

29. Tao Sun (2013), "Three-dimensional numerical simulation of nucleate boiling bubble by lattice Boltzmann method", *Computers and Fluids*, Vol. 88, 15 December 2013, pp. 400-409.

30. Xiumin Chen Haibo Y U, Huang Haizhen and Chen QunGao Ying (2008), "Development of Numerical Simulation on Flow and Heat Transfer in Internal Combustion Engine Cooling System", *Chinese Journal of Mechanical Engineering*, No. 10, pp. 162-167.

31. Wadim Jaeger, Jorge Perez Manes, Uwe Imke, Javier Jimenez Escalante and Victor Sanchez Espinoza (2013), "Validation and comparison of two-phase flow modeling capabilities of CFD, sub channel and system codes by means of post-test calculations of BFBT transient tests", *Nuclear Engineering and Design*, Vol. 263, October, pp. 313-326.

32. Zhang H P, Fu X, Jiang S C and Jia W (2014), "A numerical investigation of nucleate boiling at a constant surface temperature", *Applied Thermal Engineering*, In Press, Corrected Proof, Available online, 18 September 2014.

33. Zhihao Chen and Yoshio Utaka (2015), "On heat transfer and evaporation characteristics in the growth process of a bubble with microlayer structure during nucleate boiling", *International Journal of Heat and Mass Transfer*, Vol. 81, February 2015, pp. 750-759,

International Journal of Engineering Research and Science & Technology
Hyderabad, INDIA. Ph: +91-09441351700, 09059645577
E-mail: editorijlerst@gmail.com or editor@ijerst.com
Website: www.ijerst.com

ISSN 2319-5991