Crossref Similarity Check Powered by iThenticate

Reason- A Technical Journal
ISSN 2277-1654
Volume XVIII 2019

NUMERICAL VALIDATION OF FLOW BOILING ANALYSIS IN ANSYS FLUENT WITH VOF APPROACH

MD Shahbaz, Md Naim Hossain* and Nripen Mondal

Department of Mechanical Engineering, Jalpaiguri Govt. Engineering College, Jalpaiguri- 735102, India.
*Corresponding author: naimhossain6@gmail.com

Paper received on: April 04, 2020, accepted on: April 11, 2020
10.21843/reas/2019/8-16/196162

Abstract: The present paper represents a numerical analysis of flow boiling with volume of fluid (VOF) approach. The analysis has been carried out with the commercial CFD solver ANSYS Fluent 14.5. The said work is focused to demonstrate the suitability of the existing models in ANSYS Fluent to simulate precisely the boiling phenomenon when the fluid is in motion inside heated pipe. The proposed boiling dynamics is carried out with the available VOF approach in ANSYS Fluent. The boiling process is handled with the evaporation and condensation model. The said work is validated with an existing experimental work and the validation shows good agreement. There is some error in this validation. That is because boiling phenomenon is highly turbulent and transient phenomenon. Boiling process deviates from the expected behaviour with little perturbation of heat transfer, surface temperature and flow field etc. Another reason of error is that the boiling phenomenon is a highly localized process. The same type of flow field may not present if authors replicate the same flow boiling process. Overall, the proposed work shows satisfactory match with the said experimental work. This points out that one can use VOF model with evaporation-condensation boiling phenomenon for further hydro-geometric analyses of flow boiling.

Keywords: Flow boiling, VOF approach, evaporation-condensation, pipe flow, boiling.

NOMENCLATURE

$\bar{v}$	Velocity	σ_ε	Turbulent prandtl for ε
t	Time	S_k	Source term of k
ρ	Density	S_ε	Source term of ε
μ	Viscosity	μ_t	Turbulent (eddy) viscosity
T	Temperature	A_i	Interfacial area density
E	Total Energy	h	enthalpy
k_{eff}	Thermal conductivity	g	Gravitational acceleration
$Coeff$	Coefficient of relaxation time	P	Pressure
d_b	Bubble diameter	P^*	Vapor partial pressure at interface of gas side
α	Volume fraction	σ	Surface tension
$\dot{m}_{lv}$	Rate of mass transfer due to evaporation	D	Tube diameter
$\dot{m}_{vl}$	Rate of mass transfer due to condensation	L_{tube}	Tube length
L	Latent heat	h_{local}	Local heat transfer coefficient
k	Turbulence kinetic energy		**Subscript**
ε	Rate of dissipation of k	v	Vapor
G_k	Generation of turbulent kinetic energy due to the mean velocity gradients	l	Liquid
G_b	Generation of turbulent kinetic energy due to buoyancy	sat	Saturated
Y_M	Fluctuating dilatation in compressible turbulence	$wall$	Tube wall
σ_k	Turbulent prandtl for k		

1. INTRODUCTION

Flow boiling is associated with enormous field of practical engineering applications. These are boiler water wall tube, evaporator tubes of a refrigeration plant, riser tubes of solar water heater etc. Analysis of flow boiling is no doubt a complex process due to presence of change of phase and fluid flow inside the tube. Flow boiling is nothing but a two-phase phenomenon. During the flow of subcooled liquid inside the heated tube, the liquid initially absorbs the heat from the heated wall to become saturated. Then the boiling occurs. Boiling in stationary field or pool boiling is only associated with the vapour generation and departure of the generated bubbles from the heated surface. These bubbles may arise to the surface of the fluid or may be collapse inside the fluid itself depending upon the temperature distribution in the heated fluid. If the bubbles collapse then this is called the subcooled boiling. Flow boiling adds some velocity within the domain. This creates rather complex phenomenon. So, flow boiling is not only just bubble generation and departure method, these are also governed by the velocity of the fluid. Again the orientation of the pipe influences the flow field. This flow filed determination is an established entrusted physics. The upward vertical pipe flow boiling is initially subcooled boiling flow, followed by bubbly, slug, churn, annular, and finally mist flow. For horizontal case there is little bit different phenomenon i.e. flow stratified physics, flow waviness, vapour top incident etc. For all the flow boiling analysis there is certain incident that is called critical heat flux or sometimes departure from nucleate boiling which is nothing but continuous vapour film on the heated wall or too thin fluid layer on the wall which could incipience the dry wall situation. That is always avoided in the flow boiling operating cases to save the boiling tubes from thermal rupture.

There are some experimental analyses available in literature regarding flow boiling. Madhavi and Vivek [1] conducted an experiment to study the flow regimes variation, inception of the dry out situation, heat transfer coefficient determination, flow-instabilities and vapor quality. Remi and Thome [2] have done an experimental investigation of flow boiling of two refrigerants, R-134a and R-245fa inside microchannels. For different flow conditions optically they studied two-phase flow characterization in micro tubes. They also evaluated the coalescence rates of the bubbles leaving a micro-evaporator, and their height as well as their mean multi-phase vapor velocity. Shiferaw et al. had exited the heat transfer correlations and the model for flow boiling of R134a inside a small diameter tubes [3]. This study showed the analyses of a variety of presented correlations together with a three-zone evaporation model of heat transfer coefficient linked with flow-boiling. Syed and Ali in their paper reported the experimental appraisal of the heat transfer coefficient in a Vertical Tube which is used in rising film evaporator [4]. Their motto was to explain the dissimilarity of heat transfer coefficient against different course of action used parameters. Cristiano and Gherhardt in their paper pointed out the new experimental flow-boiling heat transfer outcome in microscale tubes [5]. Besides these experimental works, there were some numerical works also explaining the flow boiling. Das and Punekar [6] developed a novel semi mechanistic wall boiling model with the help of a mixture-multiphase flow model existing in ANSYS Fluent. The wall to fluid heat transfers due to nucleate boiling was demonstrated with the help of onedimensional model correlation which was actually modified for three-dimensional numerical era. Sweta et al. [7] proposed a work to demonstrate the modeling of forced convection sub-cooled nucleate boiling using the in-built boiling model available under Eulerian multiphase model. Both experimental and numerical works on flow boiling was conducted by Madhavi et al. on a vertical tube of 19 mm diameter with constant heat flux application. The numerical work was conducted on Eulerian multiphase concept [8]. Garma et al. had investigated the nucleate-boiling flow and bubble pump [9] computationally Computational Fluid Dynamics software ANSYS Fluent with suitable User defined functioned. They used the Eulerian-multiphase flow model for boiling phase-interaction.

In the existing literature, there are some experimental works on flow boiling but these are quite costly. Because any change of geometrical parameter forces to change the whole setup of boiling. On the other hand plenty of numerical works are there to demonstrate on flow boiling of horizontal and vertical tubes. And flow boiling analysis with VOF model is still not fully explored. VOF mechanism can track the flow regime of the mixture of vapour and fluid very accurately and in distinguished way. That is why in the present work a sole attempt has been done to investigate the suitability of the VOF model along with evaporation-condensation mass transfer model in between phases present in ANSYS Fluent 14.5.

2. Mathematical modelling

For the present numerical analysis, a pipe of 1 metre length, 18.8 mm diameter and 3.5 mm thickness are considered as shown in Fig.1 as per the work of Madhavi et al. [8] experimental analysis. The numerical work is carried out in two-dimensional domain. As per experimental set up, authors have considered constant surface heat flux of 28.2 kW/m2 and water inlet velocity inside pipe as 0.003898 m/s.

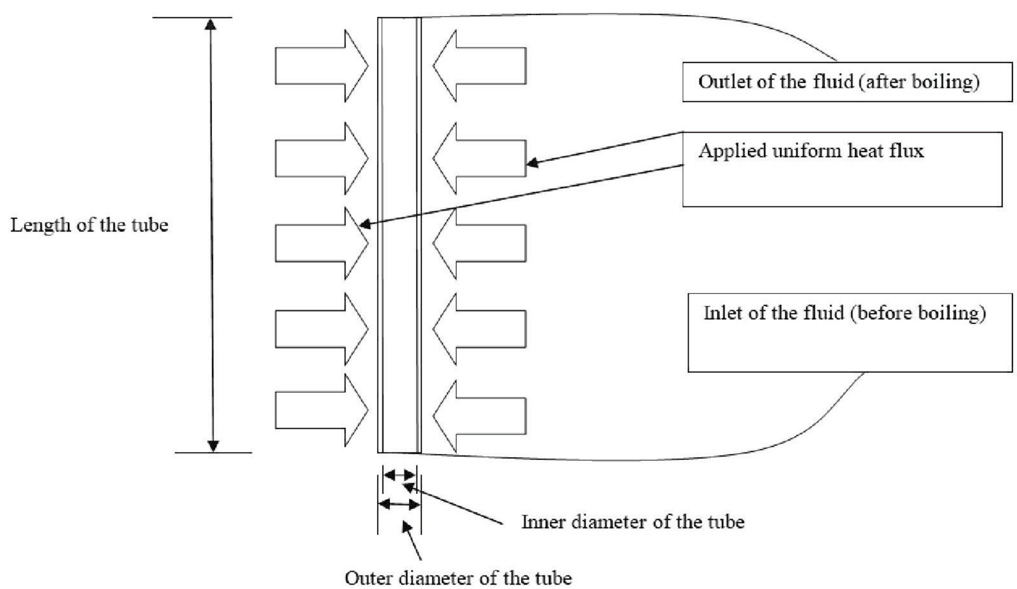

Fig.1. Schematic diagram of the computational domain

In the present analysis Volume of Fluid (VOF) model is considered for two-phase simulation. In VOF, one momentum and single energy equations are solved and volume fraction for all phase are handled by the model throughout the domain. The momentum and energy equations are shown below [10],

$$\frac{d}{dt}\left(\rho\bar{v}\right)+\nabla\left(\rho\bar{v}\bar{v}\right)=-\nabla P+\nabla\left[\mu\left(\nabla\bar{v}+\nabla\bar{v}^{Tranpose}\right)\right]+\rho\bar{q}+\bar{F} \quad (1)$$

$$\frac{d}{dt}\left(\rho E\right)+\nabla\left(\bar{v}\left(\rho E+P\right)\right)=\nabla\left(k_{eff}\nabla T\right)+S_h \quad (2)$$

Here, the energy term E and temperature T are mass-average for VOF model.

For inter-phase mass transfer, Lee model is used. This mode is based on evaporation-condensation phenomenon. The water to vapour mass interchange is described by the vapour-transport equation as shown below

$$\frac{d}{dt}\left(\alpha\rho_v\right)+\nabla\left(\alpha\rho_v\overline{v_v}\right)=\dot{m}_{lv}-\dot{m}_{vl} \quad (3)$$

For evaporation,

$$\dot{m}_{lv}=coeff\times\alpha_l\rho_l\frac{\left(T_l-T_{sat}\right)}{T_{sat}} \quad (4)$$

For condensation,

$$\dot{m}_{vl}=coeff\times\alpha_v\rho_v\frac{\left(T_v-T_{sat\,l}\right)}{T_{sat}} \quad (5)$$

For a flat interface, evaporation-condensation mass flux is based on the kinetic theory. And this is derived from the Hertz Knudsen formula as shown below.

$$F=\beta\sqrt{\frac{M}{2\pi RT_{sat}}}\left(P^*-P_{sat}\right) \quad (6)$$

Replacing the value of $\left(P^*-P_{sat}\right)$ from Clausius-Clapeyron equation as

$$\left(P^*-P_{sat}\right)=-\frac{L}{T_{sat}\left(v_g-v_l\right)}\left(T^*-T_{sat}\right) \text{ to}$$

to the Eq. (5), one gets the modified Hurtz-knudsen [10]

formula as,

$$F=\beta\sqrt{\frac{M}{2\pi RT_{sat}}}L\frac{\rho_v\rho_l}{\left(\rho_l-\rho_v\right)}\frac{\left(T^*-T_{sat}\right)}{T_{sat}} \quad (7)$$

Now assuming that all the vapour bubbles have the same dimension, the interfacial-density is presented as

$$\frac{A_i}{V_{cell}}=\frac{6\alpha_v}{d} \quad (8)$$

The phase source-term is presented as

$$F\frac{A_i}{V_{cell}}=\frac{6}{d}\beta\sqrt{\frac{M}{2\pi RT_{sat}}}L\frac{\rho_l}{\left(\rho_l-\rho_v\right)}\alpha_v\rho_v\frac{\left(T^*-T_{sat}\right)}{T_{sat}} \quad (9)$$

Here from the above equation, the coefficient (inverse of the relaxation time) is defined below,

$$coeff=\frac{6}{d}\beta\sqrt{\frac{M}{2\pi RT_{sat}}}L\frac{\rho_l}{\left(\rho_l-\rho_v\right)} \quad (10)$$

Below k–ε model equations are used to handle the turbulence situation inside the flow.

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k \tag{11}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_i}(\rho \varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon \tag{12}$$

The turbulent (eddy) viscosity, μ_t is expressed as below

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \tag{13}$$

Where, C_μ is constant.

The values of model constant, i.e., $C_{1\varepsilon}, C_{2\varepsilon}$ and C_μ are used as expressed in Table 1. These default values have been determined from analyses of experiments and used as universal standard ones and widely accepted values in turbulent study [10]. The *coeff* value is used here after trial and error value estimation for flow boiling analysis.

Table 1 Values of model constant

$C_{1\varepsilon}$	$C_{2\varepsilon}$	C_μ
1.44	1.92	0.99

σ_k	σ_ε	$coeff$
1.0	1.3	0.1

A grid independent study has been carried out with 475000, 4750000, 47500000, and 48000000 numbers of total grids within the domain and it has been found that 47500000 number of grid number case gives acceptable and stable result. The computational domain converged with this case. Also, a time step independent study has been carried out and it has been found that out of 0.01, 0.001- and 0.0001-time steps, 0.0001-time steps gives stable converged solution. So, 47500000 numbers of grid and 0.0001 time steps size are used here to simulate the problem. The numerical simulation becomes steady state after 50s time.

3. Results and Discussion

Local heat transfer coefficient in the tube wall is calculated and compared with the experimental work of Madhavi et al. [8] and the comparison is presented in the Fig.2. To calculate the local heat transfer coefficient the used expression is shown below

$$h_{local}=\frac{q}{Twall-Tsat} \tag{14}$$

In the figure, authors can see that the calculated local heat transfer coefficient from the numerical analysis is in good agreement with the experimental result through-out the tube length. The experimental values shown in work of Madhavi et al. [8] are for three particular points. The numerical results shown in the figure is estimated by section averaging method as there is a vast fluctuation in wall temperature zonal. So, the numerical results are presented here as scattered points. At length 0.43 metre and 0.7 metre the heat transfer coefficient exactly in match with the experimental results. At higher length say 1 metre there is 10% deviation of numerical result with the experimental one, which is also good agreement with the experimental result. The reason behind this deviation is because the local heat transfer coefficient is highly instantaneous phenomenon. As this is a flow boiling phenomenon, the presence of vapor bubble nearer to wall can make huge deviation in heat transfer value in between wall and flowing fluid. Overall the present numerical simulation shows good agreement with the experimental work.

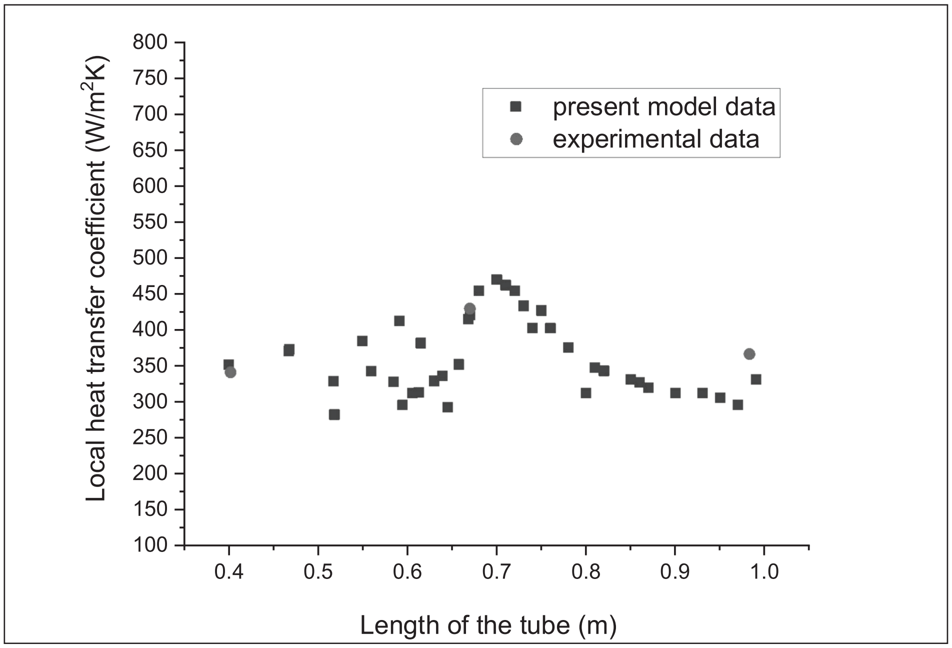

Fig.2. Comparison of local heat transfer coefficient

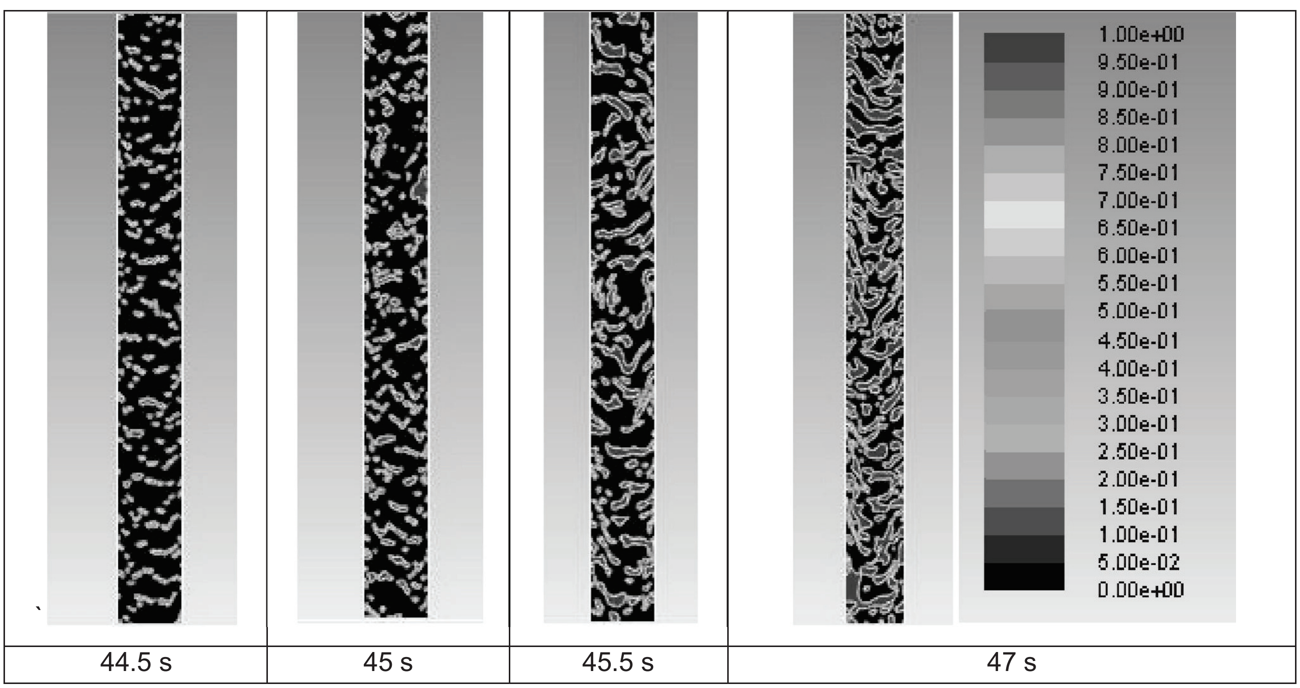

Fig.3. Volume fraction variation along tube length for various time steps

Fig.3 represents the volume fraction distribution of vapour phase along the tube for various time steps. It is observed bubbles generate and depart from the wall of the tube to the flow steam at 44.5 s time. With increasing time, the bubble generation increases. At 45 s time steps, one can see that the generated bubbles attach together at the flow steam and generate bubble slug. This trend increases up to 45.5 s time step. Full length vapour slug generates at time step 47s.

Fig.4 represents the full length flow dynamics of vapour phase along the tube length from 0.4 metre to 1 metre. Here one can see that at the beginning the subcooled liquid is entering the tube at bottom, so initially the water gets heated upto saturation point at the wall. Till then there is no bubble generation. But after that subcooled boiling happens. After sometimes full bubbly flow is clearly visible along the tube length. The flow then converts to slug flow as the generated bubbles attaches among them. The slug flow then converts to churn flow. After that as the generated slugs convert to big slugs and finally they generate annular flow, where there is vapour core along the tube length with very thin wall wetting. Finally at near the tube exit the flow is almost converted to fully vapour. That zone of flow is called mist flow zone.

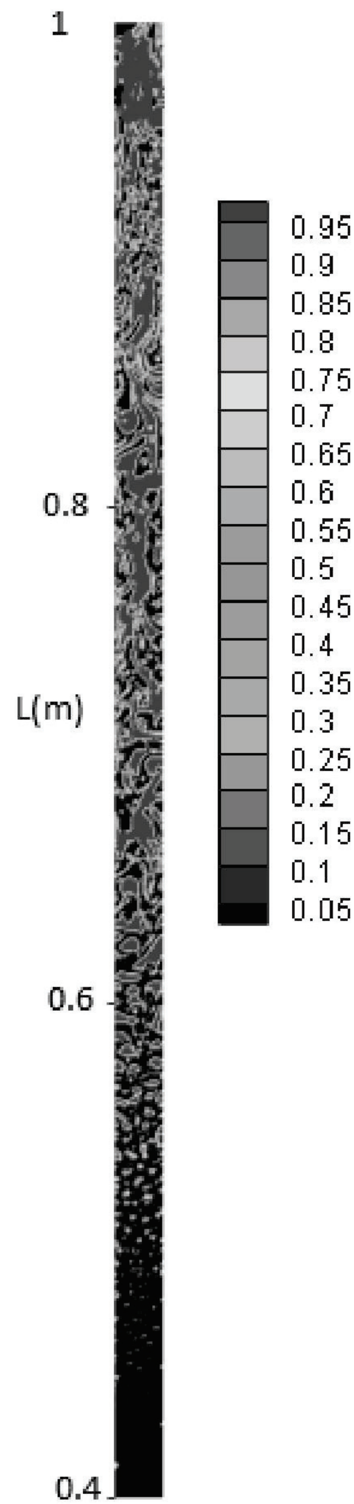

Fig.4. Volume fraction variation along tube length for steady state situation

4. CONCLUSION

The flow boiling is investigated numerically in this present work with commercial CFD solver ANSYS Fluent 14.5 with available VOF model. The boiling is handled by evaporation-condensation model. The sole attempt of the authors is to investigate the suitability of considered models to simulate the flow boiling. That is why the considered models are tested over an experimental work of flow boiling of tube 19 mm and length 1 metre with uniform heating along the tube wall. The considered model is then used to calculate the wall local heat transfer coefficient and it has been compared with the experimental results. The comparison shows good agreement. Then the model is used to analyze the vapor phase distribution prediction analysis. The calculated vapor phase along the tube length at steady state shows a very good result and clearly points out the all possible flow regimes throughout the tube length. So, authors can say after this analysis that the considered VOF model with evaporation-condensation mass transfer mechanism is suitable to analyze the flow boiling accurately.

Acknowledgement: The present paper is based on the article presented in the National Conference on Trends and Advances in Mechanical Engineering (TAME-2019) held at Kalyani Govt. Engineering College, Kalyani on February 15-16 2019 and jointly organized by The Association of Engineers, India, Kalyani Govt. Engineering College, Kalyani and Global Institute of Management and Technology, Krishnagar as a part of the Centenary Celebration of the Association of Engineers, India.

REFERENCES

[1] Madhavi, V.S. and Vivek, V.R., Two-Phase Flow Boiling in Small Channel: A Brief Review, Sadhana, Vol. 38, Part 6, pp.1083–1126, 2013.

[2] Remi, R. and John, R.T., Experimental Investigation of R-134A And R 245FA Two-Phase Flow in Microchannels for Different Flow Conditions. International Journal of Heat and Fluid Flow, Vol. 28, pp.63–71, 2007

[3] Shiferaw, D., Huo, X., Karayiannis, T.G. and Kenning, D.B.R., Examination of Heat Transfer Correlations and a Model for Flow Boiling of R134a in Small Diameter Tubes, International Journal of Heat and Mass Transfer, Vol.50, pp.5177–5193, 2007.

[4] Syed, N.H. and Ali, S., Experimental Investigation of Heat Transfer Coefficient in Vertical Tube Rising Film Evaporator, Mehran University Research Journal of Engineering & Technology, Vol. 30, 2011.

[5] Cristiano, B.T. and Gherhard, R., Flow Boiling Heat Transfer of R134a And R245fa in a 2.3 mm Tube, International Journal of Heat and Mass Transfer, Vol. 53, pp.2459–2468, 2010.

[6] Das, S. and Punekar, H., On Development of a Semimechanistic Wall Boiling

Model, Journal of Heat Transfer, Vol. 138, pp.1-10., 2016.

[7] Sweta, S., Suresh, K.B. and Santosh, D., Numerical Simulation of Nucleate Boiling Case Using Fluent, International Journal of Engineering Research and Science & Technology, Vol. 4, 2015.

[8] Madhavi, V.S., Barley, R. and Vivek, V.R., Two-Phase Flow Boiling in 19mm Tube: Experiments and CFD modelling, The Canadian Journal of Chemical Engineering, Vol. 94, 2016.

[9] Garma, R., Bourouis, M. and Bellagi, A., Numerical Investigation of Nucleate Boiling Flow in Water Based Bubble Bumps, International Journal of Fluid Mechanics & Thermal Sciences, Vol. 1, No.2, pp.36-41, 2015. DOI: 10.11648/j.ijfmts.20150102.14

[10] ANSYS Inc., ANSYS Fluent Manual Release 14.5, 2012.