International Journal of Advanced Technology in Engineering and Science www.ijates.com
Volume No.02, Issue No. 12, December 2014 ISSN (online): 2348 – 7550

NUMERICAL INVESTIGATION OF ENHANCEMENT OF HEAT TRANSFER BY TWO PHASE FLOW IN MINI CHANNELS

Srusti T[1], Anand S. N[2], Dr. G. S. V. L. Narasimham[3], P. Srikrishna[4]

[1]*M.Tech Student, EPCET Bangalore, (India)*

[2]*Associate Professor, Department of Mechanical Engineering, EPCET Bangalore, (India)*

[3]*CRS, Indian Institute of Science, Bangalore, (India)*

[4]*Scientist 'D', MTRDC, DRDO, Bangalore, (India)*

ABSTRACT

Detailed computational multi-fluid dynamics simulations have been performed to study the effect of two phase flow regime on heat transfer in mini channels. The axisymmetric model of circular pipe of 1mm diameter and 20mm length is studied at different gas volume flow ratios and constant heat flux. The Nusselt number distribution shows the taylor bubble train regime increases the transport of heat up to 0.65 to 0.71 times more as compared with thermally developing single phase flows.

Keywords: Mini Channel, Taylor Flow, Nusselt Number, Heat Transfer

I. INTRODUCTION

Multiphase flows have been part of our everyday life since a very long time. But the focus on the study of these two phase flows has attained greater importance of late due to their potential applications. High surface-to-volume rtio and the small diffusion paths in micro-structured devices make them ideal candidates for heat exchangers and microreactors. The laminar nature of the flow in microchannel helps in efficient design of the micro-structured devices as no heuristic model is required to model laminar flow. Multiphase flow can further enhance the performance of these microfluidic devices by providing large interfacial area. Transport mechanisms of heat, momentum and mass under two phase flow conditions in mini-/micro systems are greatly affected by the local distribution of phases, or flow patterns in the channel. Depending upon the properties and flow rates of the two fluids, various flow patterns such as bubbly, slug or Taylor, churn, rivulet, wavy annular, annular flow etc. occur in gas–liquid flow in microchannels. **Taylor flow** is characterized by gas bubbles that almost fill the channel, separated by liquid slugs. A thin liquid film separates these bubbles from the wall and also connects the two successive liquid slugs separated by the gas bubble. In this the inertial forces negligible compared to the surface tension forces. Flow rates are generally moderate and the flow is mostly laminar. Due to the re-circulating flow in the liquid slugs, Taylor flow improves heat and mass transfer from the liquid to the wall. This flow pattern also provides a large interfacial area and thus enhances gas–liquid mass transfer. The separation of liquid slugs by the bubble also reduces axial liquid mixing. Thus, enhanced radial mass transfer and reduced axial mass transfer make Taylor flow suitable for applications which suffer from large back-mixing. Due to its interesting flow characteristics, the Taylor flow regime has received enormous attention over the years and has been studied experimentally, analytically and using computational fluid dynamics.

II. LITERATURE SURVEY

Carlson, P. Kudinov, C. Narayanan [1] carried out numerical simulations to investigate multiphase dynamics and its characteristics for two-phase gas-liquid flow and also compare the two commonly used methods for two phase flow simulations namely VOF implemented in fluent and Levelset implemented in TransAT. They observed significant differences between the flow topologies predicted by the two codes. Both codes solve the multi-fluid Navier Stokes equations based on a finite volume method and a pressure–based solver. The codes use different discretization schemes and time integration methods, the reconstruction of the interface in VOF is performed with a CISAM scheme 2^{nd} order, and the LS function (φ) is advected using the QUICK linear upwind scheme.

Abhik Majumder, Balkrishna Mehta, Sameer Khandekar [2] experimentally investigated the enhancement of heat transfer due to taylor bubble train flow in a horizontal square channel of size 3.3 mm 3.3 mm 350 mm, heated from the bottom (heated length=175 mm), with the other three sides kept insulated, in comparison with thermally developing single-phase flows. They observed that taylor bubble train regime increases the transport of heat up to 1.2-1.6 times more as compared with laminar single-phase liquid flow.

Raghvendra Gupta, David F. Fletcher, Brian S. Haynes [3] carried out Two-dimensional, axisymmetric CFD simulations in a small diameter circular channel using two different interface capturing techniques, namely the VOF and level-set methods. They studied fully-developed flow and heat transfer for a Reynolds number of 280, Capillary number of 0.006 and homogeneous void fraction of 0.51 was studied for the constant wall heat flux and constant wall temperature boundary conditions. The average Nusselt number obtained in both cases was approximately 2.5 times higher than that for fully-developed laminar liquid-only flow.

M. K. Akbar and S. M. Ghiaasiaan [4] studied the feasibility of CFD modeling of the Taylor flow regime by using the volume-of-fluid (VOF) technique for the motion of the gas-liquid interphase was demonstrated. Steady-state Taylor bubble flow is considered. The simulation results were validated using the experimental data. The predicted bubble absolute velocity, slug length, and frictional pressure drop were compared with available relevant experimental and computational results. In addition, some improved correlations are proposed based on available experimental data.

T. Taha, Z.F. Cui [5] Understanding the motion of long gas bubbles inside capillaries is a challenging problem that is relevant in many processes of chemical and biological interests. It has been proved by many workers that such long gas bubbles can be used successfully in enhancing mass and heat transfer in many chemical and biological processes. In order to quantify and understand this enhancement a light was shed on the hydrodynamics of such a flow pattern. The volume of fluid method implemented in the commercial CFD package, Fluent, is used for this numerical study. Computed values of the bubble velocity and diameter were in excellent agreement with published experimental measurements

Taha Taha, Z.F. Cui [6] revealed the characteristics of slug flow inside circular capillaries and square micro-channels using the VOF method, showing the fundamentally different hydrodynamics of such a flow from that in larger tubes and the effect of corner flow in square capillaries. The velocity field around the bubble were shown in details for a wide range of Ca showing different flow patterns inside liquid plugs in agreement with experimental findings. The shape and terminal velocity of the Taylor bubble were predicted by the VOF method and agreed favourably with the published experimental findings. Based on hydrodynamics of slug flow, a light was shed to explain and understand the augmentation of mass and heat transfer in capillaries.

IV. PHYSICAL MODEL AND MATHEMATICAL FORMULATION

A two-phase flow consisting of air and water inside a circular pipe of Φ1mm was analysed. The 2D axis–symmetric simulations have been performed in a commercial CFD code Fluent to study the fluid flow and heat transfer. Here the conservation equations of mass and momentum in radial and axial directions were solved. The effect of surface tension is dominant in these flows through narrow channel. This is clearly shown by the fact that the Bond Number = $g\rho_L D^2/\sigma$ = 0.136 (Inertia/Surface tension) and Capillary Number = $\mu V_B/\sigma$ = 0.015 (Viscous/Surface Tension) are very small for the velocities chosen. Hence the effect of gravity in the analysis has been neglected (Bo<0.842 Ref [3]). In this approach the air is injected at the core of the pipe and water is fed at the annular space.

4.1Boundary and Initial Conditions

Boundary Condition: an axisymmetric domain with height equal to the radius of the channel (0.5mm) and length equal to 20D was modeled (Fig1). The inlet was split into two parts – one for the air inlet at the core and the other for the water inlet at the annular area. In this co-flow with different cases for different velocities of air by keeping constant void fraction as shown in Table 2 was modeled.

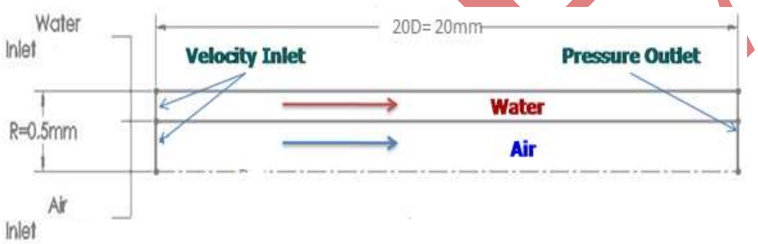

Fig1: Boundary Condition

Table 1: Details of Various Boundary Conditions

Case	R_{air}	$\alpha_G = A_G/A_T$	V_G	V_L
	mm		m/s	m/s
1	0.25	0.25	0.55	1
2	0.25	0.25	1	1
3	0.25	0.25	1.45	1
4	0.25	0.25	2	1
5	0.25	0.25	2.25	1

Table 2: Fluid Properties Used

Fluid	Density (kg/m^3)	Viscosity (kg/m s)	surface tension (N/m)
Water	997	$8.899\ 10^{-4}$	0.072
Air	1.185		

V. RESULTS AND DISCUSSION

Two dimensional, axi-symmetric simulations for flow and heat transfer in a channel of diameter 1mm and length 20d are carried out using constant property of air and water as working fluids. The water velocity at the inlet is 1m/s and air velocity is varied. The inlet temperature of both the fluids is 300K and heat flux of 159.15KW/m2 is applied on the channel wall. As the air velocity varies from 0.55 to 2.25m/s the bubble tends to change with the change in the velocity. Since the velocity of the air is varying the heat transfer coefficient is also varying. In the bubbly and slug flows, the thermal wall layer is affected by the presence of the cells, preventing it from growing naturally with the dynamic boundary layer along the pipe. Flow recirculation is visible in the wake and ahead of the cells. But the additional shear created by the defect flow field between the wall and the cell surfaces is precisely what promotes heat transfer in the two phase flow in general. In fig2 shows the stream lines and recirculation of a multiphase flow of air and water. The bulk temperature is high in the gas bubble region and low in the liquid slug region with sharp jumps in the bulk temperature observed at the nose and tail of the bubbles.

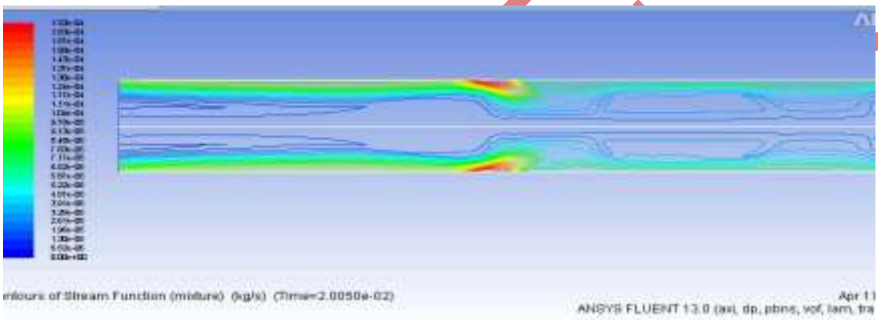

Fig2: Contours of Stream Function Showing Stream Lines and Recirculation

The volume fraction plot, stream function, velocity magnitude, pressure and temperature plot has been analysed and compared with the plots of single phase flow.

5.1 Analysis of Multiphase Flow

For, V_{water}= 1m/s, V_{air}= 1.45m/s

The twophase taylor flow topology have been investigated as shown in the figure above. The water velocity of 1m/s and air velocity of 1.45m/s are supplied at the inlet. From the contour below it can be observe that the bubble formation, detachment of the bubble due the shear force between air and water particles and the bullet shape of bubble. Due to the pressure difference between the inlet and outlet the bubbles tends to move from upstream to downstream.

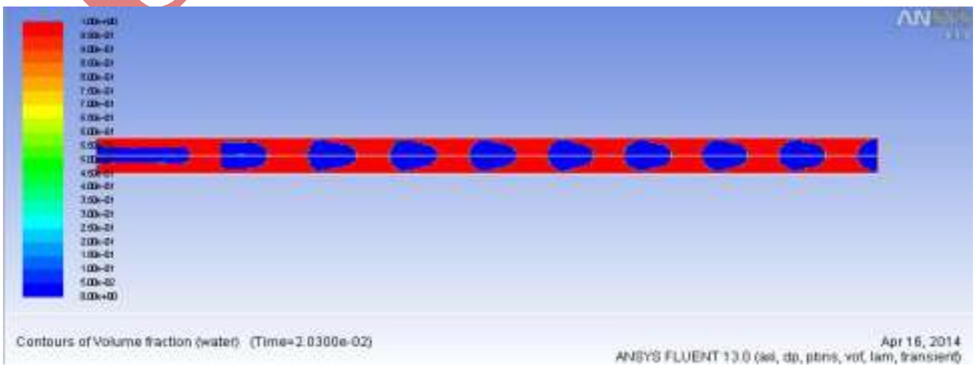

Fig3: Plot of Liquid Volume Fraction Showing Water Slug and Air Bubble. The Air Is Coloured Blue and the Water Is Coloured Red

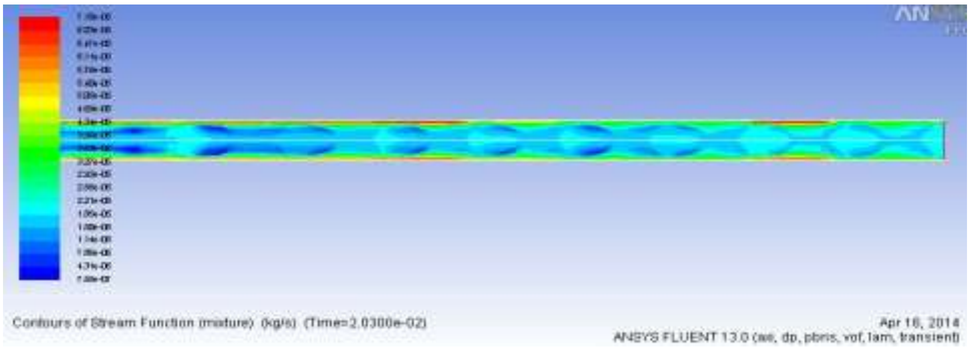

Fig4: Stream Function Plot

Fig4 shows the total flow field to be affected by the presence of the cells, the flow recirculation in particular. Flow recirculation is visible in the wake and ahead of the cells. But the additional shear created by the defect flow field between the wall and the cell surfaces is precisely what promotes heat transfer in the two-phase flow in general. The defect-flow induced shear is the main feature promoting heat transfer and preventing the wall thermal layer to naturally develop with the dynamic boundary layer.

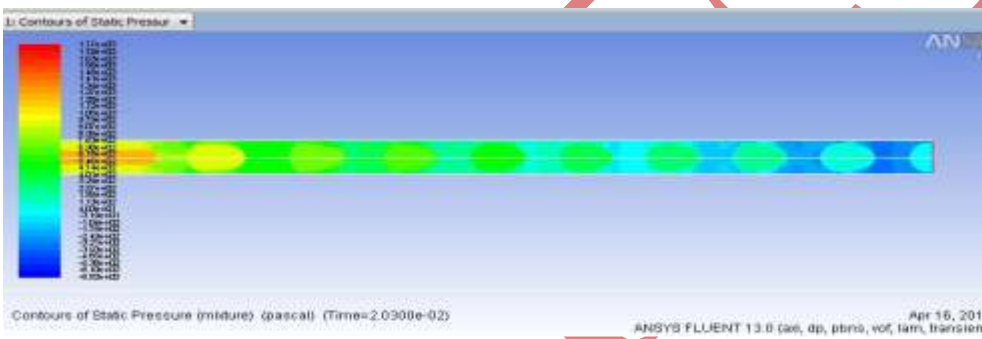

Fig5: Static Pressure Plot

The above figure depicts the pressure variation along the axis and wall. The pressure in the each gas bubble is constant and the pressure in liquid region ahead of the bubble decreases with distance. The pressure profile in the liquid slug region is complex with different pressures at the wall and the axis close to the bubble and it shows the strong pressure gradient building up at the breakup location. This is similar to pressure field observed by Raghvendra Gupta, BrianS.Haynes.

5.2 Analysis of Singlephase Flow

V_{water} = 1m/s for entire cross sectional area

Fig6: Plot of Liquid Volume Fraction Showing Only Water Slug

The above figure shows the contour of singlephase flow. Here only water is flowing at 1m/s inside the channel.

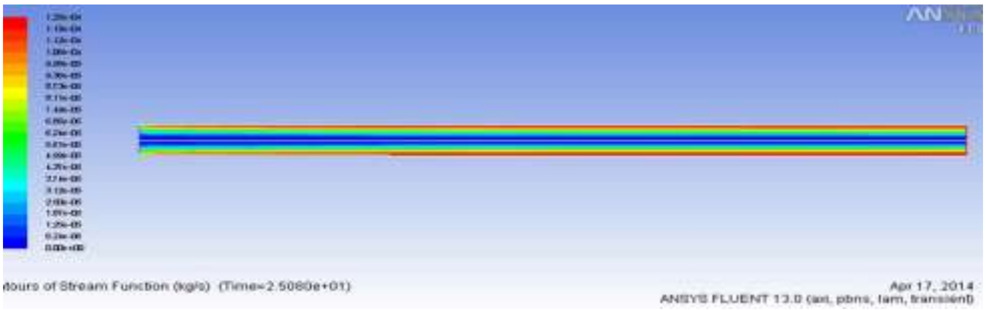

Fig7: Stream Function Plot

The fig8 shows the contours of stream function which depicting the flow of fluids in parallel layers with no disruption between the layers. At low velocities the flow will be laminar where the fluids flow without lateral mixing and adjacent layers slide over one another. There are no cross currents perpendicular to the direction of flow, nor eddies.

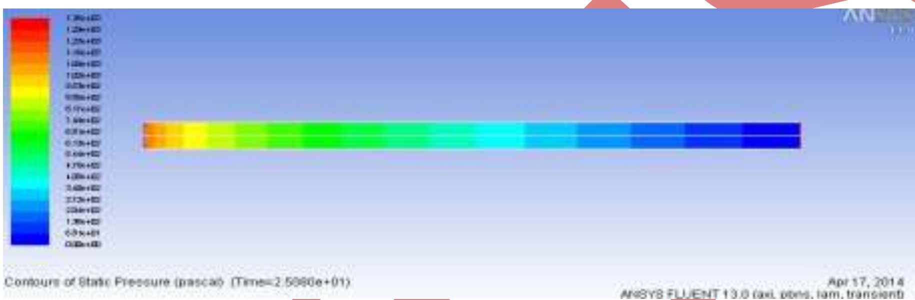

Fig8: Static Pressure Plot

The above figure shows the variation of pressure across the channel. It is seen at the inlet fluid having maximum pressure and gradually reducing when moving towards outlet and minimum at outlet. Fig9 and 10 shows the variation of the wall temperature and bulk fluid temperature across the non-dimensional axial distance. From the graph it can be observed that singlephase wall temperature gradually increases along the length of the channel, whereas the multiphase wall temperature follows a sinusoidal pattern at different velocities because of intermittent flow. The bulk fluid temperature in multiphase is increasing and higher compared to single phase bulk fluid temperature. Thus, it is evident that the multiphase wall temperature which is well below the singlephase wall temperature indicating the enhanced heat transfer rate.

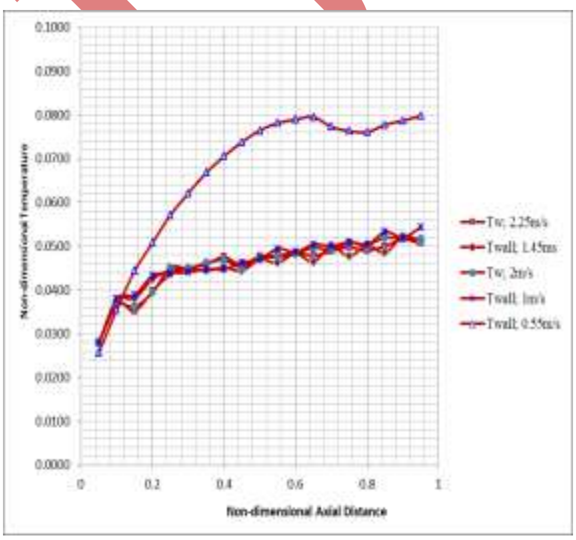

Fig9: Wall Temperature Distribution along the Axis

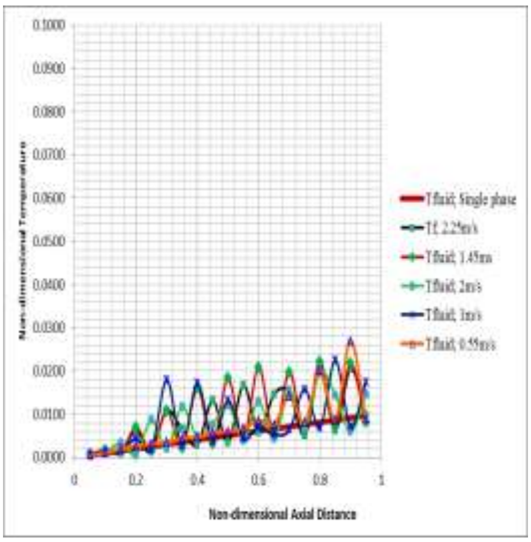

Fig10: Fluid temperature distribution along the axis

Fig11 shows the variation of Nusselt Number along the Non-Dimensional Axial distance. It can be observed that the Nusselt number is for single phase is gradually decreasing and has an average Nusselt number 14.86. Since the length of the channel is 20mm which suits the 1"x1" micro-channel, further by increasing the length of the channel the flow will be completely developed i.e Nu=4.36. The multi-phase flow has the sinusoidal pattern and the average Nusselt number is around 25.38 for the air velocity of 1.45 m/s. The percentage increase in the average Nusselt number is around 0.7079 % than single phase flow.

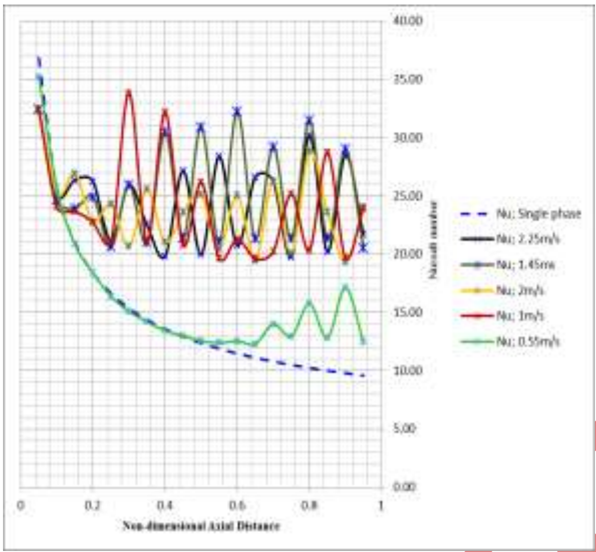

Fig11: Nusselt Number Distribution along the Axis

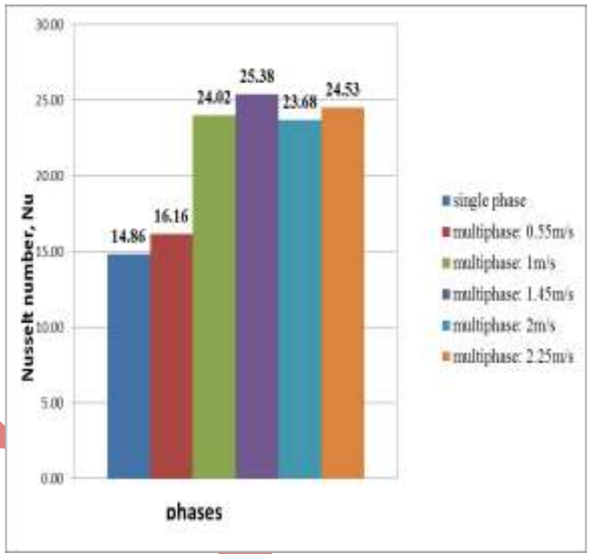

Fig12: Singlephase and multiphase Nusselt number distribution

The above graphical representation gives a clear picture of Nusselt number for single phase and multi-phase flow with various air velocities. It can be observed that the average Nusselt number is higher at the velocity of 1.45m/s which indicates the enhanced heat transfer rate. It can also be noted that after the velocity of 1.45m/s, there is no significant enhancement in the Nusselt number (or heat transfer rate). Thus the velocity of 1.45m/s can be considered as the optimum velocity for the multi-phase flow with water velocity of 1m/s.

VI. CONCLUSION

- Two-dimensional axisymmetric CFD simulations were carried out in a mini channel using VOF interface capturing technique.
- Validation of the FEM analysis model is done with respect to the available standard analytical value for Nusselt number of constant heat flux for flow through circular pipes.
- Flow recirculation is visible in the wake and ahead of the cells and the defect-flow induced by the presence of bubbles and slugs, which has been revealed to substantially increase the wall shear and in turn heat transfer in two phase flow.
- The heat transfer and average Nusselt number was studied for air velocities of 0.55, 1, 1.5, 2, 2.25m/s and water velocity of 1m/s for constant heat flux boundary condition in a thermally developing region.
- The average Nusselt number of two phase flow obtained was approximately 0.708 times higher than that for thermally developing laminar liquid-only flow.
- Also from the analysis it is evident that there is no substantial increase of Nusselt number above the air velocity of 1.45m/s.

REFERENCES

[1] A.Carlson1, P.Kudinov1, C.Narayanan., Prediction of two-phase flow in small tubes: A systematic comparison of state-of-the-art cmfd codes

[2] Abhik Majumder, Balkrishna Mehta, Sameer Khandekar., Local Nusselt number enhancement during gas liquid Taylor bubble flow in a square mini-channel: An experimental study

[3] Raghvendra Gupta, David F Fletcher, Brian S Haynes., CFD modeling of flow and heat transfer in the Taylor flow regime, School of Chemical and Biomolecular Engineering. The University of Sydney, Sydney, NSW, Australia

[4] M. K. Akbar and S. M. Ghiaasiaan., Simulation of Taylor Flow in Capillaries Based on the Volume-of-Fluid Technique,

[5] G. W. Woodruff School of Mechanical Engineering, Georgia Institute of Technology Savannah

[6] T. Taha, Z.F. Cui., Hydrodynamics of slug flow inside capillaries

[7] Department of Engineering Science, Oxford University, Parks Road, Oxford OX1 3PJ, UK

[8] Taha Taha, Z.F. Cui., CFD modelling of slug flowinside square capillaries

[9] Department of Engineering Science, Oxford University, Parks Road, Oxford OX1 3PJ, UK