DC Meeting
Venue: Online
Date: 29/07/2021
Time : 3:00 PM

COMPREHENSIVE VIVA REPORT

Numerical and Experimental Analysis of the Transient Capillary Driven Flows with Phase Change Heat Transfer and Dynamic Contact Angle Model

By
Chitransh Atre

Guided By
Prof. Arul K Prakash
Co-Guided by
Prof. A P Baburaj

Fluid Mechanics Group
Department of Applied Mechanics
Indian Institute of Technology Madras
Chennai – 600036

Data sheet

Name	: Chitransh Atre
Roll No	: AM19D017
Program	: Ph.D.
Specialization	: Fluid Mechanics
Category of Admission	: Project
Guide	: Prof. Arul K Prakash
Co-Guide	: Prof. A P Baburaj
Date of Registration	: 22/08/2019
Area of Research	: Fluid Mechanics, Heat Transfer, CFD
Date of Comprehensive Viva	: 29/07/2021

Detail of Courses

Sr. No	Course No.	Course Title	Sem/Year	Credit	Grades
Core Courses					
1	AM5530	Advanced Fluid Mechanics	July-Nov 2019	9	B
2	ME5101	Advanced Heat &Mass Transfer	July-Nov 2019	10	D
3	AM5630	Foundation of Computational Fluid Mechanics	July-Nov 2019	9	C
4	AM5570	Introduction to Turbulence	Jan-May 2020	9	A
5	ME5115	Surfaces and Interfaces	Jan-May 2020	9	B
Compulsory Courses					
1	ID6020	Introduction to Research (Institute Module)	Jan-May 2019	0	P
2	AM6021	Introduction to Research	Jan-May 2019	0	P

Date : 22/07/2021
IIT Madras, Chennai.

Signature of Guide Signature of Co-guide Signature of Scholar

Numerical and Experimental Analysis of the Transient Capillary Driven Flows with Phase Change Heat Transfer and Dynamic Contact Angle Model

---

1. Introduction

Heat pipes are the devices which is used to spread the heat and they are a popular technology for the effective heat transfer. In our daily life, we use many electronic equipment's in which heat pipe technology is being used e.g., in laptop for cooling the microprocessors, air conditioners etc. The major challenge for any electronic device is to keep it cool and hence increase its performance. Heat pipes are extensively used due to their high thermal conductivity and high heat transfer rates in geothermal energy field, aircrafts and automobile industries, solar technology fields and space applications,

There are many different types of heat pipes for example loop heat pipes, flat heat pipes now developed in the recent years and much development has been done on its structure, working fluid, wick structure etc. In a simple flat heat pipe, there are generally three chambers naming evaporator, condenser and adiabatic sections. The inner region is filled with a wick structure near the wall of heat pipe which is generally made up of sintered copper powder.

The heat pipe working is based on two phase flows. This device takes up the heat from the heat generation source which goes in to the evaporation section. The liquid inside the evaporation section heats up and evaporate. As the vapour pressure increases at one end, the vapour will travel to condenser section through adiabatic section (middle section). In condensation region, the vapour again condenses releasing the latent heat and the liquid is again absorbed by the wick structure. Now, due to the capillary pressure generated across the two ends of heat pipes, the liquid will go to evaporator section again through capillary action. In some heat pipe, fins at the end section of condenser region are used to dissipate the heat. This process continues till the vapour pressure difference at both the ends remains.

2. Literature Review

There have been many studies on the working of heat pipes. Starting with the flow through porous media, Chen et.al.[1] simulated the flow through porous media using Darcy-Brinkman-Forchhiemer model. They modeled a coupled homogeneous porous medium and non-porous region with different types of boundary condition at the interface e.g., navier slip condition and studied the flow behavior. Chen et.al. [2] extended the flow though porous region with implementation of heat transfer equation. They used backward facing step channel geometry and the passive scalar heat transport which gives the temperature. They studied the Nusselt's number variation by changing the porosity and permeability. The continuum surface force model was first introduced by Brackbill et.al.[3]. Using this model for surface tension, various researches has been performed on flow of a drop of liquid over a plate with a static contact angle at the interface of liquid and solid plate. Similar studies have been done by Sikalo et.al.[4] but using dynamic contact angle model which gives more accurate results. They compared and validated the numerical results and experimental results. Siddhartha et.al.[5] used inclined surfaces and also studied different regimes of impact of drop. Different dynamic contact angle models have been developed e.g., Cox model, klister model etc. which calculates the angle using the contact line velocity.

Hardt et.al. [6] introduced a model which incorporates the phase change at the liquid vapour interface. Rattner et.al.[7] used Volume of Fluid (VOF) method to simulate the condensation phenomena. Based on Hardt model of phase change, Kulkelmann et.al.[8] performed the simulations for boiling heat transfer using VOF method. We found that the VOF method is used extensively and easy to implement also.

A literature review for fluid flow through capillary of different geometries is also done as the requirement of this research work. D. Quere et. al. [9] investigated the capillary rise of liquid and showed the dependency of capillary height on different parameters. From the same group, Alexendre et.al.[10] developed the relation between the capillary rise height and time to rise its meniscus.

Our goal is to first simulate the capillary rise with phase change heat transfer. The geometry is having very sharp corners also which accounts for the faster movement of rise of liquid and enhanced evaporation rates. Thus, the cooling rates can be increased. Furthermore, a lot of research shall be done on the contact line evaporation model and its simulations with grid independent studies.

3. Governing Equations and Numerical Approach:

There are many interface capturing methods used in numerical modeling for example level set method, moving mesh methods and volume of fluid method (VOF). In all of these, VOF is the most extensively used method for two phase incompressible flows to detect the liquid-vapour interface. In VOF, a phase fraction value is defined, α whose value is one for liquid phase and zero for vapour phase, i.e.

$$\alpha = 1 \rightarrow liquid \text{ and } \alpha = 0 \rightarrow vapour. \text{ So, } 0 < \alpha < 1$$

Continuity equation: $$\nabla . \vec{U} = 0 \quad \text{---------- (1)}$$

Momentum Equation: $$\rho \left[\frac{\partial \vec{U}}{\partial t} + \vec{U}.\nabla\vec{U}\right] = -\nabla P + \nabla.(\mu\nabla\vec{U}) + \rho\vec{g} + \overrightarrow{F_s} \quad \text{----------(2)}$$

Where *U* and *P* are the velocity and pressure fields, and F_s includes the capillary forces. The viscosity and density is calculated by the expressions given below:

$$\mu = \alpha\mu_l + (1-\alpha)\mu_g \quad and \quad \rho = \alpha\rho_l + (1-\alpha)\rho_g \quad \text{----------(3)}$$

where μ and ρ are viscosity and density and subscripts l and g are for liquid and gas respectively.

Energy equation :

$$\frac{\partial \rho C_p T}{\partial t} + \nabla.\left(\vec{U}.\rho C_p T\right) = \nabla.(K\nabla T) + H \quad \text{----------(4)}$$

Where *K* is the thermal conductivity and *H* is the heat for evaporation.

The phase change model is based on Hardt and Wondra et. al. The evaporation mass flux at the interface of liquid and vapour is given by:

$$m_{int} = \frac{T_{interface} - T_{saturation}}{R_{interface} h_{lv}} \quad \text{----------(5)}$$

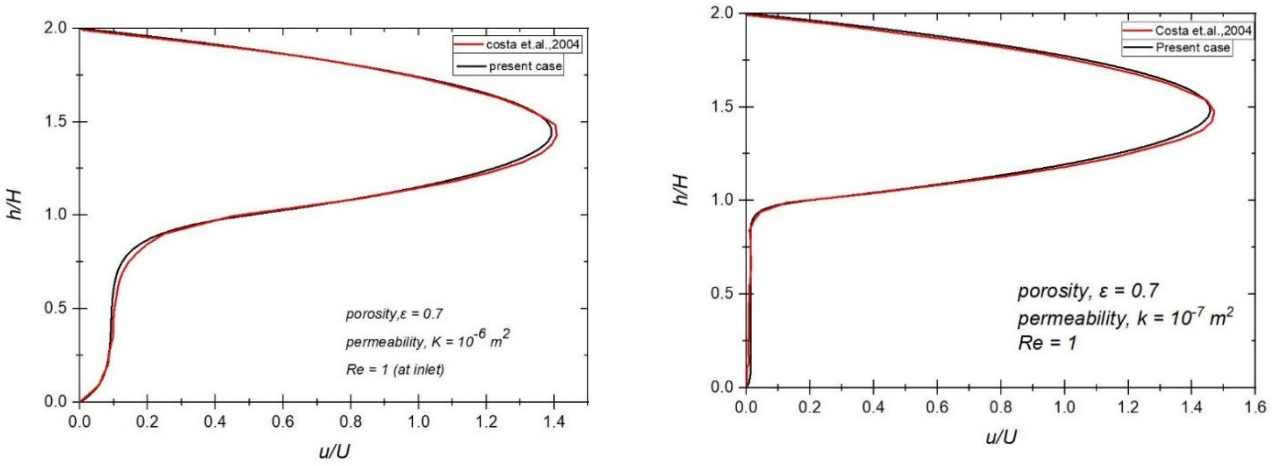
Here, $T_{interface}$= Temperature at the interface, $T_{saturation}$ = Saturation temperature

h_{lv}= latent heat of vaporization and $R_{interface}$= Heat resistance which is calculated based on the expression given by Schrage et.al.

$$R_{interface} = \frac{2-\beta}{\beta}\frac{\sqrt{2\pi R_{gas}}}{h_{lv}^2}\frac{T_{saturation}^{1.5}}{\rho_g} \quad \text{----------(6)}$$

4. Results and Discussions:

1.) For fluid flow and heat transfer inside porous media, a cavity with a homogenized porous media is modelled. Darcy-Forcheimer-Brinkman model is used in CFD simulations inside the porous region and the design parameters as porosity, permeability is set. Some benchmark simulations are also done considering the flow through the free region coupled with a porous region of same height between two parallel plates. The inlet velocity boundary condition is given with no slip conditions at the top and bottom boundaries. All simulations are done using Simple Scheme. Results are validated successfully with results of Costa et. al.[1].



(a) (b)

Fig 1: (a) and (b) showing validations of dimensionless velocity profiles with different permeability values

2.) The energy equation is added in the model for temperature calculations. This equation is responsible for heat flow in the solid (conduction) as well as in fluid region (convection) inside porous region. The validation of the results has been done and matched with the results of Chen et. al.[2]

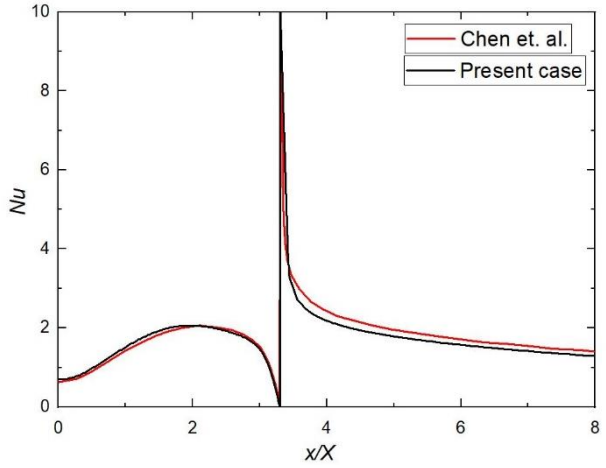


Fig 2: The Nusselt's number variation validation

3.) The geometry of capillary and its interstice is created and mesh is generated ANSYS All the simulation are being carried out in OpenFOAM in which interFOAM is the base solver used. The PISO scheme is used for the simulations of fluid flow in interstice. Fig 3 shows bundle of capillaries. There can be two arrangements possible for these capillaries stacked in vertical fashion. Fig 4 shows a unit cell when bundle of capillaries is put in uniform arrangement one above another. The dark shaded region in Fig 4 is the cross section of interstice created between the circular fine tubes.

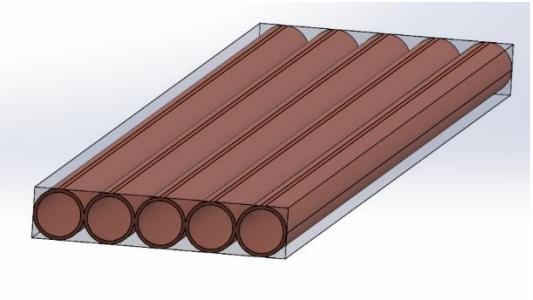


Fig 3: Bundle of capillaries

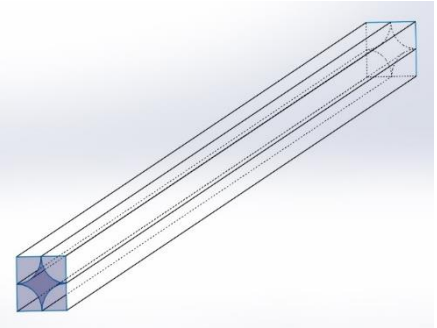


Fig 4: Unit cell of capillary for a uniform arrangement

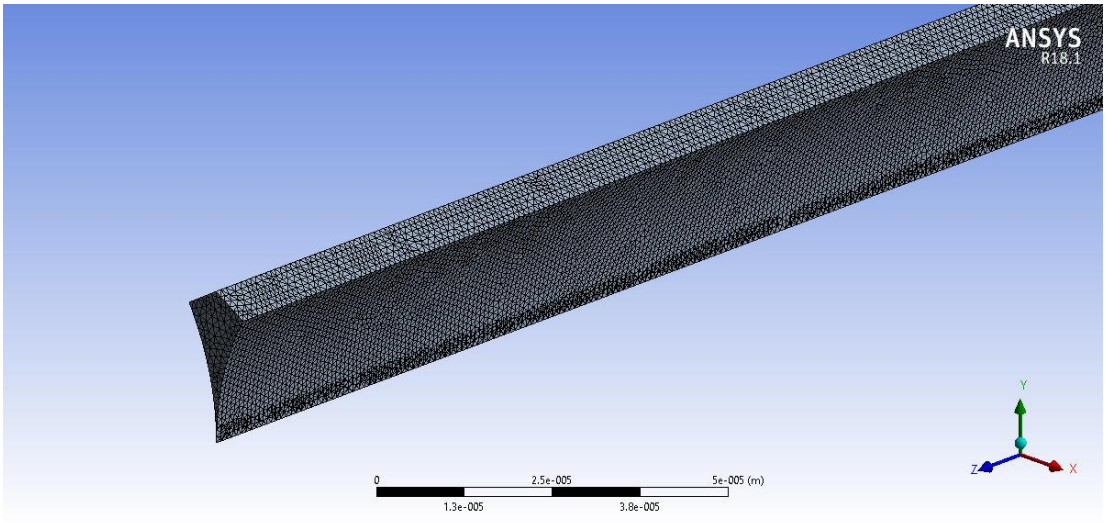


Fig 5: 3-D view of one third portion of the interstice.

4.) The above figure 5 shows the one third part of the interstice. This interstice is created when the capillaries are kept in staggered manner. For this simulations, symmetric boundary conditions are used.

5. Objectives:

Our objective is to do the numerical analysis of the fluid flow through a unit capillary-interstice which contains sharp corners with dynamic contact angle model including the evaporation and boiling heat transfer phenomena to find the various final parameters based on the given heat input which will decide the final capillary length and geometry for the experiments. Further, the contact line evaporation concept will also be applied and the experiments will be done for bundle to capillaries.

5.1: Proposed plans of Action:

- To validate the numerical simulation of fluid flow through porous media with different boundary conditions.
- To simulate and to do the grid independence study for the fluid flow in porous media with heat transfer and compare with the benchmark simulation results.
- To perform 3-D transient simulation of flow inside a micro pipe (interstice) using VOF modeling including phase change (evaporation and condensation) with dynamic contact angle model and validate with the experimental results.

References:

1) V. A. F. Costa , L. A. Oliveira , B. R. Baliga & A. C. M. Sousa, **Simulation Of Coupled Flows In Adjacent Porous And Open Domains Using A control-volume Finite element Method**, *Numerical Heat Transfer, Part A: Applications: An International Journal of Computation and Methodology*,2004; 45:7, 675-697

2) X. B. Chen, P. Yu, S. H. Winoto, H. T. Low, **Forced Convection Over a Backward-Facing Step with a Porous Floor Segment**, *Numerical Heat Transfer, Part A: Applications: An International Journal of Computation and Methodology*, 2008; 53:11, 1211-1230

3) J U Brackbill, D B Kothe, and Y Zemach, **A Continuum Method for Modeling Surface Tension**, *Journal Of Computational Physics*,1992; 100: 335354

4) Š. Šikalo, H.D. Wilhelm, I. V. Roisman, S. Jakirlić, and C. Tropea**, Dynamic contact angle of spreading droplets: Experiments and simulations,** PHYSICS OF FLUIDS, 2005; 17:062103

5) Siddhartha F. Lunkad, Vivek V. Buwa, K.D.P. Nigam**, Numerical simulations of drop impact and spreading on horizontal and inclined surfaces**, *Chemical Engineering Science*,2007; 62:7214 – 7224

6) S. Hardt a, F. Wondra, **Evaporation model for interfacial flows based on a continuum-field representation of the source terms**, *Journal of Computational Physics*, 2008;227 :5871–58951

7) Alexander S. Rattner, Srinivas Garimella, **Simple Mechanistically Consistent Formulation for Volume-of-Fluid Based Computations of Condensing Flows**, *Journal of Heat Transfer*, 2011;136: 071501-9

8) Christian Kunkelmann & Peter Stephan, **CFD Simulation of Boiling Flows Using the Volume-of-Fluid Method within OpenFOAM**, *Numerical Heat Transfer, Part A: Applications*, 2009; 56:8, 631-646

9) Jose Bico and David Quere, **Rise of Liquids and Bubbles in Angular Capillary Tubes**, *Journal of Colloid and Interface Science*,2002; 247:162–166

10) Alexandre Ponomarenko, David Quere ,Christophe Clanet, **A universal law for capillary rise in corners**, *Journal of Fluid Mechanics*,2011;666:146–154.