Single and two phase heat transfer in novel porous foams

Li, Hongying

2011

Li, H. (2011). Single and two phase heat transfer in novel porous foams. Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/46544>

<https://doi.org/10.32657/10356/46544>

**NANYANG
TECHNOLOGICAL
UNIVERSITY**

**SINGLE AND TWO PHASE HEAT TRANSFER IN
NOVEL POROUS FOAMS**

**LI HONGYING
SCHOOL OF MECHANICAL AND AEROSPACE
ENGINEERING
2011**

**SINGLE AND TWO PHASE HEAT TRANSFER IN
NOVEL POROUS FOAMS**

LI HONGYING

**SCHOOL OF MECHANICAL AND AEROSPACE
ENGINEERING**

A thesis submitted to the
Nanyang Technological University
in fulfilment of the requirements for the degree of
Doctor of Philosophy

2011

ABSTRACT

The rapid development of technology in the last decades brings increased challenges in the cooling of electronic devices. Densely packed electronic systems require more effective methods to dissipate the heat generated by the chips and circuits. Porous media with high effective thermal conductivity and large contact surface area between the solid and fluid phases are well recognised as good heat sinks. However, given the complex micro-scale structures involved in fluid flow and heat transfer in porous media, numerical modelling plays an important role in understanding the physics in both single and two-phase heat transfer processes.

The present study develops in-house codes to investigate fluid flow and heat transfer in porous media. Two-dimensional (2D) and three-dimensional (3D) codes based on the finite volume method (FVM), for both single and two phases are developed. These codes are validated against the existing published works. With the validations made, single-phase flow and heat transfer in porous media of different configurations are investigated. The predicted pressure drops and heat transfer performance are compared favourably with the experimental data. The codes for two-phase heat and fluid flow are used to investigate flow boiling characteristics in both horizontal and vertical channels fully-filled with porous media. The experiments are performed with water as the working fluid. The numerical predictions are in reasonable agreement with those of the experiments.

In single-phase flow, a simplified numerical procedure is proposed to deal with the interface between the porous and open regions. This procedure is applied successfully to a variety of well-known problems in a 2D setting for validation purpose. With this achieved, the procedure is extended to a 3D setting in a dimension-by-dimension manner. Such a 3D code is employed to study fluid flow and heat transfer in the configurations of zigzag and baffle graphite foams which is a novel open-cell porous medium. This 3D code is aimed at the simulation of the transport phenomena in porous media with complex structures which serve as heat sinks in thermal management systems.

For further validation of the developed codes for simulation of single-phase flow and heat transfer in porous media, the experimental studies are performed to compare with the simulation results. The configurations of zigzag and baffle are manufactured based on graphite foam. The physical properties of this graphite foam are obtained from the experiments. The temperature distributions at the substrate of different configurations of graphite foam are measured to compare with the simulation results. Good agreements are achieved between the simulation and experimental results.

In two-phase flow, a numerical procedure is presented to study flow boiling process in porous media. The “modified” Kirchhoff method is introduced to treat the discontinuity of the diffusion coefficient. The numerical procedure proposed is validated against the existing experimental data and then applied successfully to horizontal and vertical porous channels to reveal the evolutions of the boiling process. Upon achieving that, a 3D code is developed for a more realistic simulation of two-phase flow and heat transfer in porous channels. This 3D code

is aimed at the simulation of a large class of phase change flow problems to compare with the actual situations.

In addition to the numerical simulation of phase change in porous media, the experimental study for flow boiling is also carried out. The heat flux for the onset of nucleate boiling and the local temperature distribution measured from the experiments are compared against the simulation results. The reasonable agreements demonstrate the capability of the developed code in the prediction of two-phase heat transfer in porous media.

It is hoped that the developed codes for both single and two-phase flow and heat transfer in porous media can be used as guidelines for the preliminary design of heat sinks in thermal systems.

ACKNOWLEDGEMENT

First of all, I would like to express my sincere gratitude to my supervisor, Prof. Leong Kai Choong, for his invaluable advice, helpful guidance, patience and encouragement throughout the course of my study. I considered myself very fortunate to be his student. The lengthy discussions with Prof. Leong are always helpful and inspiring. Without his scholarly advice, this work would not have been possible.

I would also like to express a deep sense of gratitude and appreciation to Dr. Jin Liwen. He helped me greatly in both my studies and well-being.

To Dr. Yap Yitfatt, I wish to express my heartfelt gratitude. He has been a constant supporter of this research. His valuable advice on coding the program throughout the last four years is crucial for me.

I wish to thank Prof. John Chai for his help in this research. I would also like to thank technical staff, Mr. Yuan Kee Hock for his assistance in the experiments. The financial support of Nanyang Technological University (NTU) is gratefully acknowledged.

I am fortunate to have my friends in the Thermal and Fluids Research Laboratory of NTU who have rendered much help to me during the past years.

Special thanks go to my parents, my brother and his wife for their faith in me and encouragement through these years; to them this work is humbly dedicated.

Lastly, I wish to thank my husband for his love and understanding throughout these years. He had always used every imaginable way to support my research. Without him, this dissertation would not be completed in time.

CONTENTS

ABSTRACT.....	I
ACKNOWLEDGEMENT	IV
CONTENTS.....	V
NOMENCLATURE	XIII
LIST OF FIGURES	XVII
LIST OF TABLES.....	XXVI
PUBLICATIONS ARISING FROM THIS THESIS.....	XXVII
CHAPTER 1 INTRODUCTION	1
1.1 BACKGROUND	1
1.1.1 Heat Density Issue	1
1.1.2 Development of Cooling Schemes	2
1.1.3 Porous Media	5
1.2 OBJECTIVE AND SCOPE	7
1.3 OUTLINE OF THE THESIS.....	9
CHAPTER 2 LITERATURE REVIEW	10
2.1 LOCAL VOLUME-AVERAGING METHOD.....	11
2.2 MATHEMATICAL FORMULATION IN POROUS MEDIA.....	13
2.2.1 Conservation Equations for Single-Phase Flow and Heat Transfer...13	
2.2.1.1 <i>The Continuity Equation</i>	13
2.2.1.2 <i>The Momentum Equations</i>	14
2.2.1.3 <i>The Energy Equations</i>	17
2.2.2 Conservation Equations for Two-Phase Flow and Heat Transfer	20
2.2.2.1 <i>Conservation Equations Based on the Separated Flow Model</i> ..	21

2.2.2.2 <i>Conservation Equations Based on the Two-Phase Mixture Model</i>	26
2.3 SINGLE-PHASE FLOW AND HEAT TRANSFER IN OPEN-CELL FOAMS	29
2.4 TWO-PHASE FLOW AND HEAT TRANSFER IN POROUS MEDIA	34
2.5 SUMMARY	41
CHAPTER 3 NUMERICAL SCHEME FOR SINGLE-PHASE FLOW AND HEAT TRANSFER IN POROUS MEDIA	43
3.1 GOVERNING EQUATIONS IN THREE-DIMENSIONS	43
3.2 THE FINITE VOLUME DISCRETISATION	46
3.2.1 Discretisation of Conservation Equations	47
3.2.1.1 <i>Discretisation Equations for Dependent Variables</i>	48
3.2.1.2 <i>SIMPLER Method for the Momentum Equation</i>	51
3.2.2 Implementation of Boundary Conditions	55
3.2.2.1 <i>Implementation for the Velocity Boundary Condition</i>	55
3.2.2.2 <i>Implementation for the Energy Boundary Condition</i>	56
3.3 SOLUTION PROCEDURE	58
3.4 CONVERGENCE CRITERION	59
3.5 SUMMARY	60
CHAPTER 4 CASE STUDIES IN A 2D DOMAIN FOR SINGLE-PHASE FLOW AND HEAT TRANSFER	61
4.1 BEAVERS - JOSEPH FLOW PROBLEM	61
4.1.1 Problem Specification	61
4.1.2 Numerical Details	62
4.1.3 Results and Discussion	62

4.2 POROUS PLUG FLOW	64
4.2.1 Problem Specification.....	64
4.2.2 Numerical Details	64
4.2.3 Results and Discussion	64
4.3 HEAT TRANSFER BY USING THE LOCAL THERMAL EQUILIBRIUM (LTE) MODEL.....	67
4.3.1 Problem Specification.....	67
4.3.2 Numerical Details	68
4.3.3 Results and Discussion	68
4.4 HEAT TRANSFER BY USING THE LOCAL THERMAL NON- EQUILIBRIUM (LTNE) MODEL.....	69
4.4.1 Problem Specification.....	69
4.4.2 Numerical Details	70
4.4.3 Results and Discussion	70
4.5 FLOW AND HEAT TRANSFER IN A CHANNEL WITH STAGGERED POROUS BLOCKS	72
4.5.1 Problem Specification.....	72
4.5.2 Numerical Details	73
4.5.3 Results and Discussion	74
4.5.3.1 Basic Test.....	74
4.5.3.2 Effect of Darcy Number	76
4.5.3.3 Effect of Reynolds Number	79
4.5.3.4 Effect of Porous Block Height	83
4.5.3.5 Effect of Porous Block Width.....	86
4.5.3.6 Effect of Thermal Conductivity Ratio	88

4.5.3.7 <i>Effect of the Porous Blocks on Pressure Drop</i>	89
4.6 SUMMARY	91
CHAPTER 5 CASE STUDIES IN A 3D DOMAIN FOR SINGLE-PHASE FLOW AND HEAT TRANSFER	93
5.1 RELEVANT RELATIONS	93
5.1.1 Effective Thermal Conductivity	93
5.1.1.1 <i>Effective Thermal Conductivity of Upper Portion</i>	97
5.1.1.2 <i>Effective Thermal Conductivity of Middle Portion</i>	98
5.1.1.3 <i>Overall Effective Thermal Conductivity of the Unit Cell</i>	100
5.1.1.4 <i>Results and Discussion</i>	101
5.1.2 Convective Heat Transfer	102
5.1.3 Boundary Conditions	103
5.2 FLOW AND HEAT TRANSFER IN BLOCK (BLK) GRAPHITE FOAM	103
5.2.1 Problem Specification	104
5.2.2 Numerical Details	104
5.2.3 Results and Discussion	105
5.2.3.1 <i>Validation Test</i>	106
5.2.3.2 <i>Velocity Field</i>	106
5.2.3.3 <i>Temperature Field</i>	107
5.3 FLOW AND HEAT TRANSFER IN ZIGZAG (ZZG) GRAPHITE FOAM	108
5.3.1 Problem Specification	108
5.3.2 Numerical Details	110
5.3.3 Results and Discussion	110

5.3.3.1 <i>Velocity Field</i>	110
5.3.3.2 <i>Temperature Field</i>	114
5.4 FLOW AND HEAT TRANSFER IN BAFFLE (BAF) GRAPHITE FOAM	115
5.4.1 Problem Specification	115
5.4.2 Numerical Details	115
5.4.3 Results and Discussion	116
5.4.3.1 <i>Velocity Field</i>	116
5.4.3.2 <i>Temperature Field</i>	119
5.5 SUMMARY	120
CHAPTER 6 EXPERIMENTAL STUDIES FOR SINGLE-PHASE FLOW AND HEAT TRANSFER	121
6.1 TEST SPECIMENS AND EXPERIMENTAL SETUP	121
6.1.1 Different Configurations of Graphite Foams	121
6.1.2 Test Section	121
6.2 MEASURING SYSTEM	123
6.2.1 Test Sensors	123
6.2.2 Data Acquisition System	124
6.3 EXPERIMENTAL PROCEDURE	125
6.3.1 Calibration of Sensors	125
6.3.2 Experimental Procedures	128
6.4 DETERMINATION OF PROPERTIES OF GRAPHITE FOAM	129
6.5 UNCERTAINTY ANALYSIS	131
6.6 COMPARISON BETWEEN EXPERIMENTAL AND NUMERICAL RESULTS	132

6.6.1 Heat Transfer Performance	132
6.6.2 Pressure Drop.....	135
6.7 SUMMARY	136
CHAPTER 7 NUMERICAL SCHEME FOR TWO-PHASE FLOW AND HEAT TRANSFER IN POROUS MEDIA	137
7.1 GOVERNING EQUATIONS IN THREE-DIMENSIONS.....	137
7.2 FINITE VOLUME DISCRETISATION	138
7.2.1 Discretisation of Conservation Equations.....	138
7.2.1.1 Discretisation Procedure for Momentum Equations	138
7.2.1.2 Discretisation Procedures for Energy Equation	140
7.2.2 Implementation of Boundary Conditions	140
7.2.2.1 Implementation of the Velocity Boundary Condition	140
7.2.2.2 Implementation of the Enthalpy Boundary Condition	141
7.3 SOLUTION PROCEDURE.....	141
7.4 CONVERGENCE CRITERION	142
7.5 SUMMARY	143
CHAPTER 8 TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT TRANSFER IN POROUS MEDIA	144
8.1 RELEVANT RELATIONS	144
8.1.1 Relative Permeability.....	144
8.1.2 Capillary Pressure	145
8.1.3 Treatment of the Diffusion Coefficient	146
8.2 TRANSIENT TWO-PHASE FLOW AND HEAT TRANSFER IN A 2D HORIZONTAL POROUS CHANNEL.....	149
8.2.1 Problem Specification.....	150

8.2.2 Numerical Details	150
8.2.3 Results and Discussion	152
8.2.3.1 Code Validation	152
8.2.3.2 Discrete Heat Flux at the Lower Wall - the BH Case	153
8.2.3.3 Discrete Heat Flux at the Upper Wall - the TH Case.....	161
8.2.3.4 Discrete Heat Flux at Both the Lower and Upper Walls - the BTH Case	166
8.2.3.5 Variation of the Vapour Volume Fraction.....	172
8.3 TRANSIENT TWO-PHASE FLOW AND HEAT TRANSFER IN A 2D VERTICAL POROUS CHANNEL.....	173
8.3.1 Problem Specification.....	173
8.3.2 Numerical Details	174
8.3.3 Results and Discussion	175
8.3.3.1 Effects of Rayleigh Number in Aiding Flow	176
8.3.3.2 Effects of Peclet Number in Aiding Flow.....	183
8.3.3.3 Effects of Rayleigh Number in Opposing Flow	186
8.3.3.4 Effects of Peclet Number in Opposing Flow.....	192
8.3.3.5 Liquid Saturation along the Heat Wall.....	195
8.4 TWO-PHASE FLOW AND HEAT TRANSFER IN A 3D ASYMMETRICALLY HEATED POROUS CHANNEL	197
8.4.1 Problem Specification.....	197
8.4.2 Numerical Details	198
8.4.3 Results and Discussion	199
8.4.3.1 Effects of Peclet Number on Fluid Flow and Heat Transfer	199
8.4.3.2 Effects of Rayleigh Number on Fluid Flow and Heat Transfer	211

8.5 SUMMARY	216
CHAPTER 9 EXPERIMENTAL STUDIES OF FLOW BOILING IN POROUS MEDIA	217
9.1 TEST SPECIMEN AND EXPERIMENTAL SETUP	217
9.1.1 Test Specimen	217
9.1.2 Experimental Setup	218
9.1.3 Experimental Procedure	220
9.1.4 Uncertainty Analysis	221
9.2 RESULTS AND DISCUSSION	221
9.2.1 Nucleate Boiling	221
9.2.2 Hysteresis Effect	224
9.2.3 Mass Fraction of Vapour	226
9.3 SUMMARY	228
CHAPTER 10 CONCLUSIONS AND RECOMMENDATIONS FOR FUTURE WORK	229
10.1 CONCLUSIONS	229
10.1.1 Single-Phase Flow and Heat Transfer	229
10.1.2 Two-Phase Flow and Heat Transfer	231
10.2 CONTRIBUTIONS	233
10.3 RECOMMENDATIONS FOR FUTURE WORK	236
REFERENCES	238
APPENDIX: LIBRARY OF COMPUTER PROGRAMS	252

NOMENCLATURE

a	side length of the wall (m)
A_c	heated area of cross section (m ²)
a_B	coefficient in the discretized general transport equation
a_E	coefficient in the discretized general transport equation
a_N	coefficient in the discretized general transport equation
a_P	coefficient in the discretized general transport equation
a_S	coefficient in the discretized general transport equation
a_T	coefficient in the discretized general transport equation
a_W	coefficient in the discretized general transport equation
b	coefficient in the discretized transport equation
$\mathbf{B}$	total drag force per unit volume (N/m ³)
C_E	inertial coefficient of porous media
C_p	specific heat of solid (J/kg·K)
d_c	diameter of the opening (m)
d_l	ligament diameter of metal foam (m)
d_p	pore diameter of porous media (m)
Da	Darcy number based on channel height
D_h	hydraulic diameter of channel (m)
D_s	capillary diffusion coefficient (m ² /s)
f	body force per unit mass (m/s)
$f(s)$	hindrance function
F	flow rate through control volume (kg/s)
$\mathbf{g}$	gravity vector (m/s ²)
h	heat transfer coefficient (W/m ² ·K); enthalpy (J/m ³)
h_{fg}	latent heat of phase change (J/kg)
H	height of channel (m)
$\mathbf{j}$	diffusive mass flux (kg/m ² ·s)
$J(s)$	capillary pressure function
J	total flux
k	relative permeability
k_d	thermal dispersion conductivity (W/m·K)

k_e	effective thermal conductivity (W/m·K)
k_f	thermal conductivity of fluid phase (W/m·K)
k_{fe}	effective thermal conductivity of fluid (W/m·K)
k_{se}	effective thermal conductivity of solid (W/m·K)
k_{sl}	longitudinal thermal conductivity of solid phase (W/m·K)
k_{st}	transverse thermal conductivity of solid phase (W/m·K)
k_{sj}	thermal conductivity of solid phase at the juncture (W/m·K)
K	permeability of the porous medium (m ²)
L	length of the test section (m)
m	constant
m_γ	dimensionless parameter $m_\gamma = \gamma_{ad}(\lambda-1)$
m_ζ	dimensionless parameter $m_\zeta = \zeta_{ad}(\lambda-1)$
Nu_{sf}	Nusselt number based on the ligament diameter
p	pressure (Pa)
p'	correction for pressure (Pa)
p^*	guessed pressure (Pa); dimensionless pressure
P	perimeter (m)
Pe	Peclet number, $Pe = u_{in}W/\alpha$
Pr	Prandtl number
Pr_e	effective Prandtl number
q	power input (W)
q''	heat flux (W/cm ²)
Ra	Rayleigh number, $Ra = KWg\rho_l C_{pl}/\nu_l k_{eff}$
Re_d	Reynolds number based on the ligament diameter
Re_H	Reynolds number based on the channel height
Re_K	Reynolds number based on the permeability
Re_p	Reynolds number based on the pore diameter
s	liquid saturation
S	source term
t	time (s); thickness of solid component (m)
T	temperature (°C)
u	velocity component (m/s)
U	average flow velocity (m/s)
V	volume (m ³)

W	width (m)
x, y, z	coordinates

Greek Symbols

α_{sf}	specific surface area of porous media (m^2/m^3)
β	thermal expansion coefficient (1/K)
β_{ad}	dimensionless parameter $\beta_{ad} = d_c/(a-2t)$
β_{at}	dimensionless parameter $\beta_{at} = t/(a-2t)$
γ	advection correction coefficient
γ_{ad}	dimensionless parameter $\gamma_{ad} = d_c/a$
γ_{at}	dimensionless parameter $\gamma_{at} = t/a$
$\Delta\rho$	$\rho_l - \rho_v$ (kg/m^3)
ρ	density (kg/m^3)
ε	porosity
λ	relative mobility
λ_m	dimensionless parameter $\lambda_m = k_{em1}/k_{em2}$
μ	dynamic viscosity ($\text{kg}/\text{m}\cdot\text{s}$)
ν	kinetic viscosity (m^2/s)
σ	surface tension (N/m)
Ω	effective heat capacitance ratio
Γ	diffusion coefficient (m^2/s)
ϕ	quantity being averaged
Γ	diffusion coefficient (m^2/s)
θ	angle
ζ_{ad}	dimensionless parameter $\zeta_{ad} = d_c/(a-2t)$

Subscripts

ave	average
b,e,w,n,s,t	control volume faces
B,E,W,N,S,T	grid nodes
f	fluid phase
i	direction of component
in	inlet
P	central grid point
s	solid phase

sf	solid and fluid
w	wall
x	local value

Superscripts

o	former iteration value
n	updated value
*	dimensionless component

LIST OF FIGURES

Fig. 1-1 Heat density trend for different products (Kenneth, 2006)	2
Fig. 1-2 Comparison of cooling characteristics of single and two-phase heat convection heat transfer (Mudawar, 2001).....	5
Fig. 1-3 Typical internal structures of (a) aluminium foam and (b) graphite foam	7
Fig. 2-1 Schematic of a REV	11
Fig. 2-2 Schematic diagram of the channel with porous and open layers	34
Fig. 2-3 Schematic diagram of the channel with staggered porous blocks.....	34
Fig. 3-1 Sketch of control volume	47
Fig. 3-2 Sketch of staggered grid in a control volume.....	47
Fig. 3-3 Location of staggered points for u velocity.....	52
Fig. 3-4 Location of staggered points for v velocity	52
Fig. 3-5 Velocity at the outlet boundary	56
Fig. 3-6 Half control volume near left boundary	57
Fig. 4-1 Schematic diagram of open and porous conjugated domains (not to scale)	62
Fig. 4-2 Velocity profile in a conjugated domain with $Re_H = 1$ and $Da = 10^{-2}$.	63
Fig. 4-3 Velocity profile in a conjugated domain with $Re_H = 1$ and $Da = 10^{-3}$.	63
Fig. 4-4 Schematic diagram of full porous domain (not to scale)	64
Fig. 4-5 Dimensionless velocity at the centreline of full porous domain with $Re_H = 1$ and $Da = 10^{-2}$	65

Fig. 4-6 Dimensionless pressure drop in full porous domain with $Re_H = 1$ and $Da = 10^{-2}$	65
Fig. 4-7 (a) Betchen <i>et al.</i> (2006)'s results (b) present results in the porous plug domain	66
Fig. 4-8 Schematic diagram of simulation domain (not to scale)	67
Fig. 4-9 Comparison of temperatures obtained from the current and those of Alazmi and Vafai (2001).	68
Fig. 4-10 Schematic diagram of simulation domain	69
Fig. 4-11 Temperature difference contours in (a) solid phase and (b) fluid phase (dash lines for current simulation and solid lines for Betchen <i>et al.</i> 's (2006) results).....	71
Fig. 4-12 Schematic diagram of the channel with staggered porous blocks.....	73
Fig. 4-13 (a) Velocity vectors (b) Local Nusselt number distribution at lower and upper walls with $Re = 100$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.8$, $w/H = 0.2$ and $k_e/k_f = 1$	75
Fig. 4-14 (a) Velocity field with $Re = 100$, $a/H = 0.8$, $w/H = 0.2$ and $k_e/k_f = 1$ (a) $Da = 10^{-3}$, $\Lambda_H = 5.22$, (b) $Da = 10^{-5}$, $\Lambda_H = 52.2$, (c, d) close-up views of circulatory region at $Da = 10^{-5}$ and $\Lambda_H = 52.2$	77
Fig. 4-15 Effects of Darcy number on local Nusselt number with $Re = 100$, $a/H = 0.8$, $w/H = 0.2$ and $k_e/k_f = 1$ at (a) lower wall (b) upper wall	79
Fig. 4-16 Velocity field with $Re = 100$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.4$, $w/H = 0.2$ and $k_e/k_f = 1$ (a) velocity field (b, c) close-up views of circulatory region..	81
Fig. 4-17 Velocity field with $Re = 200$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.4$, $w/H = 0.2$ and $k_e/k_f = 1$ (a) velocity field (b, c) close-up views of circulatory region..	82

Fig. 4-18 Velocity field with $Re = 300$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.4$, $w/H = 0.2$ and $k_e/k_f = 1$ (a) velocity field (b, c) close-up views of circulatory region..	82
Fig. 4-19 Effects of Reynolds number on local Nusselt number with $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.4$, $w/H = 0.2$ and $k_e/k_f = 1$ at (a) lower wall (b) upper wall	83
Fig. 4-20 Velocity field with $Re = 100$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.6$, $w/H = 0.2$ and $k_e/k_f = 1$ (a) velocity field (b, c) close-up view of circulatory region ...	84
Fig. 4-21 Effects of the porous block height on local Nusselt number with $Re = 100$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $w/H = 0.2$ and $k_e/k_f = 1$ at (a) lower wall (b) upper wall.....	85
Fig. 4-22 Velocity field with $Re = 100$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.8$, $k_e/k_f = 1$ (a) $w/H = 0.4$ (b) $w/H = 0.8$ (c, d) close-up view of circulatory region.....	87
Fig. 4-23 Effect of the porous block width on local Nusselt number with $Re = 100$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.8$ and $k_e/k_f = 1$ at (a) lower wall (b) upper wall.....	88
Fig. 4-24 Effect of the thermal conductivity ratio on local Nusselt number with $Re = 100$, $Da = 10^{-5}$, $\Lambda_H = 52.2$, $a/H = 0.8$ and $w/H = 0.2$ at (a) lower wall (b) upper wall	89
Fig. 4-25 Normalised pressure drop variation with (a) Darcy number for $Re = 100$, (b) Reynolds number for $Da = 10^{-4}$, $\Lambda_H = 16.5$ at $a/H = 0.8$, $w/H = 0.2$ and $k_e/k_f = 1$	91
Fig. 4-26 Normalised pressure drop variation with $Re = 100$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $k_e/k_f = 1$ (a) height with $w/H = 0.2$ (b) width with $a/H = 0.8$	91
Fig. 5-1 Pore structures of graphite foam (a) under SEM (b) proposed unit cell	95

Fig. 5-2 (a) Upper part; (b) middle part and (c) bottom part of the proposed unit cell.....	96
Fig. 5-3 Heat conduction path for the upper part.....	97
Fig. 5-4 Heat conduction path for layer 1 in the upper part.....	98
Fig. 5-5 Variations of effective thermal conductivity with porosity	102
Fig. 5-6 Schematic diagram of BLK foam domain	104
Fig. 5-7 Comparison of 2D and 3D results.....	106
Fig. 5-8 Velocity profiles at different locations.....	107
Fig. 5-9 (a) Solid and (b) fluid surface temperature distribution at $Re_p = 40$ under $q'' = 0.8 \text{ W/cm}^2$	108
Fig. 5-10 (a) Schematic diagram of ZZG foam (b) dimensions of ZZG foam	109
Fig. 5-11 Dimensionless velocity profiles in ZZG foam at (a) representative locations (b) velocity profile before the 1 st foam wall; (c) velocity profile in the 1 st foam wall; (d) velocity profile in the slot between the 1 st and 2 nd foam wall; (e) velocity profile in the 2 nd foam wall.....	113
Fig. 5-12 Velocity in xz plane at $y/D_h = 0.12$ parallel to the flow direction	113
Fig. 5-13 (a) Solid and (b) fluid temperatures at selected slices in ZZG foam	114
Fig. 5-14 (a) Schematic diagram of BAF foam (b) dimensions of BAF foam	115
Fig. 5-15 Dimensionless velocity profiles in BAF foam at (a) representative locations; (b) velocity profile before the 1 st foam wall; (c) velocity profile in the 1 st foam wall; (d) velocity profile in the slot between the 1 st and 2 nd foam wall; (e) velocity profile in the 2 nd foam wall.....	118
Fig. 5-16 (a) Solid and (b) fluid temperatures in the selected slices in the BAF foam	119

Fig. 6-1 Photographs of (a) BLK (b) ZZG and (c) BAF foams.....	121
Fig. 6-2 (a) Experimental facility (b) cross-sectional view of test section (c) photograph of the test section	123
Fig. 6-3 Calibration curve of the thermocouple.....	126
Fig. 6-4 Calibration curve of pressure sensor	127
Fig. 6-5 Calibration curve of hot-wire	128
Fig. 6-6 Pressure drop versus velocity for 75% porosity graphite foam	130
Fig. 6-7 Temperature distributions at the heated surface with $Re_p = 50$ in (a) BLK foam; (b) BAF foam; (c) ZZG foam.....	133
Fig. 6-8 Variations of Nu_{ave} with Re_p in different configurations of graphite foams.....	135
Fig. 6-9 Pressure drop in different configurations of graphite foams.....	136
Fig. 7-1 Sketch of boundary control volumes.....	141
Fig. 7-2 General structure of the program	143
Fig. 8-1 Variations of diffusion coefficient with s	146
Fig. 8-2 Temperature contours obtained using the (a) harmonic mean method, and (b) “modified” Kirchhoff method	149
Fig. 8-3 Schematic diagram of the problem (a) BH; (b) TH; (c) BTH.....	151
Fig. 8-4 Comparison of the current simulation and the experimental results of Easterday <i>et al.</i> (1995) for (a) $x = 1/3L$; (b) $x = 1/2L$	153
Fig. 8-5 Liquid velocity vectors at (a) 2 s; (b) 10 s; (c) 15 s (d) 18 s; (e) 40 s; (f) 60 s; (g) 150 s; (h) 190 s; (i) 250 s; (j) 1000 s	158
Fig. 8-6 Vapour velocity vectors at (a) 18 s; (b) 25 s; (c) 40 s; (d) 60 s; (e) 150 s; (f) 190 s; (g) 250 s; (h) 1000 s	159

Fig. 8-7 Temperature and liquid saturation contours at (a) 2 s; (b) 10 s (c) 15 s; (d) 18 s; (e) 40 s; (f) 60 s; (g) 150 s; (h) 190 s; (i) 250 s; (j) 1000 s	161
Fig. 8-8 Liquid velocity vectors at (a) 2 s; (b) 10 s; (c) 15 s; (d) 21 s; (e) 40 s; (f) 60 s; (g) 150 s; (h) 190 s; (i) 250 s; (j) 1000 s	164
Fig. 8-9 Vapour velocity vectors at (a) 28 s; (b) 30 s; (c) 40 s; (d) 60 s; (e) 150 s; (f) 190 s; (g) 250 s; (h) 1000 s	165
Fig. 8-10 Temperature and liquid saturation contours at (a) 2 s; (b) 10 s; (c) 15 s; (d) 21 s;(e) 40 s; (f) 60 s; (g) 150 s; (h) 190 s; (i) 250 s; (j) 1000 s	166
Fig. 8-11 Liquid velocity vectors at (a) 10 s; (b) 15 s; (c) 50 s; (d) 75 s; (e) 85 s; (f) 120 s; (g) 150 s; (h) 190 s; (i) 250 s; (j) 1000 s	170
Fig. 8-12 Vapour velocity vectors at (a) 85 s; (b) 90 s; (c) 120 s; (d) 150 s; (e) 190 s; (f) 250 s; (g) 1000 s	170
Fig. 8-13 Temperature and liquid saturation contours at (a) 10 s; (b) 15 s; (c) 50 s; (d) 75 s; (e) 85 s; (f) 120 s; (g) 150 s; (h) 190 s; (i) 250 s; (j) 1000 s	172
Fig. 8-14 Time variation of vapour fraction for different cases	173
Fig. 8-15 Schematic of the problem and the coordinate systems for (a) aiding flow, and (b) opposing flow	175
Fig. 8-16 Liquid velocity vectors for aiding flow at $Pe = 0.24$ for (a) $Ra = 113$; (b) $Ra = 226$; (c) $Ra = 339$	180
Fig. 8-17 Vapour velocity vectors for aiding flow at $Pe = 0.24$ for (a) $Ra = 113$; (b) $Ra = 226$; (c) $Ra = 339$	181
Fig. 8-18 Temperature and liquid saturation contours for aiding flow at $Pe = 0.24$ for (a) $Ra = 113$; (b) $Ra = 226$; (c) $Ra = 339$	182
Fig. 8-19 Liquid velocity vectors for aiding flow at $Ra = 226$ for (a) $Pe = 0.12$; (b) $Pe = 0.48$	184

Fig. 8-20 Vapour velocity vectors for aiding flow at $Ra = 226$ for (a) $Pe = 0.12$; (b) $Pe = 0.48$	185
Fig. 8-21 Temperature and liquid saturation contours for aiding flow at $Ra =$ 226 for (a) $Pe = 0.12$; (b) $Pe = 0.48$	186
Fig. 8-22 Liquid velocity vectors for opposing flow at $Pe = 0.24$ for (a) $Ra =$ 113 ; (b) $Ra = 226$; (c) $Ra = 339$	189
Fig. 8-23 Vapour velocity vectors for opposing flow at $Pe = 0.24$ for (a) $Ra =$ 113 ; (b) $Ra = 226$; (c) $Ra = 339$	190
Fig. 8-24 Temperature and liquid saturation contours for aiding flow at $Pe =$ 0.24 for (a) $Ra = 113$; (b) $Ra = 226$; (c) $Ra = 339$	191
Fig. 8-25 Liquid velocity vectors for opposing flow at $Ra = 226$ for (a) $Pe =$ 0.12 ; (b) $Pe = 0.48$	193
Fig. 8-26 Vapour velocity vectors for opposing flow at $Ra = 226$ for (a) $Pe =$ 0.12 ; (b) $Pe = 0.48$	194
Fig. 8-27 Temperature and liquid saturation contours for opposing flow at $Ra =$ 226 for (a) $Pe = 0.12$; (b) $Pe = 0.48$	194
Fig. 8-28 The variation of liquid saturation s along the heated wall for aiding and opposing flows under different Rayleigh numbers	196
Fig. 8-29 Variation of liquid saturation s along the heated wall for aiding and opposing flows under different Peclet numbers	196
Fig. 8-30 Comparison of 3D results under symmetric boundary conditions with 2D results (a) $x = 1/3L$; (b) $x = 1/2L$	198
Fig. 8-31 Schematic of the problem and the coordinate systems (a) the simulation domain; (b) selected planes.....	199

Fig. 8-32 Liquid velocity in different planes for $Pe = 0.06$ and $Ra = 226$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$	202
Fig. 8-33 Vapour velocity in different planes for $Pe = 0.06$ and $Ra = 226$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$	203
Fig. 8-34 Liquid velocity in different planes for $Pe = 0.12$ and $Ra = 226$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$	205
Fig. 8-35 Vapour velocity in different planes for $Pe = 0.12$ and $Ra = 226$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$	206
Fig. 8-36 Liquid velocity in different planes for $Pe = 0.24$ and $Ra = 226$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$	206
Fig. 8-37 Vapour velocity in different planes for $Pe = 0.24$ and $Ra = 226$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$	207
Fig. 8-38 Temperature distributions in the 3D domain for $Ra = 226$ and (a) $Pe = 0.06$; (b) $Pe = 0.12$; (c) $Pe = 0.24$	209
Fig. 8-39 Liquid saturation on the wall with heated section for $Ra = 226$ and (a) $Pe = 0.06$; (b) $Pe = 0.12$; (c) $Pe = 0.24$	210
Fig. 8-40 Liquid velocity in different planes for $Pe = 0.06$ and $Ra = 113$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$	212

Fig. 8-41 Vapour velocity in different planes for $Pe = 0.06$ and $Ra = 113$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$	213
Fig. 8-42 Liquid velocity in different planes for $Pe = 0.06$ and $Ra = 56$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$	213
Fig. 8-43 Vapour velocity in different planes for $Pe = 0.06$ and $Ra = 56$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$	214
Fig. 8-44 Temperature distributions in the 3D domain for $Pe = 0.06$ and (a) $Ra = 113$; (b) $Ra = 56$	215
Fig. 8-45 Liquid saturation on the wall with heated section for $Pe = 0.06$ and (a) $Ra = 113$; (b) $Ra = 56$	216
Fig. 9-1 (a) Tested aluminium foams and (b) pores under SEM	218
Fig. 9-2 (a) Schematic diagram of experimental setup (b) Sectional view of the test channel	220
Fig. 9-3 Boiling curves in aluminium foam with 40 PPI.....	223
Fig. 9-4 Comparisons of the temperatures at the bottom and the centre of the test sample for water	223
Fig. 9-5 (a) The variation of temperature along flow direction at heat flux $q'' = 10 \text{ W/cm}^2$; (b) velocity distribution at xy plane	225
Fig. 9-6 Hysteresis effects in aluminium foam with 40 PPI.....	225
Fig. 9-7 Variation of mass fraction of vapour with heat flux	227
Fig. 9-8 Variation of mass fraction of vapour with flow rate	228

LIST OF TABLES

Table 2-1 Variables in the two-phase mixture model.....	28
Table 4-1 Characteristic properties of metal foam for sample #2	69
Table 5-1 Characteristic properties of graphite foams.....	105
Table 9-1 Characteristic properties of metal foams.....	218
Table A-1 List of FORTRAN programs for single-phase flow and heat transfer	252
Table A-2 List of FORTRAN programs for two-phase flow and heat transfer	253

PUBLICATIONS ARISING FROM THIS THESIS

Journal Papers

1. Leong, K.C., **Li, H.Y.**, Jin, L.W., and Chai, J.C., "Numerical and experimental study of forced convection in graphite foams of different configurations", *Applied Thermal Engineering*, Vol. 30, No. 5, pp. 520 – 532, 2010.
2. **Li, H.Y.**, Leong, K.C., Jin, L.W., and Chai, J.C., "Analysis of fluid flow and heat transfer in a channel with staggered porous blocks", *International Journal of Thermal Sciences*, Vol. 49, No. 6, pp. 950 – 962, 2010.
3. **Li, H.Y.**, Leong, K.C., Jin, L.W., and Chai, J.C., "Transient two-phase flow and heat transfer with localized heating in graphite foams", *International Journal of Thermal Sciences*, Vol. 49, No. 7, pp. 1115 – 1127, 2010.
4. **Li, H.Y.**, Leong, K.C., Jin, L.W., and Chai, J.C., "Transient behavior of fluid flow and heat transfer with phase change in vertical porous channels", *International Journal of Heat and Mass Transfer*, Vol. 53, Nos. 23 – 24, pp. 5209–5222, 2010.
5. **Li, H.Y.**, Leong, K.C., Jin, L.W., and Chai, J.C., "Three-dimensional numerical simulation of fluid flow with phase change heat transfer in an asymmetrically heated porous channel", *International Journal of Thermal Sciences*, Vol. 49, No. 12, pp. 2363 – 2375, 2010.
6. Leong, K.C., **Li, H.Y.**, Jin, L.W., and Chai, J.C., "Convective heat transfer in graphite foams with baffle and stagger structures", *ASME Journal of Heat Transfer*, Vol. 133, No. 6, pp. 060902-1 - 060902-9, 2011.
7. **Li, H.Y.**, and Leong, K.C., "Experimental and numerical study of single and two-phase flow and heat transfer in aluminum foams", *International Journal of Heat and Mass Transfer*, Vol. 54, Nos. 23 – 24, pp. 4904 – 4912, 2011.
8. Leong, K.C., **Li, H.Y.**, "Theoretical study of the effective thermal conductivity of graphite foam based on a unit cell model", *International Journal of Heat and Mass Transfer*, Vol. 54, Nos. 25 – 26, pp. 5491–5496, 2011.
9. Leong, K.C., Jin, L.W., Pranoto, I., **Li, H.Y.**, and Chai, J.C., "Experimental study of enhanced pool boiling heat transfer using graphite foam inserts", *Defect and Diffusion Forum*, Vols. 312-315, pp. 352 – 357, 2011.
10. Jin, L.W., Leong, K.C., Pranoto, I., **Li, H.Y.**, and Chai, J.C., "Experimental study of a two-phase thermosyphon with porous graphite foam insert", *ASME Journal of Thermal Science and Engineering Applications*, Vol. 3, No.2, pp. 024502-1-024502-6, 2011.

Conference Papers

1. Leong, K.C., Jin, L.W., **Li, H.Y.**, and Chai, J.C., "Forced convection air cooling in porous graphite foam for thermal management applications", *Proceedings of ITherm 2008: Eleventh Intersociety Conference on Thermal and Thermomechanical Phenomena in Electronic Systems*, 28 - 31 May 2008, Orlando, Florida, USA, pp. 57 – 64.
2. Jin, L.W., **Li, H.Y.**, Leong, K.C., and Chai, J.C., "Heat transfer and air flow through highly conductive graphite foam heat sinks", *Proceedings of the 7th International Symposium on Heat Transfer (IHST7)*, 26 - 29 October 2008, Beijing, China.
3. Leong, K.C., **Li, H.Y.**, Jin, L.W., and Chai, J.C., "Convective heat transfer in graphite foams with complex structures", *IEEE Proceedings of the Second International Conference on Thermal Issues in Emerging Technologies, Theory and Applications - ThETA 2*, 17 – 20 December 2008, Cairo, Egypt, pp. 489 – 497.
4. Leong, K.C., **Li, H.Y.**, Jin, L.W., and Chai, J.C., "Convective heat transfer and fluid flow characteristics of graphite foam heat sinks with complex structures", *Proceedings of the 7th World Conference on Experimental Heat Transfer, Fluid Mechanics and Thermodynamics*, 28 June – 3 July 2009, Krakow, Poland.
5. Leong, K.C., **Li, H.Y.**, Jin, L.W., and Chai, J.C., "Two-phase flow and heat in porous graphite foams", *Proceedings of the 11th UK National Heat Transfer Conference*, 6 – 8 September 2009, London, UK.
6. Jin, L.W., **Li, H.Y.**, Leong, K.C., and Chai, J.C., "Experimental and numerical study of forced convection in novel graphite foam heat sinks", *Proceedings of the 11th UK National Heat Transfer Conference*, 6 – 8 September 2009, London, UK.
7. **Li, H.Y.**, Leong, K.C., Jin, L.W., and Chai, J.C., "Analysis of flow and heat transfer in a channel with staggered porous blocks", *Proceedings of the 11th UK National Heat Transfer Conference*, 6 – 8 September 2009, London, UK.
8. Leong, K.C., Jin, L.W., Pranoto, I., **Li, H.Y.**, and Chai, J.C., "Experimental study of enhanced pool boiling heat transfer using graphite foam inserts", *6th International Conference on Diffusion in Solids and Liquids: Mass Transfer, Heat Transfer and Microstructure and Properties, DSL-2010*, 5 – 7 July 2010, Paris, France.
9. Leong, K.C., Jin, L.W., Pranoto, I., **Li, H.Y.**, and Chai, J.C., "Experimental study of a two-phase thermosyphon with graphite foam insert", *14th International Heat Transfer Conference, IHTC14*, August 8 – 13, 2010, Washington, DC, USA.
10. **Li, H.Y.**, Jin, L.W., and Leong, K.C., "Experimental and numerical study on flow boiling heat transfer in aluminium foams", *5th International Asia-Oceania Top University League on Engineering (AOTULE) International*

PUBLICATIONS ARISING FROM THIS THESIS

Postgraduate Students Conference on Engineering, 1 – 2 November 2010,
Bandung, Indonesia.

CHAPTER 1 INTRODUCTION

1.1 BACKGROUND

1.1.1 Heat Density Issue

With the rapid development of the Information Technology (IT) industry during the past few years, the density of heat dissipation by the communication and computer hardware products has increased significantly. Much attention is given to the heat generated by the equipment from technology spaces, data centres and other computing facilities. Figure 1-1 gives the heat density trend chart from 1992 to 2010 based on products from 15 companies provided by the Uptime Institute Inc. (Kenneth, 2006). These companies include Amdahl, Cisco Systems Company, Dell Computers and other companies. The data shown in Fig. 1-1 before 2005 are the past actual heat densities provided by these companies while the data after 2005 are the heat densities projected by these companies. The estimation is based on the technology trends of their products. These data give a general overview of the real heat density consumed and dissipated by the data processing and communications equipment. In the case of communication equipment, the rate of increase in heat density is more than 1300% over the years from 1992 to 2010. Workstations in this figure are possibly desktops in an office situation. Their heat density increase is more than 200 W/m² from 2008 to 2010. According to the IBM data in 2011, the surface temperature for its chip would reach 6000°C in operation without any cooling! In the Uptime Institute Symposium in 2011, Docca (2011) reported that the heat flux in some IT product can be as high as 1679 W/m². Such a dramatic increase

in heat densities will continue in the future driven by the underlying technology of the semiconductor industry. This presents unprecedented challenges for the thermal management of electronic equipment. While many existing data equipment appear to be able to provide sufficient electrical power, most may not be able to provide sufficient cooling capacity especially in the cases where a great number of future high-performance IT products are installed. Particular attention, therefore, must be paid to the cooling issue.

Fig. 1-1 Heat density trend for different products (Kenneth, 2006)

1.1.2 Development of Cooling Schemes

As mentioned above, rapid development of technology equipment presents increased challenges in the cooling of electronic devices. Densely packed electronic systems require more effective methods to dissipate the heat generated by the chips and circuits. Therefore, thermal management becomes more important for an optimised and successful electronic design. The cooling requirements for electronic equipment pose many challenges due to space, weight, capability of heat dissipation and other constraints. Many innovative techniques such as heat pipes, micro-channels and other exotic designs (Vadakkan *et al.*, 2004; Garimella, 2004) have been explored to improve the

efficiencies of heat transfer devices over the years. The primary concern in thermal management is to find the best cooling schemes to cater to the special needs of individual applications.

There are two main schemes for cooling the electronic equipment. These are direct cooling and indirect cooling. In direct cooling, the surface of the electronic device is directly immersed into a coolant. This reduces the thermal resistance between the device surface and the coolant. It helps the dissipation of high heat flux from the electronic device to the coolant. However, there is a need to ensure both electrical and chemical compatibility between the device and coolant. Air cooling is the most widely used direct cooling scheme for most applications. It is only limited to low heat flux devices. The use of forced air convection can only remove 0.1 W/cm^2 of heat while maintaining the electronic device safely below the industry limit of 85°C (Mudawar, 2001). Indirect cooling uses a heat sink to conduct the heat away from the device to a coolant. This scheme provides more flexibility in the selection of the coolant. However, it increases the thermal resistance between the device surface and the coolant. Liquid cooling is popularly used because remarkable improvement in cooling performance can be attained compared with air cooling. Figure 1-2 shows the relationship between the heat flux and device temperature obtained by using air cooling and liquid cooling schemes (Mudawar, 2001). The data for liquid cooling were obtained using Fluorinert FC-72 as the coolant (Anderson and Mudawar, 1989). Liquid cooling could be implemented with or without boiling. Single-phase liquid natural convection can remove heat fluxes of up to 3 W/cm^2 , displaying a linear increase in the device temperature with the heat flux. Phase change can achieve heat fluxes as high as 50 W/cm^2 with a device

temperature of only 70°C. These are much higher than the heat flux removed by the forced air convection. Phase change can be easily implemented in thermosyphon systems (Farsi *et al.*, 2003; Palm and Khodabandeh, 2003; Gima *et al.*, 2005; Leong *et al.*, 2010a), channel flow boiling systems (Samant and Simon, 1989; Willingham and Mudawar, 1992), jet impingement cooling systems (Mudawar and Wadsworth, 1991; Estes and Mudawar, 1995), and spray cooling systems (Mudawar and Valentine, 1989; Mudawar and Estes, 1996). However, the primary concern with phase change cooling systems is the ability to predict critical heat flux (CHF) in order to ensure that the device operates safely.

The cooling of a thermal system is a problem-dependent issue. Selecting a suitable cooling scheme for a particular application involves the consideration of various criteria to address different concerns. Different schemes can be tailored to the specific needs of the different applications. Although extensive developments of electronic cooling have been achieved during the past few decades, there is still a growing need for the development of innovative cooling schemes and effective heat sinks to dissipate the high heat fluxes generated by the electronic devices.

Fig. 1-2 Comparison of cooling characteristics of single and two-phase heat convection heat transfer (Mudawar, 2001)

1.1.3 Porous Media

The most important issue in indirect cooling scheme is the selection of the heat sinks. There is a variety of materials which can be used as heat sinks, among which porous media are popularly used. It has been demonstrated that heat sinks made of porous media of high thermal conductivity and large surface area can improve heat transfer performance. They are widely used in various industry applications such as heat exchangers, chemical reactors and pipes of different arrangements (Koh and Stevens, 1975; Bubnovich and Toledo, 2007; Jiang *et al.*, 2004). In recent years, novel porous materials with open-cell structure such as metal and graphite foams have emerged as good candidates for cooling electronic components. The premium thermal properties of these foams lend themselves to many applications (Fu *et al.*, 2001; Bhattacharya and Mahajan, 2002; Boomsma *et al.*, 2003; Williams and Roux, 2006).

Metal foams are relatively new materials with low density and good thermal, mechanical and electrical properties. Figure 1-3(a) shows typical internal structures of aluminium foams observed under a Scanning Electron Microscope (SEM). The foams have reticulated structures of open, shaped cells connected

by continuous solid metal ligaments. The experimental study of Kim *et al.* (2005) shows that aluminium foam heat sinks gave 8 ~ 33% higher thermal performance compared to conventional plate fin heat sinks for a single jet impingement. However, the bulk thermal conductivity of the metal foam is far less than its corresponding solid block. For aluminium foam, the bulk thermal conductivity is about 10 W/m·K compared with 180 W/m·K for its ligament component. The graphite foam developed at Oak Ridge National Laboratory (ORNL), USA in 1997 (Klett, 2000) has extremely high ligament thermal conductivity. It possesses higher bulk thermal conductivity than metal foams. Figure 1-3(b) shows the internal structure of the graphite foam. It has predominantly spherical pores with smaller openings between the ligaments. The role of the structure on the thermal properties of graphite foams was elaborated by Klett *et al.* (2004). The thermal conductivity of the solid component of graphite foam can be as high as 1700 W/m·K with a bulk thermal conductivity greater than 150 W/m·K which is almost equivalent to that of dense aluminium alloys (Klett *et al.*, 2000). However, the mass density of graphite foam is only about 20% that of aluminium. The advantages of the graphite foam's thermal properties and its potential for improved heat transfer performance have led to applications in automobile radiators, heat sinks and even in spacecraft (Ott *et al.*, 2002; Gallego and Klett, 2003; Straatman *et al.*, 2006; Straatman *et al.*, 2007b). Some research investigations have been performed for graphite foams in thermal management applications (Straatman *et al.*, 2006; Williams and Roux, 2006; Leong *et al.*, 2010a). The results from these studies showed that better heat transfer performance can be achieved by using graphite foam as a replacement material for conventional heat sinks.

Fig. 1-3 Typical internal structures of (a) aluminium foam and (b) graphite foam

Unlike packed beds or granular porous media, metal and graphite foams have a rigid and continuous solid phase. Their superior heat transfer performance makes them popular as heat sinks for electronic cooling applications. Given the new structures for these open-cell porous media, past studies of packed beds may not be applicable. Thus, there is a need to develop fundamental understanding of the transport phenomena in metal and graphite foams.

1.2 OBJECTIVE AND SCOPE

Given the micro-scale structures involved in fluid flow and heat transfer in metal and graphite foams, numerical modelling plays an important role in understanding the physics in both the single and two-phase heat transfer processes. The aim of the present study is not to delve deeply into the physics behind these processes but to provide the means to facilitate the investigation of such processes. The objective of this research is to develop and implement numerical procedures for the prediction of single and two-phase flow and heat transfer in open-cell porous media. The modules in the developed programs are listed in the Appendix. With these developed codes, the transport phenomena of fluid flow and heat transfer characteristics associated with phase change in

porous media are investigated. It is hoped that the numerical procedures developed in the present study as well as the transport phenomena revealed in the case studies can be used as guidelines in the design of porous heat sinks for practical engineering applications.

The scope of the thesis is elaborated as follows:

(1) Two-dimensional (2D) and three-dimensional (3D) numerical models will be developed to study single-phase flow and heat transfer in a porous channel based on the framework of the finite volume method (FVM). The model should be able to handle porous media of different configurations. Validation of the developed model against published works will be performed. The single-phase flow and heat transfer characteristics in porous configurations of block, zigzag and baffle will be investigated.

(2) Experiments on flow in a channel filled with different configurations of graphite foam will be performed. The physical properties of graphite foams such as the porosity, pore diameter, density, permeability and Forchheimer coefficient has to be measured for the purpose of simulation. The temperature distributions at the substrate of the graphite foams and pressure drops for different configurations will be obtained. These data will be compared against the numerical predictions. Based on the numerical and experimental results, a general principle on choosing the best design of the porous heat sinks will be proposed.

(3) Both 2D and 3D numerical models to study two-phase flow and heat transfer in porous channels which aimed at removing a large amount of heat in a cooling system will be developed. The validation of the codes will be performed. With the developed code, the transient boiling process in porous

media in both horizontal and vertical porous channels will be investigated to understand the flow boiling characteristics in porous media.

(4) Experiments on flow boiling in aluminium foams will be performed. The nucleate boiling process and the hysteresis effect will be investigated. Comparisons between the experimental and numerical results of the temperature distribution at the substrate of the porous medium will be made. The prediction of the heat flux for the onset of nucleate boiling will be validated against the simulation results.

1.3 OUTLINE OF THE THESIS

There are ten chapters in this thesis. A brief introduction is presented in Chapter 1. This is followed by a literature review in Chapter 2. The mathematical formulations for both single and two-phase flow and heat transfer in porous media are presented in this chapter. The numerical solution procedure of the single-phase flow and heat transfer in porous media is proposed in Chapter 3. In Chapter 4, the validation of the numerical code and 2D case studies are presented. A 3D numerical study in different configurations of porous media is given in Chapter 5. Chapter 6 describes the experimental studies carried out to validate the simulation results presented in Chapter 5. A numerical scheme for two-phase flow and heat transfer in porous media is proposed in Chapter 7. This is followed by case studies in 2D and 3D settings in Chapter 8. An experimental study to validate the 3D code is described in Chapter 9. Finally, the key results of the present study are summarised in Chapter 10 with future research directions suggested.

CHAPTER 2 LITERATURE REVIEW

Flow and heat transfer in porous media can be modelled via two approaches, *viz.* microscopic approach and macroscopic approach. In the microscopic approach, the simulation is carried out based on the pore structures of the porous media. Using this approach, the transport phenomena in the intricate geometry of the porous medium can be revealed. This approach was employed by many investigators including Boomsma *et al.* (2003), Krishnan *et al.* (2006, 2008), Karimian and Straatman (2009) and Kopanidis *et al.* (2010). Although detailed information on the flow and heat transfer in the pores can be uncovered based on the microscopic approach, it is not feasible to extend it to the entire physical domain due to the excessive computational time and resources required. Besides, the extension of the results for comparison with an actual test domain needs to be further verified. The macroscopic approach is a more practical approach compared to the microscopic approach. By employing this approach, the porous material can be regarded as a continuum and its structural complexity can be ignored. Generally, the use of this method loses the intrinsic characteristics occurring at the pore level. It, however, makes the study of the transport phenomenon in an entire porous physical domain feasible (Vafai and Kim, 1989; Vafai and Sozen, 1990; Ramesh and Torrance, 1990; Ramesh and Torrance, 1993). Whitaker (1986a, 1986b) introduced the procedures to derive the governing equations in porous media based on the volume-averaging method. Comprehensive reviews on this topic have been well documented in the monographs of Kaviany (1995), Vafai (2005), Nield (2006) and Vadász (2008).

2.1 LOCAL VOLUME-AVERAGING METHOD

The central idea of the volume-averaging method is to use the traditional governing equations (e.g. Navier-Stokes equation) to derive the macroscopic conservation equations. These macroscopic governing equations are based on the averages taken over a representative elementary volume (REV) which is the smallest volume possesses statistically meaningful local average properties in the porous media. Once the REV is appropriately chosen, it is hoped that adding extra volumes around it will not result in the change of the local properties as well as the governing equations. Therefore, the selection of the REV is particularly important for the volume-averaging method.

The schematic of a REV is shown in Fig. 2-1. It is considered to be filled with both the solid and fluid phases of a porous medium. Traditionally, the method of volume-averaging is assumed to be applicable for a system in which the length scales are constrained by

$$l \gg d \quad (2-1)$$

Fig. 2-1 Schematic of a REV

In reality, this inequality is satisfied by many systems of practical importance.

If ϕ (scalar, vector or tensor) is the quantity being averaged, then its volume-averaging is defined as

$$\langle \phi \rangle = \frac{1}{V} \int_V \phi dV \quad (2-2)$$

where $\langle \rangle$ stands for the volume-averaging function. For the quantity associated with the fluid, the corresponding volume average is

$$\langle \phi \rangle = \frac{1}{V} \int_{V_f} \phi dV = \varepsilon \frac{1}{V_f} \int_{V_f} \phi dV \quad (2-3)$$

where ε is the porosity of the porous media which can be expressed as

$$\varepsilon = \frac{V_f}{V} \quad (2-4)$$

V and V_f are the total and void volumes in the porous medium, respectively.

Similarly, the volume averaged quantity for the solid phase can be obtained easily.

Apart from Eqs. (2-2) and (2-3), the average of the gradient (or divergence) has to be replaced with the gradient (or divergence) of the average. Slattery (1969) developed this transformation which is presented by Whitaker (1969) and Gray *et al.* (1993) as

$$\langle \nabla \phi \rangle = \nabla \langle \phi \rangle + \frac{1}{V} \int_{A_{sf}} \phi dA \quad (2-5)$$

For the divergence of the volume averaging vector or tensor $\mathbf{b}$, the theorem is

$$\langle \nabla \cdot \mathbf{b} \rangle = \nabla \cdot \langle \mathbf{b} \rangle + \frac{1}{V} \int_{A_{sf}} \mathbf{b} dA \quad (2-6)$$

where A_{sf} is the interfacial area between solid and fluid phases in the porous medium and $d\mathbf{A}$ is the surface vector. Based on Eqs. (2-2) ~ (2-6), the macroscopic continuity, momentum and energy equations in porous media can be deduced.

Generally, the conservation equations in the porous media can be derived starting from the Navier-Stokes equations based on the local volume-averaging method as introduced previously. However, in the averaging process, the detailed transport information at the pore level especially at the pore to pore fluid dynamic interactions are lost. At the same time, a large number of unknowns are introduced. These unknowns need to be obtained from semi-empirical equations obtained from experiments. Therefore, integration of the conservation equations over the REV associated with the empiricism to various extents has to be applied to arrive at the local volume-averaging conservation equations in porous media.

2.2 MATHEMATICAL FORMULATION IN POROUS MEDIA

2.2.1 Conservation Equations for Single-Phase Flow and Heat Transfer

2.2.1.1 The Continuity Equation

The microscopic continuity equation for incompressible flow is given by

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \mathbf{u}_f = 0 \quad (2-7)$$

where $\mathbf{u}_f$ is the velocity vector inside the porous media. Integrating Eq. (2-7) with respect to a REV in a porous medium results in the macroscopic continuity equation given by

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \mathbf{u} = 0 \quad (2-8)$$

where $\mathbf{u} = \varepsilon \mathbf{u}_f$ is the Darcy velocity vector. It is also called the superficial velocity which can be measured as the fluid emerges out of the porous medium.

2.2.1.2 The Momentum Equations

The microscopic momentum equation for an incompressible flow in a porous medium is given by the Navier-Stokes equation

$$\rho_f \frac{\partial \mathbf{u}_f}{\partial t} + \rho_f \nabla \cdot (\mathbf{u}_f \mathbf{u}_f) = -\nabla p_f + \mu \nabla^2 \mathbf{u}_f \quad (2-9)$$

where ρ_f , μ and p_f are the density, dynamic viscosity and pressure of the fluid, respectively. Integrating the above equation with respect to a REV and applying Eqs. (2-5) and (2-6) yield

$$\frac{\rho_f}{\varepsilon} \frac{\partial \mathbf{u}}{\partial t} + \frac{\rho_f}{\varepsilon^2} \nabla \cdot (\mathbf{u} \mathbf{u}) = -\nabla p + \frac{\mu}{\varepsilon} \nabla^2 \mathbf{u} + \mathbf{B} \quad (2-10)$$

where $p = \varepsilon p_f$ and $\mathbf{B}$ is the total drag force per unit volume due to the presence of the solid phase. It is expressed as (Hsu and Cheng, 1990)

$$\mathbf{B} = -\frac{1}{V} \int_{A_{fs}} p dA + \frac{\mu}{V} \int_{A_{fs}} \nabla \mathbf{u} \cdot dA \quad (2-11)$$

In the porous medium, the existence of the solid material impedes the motion of the fluid and results in flow resistance. Theoretical provision of $\mathbf{B}$ is impossible as it is determined by the geometries of the porous media. A great deal of effort has been spent on this endeavour as seen from the empirical studies carried out by previous investigators (Ergun, 1952; Joseph *et al.*, 1984; Vafai and Tien, 1981 and 1982; Hsu and Cheng, 1990).

Generally, the pressure gradient is also interpreted as a measure of the flow resistance in a porous medium. Therefore, most studies have used the pressure gradient to represent the empirical equations of $\mathbf{B}$. One of the earliest works is by Darcy (1856). He found a proportional relationship between the velocity and the applied pressure difference based on his experimental results. This is given by

$$\nabla p = -\frac{\mu}{K} \mathbf{u} \quad (2-12)$$

where μ is the dynamic viscosity of the fluid and K is the permeability of the porous medium. K is considered to be independent of the nature of the fluid but dependent on the geometry of the porous medium. The calculations of K for different porous media are well documented in the monograph by Dullien (1992).

Darcy's equation is only applicable when the pore or particle size based Reynolds number is sufficiently small. For Reynolds number larger than 1, a breakdown in linearity between the pressure gradient and velocity appears. Ergun (1952) proposed another equation which is used to calculate the pressure drop in porous media at high Reynolds number. His correlation for unidirectional flow is

$$\frac{dp}{dx} = -\frac{150\mu(1-\varepsilon)^2}{d_p^2\varepsilon^3}u - \frac{1.75\rho_f(1-\varepsilon)}{d_p\varepsilon^3}u^2 \quad (2-13)$$

Joseph *et al.* (1984) modified Darcy's equation to

$$\nabla p = -\frac{\mu}{K} \mathbf{u} - \frac{C_E \rho_f}{\sqrt{K}} |\mathbf{u}| \mathbf{u} \quad (2-14)$$

where C_E is the inertial coefficient which varies with the nature of the porous medium. Equation (2-14) is known as the Forchheimer equation. The second term on the right hand side of Eq. (2-14) is called the Forchheimer term.

Brinkman's equation is another well-known equation which is expressed as

$$\nabla p = -\frac{\mu}{K} \mathbf{u} + \tilde{\mu} \nabla^2 \mathbf{u} \quad (2-15)$$

where $\tilde{\mu}$ is the effective viscosity. Based on the averaging process, it is found that $\tilde{\mu}/\mu = 1/\varepsilon T^*$ where T^* represents the tortuosity of the porous media (Bear and Bachmat, 1990). Many investigators have tried to explore the relationship between $\tilde{\mu}$ and μ (Martys *et al.*, 1994; Ochoa-Tapia and Whitaker, 1995; Liu and Masliyah, 2005). It is now accepted that $\tilde{\mu}$ is equal to μ for high porosity porous media.

For high porosity porous media, most researchers recommended the Forchheimer equation for the form of the drag force $\mathbf{B}$. Vafai and Tien (1981, 1982) and Hsu and Cheng (1990) proposed the following equation which is popularly used as the momentum equation in a porous medium:-

$$\rho_f \frac{\partial \mathbf{u}}{\partial t} + \rho_f \nabla \cdot \left(\frac{\mathbf{u} \cdot \mathbf{u}}{\varepsilon} \right) = -\varepsilon \nabla p + \mu_f \nabla^2 \mathbf{u} - \left(\frac{\mu_f \varepsilon \mathbf{u}}{K} + \rho_f \frac{C_E \varepsilon \mathbf{u} |\mathbf{u}|}{\sqrt{K}} \right) \quad (2-16)$$

The second term on the right hand side of Eq. (2-16) stands for the Brinkman viscous term. It represents the bulk viscous shear stress diffusion. The third term is the microscopic viscous shear stress while the fourth term is the microscopic inertial force. It is also referred to as the Ergun inertial term.

2.2.1.3 The Energy Equations

For the energy equation in the porous region, two different models can be employed. They are the local thermal equilibrium (LTE) and the local thermal non-equilibrium (LTNE) models. The LTE model assumes that the solid temperature is equal to the liquid temperature at any location. It neglects the heat transfer between the solid and fluid phases. With this assumption and taking the averages over a REV, the energy equation for both the solid and fluid phases is

$$\varepsilon(\rho C_p)_m \frac{\partial T}{\partial t} + (\rho C_p)_m \nabla \cdot (\mathbf{u}T) = \nabla \cdot [(k_e + k_d) \nabla T] \quad (2-17)$$

where $(\rho C_p)_m$ is the overall heat capacity per unit volume. This can be calculated by

$$(\rho C_p)_m = (1 - \varepsilon)(\rho C_p)_s + \varepsilon(\rho C_p)_f \quad (2-18)$$

where k_e is the effective thermal conductivity. It is a function of both the thermal conductivities of fluid and solid phases as well as the structure of the porous media. Many studies have been reported on the calculations of the effective thermal conductivity based on different porous media (Zehner and Schlunder, 1970; Hsu *et al.*, 1995; Calmidi and Mahajan, 1999; Tee, 1999; Yu *et al.*, 2006a).

k_d in Eq. (2-17) is the dispersion thermal conductivity. It is caused by the hydrodynamic mixing of the interstitial fluid at the pore scale. Thermal dispersion would enhance the heat transfer at the pore level. However, it cannot be easily quantified due to the complexity of the thermal dispersion mechanism, especially when the solid phase conductivity is much larger than that of fluid

phase. Hsu and Cheng (1990) concluded that the dispersion conductivity can be determined by empirical relations associated with a curve-fitting approach based on experimental data. Several studies have revealed that thermal dispersion has negligible effect in convection heat transfer in most practical cases (Vafai and Kim, 1989; Calmidi and Mahajan, 2000; Jiang and Ren, 2001). The above LTE model does not consider heat transfer between the solid and fluid phases. It would underestimate the solid and fluid temperature difference in the porous medium especially in the case when the thermal conductivity ratio between these two phases is large (Amiri and Vafai, 1994; Jiang *et al.*, 1999; Leblond and Gosselin, 2008). Hence, much attention has been paid to the LTNE model in recent years. In the LTNE model, there is an additional term which accounts for the convection heat transfer between the solid and fluid phases. In this case, the temperature of the fluid phase is generally not equal to the solid phase. Without considering the effect of thermal dispersion, the energy equations using the LTNE model for both the solid and fluid phases can be written in the following forms:-

For the solid phase,

$$(1-\varepsilon)(\rho C_p)_s \frac{\partial T_s}{\partial t} = \nabla \cdot (k_{se} \nabla T) - S_T \quad (2-19)$$

For the fluid phase,

$$\varepsilon(\rho C_p)_f \frac{\partial T_f}{\partial t} + (\rho C_p)_f \nabla \cdot (\mathbf{u} T_f) = \nabla \cdot (k_{fe} \nabla T) + S_T \quad (2-20)$$

where k_{se} and k_{fe} are the effective thermal conductivities of the solid and fluid phases, respectively. S_T is the source term accounting for the convection heat transfer at the solid and fluid interface. It is expressed as

$$S_T = h_{sf} \alpha_{sf} (T_s - T_f) \quad (2-21)$$

where α_{sf} is the interior surface area to volume ratio. It depends only on the structures of the porous media. h_{sf} is the heat transfer coefficient at the interface between the two phases. A variety of models have been proposed to calculate h_{sf} based on different porous materials. Most of these are semi-empirical equations obtained from experiments.

With this LTNE model, two boundary conditions are needed for both the fluid and solid phases. Although the prescribed temperature boundary condition is generally specified for both the phases under constant wall temperature, there is no general consensus on how to allocate the quantity of heat transferred into each phase under the condition of constant heat flux. Much work has been done on the treatment of the constant heat flux boundary condition (Amiri *et al.*, 1995; Hwang *et al.*, 1995; Kuznetsov, 1996; Alazmi and Vafai, 2002; Jiang and Ren, 2001; Jiang and Lu, 2007). They can be categorised into two groups. The first group assumes equal temperature for the two phases at the heater surface while the second group distributes the heat flux into the solid and fluid phases, respectively.

Calmidi and Mahajan (2000) used the same temperature for the solid and fluid phases at the heating surface of metal foams with good agreement with experimental data. Straatman *et al.* (2007a) adopted the same boundary conditions as Calmidi and Mahajan (2000) to study heat transfer in the graphite foams. Their study yielded benchmark results for further investigations on high porosity open-cell porous media.

2.2.2 Conservation Equations for Two-Phase Flow and Heat Transfer

Two-phase flow and heat transfer in porous media is widely encountered in engineering industries such as the oil recovery process and groundwater flows (Boberg, 1988; Faghri, 1995; Chapuis *et al.*, 2008). It involves three phases in a system. These three phases could be any composition among solid, fluid and gas. The current work only focused on the combination of solid-liquid-gas as it is directly related to a phase change system. Compared with single-phase flow, two-phase flow in porous media has one significant difference which is the wetting of the solid phase by one of the fluid phases. Generally, the liquid phase is assumed to be the wetting phase and the gas phase is the non-wetting phase. The ratio of the volume occupied by the wetting phase to the total void volume is called the liquid saturation which is expressed mathematically as

$$s = \frac{\varepsilon_l}{\varepsilon} \quad (2-22)$$

where ε_l is the fraction of the volume occupied by the liquid. With this, the constraint on the volume fraction will be given by

$$\varepsilon_s + \varepsilon_l + \varepsilon_v = 1 \quad (2-23)$$

Extensive research efforts have been made by many investigators on two-phase flow and heat transfer in porous media. However, limited by the lack of empirical information provided by the experimental data, the extension of Darcy's law (Richards, 1931; Muskat and Meres, 1936) is still used for the characterisation of fluid flow in porous media. In the following section, the volume-averaging conservation equations for two-phase flow in porous media will be discussed.

To obtain the averaged forms of the governing equations for the three-phase system, the spatial averaging theorems of Eqs. (2-5) and (2-6) become

$$\langle \nabla \phi \rangle = \nabla \langle \phi \rangle + \frac{1}{V} \int_{A_{sl}} \phi d\mathbf{A} + \frac{1}{V} \int_{A_{lv}} \phi d\mathbf{A} \quad (2-24)$$

$$\langle \nabla \cdot \mathbf{b} \rangle = \nabla \cdot \langle \mathbf{b} \rangle + \frac{1}{V} \int_{A_{sl}} \mathbf{b} d\mathbf{A} + \frac{1}{V} \int_{A_{lv}} \mathbf{b} d\mathbf{A} \quad (2-25)$$

With these two equations, the volume-averaging conservation equations for two-phase in porous media can be obtained.

2.2.2.1 Conservation Equations Based on the Separated Flow Model

Traditionally, two-phase flow and heat transfer problems in porous media are solved by the separated flow model (SFM) (Albriola and Pinder, 1985; Ramesh and Torrance, 1990). In this model, various phases are regarded as distinct fluids with individual thermodynamic and transport properties. The macroscopic mathematical governing equations are formulated for each phase. The separated phases are coupled together by interfacial transfer terms, which reflect the microscopic transport phenomena at the interface. These macroscopic conservation equations are derived, among others, by Slattery (1970), Gray (1983), Whitaker (1986b) and Bear and Bensabat (1989) using the volume-averaging method. The formulation of Whitaker (1986b) will be briefly reviewed here.

With the assumption of creeping flow for both the incompressible liquid and vapour, the microscopic continuity and momentum equations are given as

$$\nabla \cdot \mathbf{u}_l = 0 \quad (2-26)$$

$$-\nabla p_l + \rho_l \mathbf{g} + \mu_l \nabla^2 \mathbf{u}_l = 0 \quad (2-27)$$

$$\nabla \cdot \mathbf{u}_v = 0 \quad (2-28)$$

$$-\nabla p_v + \rho_v \mathbf{g} + \mu_v \nabla^2 \mathbf{u}_v = 0 \quad (2-29)$$

The boundary conditions are

$$\mathbf{u}_l = 0 \text{ at } A_{ls} \quad (2-30)$$

$$\mathbf{u}_v = 0 \text{ at } A_{vs} \quad (2-31)$$

$$\mathbf{u}_l = \mathbf{u}_v \text{ at } A_{lv} \quad (2-32)$$

$$-p_l \mathbf{n}_{lv} + \boldsymbol{\tau}_l \cdot \mathbf{n}_{lv} = -p_v \mathbf{n}_{lv} + \boldsymbol{\tau}_v \cdot \mathbf{n}_{lv} + 2\sigma H \mathbf{n}_{lv} \text{ at } A_{lv} \quad (2-33)$$

where $\boldsymbol{\tau}$ is the shear stress tensor, σ is the surface tension and H is the mean curvature of the interface.

If the local velocity on the liquid and vapour phases is $\mathbf{w}$, then

$$\frac{\partial \varepsilon_l}{\partial t} = \varepsilon \frac{\partial s}{\partial t} = \frac{1}{V} \int_{A_{lv}} \mathbf{n}_{lv} \cdot \mathbf{w} dA \quad (2-34)$$

Combining this with Eq. (2-24), the continuity equations for the liquid and vapour phases become

$$\rho_l \left(\frac{\partial \varepsilon_l}{\partial t} + \nabla \cdot \mathbf{u} \right) = \dot{m}_l \quad (2-35)$$

$$\rho_v \left(\frac{\partial \varepsilon_v}{\partial t} + \nabla \cdot \mathbf{u}_v \right) = \dot{m}_v \quad (2-36)$$

where $\dot{m}_l$ and $\dot{m}_v$ are the mass sources of liquid and vapour due to phase change, respectively. If there is no external mass source or sink, the relationship between these two parameters is given by

$$\dot{m}_l + \dot{m}_v = 0 \quad (2-37)$$

The derivations of the momentum equations for the two fluids will not be repeated here since they are well documented in the work of Whitaker (1986b). Surface tension gradient is usually small compared with the pressure gradient

term in Eqs. (2-27) and (2-29) at high permeability porous media (Whitaker, 1986; Bear and Bensabat, 1989), therefore, the effect of surface tension is neglected in the current work. The mathematical formulations of the momentum equations are

$$\mathbf{u}_l = -\frac{\mathbf{K}_l}{\mu_l}(-\nabla p_l - \rho_l \mathbf{g}) + \mathbf{K}_{lv} \cdot \mathbf{u}_v \quad (2-38)$$

$$\mathbf{u}_v = -\frac{\mathbf{K}_v}{\mu_v}(-\nabla p_v - \rho_v \mathbf{g}) + \mathbf{K}_{vl} \cdot \mathbf{u}_l \quad (2-39)$$

where $\mathbf{K}_l$ and $\mathbf{K}_v$ are the permeabilities of the liquid and vapour phases, respectively. For the one-dimensional setting, the phase permeability is related to the absolute permeability by the relative permeability according to

$$K_l = k_{rl} K \quad (2-40)$$

$$K_v = k_{rv} K \quad (2-41)$$

where K is the absolute permeability used in the single-phase flow and k_{rl} and k_{rv} are the relative permeabilities of the liquid and vapour phases, respectively. These relative permeabilities are highly dependent on the structure of the porous medium, the liquid saturation and the surface tension. Numerous experimental studies have been carried out to determine k_{rl} and k_{rv} (Muskat and Meres, 1936; Wyllie, 1962; Scheidegger, 1974; Mualem, 1976; Levec *et al.*, 1986). Muskat and Meres (1936) simplified these two relative permeabilities to be independent of other parameters except for the local saturation s .

In Eqs. (2-38) and (2-39), $\mathbf{K}_{lv}$ and $\mathbf{K}_{vl}$ are the viscous drag tensors for the liquid and vapour phase equations, respectively. These two tensors represent the influence of the viscous drag which exists between the liquid and vapour phases.

In most cases, $\mu_l \gg \mu_v$ and hence, the last terms in Eqs. (2-38) and (2-39) are negligible and can be discarded.

Based on order-of-magnitude comparisons, the viscous stress terms in Eq. (2-33) have been dropped. For closure of the momentum equations, the terms $p_l - p_v$ and $2\sigma H$ in Eq. (2-33) in the boundary conditions have to be solved.

The interfacial normal force balance is

$$p_c = -p_l + p_v = 2\sigma H_{lv} \quad (2-42)$$

where p_c is the capillary pressure. H_{lv} is the curvature at the interface between liquid and vapour and it is given by

$$H_{lv} = \frac{1}{A_{lv}} \int_{A_{lv}} H dA \quad (2-43)$$

With the assumption that the mean curvature H is constant within the averaging volume, H_{lv} is equal to H . The capillary pressure p_c is then determined by the mean curvature H which depends on the phase distribution. Many studies have been attempted to explore the mean curvature H as well as the capillary pressure p_c (Leverett, 1941; Van Genuchten, 1980; Udell, 1985; Pavone, 1989). One of the popularly used equations is the Leverett reduced function (Leverett, 1941). It is expressed as

$$\frac{p_c (K/\epsilon)^{0.5}}{\sigma} = J(s) \quad (2-44)$$

where $J(s)$ is called the Leverett J -function. It can only be obtained from experimental data. The Leverett J -function is not universal and depends on the wettability and surface tension. Wettability is the extent to which the wetting phase spreads over the solid surface. In most liquid-gas systems, the liquid phase is usually regarded as the wetting phase while the gas is the non-wetting

phase. With the Leverett J -function solved, the momentum equation can then be closed.

The two-phase energy equation based on the local volume-averaging method was developed by Whitaker (1977) and Cheng (1978). The LTE model is usually adopted due to the lack of knowledge of the coupled phase change and heat transfer mechanisms among the different phases. Although the validity of LTE model needs to be explored further, it is valid for most phase change conditions (Whitaker, 1977). With the assumption of LTE among the three phases, i.e. $T = T_s = T_l = T_v$, the energy equation becomes

$$\begin{aligned} \frac{\partial}{\partial t} \left((1-\varepsilon) \rho_s h_s + \varepsilon s \rho_l h_l + \varepsilon (1-s) \rho_v h_v \right) \\ + \nabla \cdot (\rho_l \mathbf{u}_l h_l + \rho_v \mathbf{u}_v h_v) = \nabla \cdot (k_e \nabla T) + \dot{Q} \end{aligned} \quad (2-45)$$

where k_e is the effective thermal conductivity with consideration of the three phases in the porous medium. Different models have been proposed to calculate k_e for various porous media. These information are given in Section 2.3. $\dot{Q}$ is the volumetric heat source. s is the liquid saturation which has been introduced in Eq. (2-22). The enthalpies in Eq. (2-45) are calculated based on the temperature as

$$h_s = C_{ps} T + h_s^0 \quad (2-46)$$

$$h_l = C_{pl} T \quad (2-47)$$

$$h_v = C_{pv} T + (C_{pl} - C_{pv}) T_{sat} + h_{fg} \quad (2-48)$$

where T_{sat} is the saturation temperature and h_{fg} is the latent heat. h_s^0 is taken at 0°C.

The governing conservation equations given by the SFM are rather straightforward. However, it leads to a large number of differential equations which need to be solved. This is inconvenient for direct use in numerical simulation especially for three-dimensional simulation. In order to reduce the number of equations, modified formulations are needed.

2.2.2.2 Conservation Equations Based on the Two-Phase Mixture Model

Given the excessive computations caused by the SFM, an alternative approach, the two-phase mixture model (Wang and Beckermann, 1993a) is popularly used in the study of two-phase flow in porous media. The governing equations in the two-phase mixture model are rigorously derived from the SFM without any approximation involved. In this model, the two-phase system is treated as a binary mixture. All the thermophysical properties of the multiphase mixture are functions of the properties of its constituents. These mixture properties are obtained from the derivation process based on the SFM. The governing equations are identically reduced to the single-phase transport equations. However, the formulations do not lose the intrinsic characteristics of the individual phases. Compared with the SFM, the two-phase mixture model is easier to implement numerically. Hence, it is popularly used in the study of two-phase flow in porous media.

The governing equations by adopting the two-phase mixture model (Wang, 1997) are

$$\varepsilon \frac{\partial p}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (2-49)$$

$$\mathbf{u} = -\frac{K}{\mu} [\nabla p - (\rho_k - \rho_o) \mathbf{g}] \quad (2-50)$$

$$\Omega \frac{\partial h}{\partial t} + \nabla \cdot (\gamma_h \mathbf{u} h) = \nabla \cdot (\Gamma_h \nabla h) + \nabla \cdot \left[f(s) \frac{K \Delta \rho h_{fg}}{\nu_v} \mathbf{g} \right] \quad (2-51)$$

ρ_k is the kinetic mixture density which is dependent on the liquid saturation. ρ_o is the liquid density. Ω is the heat capacitance ratio. γ_h is the advection correction coefficient and Γ_h is the diffusion coefficient. $f(s)$ is the hindrance function which is used to describe the flow resistance for liquid due to the existence of vapour. $\Delta \rho$ is the density difference between the liquid and vapour. The mathematical expressions of these mixture variables and the properties in Eqs. (2-49) - (2-51) are listed in Table 2-1. The temperature T and the liquid saturation s can be calculated from the enthalpy as

$$T = \begin{cases} \frac{h + 2\rho_l h_{vsat}}{\rho_l C_{pl}} & h \leq -\rho_l (2h_{vsat} - h_{lsat}) \\ T_{sat} & -\rho_l (2h_{vsat} - h_{lsat}) < h \leq -\rho_v h_{vsat} \\ T_{sat} + \frac{h + \rho_v h_{vsat}}{\rho_v C_{pv}} & -\rho_v h_{vsat} < h \end{cases} \quad (2-52)$$

$$s = \begin{cases} 1 & h \leq -\rho_l (2h_{vsat} - h_{lsat}) \\ -\frac{h + \rho_v h_{vsat}}{\rho_l h_{fg} + (\rho_l - \rho_v) h_{vsat}} & -\rho_l (2h_{vsat} - h_{lsat}) < h \leq -\rho_v h_{vsat} \\ 0 & -\rho_v h_{vsat} < h \end{cases} \quad (2-53)$$

The individual velocities of the liquid and vapour can be recovered from

$$\rho_l \mathbf{u}_l = \lambda_l \rho \mathbf{u} + \mathbf{j} \quad (2-54)$$

$$\rho_v \mathbf{u}_v = \lambda_v \rho \mathbf{u} - \mathbf{j} \quad (2-55)$$

where $\mathbf{j}$ is the total mass flux which is expressed as

$$\mathbf{j} = -\rho_l D(s) \nabla s + f(s) \frac{K \Delta \rho}{\nu_v} \mathbf{g} \quad (2-56)$$

Table 2-1 Variables in the two-phase mixture model

Variables	Expressions
density	$\rho = \rho_l s + \rho_v (1 - s)$
velocity	$\rho \mathbf{u} = \rho_l \mathbf{u}_l + \rho_v \mathbf{u}_v$
enthalpy	$\rho h = \rho_l s h_l + \rho_v (1 - s) h_v$
kinetic density	$\rho_k = \rho_l [1 - \beta_l (T - T_o)] \lambda_l(s) + \rho_v [1 - \beta_v (T - T_{sat})] \lambda_v(s)$
viscosity	$\mu = \frac{\rho_l s + \rho_v (1 - s)}{k_{rl}/v_l + k_{rv}/v_v}$
advection correction coefficient	$\gamma_h = \frac{[(\rho_v/\rho_l)(1 - s) + s][h_{vsat}(1 + \lambda_l) - h_{lsat}\lambda_l]}{(2h_{vsat} - h_{lsat})s + (\rho_v h_{vsat}/\rho_l)(1 - s)}$
effective heat capacitance ratio	$\Omega = \varepsilon + \rho_s C_{ps} (1 - \varepsilon) \frac{dT}{dH}$
effective diffusion coefficient	$\Gamma_h = \frac{1}{1 + (1 - \rho_v/\rho_l) h_{vsat}/h_{fg}} D + k_{eff} \frac{dT}{dH}$
capillary diffusion coefficient	$D_s = \frac{(\varepsilon K)^{0.5} \sigma}{\mu_l} \frac{k_{rl} k_{rv}}{(v_v/v_l) k_{rl} + k_{rv}} [-J'(s)]$
relative motilities	$\lambda_l(s) = \frac{k_{rl}/v_l}{k_{rl}/v_l + k_{rv}/v_v}, \quad \lambda_v(s) = \frac{k_{rv}/v_v}{k_{rl}/v_l + k_{rv}/v_v}$
hindrance function	$f(s) = \frac{k_{rv} k_{rl}/v_l}{k_{rl}/v_l + k_{rv}/v_v}$

The two-phase mixture model is valid throughout the entire domain including the sub-cooled, two-phase mixture and superheated regions. The governing equations in the two-phase mixture model are almost half of those in the SFM. Compared with the SFM, it is more suitable for numerical simulation as it reduces the computational burden significantly. The model also provides new insights for understanding the complicated two-phase flow in porous media such as the combination of single and two phases in the same domain.

Moreover, it preserves the flow characteristics for each individual phase. Therefore, it is widely used for the study of two-phase flow in porous media (Wang *et al.*, 1994; Zhao *et al.*, 1999; Waite and Amin, 1999; Najjari and Nasrallah, 2003). A detailed literature survey on the two-phase mixture model will be presented in Section 2.4.

2.3 SINGLE-PHASE FLOW AND HEAT TRANSFER IN OPEN-CELL FOAMS

During the past two decades, single-phase flow in open-cell foams has been studied extensively. Compared with traditional porous media of low porosity such as packed beds or granular substances, these open-cell foams have open-celled structures with continuous and rigid solid ligaments. Generally, they possess high porosity ($\varepsilon > 0.6$) and large solid-fluid interface. The bulk thermal conductivity is at least an order of magnitude greater than the traditional porous media. These unique properties render them to be popularly used as heat sinks in the cooling of electronic components and in heat exchangers (Kim *et al.*, 2003; Gallego and Klett, 2003; Yu *et al.*, 2006b; Williams and Roux, 2006).

Extensive research have been conducted on the comparison of the heat transfer performance and pressure drop between graphite foams and other porous media. The initial study was performed by Gallego and Klett (2003). They provided some comparative experimental data of the pressure drop and heat transfer coefficients for air and water flows through aluminium and graphite foams. Their results showed that graphite foam heat sinks have better heat transfer enhancement compared to aluminium foams as well as conventional heat sinks

at the expense of high pressure drop. The same conclusion was drawn by Straatman *et al.* (2007b). To obtain a proper design for engineering applications which has high heat transfer coefficient but low pressure drop, Gallego and Klett (2003) evaluated different configurations of graphite foams, including finned, pin-finned, blind holes in both perpendicular and parallel directions with respect to the fluid flow. They showed that the pressure drop is significantly reduced by modifying the configurations of graphite foams. However, the heat transfer performance under any graphite foam configurations is still better than those obtained from aluminium foams. Garman and Elwell (2002) obtained the heat transfer performance for three different configurations of graphite foam, viz. solid block foams of low density, high density with vertical blind holes and high density with horizontal blind holes. Their results showed that the high density graphite foam with horizontal blind holes had the best heat transfer performance with considerably low pressure drop. Straatman *et al.* (2007a) used water as the coolant to compare the heat transfer performance and pressure drop between graphite foam with other carbon foams. Their results indicated that graphite foams gave the best heat transfer performance.

For evaluation of graphite foams as heat sinks for electronic systems, Williams and Roux (2006, 2007) conducted experiments for cooling an array of power amplifier modules using both air and water. Three different materials of heat sinks were used as test samples. These included graphite foam for different configurations, copper fins and polymer fibre made of nickel. The configurations of graphite foam include zigzag, corrugated and inline shapes. Williams and Roux (2006, 2007) reported on the heat transfer performance

based on the surface temperature of the modules for all the test samples. They concluded that graphite foam had strong potential to be used in the cooling of electronic components. In another application, Yu *et al.* (2006b) implemented graphite foams as heat sinks in an air-water heat exchanger. They also proposed an engineering model involving several empirical equations to predict the pressure drop and heat transfer performance. Reasonable agreement was achieved between the experimental data and the predicted results based on their engineering model.

The above experimental results indicate a much brighter future for the use of graphite foams as heat sinks in different engineering applications compared with metal foam. Therefore, the current work will focus on the characteristics of graphite foam for single-phase. However, the large pressure drop when fluid flows through graphite foam will always be a major concern. Under this circumstance, graphite foams of different configurations become more attractive compared with a solid block graphite foam. Numerical simulation plays an important role in the design of the configurations. It can provide preliminary results before the configurations are used in actual applications. For numerical simulation, the thermophysical properties of graphite foam and related empirical equations have to be specified. These parameters include porosity ε , permeability K , inertial coefficient C_E , effective thermal conductivity k_{se} and k_{fe} , heat transfer coefficient h_{sf} and interior surface area α_{sf} . Among them, the first three parameters were obtained from the experimental studies in the current work. A unit cubic model to determine the effective thermal conductivity of graphite foams is also developed in this study.

Many researchers have attempted to determine the effective thermal conductivity of graphite foams. Tee *et al.* (1999) was the first to propose a cubic cell model to estimate the effective thermal conductivity of a graphite foam. They used the electrical analogy to obtain the thermal resistance of the proposed model. By integrating the probability density function of the cell along different directions, the overall thermal conductivity was obtained. The detailed derivation can be found in another work of Tee *et al.* (2008). Klett *et al.* (2004) proposed a semi-empirical model by accounting for the tortuosity of the heat flow paths in their equation for graphite foam thermal conductivity. Although their model gave good predictions of the effective thermal conductivity against the available data, a fitting parameter in the equation had to be calibrated against experimental data. Alam *et al.* (2004) developed a finite element model to predict the thermal conductivity of graphite foam with an anisotropic strut node structure. Their results showed that the ligament thermal conductivity affected the effective thermal conductivity significantly. However, the results obtained from their model were only valid for limited ranges of porosities. The experimental study of Alam and Maruyama (2004) demonstrated that graphite foams have a wide range of effective thermal conductivities due to differences in the manufacturing process and subsequent heat treatment. Yu *et al.* (2006a) proposed a thermal and hydrodynamic model of graphite foam based on a unit cubic cell. The effective thermal conductivity predicted by this model agreed well with the experimental data of Klett *et al.* (2004).

For the closure of the energy equations, one would still need an empirical relationship to account for the heat transfer between the solid ligament and the

fluid in the graphite foam. Zukauskas (1987) proposed a correlation for cylinders in cross-flow. Klett *et al.* (2002) obtained the coefficients based on their experiments for graphite foam. This equation and its associated coefficients will be given in Chapter 5.

With all the empirical parameters involved in the governing equations solved, the numerical simulation of flow and heat transfer in graphite foam can be realised. However, the available numerical work on single-phase flow in porous graphite foam is limited. Straatman *et al.* (2007a) adopted the numerical model proposed by Betchen *et al.* (2006) to simulate the flow and heat transfer when water is forced through block graphite foams. To the best knowledge of the author, there is no existing numerical study for different configurations of graphite foams.

In the graphite foams with different configurations, the channel would usually be inserted with porous graphite foam layers (Fig. 2-2) or discrete porous blocks graphite foams (Fig. 2-3). Many experimental and numerical studies have been performed to investigate the transport phenomena in such configurations (Poulikakos and Kazmierczak, 1987; Huang and Vafai, 1994; Sung *et al.*, 1995; Chikh *et al.*, 1995; Fu *et al.*, 1996; Zhang and Zhao, 2000; Ko and Anand, 2003; Miranda and Anand, 2004; Targui and Kahalerras, 2008). These studies were limited only to a 2D setting. There is no study in a 3D domain which is more realistic for comparison with the experimental results. In particular, there is no reported numerical study of forced convection in different configurations of graphite foams.

Fig. 2-2 Schematic diagram of the channel with porous and open layers

Fig. 2-3 Schematic diagram of the channel with staggered porous blocks

For simulation of the transport phenomena in the above channels which contain both porous and open regions, two different sets of the governing equations are needed. The traditional Navier-Stokes momentum equations can be used directly in the open region. For the porous region, the macroscopic momentum and energy equations derived based on the volume-averaging method is usually adopted. In order to achieve continuity of the velocities and temperatures at the interface between the two regions, special efforts are needed. The current work proposed one set of conservation equations which can be used for both the open and porous regions. With this formulation, the efforts to deal with the interface between the open and porous regions are eliminated. This set of the conservation equations will be given in Chapter 3.

2.4 TWO-PHASE FLOW AND HEAT TRANSFER IN POROUS MEDIA

Extensive experimental studies have been carried out on two-phase heat transfer in porous media (Sondergeld and Turcotte, 1978; Bau and Torrance, 1982a; Bau

and Torrance, 1982b; Bau and Torrance, 1982c; Chen *et al.*, 2000). It is commonly accepted that the phase structure in a fluid-saturated porous medium heated from below and cooled from above, is layered after the onset of boiling. A superheated vapour zone lies above a two-phase zone and a sub-cooled liquid zone. In the two-phase zone, both the liquid and vapour phases coexist (Sondergeld and Turcotte, 1977). The visualisation experiments of Sondergeld and Turcotte (1978) reveal that within the liquid zone, heat transfer can occur by both conduction and convection, whereas, in the two-phase zone, heat transfer occurs mainly due to the counter-percolation between the liquid and vapour. Bau and Torrance (1982a; 1982b; 1982c) performed a series of experiments to investigate natural convection with phase change in low-permeability porous media. Their results showed that the two-phase zone is essentially isothermal at the saturation temperature.

Phase change in metal foam and graphite foams has been studied extensively during the last decade. For pool boiling systems, Athreya and Mahajan (2002) investigated the boiling process of FC-72 in aluminium foams. They studied the effects of the foam orientation and structures on the nucleate boiling as well as CHF. Their results show that the foam with high PPI (pores per linear inch) in the vertical orientation had much potential in the phase change cooling system. Leong *et al.* (2011) investigated the phase change in a thermosyphon system with porous graphite foam insert. HFE-7000 was used as the coolant. It was established that their thermosyphon was able to remove heat fluxes of up to 112 W/cm² with the maximum heater temperature below 85°C. Jin *et al.* (2011) investigated the heat transfer performance of different configurations of

graphite foams in a pool boiling system. The effects of foam configuration and coolant filling volume on the pool boiling heat transfer were analysed. Their results showed that the coolant filling volume had negligible effect on the heat transfer performance while the foam configuration has significant effect on the heat transfer performance. Unlike pool boiling heat transfer, experimental work reported on the flow boiling in metal and graphite foams is rather scarce. Given the high pressure drop caused by graphite foams due to its low permeability, the current study on two-phase flow will choose metal foam as heat sink instead of graphite foam. Lu and Zhao (2009) studied flow boiling in metal foams. Their results show that metal foam with high pore density could give high heat transfer performance. Zhao *et al.* (2009) carried out experimental study for boiling heat transfer in a horizontal tube with copper foams. The results show that the heat transfer performance by using copper foam is three times higher than that of the plain tubes.

Ramesh and Torrance (1990) performed the pioneer work on the numerical simulation of boiling heat transfer in porous media. They proposed a fully implicit numerical scheme to solve the coupled momentum and energy equations based on the SFM by using the Finite Volume Method (FVM). Given the mobile interface between the sub-cooled and two-phase zones during the phase change process, coordinate transformation has to be performed before discretising the governing equations. This approach was successfully employed by Ramesh and Torrance (1990) in their investigation of boiling and natural convection in a horizontal porous channel. The coordinate transformation involved in the numerical procedure introduces nonlinearity into the formulations, resulting in pseudo-convection and additional diffusion-type

terms in both the momentum and energy equations (Hsu *et al.*, 1981). Special treatments on the two terms are required in the numerical implementation to pre-empt solution inaccuracy and termination of the simulation. Such difficulties were encountered by Chung and Catton (1991) when they tried to simulate phase change in a porous bed. They found that convergence could only be achieved for a very narrow range of iteration parameters and grid distribution.

Unlike the SFM, the two-phase mixture model gives a unified formulation for the entire domain including both the single and two-phase zones. This avoids any special treatment for tracking the interface between different zones, such as implementing moving numerical grids or performing coordinate transformation. It facilitates numerical simulations better when compared with the SFM. The two-phase mixture model was successfully adopted by many researchers who studied two-phase flow in porous media. A detailed literature survey is presented in the following sections.

Within the framework of the two-phase mixture formulation developed by Wang and Beckermann (1993a), the same authors later obtained a semi-analytical solution on the two-phase boundary layer for flow boiling in a vertical plate (Wang and Beckermann, 1993b). The capability of the numerical code developed by Wang and Beckermann (1993a) was showcased via an example of boiling and natural convection in capillary porous media by Wang *et al.* (1994) who compared their numerical results with the experimental work done by (Sondergeld and Turcotte, 1977) for four flow patterns under different permeabilities of porous media. Good agreement was achieved. Easterday *et al.* (1995) successfully employed their numerical code to predict the temperature

distributions and flow patterns in a horizontal porous channel. Visualisation of the phase distributions in their experiments showed general agreement with their numerical predictions. Qualitative identification on the local temperature distribution was achieved between experimental measurements and the numerical results.

One disadvantage of the previous two-phase mixture model (Wang and Beckermann, 1993a) is that it cannot be applied to the superheated vapour zone where $s = 0$ due to the infinite value of the advection correction coefficient. Given this limitation, Wang (1997) developed another set of numerical formulation. This formulation was aimed at solving all the zones involved during the phase change process in porous media. With this formulation, Wang (1997) proposed a primitive-variable approach based on the SIMPLE algorithm of Patankar (1980) to solve the coupling of the pressure and velocity. Such a new formulation and numerical scheme was successfully implemented to predict the post dry-out problem in porous media. The predictions compared well with the available experimental data. This new formulation was popularly used for flow boiling in porous media. For the sake of the brevity, the term, two-phase mixture model, in subsequent sections refers to this new set of formulation unless specified otherwise.

Waite and Amin (1999) adopted the two-phase mixture model to study boiling and natural convection in a square porous enclosure with a heated side. The transient behaviour of the convective flow was explored in detail. Their results showed that an increase of the heat flux or decrease of the permeability can accelerate the fluid motion as well as the conversion speed from liquid to vapour. Najjari and Nasrallah (2008, 2009) adopted the two-phase mixture model to

investigate the effect of the porous layer in a conjugated domain. Their results showed that the porous layer can either cool the hot fluid or pre-heat the cold fluid in the open layer. A modified two-phase mixture model was developed by Yuki *et al.* (2008) to simulate high velocity flow boiling under high heat flux in a porous medium. These modifications include adding the inertial force in the momentum equation and adopting the LTNE model in the energy equation. One of their major findings is that there was no significant temperature difference between the LTE and LTNE models unless the heat flux is above 10^6 W/m^2 . Although the authors argued that more realistic and reasonable results could be obtained for the two-phase flow under high heat flux and high velocity by using their models, the involvement of a large number of empirical equations in the unknowns needs to be further verified by experiments.

Another application of the two-phase mixture model is to study the mixed boiling convection in a porous channel. In mixed boiling convection, both free and forced convection associated with phase change occur. This is also widely encountered in engineering applications such as in nuclear reactors (Schäfer and Lohnert, 2006) and PEM fuel cells (You and Liu, 2002; Afshari and Jazayeri, 2009; Karimi *et al.*, 2009). In these applications, either upward flow (aiding flow) or downward flow (opposing flow) may be encountered. Therefore, a better understanding of the transport mechanisms for aiding and opposing flows in such systems is important. Special attention should be paid to the phase change process in vertical channels where the buoyancy-induced flow could become important.

Zhao and Liao (1999) used the two-phase mixture model to study the boiling heat transfer in a vertical packed bed with one side heated. Both aiding and opposing

flows were investigated. Their results showed that the direction of the incoming fluid had great influence on both the flow field and the temperature distributions. Besides single-sided heating in vertical porous channels, Zhao *et al.* (2000) also successfully implemented the two-phase mixture model into the problem of the buoyancy-induced flow in a porous vertical channel with symmetric heating. They discovered that the inlet mass flux was not a monotonic function of the imposed heat flux. This finding is consistent with their previous experimental results (Zhao *et al.*, 1998). They also found that the larger particles of the porous packed beds could give a higher dry-out heat flux. Najjari and Nasrallah (2003) adopted the two-phase mixture model to investigate the flow boiling in an inclined porous channel. Apart from the effects of permeability and inlet velocity, different inclinations of the porous channel were considered in their work. Their results showed that a critical value of the inclination angle exists which corresponded to a maximum value of the evaporated volume at low inlet velocity and high permeability. In another work, the same authors (Najjari and Nasrallah, 2002) investigated transient flow boiling in a porous channel under the effect of gravity for an opposing flow configuration. The effects of parameters, such as permeability, inlet velocity and heat flux on the flow fields were studied. Their results indicated that the vapour volume decreases with both the increase of permeability and inlet velocity. Most of their discussions were focused on the two-phase steady-state solutions. Evolutions from single-phase to two-phase fluid flow and heat transfer were only discussed briefly.

From the above literature review, it can be seen that numerical simulations were performed only for two-dimensional (2D) configurations. Generally, simulations in a 2D setting cannot capture the three-dimensional (3D) effects which may be

important under certain circumstances. Furthermore, large discrepancies between numerical predictions and experimental data may arise from 2D simulations. Although the assumption of a 2D setting in the numerical simulation is applicable in some very specific cases, a 3D numerical simulation is more desirable since it is more realistic for comparison with actual experimental data. It is especially important for understanding and predicting the development of complex flow structures and the two-phase zone for cases which cannot be simplified to a 2D setting. However, to the best of the knowledge of the author, there is no 3D numerical study of phase change heat transfer in porous media based on the two-phase mixture model. Meanwhile, limited information can be found on the transient boiling process in the porous media. Although steady-state investigations can provide substantial information for phase change problems, transient studies would be more useful since it can reveal the evolution of the phase change process and the interactions between the different phases. The current study will focus on a 3D study of two-phase flow and heat transfer in porous media based on the two-phase mixture model. The transient boiling process in porous media will also be studied.

2.5 SUMMARY

A brief review of single and two-phase heat transfer in porous media is presented in this chapter. The local volume-averaging method is introduced. Mathematical formulations for both single and two-phase flows in porous media are discussed. A thorough literature review on both experimental and numerical study of graphite foam and metal foam is conducted.

Driven by the premium heat transfer performance of graphite foams reported in the literature, the current work will focus on the study of its characteristics for single-phase flow. The single-phase flow and heat transfer in different configurations of graphite foams will be investigated experimentally and numerically. The Forchheimer-Brinkman and Navier-Stokes equations introduced in this chapter will be used to deal with the fluid flow in porous and open domain, respectively. Because of the high pressure drop caused by the graphite foams, aluminium foam is chosen to be used in heat sinks for two-phase flow instead of graphite foams. The flow boiling characteristics in aluminium foams will be experimentally studied. The two-phase mixture model presented in this chapter will be adopted to simulate the phase change process in porous media.

CHAPTER 3 NUMERICAL SCHEME FOR SINGLE-PHASE FLOW AND HEAT TRANSFER IN POROUS MEDIA

This chapter first presents the governing equations for single-phase flow and heat transfer in a channel with both porous and open regions. The governing equations for a 3D setting are presented. This is followed by the discretisation of these equations and the associated numerical solution procedures. Finally, some concluding remarks are given.

3.1 GOVERNING EQUATIONS IN THREE-DIMENSIONS

With reference to the configurations proposed in Chapter 2, the computational domain contains both the porous and open regions. For the open region, the traditional Navier-Stokes momentum equations can be used directly. For the porous region, the governing equations derived based on the volume-averaging method are used.

With the assumptions of laminar flow, constant fluid and solid properties, negligible gravitational force in both the porous and open regions, the steady-state mass and momentum conservation equations for fluid flow can be cast into the general forms as follows:-

Continuity Equation

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (3-1)$$

x-momentum Equation

$$\begin{aligned} \frac{\rho_f}{\varepsilon} \frac{\partial(uu)}{\partial x} + \frac{\rho_f}{\varepsilon} \frac{\partial(uv)}{\partial y} + \frac{\rho_f}{\varepsilon} \frac{\partial(uw)}{\partial z} = -\varepsilon \frac{\partial p}{\partial x} \\ + \mu_f \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) + S_{fx} \end{aligned} \quad (3-2)$$

y-momentum Equation

$$\begin{aligned} \frac{\rho_f}{\varepsilon} \frac{\partial(uv)}{\partial x} + \frac{\rho_f}{\varepsilon} \frac{\partial(vv)}{\partial y} + \frac{\rho_f}{\varepsilon} \frac{\partial(vw)}{\partial z} = -\varepsilon \frac{\partial p}{\partial y} \\ + \mu_f \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) + S_{fy} \end{aligned} \quad (3-3)$$

z-momentum Equation

$$\begin{aligned} \frac{\rho_f}{\varepsilon} \frac{\partial(uw)}{\partial x} + \frac{\rho_f}{\varepsilon} \frac{\partial(vw)}{\partial y} + \frac{\rho_f}{\varepsilon} \frac{\partial(ww)}{\partial z} = -\varepsilon \frac{\partial p}{\partial z} \\ + \mu_f \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) + S_{fz} \end{aligned} \quad (3-4)$$

Using Eq. (2-15), the source terms S_f become

$$S_{fx} = - \left(\frac{\mu \varepsilon}{K} u + \frac{\rho_f \varepsilon C_E}{\sqrt{K}} |u| u \right) \quad (3-5)$$

$$S_{fy} = - \left(\frac{\mu \varepsilon}{K} v + \frac{\rho_f \varepsilon C_E}{\sqrt{K}} |u| v \right) \quad (3-6)$$

$$S_{fz} = - \left(\frac{\mu \varepsilon}{K} w + \frac{\rho_f \varepsilon C_E}{\sqrt{K}} |u| w \right) \quad (3-7)$$

Energy Equation for LTE model

$$(\rho C_p)_f \left(u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} \right) = k_e \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) \quad (3-8)$$

Energy Equations for LTNE model

Solid phase

$$k_{se} \frac{\partial^2 T_s}{\partial x^2} + k_{se} \frac{\partial^2 T_s}{\partial y^2} + k_{se} \frac{\partial^2 T_s}{\partial z^2} - S_T = 0 \quad (3-9)$$

Fluid phase

$$\begin{aligned} & (\rho C_p)_f \left(u \frac{\partial T_f}{\partial x} + v \frac{\partial T_f}{\partial y} + w \frac{\partial T_f}{\partial z} \right) \\ & = k_{fe} \left(\frac{\partial^2 T_f}{\partial x^2} + \frac{\partial^2 T_f}{\partial y^2} + \frac{\partial^2 T_f}{\partial z^2} \right) + S_T \end{aligned} \quad (3-10)$$

where k_{se} and k_{fe} are the effective thermal conductivities for the solid and fluid phases, respectively. S_T is the source term which accounts for the convection heat transfer at the solid and fluid interface. It is mathematically expressed as

$$S_T = h_{sf} \alpha_{sf} (T_s - T_f) \quad (3-11)$$

With this formulation, only one set of governing equations is solved in the entire calculation domain. The additional source terms S_f and S_T are non-zero only within the porous regions. For LTNE model, two different energy equations have to be solved separately and they are coupled together with the source term in Eq. (3-11). The appropriate properties such as the porosity, density, viscosity, specific heat and thermal conductivity are specified in both the porous and open regions. The porosity in the open region was set to 1. Since the open region contains only the fluid phase, the solid temperature in this region is meaningless. Thus, an extremely small value of 10^{-17} W/m·K was prescribed for the thermal conductivity of the solid phase in the open region to avoid heat conduction to the porous region. The harmonic-mean method (Patankar, 1980) was used to treat the thermophysical properties at the interface

between the porous and open regions, no other special efforts are needed to deal with the interface between different regions.

3.2 THE FINITE VOLUME DISCRETISATION

The finite volume method (FVM) proposed by Patankar (1980) is used to solve the governing equations stated above. In FVM, the calculation domain is divided into a number of control volumes (CVs) such that there is one control volume surrounding each grid point (Patankar, 1980; Versteeg and Malalasekera, 2007). All the unknowns ϕ in the differential equations are at the chosen discrete locations. The differential equation is integrated over each control volume. The most attractive feature of FVM is that integral conservation is satisfied for any number of grid points. Hence, even the coarse grid solution exhibits exact conservation balance.

A typical control volume is illustrated in a 2D setting as shown in Fig. 3-1. A node labelled P is located at the centre of the CV. The neighbouring nodes are denoted as W , E , N and S , respectively. As seen from the figure, the CV has four boundaries with lower-case designations of e , w , n , s , of which the areas of the four faces are A_e , A_w , A_n and A_s , respectively. Δx and Δy are the corresponding lengths of the CV, while δx and δy are the distances between two neighbouring grid points. The scalar variables such as temperature and pressure are stored in the nodes represented by upper-case letters. However, the velocities are stored at the boundaries of each CV. This creation method to form the grid points is called staggered grid as shown in Fig. 3-2. It is used to avoid the checker-board distribution of the pressure field.

Fig. 3-1 Sketch of control volume

Fig. 3-2 Sketch of staggered grid in a control volume

3.2.1 Discretisation of Conservation Equations

Equations (3-1) ~ (3-4) and (3-8) ~ (3-10) can be recast into a general convective-diffusive equation of the following form

$$\begin{aligned} & \frac{\partial}{\partial x}(\rho u \phi) + \frac{\partial}{\partial y}(\rho v \phi) + \frac{\partial}{\partial z}(\rho w \phi) \\ &= \frac{\partial}{\partial x} \left(\Gamma \frac{\partial \phi}{\partial x} \right) + \frac{\partial}{\partial y} \left(\Gamma \frac{\partial \phi}{\partial y} \right) + \frac{\partial}{\partial z} \left(\Gamma \frac{\partial \phi}{\partial z} \right) + S \end{aligned} \quad (3-12)$$

where ϕ , Γ and S are the dependent variables, the diffusion coefficient and the source term, respectively. The three terms on the left hand side of Eq. (3-12) represent the convection terms and the first three terms on the right hand side of Eq. (3-12) stand for the diffusion terms. The source term is the last term on the

right hand side of Eq. (3-12), which can be any term that cannot be directly included into the convection or diffusion terms.

3.2.1.1 Discretisation Equations for Dependent Variables

For FVM, the differential equation is integrated over the CVs. The following discretisation procedure is employed to obtain a series of algebraic equations with the dependent variables at the grid points.

Convection terms

$$\begin{aligned} \int_{\Delta V} \left(\frac{\partial}{\partial x}(\rho u \phi) + \frac{\partial}{\partial y}(\rho v \phi) + \frac{\partial}{\partial z}(\rho w \phi) \right) dV = \\ \int_{\Delta A} (\rho u \phi) dA + \int_{\Delta A} (\rho v \phi) dA + \int_{\Delta A} (\rho w \phi) dA \end{aligned} \quad (3-13)$$

Diffusion terms

$$\begin{aligned} \int_{\Delta V} \frac{\partial}{\partial x} \left(\Gamma \frac{\partial \phi}{\partial x} \right) + \frac{\partial}{\partial y} \left(\Gamma \frac{\partial \phi}{\partial y} \right) + \frac{\partial}{\partial z} \left(\Gamma \frac{\partial \phi}{\partial z} \right) dV = \\ \int_{\Delta A} \left(\Gamma \frac{\partial \phi}{\partial x} \right) dA + \int_{\Delta A} \left(\Gamma \frac{\partial \phi}{\partial y} \right) dA + \int_{\Delta A} \left(\Gamma \frac{\partial \phi}{\partial z} \right) dA \end{aligned} \quad (3-14)$$

Source terms

$$\int_{\Delta V} S dV = \int_{\Delta V} (S_C + S_P \phi_P) dV \quad (3-15)$$

The source term S is always a function of the dependent variable ϕ , so that it is desirable to acknowledge this dependence in constructing the discretisation equations. However, the algebraic equations require the linear function of the unknown variables, and hence, the source term can be dealt with by a linear procedure which is given by

$$S = S_C + S_P \phi_P \quad (3-16)$$

**NUMERICAL SCHEME FOR SINGLE-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

Performing the integration for the above terms, the differential equation becomes

$$J_e + J_w + J_n + J_s + J_t + J_b = (S_C + S_P \phi_p) \Delta x \Delta y \Delta z \quad (3-17)$$

where

$$J_e = \Delta y \Delta z \left(\rho u \phi - \Gamma \frac{\partial \phi}{\partial x} \right)_e \quad (3-18)$$

$$J_w = \Delta y \Delta z \left(\rho u \phi - \Gamma \frac{\partial \phi}{\partial x} \right)_w \quad (3-19)$$

$$J_n = \Delta x \Delta z \left(\rho v \phi - \Gamma \frac{\partial \phi}{\partial y} \right)_n \quad (3-20)$$

$$J_s = \Delta x \Delta z \left(\rho v \phi - \Gamma \frac{\partial \phi}{\partial y} \right)_s \quad (3-21)$$

$$J_t = \Delta x \Delta y \left(\rho w \phi - \Gamma \frac{\partial \phi}{\partial z} \right)_f \quad (3-22)$$

$$J_b = \Delta x \Delta y \left(\rho w \phi - \Gamma \frac{\partial \phi}{\partial z} \right)_b \quad (3-23)$$

The J terms in the above equations are evaluated at the surfaces of the CV.

The power law scheme is employed to illustrate this term, which is defined as

$$J_e = \begin{cases} F_e \phi_E & \text{Pe} < -10 \\ F_e [\phi_E - \beta_e (\phi_E - \phi_P)] & -10 \leq \text{Pe} < 0 \\ F_e [\phi_P - \beta_e (\phi_E - \phi_P)] & 0 \leq \text{Pe} \leq 10 \\ F_e \phi_E & \text{Pe} > 10 \end{cases} \quad (3-24)$$

where β_e and Pe_e are

$$\beta_e = \frac{(1 - 0.1 |\text{Pe}_e|)^{0.5}}{\text{Pe}_e} \quad (3-25)$$

**NUMERICAL SCHEME FOR SINGLE-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

$$\text{Pe}_e = \frac{F_e}{D_e} \quad (3-26)$$

J_w, J_n, J_s, J_t, J_b are defined in a similar manner as J_e . In the calculation of the Peclet number, F and D are given as

$$F_e = (\rho u)_e \Delta y \Delta z \quad D_e = \frac{\Gamma_e \Delta y \Delta z}{(\delta x)_e} \quad (3-27)$$

$$F_w = (\rho u)_w \Delta y \Delta z \quad D_w = \frac{\Gamma_w \Delta y \Delta z}{(\delta x)_w} \quad (3-28)$$

$$F_n = (\rho v)_n \Delta x \Delta z \quad D_n = \frac{\Gamma_n \Delta x \Delta z}{(\delta y)_n} \quad (3-29)$$

$$F_s = (\rho v)_s \Delta x \Delta z \quad D_s = \frac{\Gamma_s \Delta x \Delta z}{(\delta y)_s} \quad (3-30)$$

$$F_t = (\rho w)_t \Delta x \Delta y \quad D_t = \frac{\Gamma_t \Delta x \Delta y}{(\delta z)_t} \quad (3-31)$$

$$F_b = (\rho w)_b \Delta x \Delta y \quad D_b = \frac{\Gamma_b \Delta x \Delta y}{(\delta z)_b} \quad (3-32)$$

Here, the harmonic mean of the diffusion coefficient is adopted to avoid the sudden change in the neighbouring grid points. It is given by (Patankar, 1980)

$$\frac{(\delta x)_e}{\Gamma_e} = \frac{(\delta x)_{e^-}}{\Gamma_P} + \frac{(\delta x)_{e^+}}{\Gamma_E} \quad (3-33)$$

The final discretisation equation for Eq. (3-12) can be written in a compact form of

$$a_P \phi_P = a_E \phi_E + a_W \phi_W + a_N \phi_N + a_S \phi_S + a_T \phi_T + a_B \phi_B + b \quad (3-34)$$

where

$$a_E = D_e A \left(|\text{Pe}_{\Delta e}| \right) + \left[-F_e, 0 \right] \quad (3-35)$$

$$a_W = D_w A(|\text{Pe}_{\Delta w}|) + [F_w, 0] \quad (3-36)$$

$$a_N = D_n A(|\text{Pe}_{\Delta n}|) + [-F_n, 0] \quad (3-37)$$

$$a_S = D_s A(|\text{Pe}_{\Delta s}|) + [F_s, 0] \quad (3-38)$$

$$a_T = D_t A(|\text{Pe}_{\Delta t}|) + [-F_t, 0] \quad (3-39)$$

$$a_B = D_b A(|\text{Pe}_{\Delta b}|) + [F_b, 0] \quad (3-40)$$

$$b = S_C \Delta x \Delta y \Delta z \quad (3-41)$$

$$a_P = a_E + a_W + a_N + a_S + a_T + a_B - S_P \Delta x \Delta y \Delta z \quad (3-42)$$

In the above equations, the power law has been written as

$$A(|\text{Pe}|) = \left\| 0, (1 - 0.1|\text{Pe}|)^5 \right\| \quad (3-43)$$

Here, $\|A, B\|$ is defined as the greater of the numbers A and B .

3.2.1.2 SIMPLER Method for the Momentum Equation

The SIMPLER method is adopted to avoid the checker-board field for the pressure. Staggered grids for the velocity components are employed in the calculation domain. In the staggered grid, the velocity components are calculated for the points that lie on the faces of the CV. Figures 3-3 and 3-4 give the illustration for staggered grids in the x and y directions, respectively.

Although the unknown variables are not in the same grid points, the resulting discretisation equation has the same form as Eq. (3-34)

$$a_e u_e = \sum a_{nb} u_{nb} + b + (p_P - p_E) A_e \quad (3-44)$$

Equation (3-44) can also be written in the following form for simplicity

$$u_e = \frac{\sum a_{nb} u_{nb} + b}{a_e} + (p_P - p_E) d_e \quad (3-45)$$

where d_e is defined as

$$d_e = \frac{A_e}{a_e} \quad (3-46)$$

Fig. 3-3 Location of staggered points for u velocity

Fig. 3-4 Location of staggered points for v velocity

A pseudo velocity $\hat{u}_e$ is defined by

$$\hat{u}_e = \frac{\sum a_{nb} u_{nb} + b}{a_e} \quad (3-47)$$

Hence, Eq. (3-44) becomes

$$u_e = \hat{u}_e + (p_P - p_E) d_e \quad (3-48)$$

Similarly, for the other coordinates,

$$v_n = \hat{v}_n + (p_P - p_N) d_n \quad (3-49)$$

$$w_t = \hat{w}_t + (p_P - p_T) d_t \quad (3-50)$$

The velocity and pressure are coupled together. For the unknown pressure field, the continuity equation can be used to calculate the required pressure field. Substituting Eqs. (3-48) ~ (3-50) into the discretised continuity equation result in

$$a_P p_P = a_E p_E + a_W p_W + a_N p_N + a_S p_S + a_T p_T + a_B p_B + b \quad (3-51)$$

where

$$a_E = \rho_e d_e \Delta y \Delta z \quad (3-52)$$

$$a_W = \rho_w d_w \Delta y \Delta z \quad (3-53)$$

$$a_N = \rho_n d_n \Delta x \Delta z \quad (3-54)$$

$$a_S = \rho_s d_s \Delta x \Delta z \quad (3-55)$$

$$a_T = \rho_t d_t \Delta x \Delta y \quad (3-56)$$

$$a_B = \rho_b d_b \Delta x \Delta y \quad (3-57)$$

$$a_P = a_E + a_W + a_N + a_S + a_T + a_B \quad (3-58)$$

$$\begin{aligned} b = & \left[\left(\hat{\rho u} \right)_w - \left(\hat{\rho u} \right)_e \right] \Delta y \Delta z + \left[\left(\hat{\rho v} \right)_s - \left(\hat{\rho v} \right)_n \right] \Delta x \Delta z \\ & + \left[\left(\hat{\rho w} \right)_b - \left(\hat{\rho w} \right)_t \right] \Delta y \Delta x \end{aligned} \quad (3-59)$$

No approximation is introduced in the derivation of the pressure equation.

Thus, a correct velocity field is used to calculate the pseudo velocities and the pressure equation would give the correct pressure for any iteration.

**NUMERICAL SCHEME FOR SINGLE-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

During the iterative process, the velocities and pressure are taken as temporal values; a small correction may be needed to verify the velocity and pressure.

$$u = u^* + u' \quad (3-60)$$

$$v = v^* + v' \quad (3-61)$$

$$w = w^* + w' \quad (3-62)$$

$$p = p^* + p' \quad (3-63)$$

where u^*, v^*, w^*, p^* are the starred values which may not be the exact values for the corresponding components. u', v', w', p' are corrections to be added to the starred values and u, v, w, p are the correct velocity and pressure fields.

After performing the mathematical transformations, the velocity-correction formulae can be written as

$$u_e = u_e^* + d_e (p'_P - p'_E) \quad (3-64)$$

$$v_n = v_n^* + d_n (p'_P - p'_N) \quad (3-65)$$

$$w_t = w_t^* + d_t (p'_P - p'_T) \quad (3-66)$$

The above correction formulae are substituted into the continuity equation to obtain the pressure correction p'

$$a_P p'_P = a_E p'_E + a_W p'_W + a_N p'_N + a_S p'_S + a_T p'_T + a_B p'_B + b \quad (3-67)$$

where the coefficients have the same meanings as Eqs. (3-52) ~ (3-58). For the last term b in Eq. (3-67), the expression is given by

$$b = \left[(\rho u^*)_w - (\rho u^*)_e \right] \Delta y \Delta z + \left[(\rho v^*)_s - (\rho v^*)_n \right] \Delta x \Delta z + \left[(\rho w^*)_b - (\rho w^*)_t \right] \Delta y \Delta x \quad (3-68)$$

The sequence of the operations for SIMPLER is stated as follows:-

- (1) Set a guessed velocity field.

(2) Calculate the coefficients for the momentum equations and hence calculate

$$\hat{u}_e, \hat{v}_e \text{ and } \hat{w}_e.$$

(3) Calculate the coefficients for the pressure Eq. (3-51) to obtain the pressure field.

(4) Solve the discretised momentum equations and set the velocity field to obtain u^*, v^* and w^* .

(5) Solve the equation to obtain pressure correction p' .

(6) Correct the velocity field.

(7) Return to step 2 and repeat until convergence.

3.2.2 Implementation of Boundary Conditions

3.2.2.1 Implementation for the Velocity Boundary Condition

The treatment for the boundary conditions needs to be considered at the outflow boundary, where the fluid leaves the calculation domain. If the velocities are known at the outlet boundary, no special treatment is needed. However, neither the value nor the flux is known for most cases. Therefore, the outlet velocity as shown in Fig. 3-5 is imposed to satisfy mass conservation in the entire calculated domain. It is given by

$$u_{B+1} = u_B + C \quad (3-69)$$

Equating the inlet mass flow rate to the outlet mass flow rate gives

$$C = \frac{\dot{m}_{in} - \rho u_B A_B}{\rho A_B} \quad (3-70)$$

where $\dot{m}_{in}$ is the inlet mass flow rate.

For velocities v and w in the y and z directions, the outflow boundary conditions are

$$v_B = v_{B-1} \quad (3-71)$$

$$w_B = w_{B-1} \quad (3-72)$$

For the pressure and pressure correction equations, all boundary values are specified and hence, the corresponding coefficients are set to zero.

Fig. 3-5 Velocity at the outlet boundary

3.2.2.2 Implementation for the Energy Boundary Condition

The discretised equation requires boundary conditions to obtain a unique solution. Typically, three kinds of boundary conditions are encountered in heat conduction. These are

- (1) Boundary temperature is given.
- (2) Boundary heat flux is given.
- (3) Boundary heat flux is specified through a heat transfer coefficient and the temperature of the surroundings.

Obviously, no additional equations are required if the boundary temperature is given. For the case of given boundary flux, the boundary conditions can be imposed by modifying the discretised equations at the boundary. The additional source term method is used to deal with the given heat flux boundary condition.

For example, assuming q_B'' is given as the heat flux, as shown in Fig. 3-6, the required equation for T_B becomes

$$a_B T_B = a_I T_I + b \quad (3-73)$$

where

$$a_I = \frac{k_i}{(\delta x)_i} \quad (3-74)$$

$$b = S_C \Delta x + q_B'' \quad (3-75)$$

$$a_B = a_I - S_p \Delta x \quad (3-76)$$

If the heat flux q_B'' is specified through a heat transfer coefficient h and a surrounding fluid temperature T_f , the boundary condition can be imposed as

$$q_B'' = h(T_f - T_B) \quad (3-77)$$

Then the coefficients for Eq. (3-73) become

$$a_I = \frac{k_i}{(\delta x)_i} \quad (3-78)$$

$$b = S_C \Delta x + h T_f \quad (3-79)$$

$$a_B = a_I - S_p \Delta x + h \quad (3-80)$$

Fig. 3-6 Half control volume near left boundary

After the above transformation, the required number of equations for the unknown temperatures is sufficient to solve the algebraic equations.

3.3 SOLUTION PROCEDURE

The discretised equation (Eq. 3-34), in general, is non-linear and interlinked. The coefficients of the equations may depend on the dependent variables ϕ . An iterative method is adopted here to solve these equations. First, the coefficients in the equations are calculated from the initial values (or previous iteration values) of ϕ . With these known values of the coefficients, the equations become linear. The linearised equations are solved by the line-by-line method to obtain the updated values of ϕ . Then the coefficients are recalculated using these new values of ϕ and again the linearised equations are solved. Such a process is repeated until the convergence of ϕ is achieved.

The general structure of the program can be summarised as follows:-

- (1) Set a guessed velocity and temperature fields for both the solid and liquid phases.
- (2) Use the SIMPLER method to calculate the velocity and pressure.
- (3) Calculate the coefficients and source term for the temperature of the solid phase.
- (4) Calculate the coefficients and source term for the temperature of the fluid phase.
- (5) Solve Eqs. (3-9) and (3-10) for the solid and fluid temperatures.
- (6) Return to steps (3) through (5) until the solution converges.

In the iterative solution of the algebraic equations, under-relaxation is usually employed for the sake of stability. The solution for a dependent variable ϕ in the n^{th} iteration is

$$a_p \phi_p^n = \sum a_{nb} \phi_{nb}^n + b \quad (3-81)$$

$$\phi_p^n = \frac{\sum a_{nb} \phi_{nb}^n + b}{a_p} \quad (3-82)$$

where a_{nb} are the coefficients at the neighbouring nodes, viz. W, E, N, S, T, B .

With an under-relaxation factor α introduced, Eq. (3-82) becomes

$$\phi_p^n = \alpha \phi_p^n + (1 - \alpha) \phi_p^{n-1} \quad (3-83)$$

For under-relaxation

$$0 \leq \alpha \leq 1 \quad (3-84)$$

Substituting Eq. (3-83) into Eq. (3-82), the dependent variable is

$$\frac{a_p}{\alpha} \phi_p^n = \sum a_{nb} \phi_{nb}^n + b + (1 - \alpha) \frac{a_p}{\alpha} \phi_p^{n-1} \quad (3-85)$$

There are no general rules to choose the optimum value of α . This is a problem-dependent issue. The optimum α is dependent on the CVs, the iteration procedures, the grid spacing and other parameters. Generally, a suitable under-relaxation can be obtained from experience or exploratory computations for a given problem.

3.4 CONVERGENCE CRITERION

In the iterative procedures for a steady state problem, the solution is considered to have converged if the maximum change of ϕ for two successive iterations is less than a prescribed value η . The convergence criterion is defined as

$$\left| \frac{\phi_i - \phi_{i-1}}{\phi_i} \right|_{\max} \leq \eta \quad (3-86)$$

η is chosen to be 10^{-5} and it is applied to all the dependent variables.

3.5 SUMMARY

In this chapter, the mathematical formulations for flow and heat transfer in channel with both porous and open regions in a 3-D setting are presented. The FVM is introduced to discretise the general equations. The SIMPLER method to solve the coupling of pressure and velocity is briefly presented. Boundary conditions for both velocity and temperature are introduced. The solution procedure and the convergence criterion are given. The numerical solution proposed in this chapter will be implemented for different cases in the next chapter.

CHAPTER 4 CASE STUDIES IN A 2D DOMAIN FOR SINGLE-PHASE FLOW AND HEAT TRANSFER

Based on the mathematical formulation and the associated solution procedure presented in Chapter 3, the 2D and 3D codes in FORTRAN are developed. In this chapter, the developed codes are tested and validated against several problems. These test problems range from a porous domain in a 2D setting to a conjugated domain which contains both porous and open regions. All the simulations in the current thesis were carried out on a desktop with Intel (R) Core (TM) 2 Quad CPU Q9450@2.66GHz and 3.25GB RAM.

In this chapter, each subsequent subsection begins with a discussion of the chosen problem. This is followed by the specifications of the problem. A brief discussion of the grids is given before the presentation of the results.

4.1 BEAVERS - JOSEPH FLOW PROBLEM

The Beavers - Joseph flow problem (1967) is a famous case which has been studied extensively (Gartling *et al.*, 1996; Alazmi and Vafai, 2001; Costa *et al.*, 2004; Betchen *et al.*, 2006). It is chosen to test the ability of the current code to deal with the parallel interface between the porous and open regions.

4.1.1 Problem Specification

The geometry of simulation domain is depicted in Fig. 4-1. The channel is divided into two regions, *viz.* open and porous regions. These two regions have the same dimensions with the interface parallel to the flow direction. A length

of $12H$ along the flow direction is chosen to obtain fully developed flow in the open region. The porosity of the porous media is 0.7 with the correlation of inertial coefficient $C_E = 1.75\varepsilon/(150\varepsilon^5)^{0.5}$ (Alazmi and Vafai, 2004). The dimensionless parameter of Reynolds number is defined as $Re_H = \rho UH/\mu$ which is chosen to be 1. U is the mean velocity in the open region. Darcy number is defined as $Da = K/H^2$.

Fig. 4-1 Schematic diagram of open and porous conjugated domains (not to scale)

4.1.2 Numerical Details

The effects of the spatial grids on the velocity fields are studied. Solutions obtained using 75×20 control volumes (CVs), 150×40 CVs and 300×80 CVs are compared. The results showed that there is not much difference between the velocity fields obtained for meshes of 150×40 CVs and 300×80 CVs. Therefore, 150×40 CVs is used for the current simulation.

4.1.3 Results and Discussion

Figures 4-2 and 4-3 show the velocity profiles obtained from the current simulation at $x = 10H$ for $Da = 10^{-2}$ and $Da = 10^{-3}$, respectively. The dimensionless velocity u^* is given by $u^* = u/U$. Generally, good agreements are achieved between the current results and the results of Costa *et al.* (2004). In the open region, the velocity shows an approximate parabolic profile while in

the porous region, it exhibits a more uniform profile. With the decrease of Da , the magnitude of the velocity in the open region increases and decreases in the porous region. Both figures indicate that most of the fluid passes through the open region and only a small portion goes through the porous medium. The quantity of the fluid flowing into the porous region is highly dependent on the permeability of the porous medium.

Fig. 4-2 Velocity profile in a conjugated domain with $Re_H = 1$ and $Da = 10^{-2}$

Fig. 4-3 Velocity profile in a conjugated domain with $Re_H = 1$ and $Da = 10^{-3}$

The good agreement between the present results and those of Costa *et al.* (2004) demonstrates that the present numerical model is capable of dealing with the parallel interface between the porous and open regions.

4.2 POROUS PLUG FLOW

This problem involves flow through a porous plug located in a horizontal channel. It was chosen to test the ability of the current code to deal with the porous and open regions' interface normal to the flow direction. This problem was studied, among others, by Costa *et al.* (2004) and Betchen *et al.* (2006).

4.2.1 Problem Specification

The test domain is shown in Fig. 4-4. Two empty ducts each of length $3H$ are imposed before and after the porous region where the length is $2H$. A fully developed velocity profile is imposed at the inlet. The parameters used in this study are similar to those in the Beavers - Joseph flow problem.

Fig. 4-4 Schematic diagram of full porous domain (not to scale)

4.2.2 Numerical Details

Meshes of 70×15 CVs, 140×30 CVs and 280×60 CVs are studied to achieve grid-independent solutions. The velocity profiles were found to overlap under meshes of 140×30 CVs and 280×60 CVs. Hence, a mesh of 140×30 CVs is used in the current simulation.

4.2.3 Results and Discussion

Figure 4-5 presents the comparison of dimensionless velocity ($u^* = u/U$) at the centreline obtained from the current simulation with the data of Costa *et al.* (2004). The results are almost identical to each other. Based on the fully

developed velocity profile given at the inlet boundary, the centreline velocity is 1.5 times larger than the average velocity in the open region. It decreases rapidly when fluid flows into the porous plug with a value about 1.27 times larger than the average velocity. As fluid flows out of the porous plug, it approaches the fully developed velocity in a very short distance. The comparison of the dimensionless pressure is shown in Fig. 4-6. The dimensionless pressure p^* is defined as $p/\rho U^2$ which appears as a linear drop in each of the three regions shown in Fig. 4-6. The porous plug gives a much higher pressure drop compared to that in the open region.

Fig. 4-5 Dimensionless velocity at the centreline of full porous domain with $Re_H = 1$ and $Da = 10^{-2}$

Fig. 4-6 Dimensionless pressure drop in full porous domain with $Re_H = 1$ and $Da = 10^{-2}$

Another case is performed to test the capability of the current code in dealing with the high velocity. It is carried out in the same domain except for the change of the dimensions. The porous plug domain is set to be $5H$. The lengths of the two ducts are chosen to be $5H$ and $50H$, respectively. With a large length of the duct when fluid flows out of the porous plug, fully developed boundary condition can be imposed at the outlet. The velocity profile at $x/H = 2.5$, $x/H = 7.5$ and $x/H = 12.5$ for $Re_H = 1000$ in the domain are chosen for comparison purpose. As seen from Figs. 4-7(a) and 4-7(b), the velocity profiles at different locations are similar. This validation confirms the capability of the present code to deal with the coupling of velocity and pressure for high Reynolds number.

Fig. 4-7 (a) Betchen *et al.* (2006)'s results (b) present results in the porous plug domain

As for the porous plug problem, both the velocity fields under low and high velocity as well as the pressure fields are compared with the existing results. The excellent agreements demonstrate the capability of the current code to deal with the perpendicular interface between the porous and open regions.

4.3 HEAT TRANSFER BY USING THE LOCAL THERMAL EQUILIBRIUM (LTE) MODEL

This problem is chosen to test the capability of the current code in modelling heat transfer by using local thermal equilibrium (LTE) model. It has been studied by Alazmi and Vafai (2001).

4.3.1 Problem Specification

The test domain of this problem is shown in Fig. 4-8. It contains a porous region located above an open region where these two regions are parallel to each other. A length of $12H$ along the flow direction is chosen to obtain the fully developed flow in the open region. The heights of the porous and open regions are $50H$ and H , respectively. The porosity of the porous media is 0.7. The Reynolds number is chosen to be 1 while the Darcy number given by $Da = K/H^2$ is prescribed as 10^{-3} . The inertial parameter is non-dimensionlised as $\Lambda = \epsilon C_E H/K^2$ which is set to be 1.0. The ratio of the thermal conductivities of the solid to fluid phases is 25.

Fig. 4-8 Schematic diagram of simulation domain (not to scale)

4.3.2 Numerical Details

A uniform velocity is imposed at the inlet and fully developed condition is imposed at the outlet. The bottom and upper walls are maintained at constant temperatures of T_w and T_{in} , respectively.

Temperature distributions are obtained using uniform grid of 100×370 CVs, 150×560 CVs and 225×840 CVs. The maximum temperature differences are 1% and 0.3% between the successive increases of the grids. Hence, 150×560 CVs is chosen in this simulation.

4.3.3 Results and Discussion

Figure 4-9 shows the comparison of the results of the current simulation and those from Alazmi and Vafai (2001). The dimensionless parameter θ is defined as $\theta = (T - T_{in}) / (T_w - T_{in})$. This temperature profile is taken at the outlet of the domain. Excellent agreement is achieved. Hence, it is shown that the current code can handle the problem of heat transfer in porous media by using the LTE model.

Fig. 4-9 Comparison of temperatures obtained from the current and those of Alazmi and Vafai (2001).

4.4 HEAT TRANSFER BY USING THE LOCAL THERMAL NON-EQUILIBRIUM (LTNE) MODEL

This problem is adopted to test the capability of the current code in modelling heat transfer in porous media by using the LTNE model. It has been studied by Calmidi and Mahajan (2000) and Betchen *et al.* (2006).

4.4.1 Problem Specification

The schematic diagram of simulation domain is shown in Fig. 4-10. A channel is fully filled with metal foam of dimensions 114 mm (L) $\times$ 45 mm (H) $\times$ 63 mm (W). The physical and thermal properties of the metal foam are listed in Table 4-1. PPI is the pore density which means pores per linear inch, d_l and d_p are the ligament diameter and the pore diameter of the foam, respectively. The other properties have the same meanings as explained in Chapter 2.

Fig. 4-10 Schematic diagram of simulation domain

Table 4-1 Characteristic properties of metal foam for sample #2

ε	PPI	d_l (mm)	d_p (mm)	K (10^{-7} m^2)	C_E	k_{se} (W/m $\cdot$ K)	k_{fe} (W/m $\cdot$ K)
0.9118	5	0.55	3.80	1.80	0.085	6.46	0.0237

4.4.2 Numerical Details

The fluid properties are evaluated at 297 K. The interfacial heat transfer coefficient h_{sf} and specific surface area α_{sf} are calculated according to the model proposed by Calmidi and Mahajan (2000). The Nusselt number is evaluated from the following proposed by Zukauskas (1987)

$$\text{Nu}_{sf} = \frac{h_{sf} d_f}{k_f} = C_T \text{Re}_d^{0.5} \text{Pr}^{0.37} \quad (4-1)$$

with $C_T = 0.52$. The Reynolds number is defined as

$$\text{Re}_d = \frac{\rho u d_l}{\epsilon \mu_f} \quad (4-2)$$

A fully developed velocity profile is set as the inlet flow boundary condition. The average velocity is equal to 1. Zero derivative conditions are prescribed at the outlet. The velocities at $y = 0$ and $y = H$ are set to be zero. For the heated wall at $y = 0$, the discrete experimental data obtained by Calmidi and Mahajan (2000) are fitted into quadratic equations as a function of x which is in the flow direction.

The simulation domain is discretised into 50×40 CVs, 100×80 CVs and 200×160 CVs, respectively to arrive at the grid-independent solution. The results show that 100×80 CVs is sufficient to obtain grid-independent solutions.

4.4.3 Results and Discussion

Figure 4-11 shows the temperature difference contours for the solid and fluid phases. This temperature difference θ_i is defined as $\theta_i = T_i - T_{in}$ where i represents either the solid or the fluid phase. Generally, good agreement is observed for both the solid and fluid temperature distributions.

Fig. 4-11 Temperature difference contours in (a) solid phase and (b) fluid phase (dash lines for current simulation and solid lines for Betchen *et al.*'s (2006) results)

In addition, the Nusselt numbers obtained at the selected points are also compared to the results obtained by Calmidi and Mahajan (2000) and Betchen *et al.* (2006). The Nusselt number is defined as

$$Nu = \frac{\bar{h} L}{k_e} = \frac{q'}{k_e \Delta T_{avg}} \quad (4-3)$$

where q' is the heat transfer rate per unit length and ΔT_{avg} is the difference between the average substrate temperature and the inlet temperature. Comparing a single point at $Re_K = 27.4$ and $Pr_e = 2.84 \times 10^{-3}$, the Nusselt number based on the present model is 5.04 and the results from Calmidi and Mahajan (2000) and Betchen *et al.* (2006) are 5.0 and 5.1, respectively which indicate excellent agreement. The results indicate that the present code can deal with heat transfer in porous media by using the LTNE model.

4.5 FLOW AND HEAT TRANSFER IN A CHANNEL WITH STAGGERED POROUS BLOCKS

In this problem, fluid flow and heat transfer in a channel with staggered porous blocks are studied. This problem is chosen to study the effects of various parameters on the fluid flow and heat transfer fields. These parameters include Darcy number, Reynolds number, porous block height and width, and the thermal conductivity ratio between the porous media and the fluid.

4.5.1 Problem Specification

The schematic diagram of this problem is shown in Fig. 4-12. The channel is filled with porous blocks on both the upper and lower walls in a staggered arrangement. The porous blocks are numbered consecutively along the flow direction. The open regions created by the tips of the porous blocks and their corresponding upper or lower walls are called gaps. The two walls of the channel are heated to a temperature of T_w . As the fluid at lower temperature T_{in} flows into the channel, convection heat transfer occurs between the walls and the fluid. Due to the arrangement of the porous blocks, the convection heat transfer is different from the case in which the fluid touches the wall or the porous blocks.

The height of the channel is H and the length is L . The porosity, height and width of the porous blocks are ε , a and w , respectively. The space between the consecutive porous blocks is b . The distances before the fluid flows into and out of the physical domain are l_1 and l_2 , respectively.

In all the cases studied, the following parameters are kept constant: $Pr = 0.7$, $(w+b)/H = 1$, $l_1/H = 4$ and l_2/H was set to 40 to ensure that the exit boundary condition has no effect on the solution. The Reynolds and Darcy numbers in the current problem are defined as $Re = \rho u_{in} H / \mu$ and $Da = K/H^2$, respectively. The porosity of the blocks is 0.75 and the familiar relation of $C_E = 1.75\epsilon/(150\epsilon^5)^{0.5}$ (Alazmi and Vafai, 2004) is used to calculate the inertial coefficient. The dimensionless inertial parameter is defined as $\Lambda_H = C_E H \epsilon / K^{0.5}$.

Fig. 4-12 Schematic diagram of the channel with staggered porous blocks

4.5.2 Numerical Details

In order to focus on the effect of other parameters on the flow field and local heat transfer, the LTE model is adopted in the present study. The highest thermal conductivity ratio of k_e/k_f was taken to be 10. For $k_e/k_f > 10$, the LTNE model may be required (Amiri and Vafai, 1994; Jiang *et al.*, 1999; Leblond and Gosselin, 2008).

Meshes of 280×50 CVs, 560×100 CVs and 1120×200 CVs are used to obtain grid-independent solutions. Both the velocity and temperature profiles have almost overlapped between the grids of 560×100 CVs and 1120×200 CVs. Therefore, the mesh of 560×100 CVs is used in the current simulation.

4.5.3 Results and Discussion

The local Nusselt number is defined as

$$\text{Nu}_x = -\frac{k_e}{k_f} \frac{H}{T_w - T_b} \frac{\partial T}{\partial y} \bigg|_{y=0, y=H} \quad (4-4)$$

It is calculated for both the upper and lower walls. In the open regions, k_e/k_f is 1.

The bulk temperature T_b is given by

$$T_b = \frac{\int_0^H |u| T dy}{\int_0^H |u| dy} \quad (4-5)$$

4.5.3.1 Basic Test

The basic study corresponds to the case of $\text{Da} = 10^{-4}$, $\Lambda_H = 16.5$, $\text{Re} = 100$, $a/H = 0.8$, $w/H = 0.2$ and $k_e/k_f = 1$. Figures 4-13(a) and 4-13(b) show the velocity field and local Nusselt number, respectively. As seen from Fig. 4-13(a), most of the fluid, referred to as the main flow henceforth, is forced to flow in a “serpentine” manner guided by the staggered porous blocks. The main fluid flows past the gap created between the first porous block and the upper wall, and is forced to flow downward due to the large flow resistance presented by the second porous block. Thereafter, the main fluid flows upward and downward alternately according to the locations of the gaps. Thus, a “serpentine” flow is formed. A large amount of fluid flows through the gaps and only a small portion of the fluid can penetrate the porous blocks. This results in the large difference of velocity in the open and porous regions. No vortex is found in the flow field. Figure 4-13(b) shows Nu_x at the upper and lower walls. Compared to the channel without porous blocks, Nu_x is significantly affected by the presence of the staggered porous blocks and a

significant enhancement of heat transfer is observed. This enhancement is due to the distorted flow which increases fluid mixing. A periodic variation of Nu_x at the two walls is observed for the channel with staggered porous blocks. The critical values of Nu_x for the upper wall are alternately changed with that of the lower wall. In each cycle of the upper wall, the maximum Nu_x occurs at the location of x which is on the top of the porous block, while the minimum Nu_x occurs roughly between the two consecutive porous blocks. The heat transfer is enhanced due to the high velocity near the wall, an indication of increased convective heat transfer. Poorer heat transfer is found at the locations with low velocity. A large decrease in Nu_x at the upper wall is observed as the fluid flows out of the last porous block. This is because only a small amount of the fluid flows across the last porous block thus reducing the convective heat transfer.

Fig. 4-13 (a) Velocity vectors (b) Local Nusselt number distribution at lower and upper walls with $Re = 100$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.8$, $w/H = 0.2$ and $k_c/k_f = 1$

4.5.3.2 Effect of Darcy Number

The Darcy number is a dimensionless parameter which is directly related to the permeability of the medium. Porous blocks of low permeability and hence, low Darcy number would present high flow resistances for the fluid flow. Conversely, the fluid can flow easily through porous blocks of high permeability under the condition of large Darcy number. The effects of Darcy number on the flow field are shown in Fig. 4-14. Comparing the velocity fields in Figs. 4-13(a), 4-14(a) and 4-14(b), the main flow becomes more serpentine-like with the decrease of Da. At high Da (10^{-3}), the fluid flows through the porous blocks easily and the velocity field is slightly distorted. As Da increases, the flow field approaches to that of the channel without porous blocks. However, at low Da (10^{-5}), the porous blocks become almost impermeable to the fluid. The low permeability of the porous blocks results in a large flow resistance. Thus, the fluid has difficulty penetrating the porous blocks. Most of the fluid flows through the gaps and the flow rate through the porous blocks is dramatically reduced. Very interesting features are observed at Da = 10^{-5} . Vortices are formed near the porous blocks. These are shown in Figs. 4-14(c) and 4-14(d). The larger vortex occurs at the rear of the first porous block (Fig. 4-14c) covering more than half of the open region between the two porous blocks. The fluid is accelerated across the gap until it hits the second porous block and is forced to flow downward. Shear stress is exerted by this stream of fluid on the fluid near the corner of the first porous block, and thus creating a circulatory flow. The smaller vortex is generated at the front corner of the second porous block as the fluid is trapped. The vortex formed after the last

porous block is shown in Fig. 4-14(d). The fluid which flows out of the last porous block pushes the vortex further downstream, shifting the centre of the circulation away.

Fig. 4-14 (a) Velocity field with $Re = 100$, $a/H = 0.8$, $w/H = 0.2$ and $k_e/k_f = 1$ (a) $Da = 10^{-3}$, $\Lambda_H = 5.22$, (b) $Da = 10^{-5}$, $\Lambda_H = 52.2$, (c, d) close-up views of circulatory region at $Da = 10^{-5}$ and $\Lambda_H = 52.2$

The variations of Nu_x along the channel at both the upper and lower walls for various Da are shown in Figs. 4-15(a) and 4-15(b), respectively. Nu_x is significantly affected by Da . At a higher Da , both the flow and heat transfer characteristics resemble closely those of a channel without porous blocks. The

peaks in the Nu_x arose from the effect of the porous blocks on the flow field which modified the heat transfer characteristics. Referring to the velocity field, it can be observed that the more distorted the flow field is, the larger would be the variation of Nu_x . With the decrease of Da , the heat transfer is increased significantly. For $Da = 10^{-5}$, a minimum heat transfer is predicted at the rear of the porous blocks due to the formation of circulatory flow in these regions. For a localised circulatory flow, there is very little mixing with the fluid in the main flow. The heat absorbed by the circulatory flow is mostly transferred to the main flow by conduction, rather than by convection. This is not favourable for heat transfer. The maximum heat transfer occurs at the wall located above the tips of the porous blocks. The existence of the porous blocks forces the fluid to bypass the blocks and thus, increases fluid mixing. The fluid which is accumulated and accelerated in the gaps enhances the convection heat transfer significantly. In Fig. 4-14(d), a large circulatory flow which is formed between $x/H = 8$ and 11 near the upper wall results in the decrease of Nu_x . A small circulatory flow between $x/H = 11$ and 13 causes the flow stream to bend upward; thus, increasing the convection heat transfer at the upper wall. Although a large variation of Nu_x is observed at low Da , it is noteworthy that the total heat transfer is sufficiently enhanced overall.

(a)

(b)

Fig. 4-15 Effects of Darcy number on local Nusselt number with $Re = 100$, $a/H = 0.8$, $w/H = 0.2$ and $k_e/k_f = 1$ at (a) lower wall (b) upper wall

4.5.3.3 Effect of Reynolds Number

The effect of Re on the flow field is investigated for the case of $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.4$, $w/H = 0.2$ and $k_e/k_f = 1$. Figures 4-16 to 4-18 show the velocity fields for $Re = 100$, 200 and 300, respectively. The streamlines for the close-up views of the velocity fields are constructed and shown in Figs. 4-16(b), 4-16(c), 4-17(b), 4-17(c) and 4-18(b). At $Re = 100$, vortices are formed at the rear of every porous block. These vortices become weaker at $Re = 200$ and ultimately vanish at $Re = 300$. As Re increases, the inertial effect becomes important. With a larger inertia force, the fluid changes its direction slowly to

bypass the porous block, and hence, more fluid would flow through the porous blocks and push the vortex further downstream. The delayed vortex is suppressed by the upcoming main stream to a small region as shown in $Re = 200$. As the Reynolds number increases to a certain value, the vortex is totally suppressed and the recirculation region disappears as in the case of $Re = 300$. However, this is not the case for the recirculation region behind the last porous blocks. As the main stream flows downstream directly through the gaps created by the last porous block and the lower wall, only a small amount of the liquid can penetrate porous block 4. Such small amount of the liquid is not sufficient to suppress the recirculation region behind the last porous block. Therefore, the recirculation region behind the last porous block remains. A closer investigation of Figs. 4-16(c), 4-17(c) and 4-18(c) shows that the centre of the circulation formed at the last porous block is shifted further downstream with the increase of Re . The centres of the circulation are located at $x/H = 8.7$ for $Re = 100$, $x/H = 10.0$ for $Re = 200$ and $x/H = 11.8$ for $Re = 300$.

The upper and lower Nu_x are shown in Figs. 4-19(a) and 4-19(b), respectively. As expected, maximum heat transfer is obtained at the highest Reynolds number of 300 out of the porous blocks region. However, this is not the case for the structures within the porous blocks. Upon careful examination of Nu_x at around $x/H = 5.0$ for the lower wall, it is found that the peak value occurs at $Re = 100$ rather than $Re = 300$. The same behaviour is observed for the upper Nu_x at around $x/H = 6.5$. These phenomena are caused by the vortices formed for different Re . The minimum heat transfer occurs at the location with a vortex, whereas, the maximum heat transfer is simultaneous produced at the opposite wall with almost the same x value. As seen from Figs. 4-16 to 4-18, the

**CASE STUDIES IN A 2D DOMAIN FOR SINGLE-PHASE FLOW
AND HEAT TRANSFER**

recirculation region between two consecutive porous blocks gradually shrinks with the increase of Reynolds number until it disappears at $Re = 300$. More fluid bypasses the vortex at lower Re . The accumulated fluid above the vortex is accelerated, and convection heat transfer at the opposite wall is increased.

Fig. 4-16 Velocity field with $Re = 100$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.4$, $w/H = 0.2$ and $k_e/k_f = 1$ (a) velocity field (b, c) close-up views of circulatory region

Fig. 4-17 Velocity field with $Re = 200$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.4$, $w/H = 0.2$ and $k_e/k_f = 1$ (a) velocity field (b, c) close-up views of circulatory region

Fig. 4-18 Velocity field with $Re = 300$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.4$, $w/H = 0.2$ and $k_e/k_f = 1$ (a) velocity field (b, c) close-up views of circulatory region

Fig. 4-19 Effects of Reynolds number on local Nusselt number with $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.4$, $w/H = 0.2$ and $k_e/k_f = 1$ at (a) lower wall (b) upper wall

4.5.3.4 Effect of Porous Block Height

The effect of the porous block height is investigated. This investigation corresponds to the case of fixed $Re = 100$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $w/H = 0.2$ and $k_e/k_f = 1$. Figures 4-13(a), 4-16 and 4-20 show the plots of the velocity fields for the cases of $a/H = 0.8$, 0.6 and 0.4, respectively. For porous blocks of small heights, a circulatory flow is formed just downstream of the first and second porous blocks. The inertia of the fluid flowing at the tip of the porous blocks is sufficiently high such that the fluid changes direction gradually. The fluid near the corner formed by the porous block and the wall behaves as if it is trapped in the corner. Driven by the larger shear stress of the main flow just past the tip of the porous block, the trapped fluid circulates at the corner. This resembles the formation of circulatory flow in the flow with a forward facing step or a sudden expansion. The fluid flowing out from the porous blocks exerts downstream-wise forces on the trapped fluid, shifting the centre of circulation downstream and away from the porous blocks. The effect of an increase in the height of the porous blocks can be best exemplified by the case of $a/H = 0.8$. The majority of

the fluid which flows past the tip of the first porous block has to turn downward as the presence of the second porous block creates a large flow resistance in the stream wise direction. The downward flowing fluid generates forces on the fluid near the corner formed by the first porous block and the lower wall, squeezing it to flow downstream. With this, the formation of a circulatory flow, as in the case of channel with smaller height of the porous block, is weakened and suppressed. A similar phenomenon leads to the absence of circulatory flow at the corner formed by the second porous block and the upper wall.

Fig. 4-20 Velocity field with $Re = 100$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.6$, $w/H = 0.2$ and $k_e/k_f = 1$ (a) velocity field (b, c) close-up view of circulatory region

The variation of Nu_x with different porous block heights is shown in Fig. 4-21.

The case of $a/H = 1$ corresponds to the situation where the porous blocks height is equal to the height of the channel. Unexpectedly, it is observed that Nu_x is not a monotonic function of the porous block height. The maximum heat transfer is observed when the porous block height is at $0.8H$ among all the

tested cases. A further increase of the height accelerates the fluid velocity in the smaller gaps which can enhance the convection heat transfer. However, with the fully inserted porous blocks, the gaps vanished and fluid mixing is weakened, resulting in reduced heat transfer. The relatively anomalous variation of the local Nusselt number is observed at $a/H = 0.4$ due to the vortices behind each porous block. It can be concluded that heat transfer can be increased by selection of an optimum porous block height.

(a)

(b)

Fig. 4-21 Effects of the porous block height on local Nusselt number with $Re = 100$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $w/H = 0.2$ and $k_e/k_f = 1$ at (a) lower wall (b) upper wall

4.5.3.5 Effect of Porous Block Width

The effects of the porous block width on the velocity field are shown in Figs. 4-22(a) and 4-22(b) by increasing the width to $w/H = 0.4$, $w/H = 0.8$, respectively. As compared with Fig. 4-13(a) at $w/H = 0.2$, it is observed that with the increase of the porous block width, the fluid flow in the open region between the two consecutive porous blocks is more “serpentine”. Unlike the case where $w/H = 0.2$, a vortex appears for both $w/H = 0.4$ and 0.8 when the fluid flows out of the last porous block. Meanwhile, the circulatory region is expanded to a larger region when the width of the porous block increased. The large flow resistance incurred by the increased width results in the decrease of the flow rate through the porous blocks and the increased flow rate in the gaps. The larger shear stresses of the main flow through the gaps force the fluid to circulate at the corner. The decreased flow rate across the porous blocks pulled the circulatory region close to the surface of the last porous block.

Fig. 4-22 Velocity field with $Re = 100$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.8$, $k_e/k_f = 1$ (a) $w/H = 0.4$ (b) $w/H = 0.8$ (c, d) close-up view of circulatory region

Figures 4-23(a) and 4-23(b) show the upper and lower Nu_x for different porous block widths. Generally, the rate of heat transfer increases with the increase of the porous block width. The decrease of the space between two consecutive porous blocks results in good mixing of the fluid thus enhancing convection heat transfer. It is noted that further increase of the width from $0.6H$ to $0.8H$ does not enhance the heat transfer significantly, and a larger pressure drop is induced instead.

Fig. 4-23 Effect of the porous block width on local Nusselt number with $Re = 100$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.8$ and $k_e/k_f = 1$ at (a) lower wall (b) upper wall

4.5.3.6 Effect of Thermal Conductivity Ratio

The results discussed previously were for the cases in which the thermal conductivity ratio k_e/k_f was fixed at 1. Further investigations were performed at higher thermal conductivity ratios of 5 and 10. These studies were performed for prescribed values of $Re = 100$, $Da = 10^{-5}$, $\Lambda_H = 52.2$, $a/H = 0.8$ and $w/H = 0.2$. Figures 4-24(a) and 4-24(b) display the upper and lower Nu_x for different thermal conductivity ratios. It can be seen that both the upper and lower Nu_x increase due to an increase in the thermal conductivity ratio between the porous blocks and the fluid. Significant enhancement of Nu_x can be found at the locations of the porous blocks when the thermal conductivity ratio is increased. Such improvements of heat transfer would become more important at high permeability as a large amount of the fluid could penetrate the porous blocks.

Fig. 4-24 Effect of the thermal conductivity ratio on local Nusselt number with $Re = 100$, $Da = 10^{-5}$, $\Lambda_H = 52.2$, $a/H = 0.8$ and $w/H = 0.2$ at (a) lower wall (b) upper wall

4.5.3.7 Effect of the Porous Blocks on Pressure Drop

The average pressure in the cross section perpendicular to the flow direction is defined as

$$\bar{p} = \frac{1}{H} \int_0^H p dy \quad (4-6)$$

The pressure drop in the channel with porous blocks is calculated by the difference between the average inlet and outlet pressures. The dimensionless pressure drop p^* is defined as $p^* = \Delta p / \rho u_{in}^2$.

Figure 4-25(a) shows the variation of the pressure drop with Darcy number for $Re = 100$, $a/H = 0.8$, $w/H = 0.2$ and $k_e/k_f = 1$. The pressure drop increases significantly with the decrease of the Darcy number. The lower the Darcy number, the higher is the flow resistance as the fluid flows through the porous blocks. Thus, a high pressure drop is produced. Figure 4-25(b) shows the variation of the pressure drop with Reynolds number at $Da = 10^{-4}$, $\Lambda_H = 16.5$, $a/H = 0.8$, $w/H = 0.2$ and $k_e/k_f = 1$. The pressure drop for $Re = 100$, Δp_{100} , is selected as the reference pressure drop for the purpose of comparison. As expected, the pressure drop increases with the increase of the Reynolds number. The effects of porous block width and height on the pressure drop are shown in Figs. 4-26(a) and 4-26(b). The pressure drop increases gradually when the height of the porous blocks is less than $0.4H$. However, significant augmentations are observed for larger heights. A higher porous block results in increase in the flow resistance so that the pressure drop is increased. An almost linear increase of the pressure drop with the width of the porous block can be seen from Fig. 4-26(b).

(a)

Fig. 4-25 Normalised pressure drop variation with (a) Darcy number for $Re = 100$, (b) Reynolds number for $Da = 10^{-4}$, $\Lambda_H = 16.5$ at $a/H = 0.8$, $w/H = 0.2$ and $k_e/k_f = 1$

Fig. 4-26 Normalised pressure drop variation with $Re = 100$, $Da = 10^{-4}$, $\Lambda_H = 16.5$, $k_e/k_f = 1$ (a) height with $w/H = 0.2$ (b) width with $a/H = 0.8$

4.6 SUMMARY

A numerical procedure for the study of fluid flow and heat transfer in channels with both open and porous regions is developed in this chapter. The capability of the codes is validated and demonstrated by a variety of 2D cases. The study of fluid flow and heat transfer in the channel with staggered porous blocks

reveal that the heat transfer performance increases with both the increase of the porous block height and width. The results can be used to establish guidelines for the design of different configurations of graphite foams in the experiments. With the 2D code, a 3D code is then developed. The capability of the 3D code will be demonstrated in the next chapter.

CHAPTER 5 CASE STUDIES IN A 3D DOMAIN FOR SINGLE-PHASE FLOW AND HEAT TRANSFER

In Chapter 4, the capability of the present numerical model is demonstrated with various 2D cases. In this chapter, the developed 2D numerical procedures are extended to three dimensions. Such an extension is straight forward and is performed in a dimension-by-dimension manner. No new additional formulation or numerical technique is involved. The capability of the developed 3D model is showcased via three examples, *viz.* (1) flow and heat transfer in block graphite foam, (2) flow and heat transfer in zigzag graphite foam and (3) flow and heat transfer in baffle graphite foam. The foams are named after their configurations. These configurations will be given in a later section. For ease of reference, they are referred to as BLK foam, ZZG foam and BAF foam hereinafter.

5.1 RELEVANT RELATIONS

In the energy equations, the LTNE model is adopted due to the large difference of thermal conductivities between the solid and fluid phases. Additional relationships are needed for the empirical parameters involved in the energy equations.

5.1.1 Effective Thermal Conductivity

The effective thermal conductivity is the thermal conductivity of the porous media which considers its pore structures as well as the saturated material in the

pores. It is highly dependent on the microstructures of the porous media. In order to calculate the effective thermal conductivity of the graphite foam, a unit cell model is proposed to represent the microscopic geometry of the graphite foam. Figure 5-1(a) shows the pore structure of a typical graphite foam manufactured by Poco Graphite, Inc., USA (Klett, 2000) and observed under a scanning electron microscope (SEM). The foam is composed of mainly spherical pores connected to each other by small openings. These small openings have different shapes with most of them being nearly circular. The diameters of these openings vary with the main pores. Even in the same main pores, the diameters of the openings would be different. Due to the uncontrollable manufacturing process in forming the pores of the graphite foams, the number of openings varies from pore to pore. Such uncertainties in the shape of the microstructure of graphite foam call for a flexible unit cell model which could represent the majority of the pores with slight changes to its structure. The model focuses mainly on this aspect.

In order to eliminate the complexity in analysing heat conduction in the curved surface of the spherical pores in a graphite foam, the proposed model assumes a plane surface instead. This gives a cube which is used to depict the main pores of the graphite foam. Figure 5-1(b) shows the unit cell model proposed in the current work. Spheres with centres located at each corner of the cube are used to cut the cube to form the small openings shown in Fig. 5-1(a). The diameters of the spheres are measured from the small openings in the SEM micrographs of graphite foams. These diameters can be different from each other. The number of spheres which cut the cube is determined from a particular pore. With these alterable parameters, this unit cell model can represent the majority

of the pores in graphite foams which are assumed to be spheres. This makes the current model more flexible than existing models in the literature (Klett *et al.*, 2004; Yu *et al.*, 2006a; Tee *et al.*, 2008).

To introduce simply the derivation of the effective thermal conductivity, the current work considers only the case with eight openings located at each corner of the cube as shown in Fig. 5-1(b). The diameters of the spheres used to cut the cube to form the openings are assumed to be the same.

Fig. 5-1 Pore structures of graphite foam (a) under SEM (b) proposed unit cell

It is assumed that the void space of the unit cell model is saturated with fluid. The side length of the proposed unit cell can be determined by the volume of the microstructure of pores in the graphite foam. The mathematical expression for calculating the side length a is

$$\frac{1}{6} \pi d_p^3 = a^3 \quad (5-1)$$

The thickness of the proposed unit cell model is determined by the porosity of the graphite foam. It is calculated as

$$\varepsilon = \frac{V_{void}}{V} = \frac{a^3 - 6 \left(a^2 - \frac{1}{4} \pi d_c^2 \right) t}{a^3} \quad (5-2)$$

where a and t are the side length and the thickness of the cube, respectively. d_p is the pore diameter of the porous graphite foam and d_c is the diameter of the spheres which is used to cut the cube to form the openings.

With the assumption of one-dimensional heat conduction in the direction where heat is imposed, the thermal resistance of the proposed model is obtained by considering the parallel or series resistances. The effective thermal conductivity can be derived from the analogy between thermal and electrical resistances. If heat is imposed at the left plane as shown in Fig. 5-1(b), the equivalent thermal circuit will consist of three parallel components as shown in Fig. 5-2. The overall effective thermal conductivity is then given by

$$k_e a = k_{eu} t + k_{em} \cdot (a - 2t) + k_{eb} t \quad (5-3)$$

where k_{eu} , k_{em} and k_{eb} are the effective thermal conductivities for the upper, middle and bottom portions, respectively. k_{eu} is equal to k_{eb} since the geometries of the upper and bottom portions are identical.

Fig. 5-2 (a) Upper part; (b) middle part and (c) bottom part of the proposed unit cell

5.1.1.1 Effective Thermal Conductivity of Upper Portion

Figure 5-3 gives the schematic diagram of the heat conduction path for the upper portion. It is divided into three layers which are parallel to each other. In layers 1 and 3, there are both solid and fluid phases. The effective thermal conductivity of the upper portion is

$$ak_{eu} = \frac{d_c}{2} k_{eu1} + (a - d_c) k_{eu2} + \frac{d_c}{2} k_{eu3} \quad (5-4)$$

where k_{eu1} , k_{eu2} and k_{eu3} are the effective thermal conductivities for layers 1, 2 and 3, respectively. With similar geometries for layers 1 and 3, it is obvious that $k_{eu1} = k_{eu3}$. k_{eu2} is equal to the solid phase thermal conductivity as there is only solid phase in layer 2.

Fig. 5-3 Heat conduction path for the upper part

The schematic diagram of the heat conduction path in layer 1 of the upper portion is shown in Fig. 5-4. The fluid and solid phases are in series. Taking a small element in this layer, the effective thermal conductivity in $d\delta$ is given by

$$\frac{a}{k_{eu1} d\delta} = \frac{2l_{eu1f}}{k_f d\delta} + \frac{l_{eu1s}}{k_s d\delta} \quad (5-5)$$

where

$$l_{eu1f} = \frac{d_c}{2} \sin \theta_u \quad (5-6)$$

$$l_{eu1s} = a - d_c \sin \theta_u \quad (5-7)$$

$$d\delta_{eu1i} = \frac{d_c}{2} \sin \theta_u d\theta_u \quad (5-8)$$

Fig. 5-4 Heat conduction path for layer 1 in the upper part

Integrating Eq. (5-5) over θ_u gives

$$k_{eu1} = \int_0^{\theta_u} \frac{k_s \sin \theta_u}{\gamma_{ad} \sin \theta_u (\lambda - 1) + 1} d\theta_u \quad (5-9)$$

where $\lambda = k_s/k_f$ and $\gamma_{ad} = d_c/a$.

Substituting Eq. (5-9) into Eq. (5-4), the effective thermal conductivity of the upper portion becomes

$$k_{eu} = \gamma_{ad} \int_0^{\theta_u} \frac{k_s \sin \theta_u}{\gamma_{ad} \sin \theta_u (\lambda - 1) + 1} d\theta_u + (1 - \gamma_{ad}) k_s \quad (5-10)$$

The integration of Eq. (5-10) will be given later.

5.1.1.2 Effective Thermal Conductivity of Middle Portion

The middle portion of the unit cell model is divided into three sections which are connected in series along the heat flow direction. These three sections are numbered as shown in Fig. 5-2. Section 2 is like a sandwiched structure where the fluid phase is clamped by two planes. As section 1 is similar to section 3, the effective thermal conductivity of the middle portion is given by

$$\frac{a}{k_{em} A} = \frac{2t}{k_{em1} A} + \frac{a - 2t}{k_{em2} A} \quad (5-11)$$

where k_{em1} and k_{em2} are the effective thermal conductivities of sections 1 and 2 of the middle portion, respectively.

For section 1, the solid and fluid phases are parallel to each other, and hence, its effective thermal conductivity is given by

$$k_{em1}A_{em1} = k_{em1s}A_{em1s} + k_{em1f}A_{em1f} \quad (5-12)$$

where A_{em1s} and A_{em1f} are the heat transfer areas for the solid and fluid phases, respectively. These are given by

$$A_{em1s} = A_{em1} - A_{em1f} = a(a-2t) - \frac{d_c^2 \theta_m}{2} + td_c \sin \theta_m \quad (5-13)$$

$$A_{em1f} = \frac{d_c^2 \theta_m}{2} - td_c \sin \theta_m \quad (5-14)$$

Introducing the dimensionless parameters of $\beta_{ad} = d_c/(a-2t)$ and $\beta_{at} = t/(a-2t)$ and substituting Eqs. (5-13) and (5-14) to Eq. (5-12), the effective thermal conductivity of either section 1 or 3 becomes

$$k_{em1} = k_s \left[1 - \frac{\theta_m}{2} \gamma_{ad} \beta_{ad} + \gamma_{at} \beta_{ad} \sin \theta_m \right] + k_f \left(\frac{\theta_m}{2} \gamma_{ad} \beta_{ad} - \gamma_{at} \beta_{ad} \sin \theta_m \right) \quad (5-15)$$

In the sandwiched structure of section 2, the two planes and the fluid phase are parallel to each other. The effective thermal conductivity of section 2 is calculated by

$$k_{em2}a(a-2t) = k_f \left[a(a-2t) - 2t(a-2t) \right] + 2tk_{em2s}(a-2t) \quad (5-16)$$

With the aid of the dimensionless parameter, $\gamma_{at} = t/a$, Eq. (5-16) becomes

$$k_{em2} = k_f [1 - 2\gamma_{at}] + 2\gamma_{at}k_{em2s} \quad (5-17)$$

The calculation procedure for k_{em2s} is the same as that of the upper layer. Its mathematical expression is

$$k_{em2s} = \zeta_{ad} \int_0^{\theta} \frac{k_s \sin \theta_m}{\zeta_{ad} \sin \theta_m (\lambda - 1) + 1} d\theta_m + (1 - \zeta_{ad}) k_s \quad (5-18)$$

where $\zeta_{ad} = d_c/(a-2t)$. Substituting Eqs. (5-15) and (5-17) into Eq. (5-11), the effective thermal conductivity of the middle portion is given by

$$k_{em} = \frac{k_{em1}}{2\gamma_{at} + (1 - 2\gamma_{at})\lambda_m} \quad (5-19)$$

where $\lambda_m = k_{em1}/k_{em2}$.

5.1.1.3 Overall Effective Thermal Conductivity of the Unit Cell

The overall effective thermal conductivity of the unit cell can be obtained by substituting Eqs. (5-19) and (5-10) into Eq. (5-3). Before this, the analytical solutions of Eqs. (5-10) and (5-18) have to be obtained. The integration of the R.H.S. of Eq. (5-10) gives

$$\gamma_{ad} \int_0^{\pi/2} \frac{k_s \sin \theta_u}{\gamma_{ad} \sin \theta_u (\lambda - 1) + 1} d\theta_u = k_s \frac{1}{(\lambda - 1)} \int_0^{\pi/2} \left(1 - \frac{1}{\gamma_{ad} \sin \theta_u (\lambda - 1) + 1} \right) d\theta_u \quad (5-20)$$

$$+ (1 - \gamma_{ad}) k_s$$

Introducing $m_\gamma = \gamma_{ad}(\lambda - 1)$, the results of Eq. (5-20) is dependent on the value of m_γ .

For $m_\gamma^2 < 1$

$$k_{eu} = \frac{\pi k_s}{2(\lambda - 1)} - \frac{k_s}{(\lambda - 1)} \frac{2}{\sqrt{1 - m_\gamma^2}} \tan^{-1} \sqrt{\frac{1 - m_\gamma}{1 + m_\gamma}} + (1 - \gamma_{ad}) k_s \quad (5-21)$$

For $m_\gamma^2 > 1$

$$k_{eu} = \frac{\pi k_s}{2(\lambda-1)} - \frac{k_s}{(\lambda-1)} \frac{1}{\sqrt{m_\gamma^2-1}} \ln\left(m_\gamma + \sqrt{m_\gamma^2-1}\right) + (1-\gamma_{ad})k_s \quad (5-22)$$

Similarly, the integration of Eq. (5-18) results in the following expressions:-

For $m_\zeta^2 < 1$

$$k_{em2s} = k_f (1-2\gamma_{at}) + 2\gamma_{at} \left[\frac{\pi k_s}{2(\lambda-1)} - \frac{k_s}{(\lambda-1)} \frac{2}{\sqrt{1-m_\zeta^2}} \tan^{-1} \sqrt{\frac{1-m_\zeta}{1+m_\zeta}} + (1-\zeta_{ad})k_s \right] \quad (5-23)$$

For $m_\zeta^2 > 1$

$$k_{em2s} = k_f (1-2\gamma_{at}) + 2\gamma_{at} \left[\frac{\pi k_s}{2(\lambda-1)} - \frac{k_s}{(\lambda-1)} \frac{1}{\sqrt{m_\zeta^2-1}} \ln\left(m_\zeta + \sqrt{m_\zeta^2-1}\right) + (1-\zeta_{ad})k_s \right] \quad (5-24)$$

where $m_\zeta = \zeta_{ad} (\lambda-1)$.

Substituting these equations into Eq. (5-3), the overall effective thermal conductivity for the unit cell can be obtained.

5.1.1.4 Results and Discussion

The diameter of the main pores and the openings were measured by “Image-Pro” software based on SEM micrographs. With built-in functions, “Image-Pro” can recognise clusters of brighter pixels as cells so that the diameter of every cell can be calculated. To reduce further the randomness of the pore diameter measurements, three samples were measured and the average value was used as the pore diameter. The average pore diameter of 75% porosity foam is about 310 μm which is close to the manufacturer’s data of 350 μm . The average diameter of the small openings is 160 μm .

Figure 5-5 shows the comparison of the overall effective thermal conductivity obtained by the current model and the experimental data Klett *et al.* (2004) for graphite foams and the model of Yu *et al.* (2006). The increase of the porosity decreases the volume occupied by the solid phase, resulting in a decrease of the effective thermal conductivity. In general, the current model gives good agreement with Klett *et al.*'s experimental data as well as the model from Yu *et al.* (2006).

Fig. 5-5 Variations of effective thermal conductivity with porosity

5.1.2 Convective Heat Transfer

The interior surface area to volume ratio quantifies the internal surface area in the contribution of the convection heat transfer in the pores level. Based on the current unit cell model, the interior surface area to volume ratio α_{sf} is given by

$$\alpha_{sf} = \frac{1}{a^3} \left(6a^2 - \frac{3}{2} \pi d_c^2 \right) \quad (5-25)$$

The interfacial heat transfer coefficient h_{sf} in the source term is obtained from the correlation proposed by Zukauskas (1987) for cylinders in cross flow i.e.

$$\text{Nu}_{sf} = \frac{h_{sf} d_p}{k_f} = m \text{Re}^n \text{Pr}^{0.36} \quad (5-26)$$

where m and n are constants. Klett *et al.* (2002) obtained the two constants through fitting of their experimental data. These are 0.0158 and 0.7225, respectively for air flow through graphite foam with 75% porosity.

5.1.3 Boundary Conditions

The LTNE model has two energy equations to describe the solid and liquid temperatures, respectively. For each equation, boundary conditions are required. Different models have been proposed for the allocation of the heat fluxes between the solid and fluid phases at the boundary. A detailed literature review on this topic is given in Chapter 2. In the current study, the boundary conditions for the two phases at the heated wall are taken as (Amiri *et al.*, 1995)

$$-(k_{fe} \frac{\partial T_f}{\partial z} + k_{se} \frac{\partial T_s}{\partial z}) = q'', T_s = T_f \quad (5-27)$$

With these supplementary correlations provided, the governing equations are closed and the numerical simulation can be realised.

5.2 FLOW AND HEAT TRANSFER IN BLOCK (BLK) GRAPHITE FOAM

This problem is chosen to study the fluid flow and heat transfer in solid block foams. It is chosen for a two-fold purpose *viz.* (1) to validate the current 3D numerical model and (2) to serve as a prototype study for the designed configurations of ZZG and BAF foams.

5.2.1 Problem Specification

The geometry of the simulation domain is depicted in Fig. 5-6. The channel is filled with BLK graphite foam of dimensions of 50 mm (L) $\times$ 50 mm (W) $\times$ 25 mm (H). Fluid flows into the channel at the inlet with a uniform velocity and constant temperature. A constant heat flux is imposed at the substrate of the graphite foam.

Fig. 5-6 Schematic diagram of BLK foam domain

5.2.2 Numerical Details

Only half of the geometry is simulated since the domain is symmetrical. The test sample is graphite foam with porosity of 75%. The properties used in the program are listed in Table 5-1. The parameters with the superscript ‘*’ are the measured data from experiments which will be discussed separately in Chapter 6.

Grid-independence study is carried out based on meshes of $20 \times 10 \times 10$ CVs, $40 \times 20 \times 20$ CVs and $80 \times 40 \times 40$ CVs. The results show that the mesh with $40 \times 20 \times 20$ CVs is sufficient to achieve spatial-independent solutions. The convergence criterion for steady state is that the maximum temperature changes

for both the fluid and solid phases should be less than 10^{-5} between successive iterations.

Table 5-1 Characteristic properties of graphite foams

ε	d_p^* (mm)	K^* (10^{-10} m^2)	C_E^*	α_{sf} (m^2/m^3)	k_{se} (W/m·K)	k_{fe} (W/m·K)
0.75	0.310	7.74	0.063	8216	16.20	0.023

5.2.3 Results and Discussion

The dimensionless parameter x/D_h is introduced to express the non-dimensional locations of the temperature measurement along the simulated configuration. D_h is the hydraulic diameter of the channel which is defined as

$$D_h = \frac{4A_c}{P} \quad (5-28)$$

where A_c and P are the cross-sectional area and perimeter of the channel, respectively. The pore diameter based Reynolds number and local Nusselt number are defined as

$$\text{Re}_p = \frac{\rho u d_p}{\mu} \quad (5-29)$$

$$\text{Nu}_x = \frac{q}{A_c (T_x - T_{in})} \frac{d_p}{k_f} \quad (5-30)$$

where q is the heating power. T_{in} and k_f are the inlet temperature and fluid thermal conductivity, respectively.

The average pressure in the cross section perpendicular to the flow direction is defined as

$$\bar{p} = \frac{1}{A_c} \int_0^W \int_0^H p dy dz \quad (5-31)$$

The pressure drops in the channel are calculated by the difference between the average pressures across the graphite foams and is expressed as

$$\Delta p = \bar{p}_{in} - \bar{p}_{out} \quad (5-32)$$

5.2.3.1 Validation Test

With symmetrical boundary conditions enforced along the width direction in the 3D domain (fluid flows along the length-wise (x) direction and heating occurs along height-wise (z) direction), the results from the present 3D code should reduce to its 2D counterpart. Such a test is used to validate the 3D numerical model as there is no existing work on the same topic to validate the current results. This comparison is given in Fig. 5-7. The local Nusselt number obtained from the 2D and 3D simulations are found to overlap at different locations. The validity of the 3D code is therefore demonstrated.

Fig. 5-7 Comparison of 2D and 3D results

5.2.3.2 Velocity Field

Figure 5-8 shows the velocity profile along the z -axis at different locations along flow direction. Only the profile in the centre of the width direction is

chosen. Generally, porous media with low permeability would result in a nearly uniform velocity. This is exemplified by the velocity profiles at the entrance region and outlet boundary. As seen from the figure, the velocity becomes nearly uniform in a very short length when the fluid flows into the domain. The thickness of the momentum boundary layer in the porous media is of the same order as $(K/\varepsilon)^{0.5}$ (Vafai and Tien, 1981). Under such a condition, an extremely thin boundary layer is formed when the fluid approaches the wall of the graphite foam because the permeability is rather small. The boundary effect plays an insignificant role in the overall flow consideration.

Fig. 5-8 Velocity profiles at different locations

5.2.3.3 Temperature Field

Figures 5-9(a) and 5-9(b) present the temperature distributions of the solid and fluid phases on the surface of the graphite foam at $Re_p = 40$ under heat flux of 0.8 W/cm^2 . The temperature contours of the solid phase are different from those of the fluid phase due to the different boundary conditions set at $x = 0$. Although the thermal conductivity difference between the solid graphite foam and the fluid is large, it does not result in a significant temperature difference

between the solid and fluid phases. This is due to the extremely large contact surface area at the pore level which enhances the convection significantly.

Fig. 5-9 (a) Solid and (b) fluid surface temperature distribution at $Re_p = 40$ under $q'' = 0.8 \text{ W/cm}^2$

5.3 FLOW AND HEAT TRANSFER IN ZIGZAG (ZZG) GRAPHITE FOAM

5.3.1 Problem Specification

Figure 5-10(a) shows the schematic diagram of the ZZG foam. This configuration is designed in such a way that part of the fluid can flow through the channel without passing through the porous foams. Several foam walls are

aligned perpendicular to the inlet flow direction. Gaps in each of the foam wall allow the fluid to flow through. The slots between successive foam walls are connected together through the gaps. The thickness of the foam walls is determined based on the parametric study in Chapter 4. For the experimental study, the manufacturing constraints on thin foam walls are also considered. A 4-mm-thick foam base is designed to conduct heat effectively from the heat surface to the foam walls above. Figure 5-10(b) shows the detailed dimensions of the ZZG foam. With this design, it is hoped that the pressure drop would be reduced without much decrease of the heat transfer performance compared with the BLK foam. All the dimensions are in millimetres.

Fig. 5-10 (a) Schematic diagram of ZZG foam (b) dimensions of ZZG foam

5.3.2 Numerical Details

Numerical simulations are performed in a domain larger than the actual dimensions of ZZG graphite foam. An empty duct with length of 25 mm located before and after the ZZG foam to remove both the inlet and outlet influence on the flow in the domain of interest. The length was chosen based on several trial and error tests. Fluid flows into the channel at the inlet with a uniform velocity profile and a constant temperature. A constant heat flux is imposed at the substrate of the graphite foam. The remaining walls of the channel are adiabatic.

Both the velocity and temperature profiles are obtained under meshes of $70 \times 24 \times 18$ CVs, $106 \times 36 \times 26$ CVs and $159 \times 54 \times 39$ CVs. The results show that there are insignificant differences in the velocity and temperature distribution for meshes of $106 \times 36 \times 26$ CVs and $159 \times 54 \times 39$ CVs. Therefore, a mesh with $106 \times 36 \times 26$ CVs gives the spatial-independent solution. The iteration is stopped when the relative maximum errors of both velocity and temperature between successive iterations are smaller than 10^{-5} .

5.3.3 Results and Discussion

5.3.3.1 Velocity Field

For the periodic configurations in the ZZG foams, only the essential features in the first flow patterns are presented. Figures 5-11(b) ~ 5-11(e) give the velocity profiles according to the selected locations shown in Fig. 5-12(a). The velocity profile before the fluid flows into the ZZG foam is shown in Fig. 5-11(b). Most of the fluid, referred to as the main flow henceforth, flows through the gap due

to the presence of the first foam wall in front. It results in the velocity difference between the orientation of the gap and the first foam wall. This velocity difference is higher as the fluid flows into the first foam wall as shown as Fig. 5-11(c). Most of the fluid flows through the gap due to the high flow resistance of the foam wall. Figure 5-11(d) shows the velocity profile in the slot between the first and second foam walls. The main flow is forced to flow to the other side of the structure where there is a slot before it hits the second foam wall. The inertia of the fluid flowing through the first gap is sufficiently high such that the fluid changes direction slowly. This results in the dominance of the main flow at the two ends along the y -direction. Figure 5-11(e) shows the velocity profile in the second foam wall. Similar to the velocity profile in Fig. 5-11(c), the velocity in the gap is larger than that in the foam wall. However, this velocity difference between the slot and the foam wall is not as large as in the first foam wall. The inertial force in the previous main stream is so large that the fluid has difficulty changing its direction in such a short slot before it hits the second foam wall. After that, the main stream flows in a zigzag manner guided by the gaps and the foam walls.

It is noted from Figs. 5-11(c), 5-11(d) and 5-11(e) that the velocity near the foam base is slightly higher than that in the upper locations. In order to present this phenomenon clearly, the enlarged velocity vectors in the ZZG foam around the foam base at the middle plane of the width direction are shown in Fig. 5-12. The solid lines represent the joint section of the foam wall and foam base. An interesting finding is observed. The fluid is forced to flow downward to the foam base due to the foam wall in front. However, it is forced to flow upward from the foam base to the slot after passing through the foam wall. Therefore,

CASE STUDIES IN A 3D DOMAIN FOR SINGLE-PHASE FLOW AND HEAT TRANSFER

the fluid velocity near the interface between the foam base and the slot increases slightly after the fluid penetrates the foam wall.

(a)

(b)

(c)

Fig. 5-11 Dimensionless velocity profiles in ZZG foam at (a) representative locations (b) velocity profile before the 1st foam wall; (c) velocity profile in the 1st foam wall; (d) velocity profile in the slot between the 1st and 2nd foam wall; (e) velocity profile in the 2nd foam wall

Fig. 5-12 Velocity in xz plane at $y/D_h = 0.12$ parallel to the flow direction

5.3.3.2 Temperature Field

The solid and fluid temperature distributions in the ZZG foam are shown in Figs. 5-13(a) and 5-13(b), respectively. The slices perpendicular to the flow direction are chosen to represent different locations of the configuration. The solid temperature in the gaps or slots is set to be the same as the fluid temperature. In ZZG, the temperature difference between the solid and fluid phases is increased compared with the solid block foam. This is caused by the reduced heat transfer area at the pore level between the solid and fluid phases. Generally, the solid phase temperature is much higher than its fluid counterpart. However, inside of foam walls, the fluid temperature increases because of the convection heat transfer at the interface between the solid and fluid phases.

(a)

(b)

Fig. 5-13 (a) Solid and (b) fluid temperatures at selected slices in ZZG foam

5.4 FLOW AND HEAT TRANSFER IN BAFFLE (BAF) GRAPHITE FOAM

5.4.1 Problem Specification

Figure 5-14(a) shows the schematic diagram of BAF foam. To minimise the surface area on the heat transfer, the BAF configurations are determined based on the fact that the designed ZZG and BAF foams should possess approximately the same total wall surface area. The detailed dimensions are shown in Fig. 5-14(b). All the dimensions are in millimetres.

Fig. 5-14 (a) Schematic diagram of BAF foam (b) dimensions of BAF foam

5.4.2 Numerical Details

The simulation of the BAF foam is also carried out in a larger domain with two empty ducts each of length 25 mm located before and after the BAF foam.

Given the symmetrical geometry of BAF foams, only half of the domain is simulated.

Grid-independence tests were carried out on meshes of $68 \times 25 \times 16$ CVs, $102 \times 38 \times 24$ CVs and $153 \times 57 \times 36$ CVs. The results show that a mesh size with $102 \times 38 \times 24$ CVs is sufficient to achieve grid-independent solutions. The steady-state solution is achieved when the two successive iterations for the maximum changes of both velocity and temperature is less than 10^{-5} .

5.4.3 Results and Discussion

5.4.3.1 Velocity Field

Four chosen representative locations of the velocity profiles in BAF foam are shown in Fig. 5-15(a). Fig. 5-15(b) shows the velocity profile at $x/D_h = -0.15$. It is the position in the added empty channel where the fluid has not entered the BAF foams. The velocity profile is slightly affected by the first foam wall and the gap. The low permeability of the foam wall results in a large flow resistance. The fluid has difficulty penetrating the foam wall. Therefore, it is forced to flow through the gap. As fluid flows into the first foam wall i.e., $x/D_h = 0.05$ as shown in Fig. 5-15(c), the main flow passes through the gap and only a small portion flows through the foam walls. The velocity profile in the slot between the first and second foam walls is shown in Fig. 5-15(d). The main flow is divided into three branches in an “E” manner guided by the gaps in the second foam wall. The main flow shown in Fig. 5-15(c) is distributed into three streams. When the fluid flows into the second foam wall as shown in Fig. 5-15(e), the flow of the middle stream decreases as there is a foam wall directly in

CASE STUDIES IN A 3D DOMAIN FOR SINGLE-PHASE FLOW AND HEAT TRANSFER

front of the gap. Guided by the foam wall, most of the fluid flows sideward into the next two gaps. The flow in the two side branches becomes dominant. The reduction of the flow rate in the foam walls decreases the heat transfer. Convection heat transfer occurs mainly in the foam wall surface rather than inside the foam walls. The slots and gaps above the foam base for both ZZG and BAF foams change the flow characteristics in the BLK foam which also modified the heat transfer. For the foam base, the heat absorbed by the solid ligament is mainly transferred by conduction to its ligament, rather than being convected away by the fluid directly. This is not favourable for heat transfer. However, the arrangements of the alternate gaps accelerate the velocity significantly, and fluid mixing is enhanced which is favourable to heat transfer.

(a)

(b)

Fig. 5-15 Dimensionless velocity profiles in BAF foam at (a) representative locations; (b) velocity profile before the 1st foam wall; (c) velocity profile in the 1st foam wall; (d) velocity profile in the slot between the 1st and 2nd foam wall; (e) velocity profile in the 2nd foam wall

5.4.3.2 Temperature Field

Figure 5-16 shows the solid and fluid temperature distributions in BAF foam at selected slices for $Re_p = 32$. These selected slices represent different locations of the slots. Both the solid and fluid temperatures increase progressively along the flow direction. The solid temperature is much higher than that of the fluid temperature at the same location which indicates poor heat transfer in the internal pores of the graphite foam.

Fig. 5-16 (a) Solid and (b) fluid temperatures in the selected slices in the BAF foam

5.5 SUMMARY

In this chapter, the proposed 3D model is first validated against its corresponding 2D model and then applied to three different problems *viz.* BLK, ZZG and BAF foams. These problems involve different configurations of graphite foam which are aimed at reducing the pressure drop. The velocity and temperature distributions in the three configurations are discussed. The temperature difference between the solid and fluid phases is also presented. Compared with BLK foam, the velocity fields in ZZG and BAF foams are affected by the location of the gaps and slots significantly. The temperature difference between the solid and fluid phases is significant in ZZG and BAF foams. The comparison of these three configurations on both the heat transfer and pressure drop will be discussed in the next chapter.

CHAPTER 6 EXPERIMENTAL STUDIES FOR SINGLE-PHASE FLOW AND HEAT TRANSFER

In this chapter, experimental studies are carried out based on the three configurations of graphite foams mentioned in Chapter 5, i.e. BLK, ZZG and BAF foams. The heat transfer performance and pressure drop of these configurations are compared with the numerical findings described previously.

6.1 TEST SPECIMENS AND EXPERIMENTAL SETUP

6.1.1 Different Configurations of Graphite Foams

The material used in the experimental studies is graphite foam which is licensed by ORNL (Klett, 2000). Three different configurations are shown in Fig. 6-1. These configurations have the same dimensions as those studied numerically in Chapter 5. These foams are fabricated by Electron Discharge Machining (EDM) in the Manufacturing Process Laboratory in NTU.

Fig. 6-1 Photographs of (a) BLK (b) ZZG and (c) BAF foams

6.1.2 Test Section

The experimental facility is shown in Fig. 6-2(a). Different configurations of the graphite foams are placed in the duct. The duct is fabricated of Teflon to

EXPERIMENTAL STUDIES FOR SINGLE-PHASE FLOW AND HEAT TRANSFER

reduce heat loss to the atmosphere. Two pressure taps are located before and after the graphite foams to measure the pressure drop across the foams. Air flow is provided by an auto-balance compressor. A hot-wire sensor is placed in the middle of the duct before the fluid flows into the test section to measure its velocity. A data acquisition system is used to gather all the data measured by the different sensors. A close-up view of the test section and its photograph are shown in Figs. 6-2(b) and 6-2(c), respectively. A film heater powered by a DC power supply is installed at the bottom of the section. A thin copper plate with ten narrow slots perpendicular to the flow direction is attached on the substrate of the graphite foams. The thermocouples are inserted into those slots to measure the local temperatures along the flow direction. Highly conductive thermal grease ($k = 8.5 \text{ W/m}\cdot\text{K}$) manufactured by Arctic Silver, Inc., USA was applied as filling material to reduce the contact resistances across the interfaces of the film heater, copper plate and test sample.

1. Test section 2. Teflon channel 3. Hot-wire 4. Mesh
5. Pressure sensor 6. Data acquisition system
(a)

Fig. 6-2 (a) Experimental facility (b) cross-sectional view of test section (c) photograph of the test section

6.2 MEASURING SYSTEM

6.2.1 Test Sensors

Three types of measuring sensors, namely, thermocouples, pressure sensors and single hot-wire sensor are used to measure the temperature, pressure drop and flow velocity in the test section, respectively.

Ten fast response type K “Omega” thermocouples are arranged perpendicular to the air flow direction. These thermocouples have wire diameters of 0.254 mm (0.010”). By clamping the channel tightly and filling the interface with highly conductive thermal grease, the temperatures of the graphite foam surface

and the copper plate at the bottom of the channel are assumed to be same in the measurements.

The differential pressure transducer used in the current study is VALIDYNE DP15. It features field-replaceable sensing diagrams so that the full scale can be changed according to the range of the measurement. The rugged construction of the DP15 makes it highly resistant to shock and vibration. To obtain standard analogue signal outputs, the DP15 pressure sensor is connected to the VALIDYNE CD15 by a four-conductor shielded cable. The CD15 is a carrier demodulator which supplies a 5 kHz excitation to the transducer and provides zero balance and span adjustments. The standard output signal option of the CD15 is ± 10 VDC with accuracy of $\pm 0.25\%$ of full-scale.

The velocity measuring system consists of a single hot-wire sensor (DANTEC 55P11) and a constant-temperature anemometer (DANTEC MiniCTA 54T30). The 55P11 single hot-wire sensor probe permits measurement from 0.05 to 300 m/s at a maximum ambient temperature of 150°C while the 54T30 is a versatile anemometer that can be used with hot-wire and fibre-film probes through a BNC connector. The bandwidth of the velocity measurement is up to 10 kHz and the overheat setup and signal conditioning can be performed via DIP switches and jumpers inside the constant-temperature anemometer.

6.2.2 Data Acquisition System

The data measured by thermocouples, pressure transducer and hot-wire sensor are linked to a data acquisition system which consists of a signal board (NI CB-68LP), analogue and digital I/O boards (NI PCI-6224) and an operating software (NI DAQmx 4.0) within a workstation. The values of the temperature,

pressure drop and velocity measured by the sensors are input into the data acquisition board through different channels on the signal board. A 32 channel data cable was used to connect the signal and I/O boards.

The National Instruments (NI) PCI-6224 is a data acquisition card with 16-bit, 250 kS /s analogue input and 1 MHz digital lines for applications including data logging and sensor measurements. NI PCI-6224 contains 32 channels inputs and integrates a cold junction compensation function which adapts to measure the temperature, velocity and pressure in the present experiments. The analogue data are transferred by the NI RC68-68 ribbon cable connected to the CB-68LP and PCI-6224 boards. The analogue inputs of PCI-6224 have calibration circuitry to correct gain and offset errors. It can be used to minimise AI and AO errors caused by time and temperature drift at run time. An internal reference ensures high accuracy and stability over time and temperature changes.

6.3 EXPERIMENTAL PROCEDURE

6.3.1 Calibration of Sensors

Before the commencement of the experiments, the thermocouples, pressure sensor and hot-wire sensor are calibrated carefully. Ten thermocouples are calibrated together with the mini-volt amplifiers. A thermal-calibrator with an adjustable pre-set temperature was used as a heat source. All thermocouples are dipped inside a cylindrical chamber in the thermal-calibrator, where they are heated up to a prescribed temperature. The other ends of thermocouples are connected to the different channels of the signal board (NI CB-68LP). The analogue data of the enlarged voltages for the different temperatures provided by thermal-calibrator are collected by the data acquisition card. The NI PCI-

6224 is configured with an integrated thermocouple calibration function including cold junction compensation. Hence, the input analogue data of voltage are converted to standard temperatures. The calibration curve of the measured temperature against corresponding output standard temperature and the fitting equation is shown in Fig. 6-3.

Fig. 6-3 Calibration curve of the thermocouple

The pressure transducer is calibrated with an inclined manometer. During calibration, the manometer is set at the “Top Inclined” position, with a scale multiplier of 0.2. The liquid level is adjusted to 0 kN/m² to eliminate any offset when the tube is horizontal. Initial pressure readings are obtained based on the pressure difference across the test section. Air is provided by an air compressor. By adjusting the “Zero & Span” Control in the demodulator, the input pressure is gradually reduced to zero and the zero setting of the demodulator is rechecked and locked. Voltages from the Demodulator and the corrected manometer’s pressure readings are carefully recorded to facilitate the plotting of the calibration curve. The calibration curve of pressure against corresponding output voltage is shown in Fig. 6-4.

Fig. 6-4 Calibration curve of pressure sensor

The velocity calibration equipment consists of three components: hot wire sensor, miniCTA and air velocity calibrator. These velocities can be determined by measuring differential pressures across nozzles in the calibrator. To assemble the calibrator, the filter/regulator assembly, which includes a pressure regulator, valve, filter, hose and fitting, is attached to the main line of air supply. The rubber hose is connected to the inlet of the calibrator to generate air of various velocities, and the hot-wire probe is attached to the top of the flow-setting chamber and is carefully oriented. The voltage output of the anemometer and the pressure drop across the nozzle are measured first and the velocity is calculated based on the pressure results. The relationship between the velocity and pressure drop is

$$u = \sqrt{\frac{2(p_t - p_s)}{\rho_f}} \quad (6-1)$$

where u is the velocity, ρ_f is the density of the air and $p_t - p_s$ is the pressure drop between exit nozzle and atmosphere. Figure 6-5 shows the curve obtained for the velocity against the voltage output.

Fig. 6-5 Calibration curve of hot-wire

6.3.2 Experimental Procedures

To investigate the pressure drop and heat transfer performance of different configurations of graphite foams subjected to various heating powers, a series of experiments is conducted. During a typical experiment run for each configuration, the heating power is set to a prescribed value and the inlet velocity is varied by adjusting the valve installed on the supply line. The temperature readings are monitored every 3 minutes to ensure that steady state had been achieved. The steady state condition is assumed to be reached if the maximum temperature variation indicated in all the thermocouples is within 0.5 °C. This usually takes about 10 to 15 minutes depending on the flow velocity. The steady state velocity, pressure drop and temperatures are then recorded by the data acquisition board. The heating power is varied from 20 to 60 W. The same experimental procedure is repeated while the heating power is increased. The graphite foam in the channel is removed after completing the experiments for different heating powers. The entire system was cooled to the ambient temperature before another configuration was studied.

6.4 DETERMINATION OF PROPERTIES OF GRAPHITE FOAM

The pore diameters of the tested graphite foam are measured by Image-Pro software based on photographs obtained by an SEM. With built-in functions, Image-Pro can recognise clusters of brighter pixels as cells and hence, calculate the diameter of every cell. To reduce further the random effect of the pore diameter measurement, three samples were measured and the average value was used as the pore diameter. It was found that the average pore diameter of 75% porosity foam is 0.31 mm which is close to the manufacturer's data of 0.35 mm. Porosity is defined as the total void volume divided by the total volume including the solid matrix and void volume, which can be expressed as

$$\varepsilon = \frac{V_t - V_s}{V_t} \quad (6-2)$$

where V_t and V_s are the total volume and the total volume occupied by the solid phase, respectively. In this study, the porosity of the test graphite foam is measured by Ultrapycnometer 1000 which can measure the density and the volume of the solid components in the porous medium. For the tested graphite foams, the sample of dimensions 2 cm × 2 cm × 2 cm is measured. It is found that the solid density and volume are 2.13 g/cm³ and 2.24 cm³, respectively. The measured porosity of 0.72 is close to the manufacturer's data of 0.75.

The permeability K and inertial coefficient C_E of the graphite foams can be determined from a plot of the pressure drop versus the velocity. It is widely accepted that the pressure gradient in a porous medium is given by

$$\frac{dp}{dx} = -\frac{\mu}{K}u - \rho_f \frac{C_E}{\sqrt{K}}u^2 \quad (6-3)$$

Equation (6-3) can be simplified as

$$-\frac{\Delta p}{L} = Au + Bu^2 \quad (6-4)$$

where

$$A = \frac{\mu}{K}, B = \rho_f \frac{C_E}{\sqrt{K}} \quad (6-5)$$

where A and B are the constants to be determined. Δp is the pressure drop and L is the length of the graphite foam.

Figure 6-6 shows the experimental results of pressure drop per unit length versus the velocity for air flow across the 75% porosity graphite foams. A least-squares fit was performed to obtain the coefficients of A and B . The regression analysis showed a fit with a regression coefficient of $R^2 > 99.95\%$ where, R^2 reveals the efficiency and accuracy of the quadratic relationship. As seen from the fitting results, the values of A and B are 23257.5 and 2520.3, respectively. Therefore, K and C_E are 7.74×10^{-10} and 0.063, respectively.

Fig. 6-6 Pressure drop versus velocity for 75% porosity graphite foam

6.5 UNCERTAINTY ANALYSIS

The uncertainty of measured data can be classified into two groups: random uncertainties, which can be treated statistically; and systematic uncertainties, which cannot be treated in the same way. With careful experimentation, systematic uncertainty was minimised. The accuracies of the thermocouple and pressure transducer readings are within $\pm 0.5^\circ\text{C}$ and $\pm 0.25\%$ of full-scale, respectively. The accuracy of the velocity measured by the hot-wire anemometer is ± 0.01 m/s. The uncertainties of the measured quantities are calculated based on the measurement accuracy. Based on the assumption that the uncertainties of the measured data are independent and random with normal distribution, the uncertainties of the calculated parameters were determined by the method described by Taylor (1997). The uncertainties of Nu_{ave} is calculated to be 8.9%.

The estimation of uncertainties of K and C_E are based on the coefficients of A and B . It can be seen that the uncertainty of K is equal to the uncertainty of A ; however, the uncertainty of C_E is not only affected by B , but also by uncertainty of K . The error in the measurement of the velocity is given by ± 0.01 m/s; the error for the pressure sensor is $\pm 0.25\%$. With these information, the error in the value of A can be found. The same procedure is used to calculate the uncertainty of B . The uncertainties of K and C_E are determined to be 3.2% and 11.2%, respectively.

6.6 COMPARISON BETWEEN EXPERIMENTAL AND NUMERICAL RESULTS

6.6.1 Heat Transfer Performance

Figure 6-7 shows the numerical results of temperature distribution on the heated surface for different configurations of air flow at $Re_p = 50$. The dash lines represent the location of the foam walls. For BLK foam (Fig. 6-7a), the temperature increases linearly along the flow direction with the lowest and highest temperatures at about 31°C and 36°C, respectively. Due to the structured passages in the BAF and ZZG configurations, the temperatures increase progressively and the lowest and the highest temperatures of these configurations are 35°C and 45°C, respectively (Figs. 6-7b and 6-7c). Therefore, the temperature distribution in the BLK foam is more uniform than those in the BAF and ZZG foams. As affected by the different local flow fields, the temperature distributions in the structured foams deviate substantially from the BLK foam with higher temperature at the location where the flow changes its direction. A closer view shows that the ZZG foam has generally lower temperatures compared to the BAF foam. Hence, if only heat transfer is considered, the BLK foam is still better than both the BAF and ZZG foams since it has lower temperatures and a more uniform temperature distribution.

Fig. 6-7 Temperature distributions at the heated surface with $Re_p = 50$ in (a) BLK foam; (b) BAF foam; (c) ZZG foam

The average Nusselt number is calculated as

$$\text{Nu}_{ave} = \frac{qd_p}{k_f A_c L} \int_0^L \frac{1}{(T_x - T_{in})} dx \quad (6-6)$$

Figure 6-8 shows Nu_{ave} versus Re_p for different configurations of graphite. Overall, the numerical results agree fairly well with the experimental data. It is obvious that Nu_{ave} in the BLK foam is the highest among all the configurations. In the calculation of Nu_{ave} , Nu_x is used to obtain Nu_{ave} . Since Nu_x is inversely proportional to T_x , a high Nu_x corresponds to low substrate temperature. Hence, the highest Nu_{ave} in the BLK foam predicts the lowest substrate temperature among all the configurations. The BLK foam possesses the largest surface area at pore level which is favourable to convection heat transfer between the solid and fluid phases. In the ZZG and BAF foams, the velocity field is distorted by the design of the foam walls and gaps. Most of the fluid flows through the gaps. The flow rate in the graphite foams is dramatically reduced due to its low permeability. This weakens the convection heat transfer at the pore level compared with the BLK foam. The fluid is accelerated in the gap which improves the convection heat transfer between the solid and fluid phases. This improvement occurs only at the interface between the foam walls and the slots. It is observed from the velocity field (Figs. 5-11 and 5-15) that there is little fluid flow through the foam base in the ZZG and BAF foams. Therefore, the mode of heat transfer in the foam base is mainly conduction rather than conduction and convection. This is also not favourable for heat transfer. However, the designed slots and gaps in ZZG and BAF foams significantly reduced the pressure drop compared with the BLK foam. The ZZG foam has slightly better heat transfer than the BAF foam. Fluid flow in the ZZG foam is

more distorted compared with the BAF foam which increases fluid mixing. This increased fluid mixing enhances the convection heat transfer, at the expense of high pressure drop.

Fig. 6-8 Variations of Nu_{ave} with Re_p in different configurations of graphite foams

6.6.2 Pressure Drop

Figure 6-9 shows the experimental and simulated pressure drops at different inlet velocities in all the configurations of graphite foams. As expected, the pressure drop increases with the increase of the inlet velocity for all the foams. Excellent agreement was achieved between the experimental and numerical results for all the studied configurations of graphite foams. The BLK foam has the highest pressure drop among the three foams. With the increase of velocity, the pressure drop in the BLK foam increases significantly while the pressure drop in the designed ZZG and BAF foams reduces significantly. The pressure drop in the ZZG foam is higher than that in the BAF foam. For the BLK foam, the low permeability results in high flow resistance as the fluid flows through it. A large amount of the fluid flows through the designed gaps in the ZZG and BAF foams which reduce the pressure drops significantly. In the ZZG foam, the

main flow needs to flow from one end to the other along the width direction, whereas, in the BAF foam, the main flow only need to flow through half the distance compared with the ZZG foam. Hence, the pressure drop in the BAF foam is lower than that in the ZZG foam.

Fig. 6-9 Pressure drop in different configurations of graphite foams

6.7 SUMMARY

This chapter describes the experimental studies that were carried out to compare with the simulation results based on the proposed three configurations, i.e. BLK, ZZG and BAF foams in Chapter 5. The physical properties of the graphite foams were measured. These properties were then used in the simulations. Generally, the experimental and numerical results for heat transfer and pressure drops show good agreement for all configurations of graphite foams. This demonstrates the capability of the developed code to predict the heat and fluid flow in channel through a porous medium serving as a heat sink. With these, the current code can be confidently used to predict flow and heat transfer in more complex geometries with porous media heat sinks generally found in practical engineering applications.

CHAPTER 7 NUMERICAL SCHEME FOR TWO-PHASE FLOW AND HEAT TRANSFER IN POROUS MEDIA

In this chapter, the discretisation of the governing equations for two-phase flow and heat transfer in porous media based on finite volume method is presented. These formulations are presented in a 3D setting.

7.1 GOVERNING EQUATIONS IN THREE-DIMENSIONS

The present model considers an isotropic and homogeneous porous medium. The local thermal equilibrium model is used here. The governing equations obtained by adopting the two-phase mixture model (Wang, 1997) can be written as follows:

Continuity Equation

$$\varepsilon \frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad (7-1)$$

x-momentum Equation

$$u = -\frac{K}{\mu} \left[\frac{\partial p}{\partial x} - (\rho_k - \rho_0) g_x \right] \quad (7-2)$$

y-momentum Equation

$$v = -\frac{K}{\mu} \left[\frac{\partial p}{\partial y} - (\rho_k - \rho_0) g_y \right] \quad (7-3)$$

z-momentum Equation

$$w = -\frac{K}{\mu} \left[\frac{\partial p}{\partial z} - (\rho_k - \rho_0) g_z \right] \quad (7-4)$$

Energy Equations

$$\Omega \frac{\partial h}{\partial t} + \nabla \cdot (\gamma_h \mathbf{u} h) = \nabla \cdot (\Gamma_h \nabla h) + \nabla \cdot \left[f(s) \frac{K \Delta \rho h_{fg}}{v_v} \mathbf{g} \right] \quad (7-5)$$

7.2 FINITE VOLUME DISCRETISATION

7.2.1 Discretisation of Conservation Equations

7.2.1.1 Discretisation Procedure for Momentum Equations

The finite volume method (FVM) is used to discretise the above mentioned equations. Integration of the continuity equation (7-1) over the control volume gives

$$\frac{\varepsilon (\rho_P - \rho_P^0) \Delta V}{\Delta t} + (\rho u A)_e - (\rho u A)_w + (\rho v A)_n - (\rho v A)_s + (\rho w A)_t - (\rho w A)_b \quad (7-6)$$

u_e , u_w , v_n , v_s , w_t and w_b can be directly obtained from Eqs. (7-2) to (7-4), respectively. They are given as

$$u_e = -\frac{K_e}{\mu_e} \frac{p_E - p_P}{\delta x_e} + S_{ue} \quad (7-7)$$

$$u_w = -\frac{K_w}{\mu_w} \frac{p_P - p_W}{\delta x_w} + S_{uw} \quad (7-8)$$

$$v_n = -\frac{K_n}{\mu_n} \frac{p_N - p_P}{\delta y_n} + S_{vn} \quad (7-9)$$

$$v_s = -\frac{K_s}{\mu_s} \frac{p_P - p_S}{\delta y_s} + S_{vs} \quad (7-10)$$

$$w_t = -\frac{K_t}{\mu_t} \frac{p_T - p_P}{\delta z_t} + S_{wt} \quad (7-11)$$

$$w_b = -\frac{K_b}{\mu_b} \frac{p_P - p_B}{\delta z_b} + S_{wb} \quad (7-12)$$

where

$$S_u = \frac{K}{\mu} (\rho_k - \rho_0) g_x \quad (7-13)$$

$$S_v = \frac{K}{\mu} (\rho_k - \rho_0) g_y \quad (7-14)$$

$$S_w = \frac{K}{\mu} (\rho_k - \rho_0) g_z \quad (7-15)$$

By substituting Eqs. (7-7) ~ (7-15) into the discretised continuity equation (7-6), the pressure based equation is obtained as

$$a_P p_P = a_E p_E + a_W p_W + a_N p_N + a_S p_S + a_T p_T + a_B p_B + b \quad (7-16)$$

where

$$a_E = -\frac{\rho_e K_e}{\mu_e \delta x_e} \Delta y \Delta z \quad (7-17)$$

$$a_W = -\frac{\rho_w K_w}{\mu_w \delta x_w} \Delta y \Delta z \quad (7-18)$$

$$a_N = -\frac{\rho_n K_n}{\mu_n \delta y_n} \Delta x \Delta z \quad (7-19)$$

$$a_S = -\frac{\rho_s K_s}{\mu_s \delta y_s} \Delta x \Delta z \quad (7-20)$$

$$a_T = -\frac{\rho_t K_t}{\mu_t \delta y_t} \Delta x \Delta y \quad (7-21)$$

$$a_B = -\frac{\rho_b K_b}{\mu_b \delta y_b} \Delta x \Delta y \quad (7-22)$$

$$b = \frac{\varepsilon (\rho_P - \rho_P^0) \Delta V}{\Delta t} + \rho_w S_{uw} \Delta y \Delta z - \rho_e S_{ue} \Delta y \Delta z + \rho_s S_{vs} \Delta x \Delta z \\ - \rho_n S_{vn} \Delta x \Delta z + \rho_b S_{wb} \Delta x \Delta y - \rho_t S_{wt} \Delta x \Delta y \quad (7-23)$$

7.2.1.2 Discretisation Procedures for Energy Equation

The energy equation (7-5) can be written as

$$\begin{aligned} \Omega \frac{\partial h}{\partial t} + \frac{\partial h}{\partial t} - \frac{\partial h}{\partial t} + \gamma_h \nabla \cdot (\mathbf{u}h) + \mathbf{u}h \cdot \nabla \gamma_h + \nabla \cdot (\mathbf{u}h) \\ - \nabla \cdot (\mathbf{u}h) = \nabla \cdot (\Gamma_h \nabla h) + \nabla \cdot \left[f(s) \frac{K \Delta \rho h_{fg}}{\mathbf{v}_v} \mathbf{g} \right] \end{aligned} \quad (7-24)$$

It can be written in a compact form as

$$\frac{\partial h}{\partial t} + \nabla \cdot (\mathbf{u}h) = \nabla \cdot (\Gamma_h \nabla h) + \nabla \cdot \left[f(s) \frac{K \Delta \rho h_{fg}}{\mathbf{v}_v} \mathbf{g} \right] + S_h \quad (7-25)$$

where

$$S_h = (1 - \Omega) \frac{\partial h}{\partial t} - \mathbf{u}h \nabla \gamma_h + (1 - \gamma_h) \nabla \cdot (\mathbf{u}h) \quad (7-26)$$

With this formulation, the energy equation becomes a general convective-diffusive equation as shown in Eq. (3-12). This equation can be solved using FVM (Patankar, 1980).

7.2.2 Implementation of Boundary Conditions

7.2.2.1 Implementation of the Velocity Boundary Condition

In order to solve the pressure-based Eq. (7-16), boundary conditions are needed.

Generally, the boundary condition of velocity is given. Under such a condition, the pressure is calculated based on the given boundary condition of $\mathbf{u}$. Figure 7-1 shows the schematic diagram of the boundary control volumes in the x -axis.

The treatment of the boundary conditions are detailed as follows:-

At $I = 2$, $u_{2,J,K} = u_{in}$, set $a_W = 0$, $\rho_w S_{uw} \Delta y \Delta z = 0$ and $b = b + \rho_{2,J,K} u_2 \Delta y \Delta z$

At $I = L_1$, $u_{L_1,J,K} = u_{L_2,J,K}$, set $a_E = 0$, $\rho_e S_{ue} \Delta y \Delta z = 0$ and $b = b - \rho_{L_1,J,K} u_{L_1} \Delta y \Delta z$

At $J = 2$, $v_{I,2,K} = 0$, set $a_S = 0$, $\rho_s S_{vs} \Delta x \Delta z = 0$ and $b = b + \rho_{I,2,K} v_2 \Delta x \Delta z$

At $J = M_1$, $v_{I,M_1,K} = 0$, set $a_N = 0$, $\rho_n S_{vn} \Delta x \Delta z = 0$ and $b = b - \rho_{I,M_1,K} v_{M_1} \Delta x \Delta z$

At $K = 2$, $w_{I,J,2} = 0$, set $a_B = 0$, $\rho_b S_{wb} \Delta x \Delta y = 0$ and $b = b - \rho_{I,J,2} w_2 \Delta x \Delta y$

At $K = N_1$, $w_{I,J,N_1} = 0$, set $a_T = 0$, $\rho_t S_{wt} \Delta x \Delta y = 0$ and $b = b - \rho_{I,J,N_1} w_{N_1} \Delta x \Delta y$

Fig. 7-1 Sketch of boundary control volumes

7.2.2.2 Implementation of the Enthalpy Boundary Condition

Generally, temperature or a prescribed heat flux is given at the boundaries. If the temperature is given, it can be easily changed into enthalpy by using Eq. (2-51). If the heat flux is given at the boundary, the mathematical expression to calculate enthalpy at the boundary is

$$-\frac{\Gamma_h}{\rho} \frac{\partial h}{\partial x_i} - f(s) \frac{K \Delta \rho h_{fg}}{v_v} \mathbf{g} = q'' \quad (7-27)$$

where i is the corresponding axis on which heat flux is imposed. For an adiabatic boundary condition, the right hand side of Eq. (7-26) becomes 0.

7.3 SOLUTION PROCEDURE

In the present problem, the momentum equations (Eqs. 7-2 to 7-4) are first substituted into the continuity equation (Eq. 7-1) to obtain the equation for pressure. The resulting pressure equation (Eq. 7-16) is solved by a line-by-line

tri-diagonal matrix algorithm. With the results of the pressure, the mixture velocity field can be calculated using the momentum equation (Eqs. 7-2 to 7-4). The individual velocities of the liquid and vapour can be obtained from the mixture velocity using Eqs. (2-53) and (2-54). These velocities are stored at the interfaces of the control volumes. The energy equation (Eq. 7-5) can be written as a general convective-diffusive equation of the form

$$\frac{\partial(\rho\phi)}{\partial t} + \frac{\partial(\rho u_j \phi)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\Gamma \frac{\partial \phi}{\partial x_j} \right) + S \quad (7-28)$$

where ϕ , Γ and S are defined as the dependent variables, diffusion coefficient and source term, respectively. These equations were solved using FVM (Patankar, 1980). The power-law was used to treat the combined convection-diffusion term. The full sequence of the numerical procedure is described in Fig. 7-2.

7.4 CONVERGENCE CRITERION

In the iterative procedures for a transient study, for time $t+\Delta t$, the solution is considered to have converged if the maximum change of ϕ for two successive iterations is less than a prescribed value η . The convergence criterion is defined as

$$\left| \frac{\phi_i^n - \phi_i^{n-1}}{\phi_i^n} \right|_{\max} \leq \eta \quad (7-29)$$

In the current study, a relative error of less than 10^{-6} is required for both the velocity and temperature fields between successive iterations to achieve convergence.

Fig. 7-2 General structure of the program

7.5 SUMMARY

The finite volume discretisation of the governing equations for two-phase flow in porous media in a 3D domain is presented in this chapter. These governing equations are based on the two-phase mixture model. The boundary conditions for both the velocity and enthalpy are introduced. The numerical procedure and the convergence criterion are briefly discussed. The numerical solution presented in this chapter will be adapted to the specific problems in the next chapter.

CHAPTER 8 TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT TRANSFER IN POROUS MEDIA

The discretised mathematical formulation and the associated solution procedure for a 3D model are presented in the previous chapter. In this chapter, the codes developed in both 2D and 3D are validated against the existing work. The capability of the codes is showcased via three examples, i.e. (1) transient two-phase flow and heat transfer in a 2D horizontal porous channel, (2) transient two-phase flow and heat transfer in a 2D vertical porous channel and (3) two-phase flow and heat transfer in a 3D asymmetrically heated porous channel.

8.1 RELEVANT RELATIONS

In deriving the conservation equations for two-phase flow and heat transfer in porous media by the local volume-averaging method, several unknowns are introduced. These unknowns are given by available empirical correlations.

8.1.1 Relative Permeability

Relative permeability is a measure of the effective permeability for one phase in a multiple phase system. It is a ratio between the effective permeability for one phase to the absolute permeability for all phases and is a complicated function of many parameters such as the structure of the porous media, the liquid saturation and surface tension. Most of the published studies had assumed that the relative permeability is only dependent on the liquid saturation s (Muskat and Meres, 1936; Wyllie, 1962; Scheidegger, 1974; Mualem, 1976;

Levec *et al.*, 1986). They proposed different kinds of correlations for the relative permeability. Among them, the correlation introduced by Wyllie (1962) for a well sorted porous media is adopted here. The relative permeabilities for the liquid and vapour phases are expressed

$$k_{rl} = s^3 \quad (8-1)$$

$$k_{rv} = (1-s)^3 \quad (8-2)$$

where k_{rl} and k_{rv} are the relative permeabilities for the liquid and vapour phases, respectively.

8.1.2 Capillary Pressure

The capillary pressure p_c is the pressure jump across the liquid and vapour phases. p_c is determined by the mean meniscus curvature H which is dependent on the phase distribution. The calculation of H should take the dynamics of the phase distribution into account within a statistical model. This requires extensive efforts. Therefore, a general theoretical prediction of p_c is impossible. Some empirical correlations have been obtained from experiments (Leverett, 1941; Van Genuchten, 1980; Udell, 1985; Pavone, 1989). Among them, the correlation proposed by Leverett (1941) is popularly used. It is applicable for systems in which liquid and gas coexist. Therefore, it is adopted here and its mathematical expression is given by

$$\frac{p_c (K/\varepsilon)^{0.5}}{\sigma} = J(s) \quad (8-3)$$

where $J(s)$ is called the Leverett J -function. Its mathematical expression is

$$J(s) = 1.417(1-s) - 2.120(1-s)^2 + 1.263(1-s)^3 \quad (8-4)$$

8.1.3 Treatment of the Diffusion Coefficient

In modelling a phase change problem, treatment of the discontinuous diffusion coefficient at the phase change boundary poses a great challenge. The variation of the diffusion coefficient with liquid saturation s in the current problem is shown in Fig. 8-1. The diffusion coefficient is a strong function of s which in turn is highly dependent on temperature, as well as enthalpy. As seen from Fig. 8-1, the diffusion coefficient encounters a sharp and discontinuous variation, viz. from around 10^{-6} at $s = 1$ to 0 for a slight change in s . The same phenomenon occurs as s changes from non-zero to 0. This type of problem requires careful consideration.

Fig. 8-1 Variations of diffusion coefficient with s

Several numerical schemes have been proposed to deal with the discontinuity in the diffusion coefficient (Patankar, 1980; Lee and Tzong, 1991; Voller and Swaminathan, 1993). One popular scheme is the harmonic mean method. A detailed description of this method can be found in the work of Patankar (1980). Voller and Swaminathan (1993) developed a new scheme based on the local Kirchhoff transformation (Crank, 1984) which is mathematically expressed as

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

$$\phi = \int_{T_{ref}}^T \Gamma(\alpha) d\alpha \quad (8-5)$$

After some mathematical manipulations, the equivalent expressions of $\Gamma(\alpha)$ are (Voller and Swaminathan, 1993)

$$\Gamma_x(\alpha) = \frac{\partial \phi}{\partial x}, \Gamma_y(\alpha) = \frac{\partial \phi}{\partial y} \quad (8-6)$$

With reference to Eqs. (8-5) and (8-6), the central difference approximations of the interface diffusion coefficient can be written as

$$\Gamma_e = \frac{\phi_E - \phi_P}{T_E - T_P}, \Gamma_w = \frac{\phi_P - \phi_W}{T_P - T_W} \quad (8-7)$$

$$\Gamma_n = \frac{\phi_N - \phi_P}{T_N - T_P}, \Gamma_s = \frac{\phi_P - \phi_S}{T_P - T_S} \quad (8-8)$$

$$\Gamma_t = \frac{\phi_T - \phi_P}{T_T - T_P}, \Gamma_b = \frac{\phi_P - \phi_B}{T_P - T_B} \quad (8-9)$$

Equations (8-7) to (8-9) are the new schemes proposed by Voller and Swaminathan (1993) who termed them collectively as the “modified” Kirchhoff method. The current work will adopt the same terminology for easy reference. However, unlike the work of Voller and Swaminathan (1993) where the integral variable is temperature, the present article adopts enthalpy h , as the integral variable since the energy equation is expressed in terms of enthalpy. The performance of the harmonic mean method and the “modified” Kirchhoff method in resolving the discontinuous diffusion coefficient will be compared. A square channel which is completely filled with porous media is used to test these two methods. Water flows through the channel with constant heat flux at the lower wall. The length and the width of the channel are both 20 mm. The porosity of the porous media is 0.75 with absolute permeability of 7.74×10^{-10}

TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT TRANSFER IN POROUS MEDIA

m². The effective thermal conductivity for water flow through the porous media is calculated based on the model proposed by the author in Chapter 5. A parabolic velocity profile with an average velocity of 1 mm/s is imposed at the inlet. The heat flux is set to 40 W/cm². A time-step size of $\Delta t = 0.5$ s with 60×60 control volumes was found to produce grid-independent results.

The steady state temperature distributions from the harmonic mean and “modified” Kirchhoff methods are shown in Figs. 8-2(a) and 8-2(b), respectively. The dashed lines in both figures give the interface of the sub-cooled liquid zone and two-phase zone. As seen from Fig. 8-2(a) for the harmonic mean method, both the 80°C and 90°C isotherms are not smooth. Although the use of the “modified” Kirchhoff method involves the integral of Eq. (8-5) which requires additional calculations, all the temperature contours are rather smooth. During the simulation, not much difference is observed between the two methods before the occurrence of phase change during the numerical iteration. However, as phase change occurs, the “modified” Kirchhoff method gives more rapid convergence and shorter computational time for achieving steady state. Hence, in the present simulation, the “modified” Kirchhoff method will be employed.

(a)

(b)

Fig. 8-2 Temperature contours obtained using the (a) harmonic mean method, and (b) “modified” Kirchhoff method

8.2 TRANSIENT TWO-PHASE FLOW AND HEAT TRANSFER IN A 2D HORIZONTAL POROUS CHANNEL

This problem is chosen for two reasons. First, it is used to validate the current 2D code. Second, it is used to reveal the effects of the locations of heat flux on the evolutions of the boiling process.

8.2.1 Problem Specification

The schematic diagram of the problem is shown in Fig. 8-3. A discrete heat flux, at the lower wall (Fig. 8-3a), the upper wall (Fig. 8-3b), and both the lower and upper walls (Fig. 8-3c), is imposed in turn on the domain. These are henceforth referred to as bottom heated (BH), top heated (TH), and bottom and top heated (BTH) cases, respectively. The non-heated portion of the wall is adiabatic. As the sub-cooled water with low temperature T_{in} flows into the channel, convection heat transfer occurs between the heated wall and the fluid. When the heat flux is increased, boiling occurs at the heated surface and thus a two-phase zone is formed. Further increase in the heat flux will lead to a superheated vapour zone. This could cause dry-out in which no liquid exists on the heated surface. The vapour layer blankets the heated surface and greatly reduces the heat transfer between the heated surface and the liquid. Consequently, the local temperature at the dry-out location increases sharply. This is unfavourable to the cooling system. Hence, the present work considers only the case without any occurrence of a superheated vapour zone.

8.2.2 Numerical Details

The height of the channel is H and the length is L . The distances before fluid flows into and out of the heated section are l_1 and l_2 , respectively. The channel is completely filled with a porous medium. For all cases studied, the height of the channel is 20 mm. l_1/H and l_2/H are set to 2 and 3, respectively to ensure that the inlet and exit boundary conditions have no effect on the solution. Water enters the domain with a parabolic velocity profile. The average velocity at the inlet is 0.2 mm/s and the inlet temperature is 22°C. For both BH and TH cases,

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

a heat flux $q'' = 10 \text{ W/cm}^2$ is imposed at the wall between the positions of l_1 and l_2 along the flow direction. For the BTH case, the heat fluxes at the bottom and the top heated sections are set to 5 W/cm^2 resulting in a total heat flux of 10 W/cm^2 . These heat fluxes are applied at the same axial location as the aforementioned two cases on the upper and lower walls.

A grid independence study shows that a mesh of 360×60 CVs with $\Delta t = 0.5\text{s}$ produces a grid-independent solution. All subsequent computations are performed using this time-step and mesh size.

Fig. 8-3 Schematic diagram of the problem (a) BH; (b) TH; (c) BTH

8.2.3 Results and Discussion

8.2.3.1 Code Validation

The present code was validated against the experimental results of Easterday *et al.* (1995). Two-phase flow through a porous channel was investigated in their work. Their schematic test section is similar to that shown in Fig. 8-3(a) except for the dimensions. The lower wall of the channel between $1/3L$ to $2/3L$ along the flow direction was subjected to a constant heat flux. Figures 8-4(a) and 8-4(b) show a comparison of the results obtained from the current simulation and the experimental data of Easterday *et al.* (1995). Reasonable agreement is achieved for $u_{in} = 0.35$ mm/s. However, large discrepancies between the experiments and numerical simulation are observed around the locations with steep temperature gradients. These can be attributed to several causes. First, the use of some constitutive relationships from the literature which include the effective thermal conductivity, the capillary pressure and others, may not correspond exactly to the conditions of the experiments. Secondly, thermal dispersion was not included in the simulation. Thirdly, steep temperature gradients exist in the thermocouple probe locations. Accurate measurement of the temperature is extremely difficult. Lastly, the three-dimensional effect may be important in the experimental setup while the current simulation is only two-dimensional.

Fig. 8-4 Comparison of the current simulation and the experimental results of Easterday *et al.* (1995) for (a) $x = 1/3L$; (b) $x = 1/2L$

8.2.3.2 Discrete Heat Flux at the Lower Wall - the BH Case

Figures 8-5 to 8-7 show the transient behaviour of the fluid flow and heat transfer for the case where the lower wall is discretely heated. In the figures, the interface between the sub-cooled liquid zone and the two-phase zone, also known as the condensation front, is shown as a bold solid line. The temperature in the two-phase zone remains at 100°C. The liquid saturation inside the sub-cooled zone is 1. In Fig. 8-6, the numbers on the isotherms in the sub-cooled

TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT TRANSFER IN POROUS MEDIA

liquid zone show the temperatures. The numbers inside the two-phase zone are the values of the liquid saturation s .

At $t = 0$, liquid flows into the channel with a parabolic velocity profile. At the same time, the discrete heat flux at the lower wall is activated. Within a short distance from the inlet, the flow becomes uniform (Fig. 8-5a). This is characteristic of a porous medium. Heat is conducted from the heated surface to the solid portion of the porous medium, and then transferred to the liquid flowing through the empty space near the heated surface. The temperature of the liquid near the heated surface increases (Fig. 8-7a) and its density decreases. Acted upon by the buoyancy force, the liquid deflects upward slightly after flowing past the heated surface. At this moment of time, since the density variation is small due to the small temperature difference, the buoyancy force remains small and its effect is less obvious. This will however change upon further heating. At $t = 10$ s, as seen from Fig. 8-5(b) and given the large temperature difference (Fig. 8-7b), the buoyancy force on the liquid becomes stronger and its effect on the flow field is now obvious. The liquid upstream of the heated surface flows downward and toward the heated surface given its lower temperature and higher density. Upon leaving the heated surface, it flows downstream upwardly after being heated. With continued heating, at $t = 15$ s, the effect of the buoyancy on the liquid becomes so significant that it changes the flow structure. The upwardly direct flow of the liquid near the heated surface and the downwardly direct flow of the liquid away from the heated surface create a circulatory flow at the rear of the heated surface (Fig. 8-5c). Prior to this time, the temperature contours have approximately the “dome” shape covering the heated surface (Figs. 8-7a and 8-7b). These profiles are

similar to the two-dimensional conduction profiles until the flow structures change. The circulatory flow which occurred at $t = 15$ s changes the shapes of the temperature contours slightly (Fig. 8-7c). The isotherms are seen to be slightly sparser at the back of the heated surface compared to those at the leading edge of the heated surface, indicating poorer heat transfer. The heat absorbed by the circulatory flow is mainly transferred to the main stream by conduction rather than by convection. This is not favourable to heat transfer.

The onset of phase change occurs around $t = 18$ s as seen from Fig. 8-5(d). A two-phase zone, although extremely thin, forms just above the heated surface. This can be seen in the view of the enlarged figure at the relevant location adjacent to the heated surface. The two-phase zone contains both liquid and vapour coexisting together. Only a small amount of liquid becomes vapour at this time. The generated vapour inside the two-phase zone covers the heated surface and heat transfer from the heated surface to the fluid is thus reduced. The quantity of vapour at this time is sufficiently small so that its effect on the velocity field is not obvious (Fig. 8-6a). At $t = 40$ s, the two-phase zone expands (Fig. 8-5e) as more liquid vaporises. The density of the vapour is much lower than the liquid. Therefore, the generated vapour flows upwards (Fig. 8-6c) due to the buoyancy force which, simultaneously, leads to the expansion of the two-phase zone. As heating continues, the density difference between the liquid from the incoming upstream and the liquid adjacent to the condensation front is further increased. Most of the incoming liquid in the sub-cooled liquid zone, referred to as the main stream from henceforth, flows downwardly to the heated surface. As it approaches the condensation front, the main stream turns upward because it is heated and becomes lighter. The formed vapour in the two-phase

zone occupies the pores partially thus reducing the permeability for liquid flow through the two-phase zone. This too helps to deflect the liquid upward. After flowing through the two-phase zone, the accelerated main stream flows downward and exerts a larger shear force on the liquid at the rear of the two-phase zone, thus, increasing the strength of the vortex. With the occurrence of the two-phase zone, the dome-shaped temperature contours changes totally. The isotherms in the leading edge of the heated surface are inclined in an upward fashion towards the outlet (Fig. 8-7e) due to the incoming sub-cooled liquid.

At $t = 60$ s, the downward flow of the incoming main stream becomes so significant that only a small portion of liquid flows near the upper wall. The liquid with low velocity adjacent to the upper wall is counteracted by the deflected main stream passing the two-phase zone, resulting in another circulatory flow (Fig. 8-5f). The vapour, driven by both capillary-induced and buoyancy-induced forces, flows partially upwards to the inlet and outlet (Fig. 8-6d). The newly emerged vortex adjacent to the upper wall forces the incoming liquid to flow downwards, and thus reduces the heat transfer in the locations near the vortex. This is reflected by the changes to the isotherms of $T = 25^{\circ}\text{C}$ and $T = 40^{\circ}\text{C}$ from Figs. 8-7(e) and 8-7(f). The minimum liquid saturation s is found to be at the location above the heated surface. With the expansion of the two-phase zone, the vortex at the rear of the two-phase zone is pushed further downstream (Fig. 8-5f). As heat is being transferred, the temperature difference of the main flow leaving the two-phase zone is reduced (Fig. 8-7g) thus reducing its density difference. This gradually suppressed the vortex at the rear of the two-phase zone (Fig. 8-5g). The vapour inside the two-phase zone splits into three streams at this time (Fig. 8-6e), with one towards the upstream at the

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

front, the other flowing downstream at the back, and the left one flowing upwards to the upper wall in the middle. At $t = 190$ s, the vortex at the rear of the two-phase zone is totally suppressed. Slightly affected by the buoyancy force, the liquid at the rear of the two-phase zone flows upward. This stream of liquid becomes parallel as the two-phase zone expands to the outlet as shown in Fig. 8-5(j) for $t = 1000$ s. At this time, steady state has been achieved. The majority of the vapour above the non-heated surface flows directly upwards because of the buoyancy force (Fig. 8-6h). Note that vapour from the heated surface flows to the condensation front in a normal direction.

Fig. 8-5 Liquid velocity vectors at (a) 2 s; (b) 10 s; (c) 15 s; (d) 18 s; (e) 40 s; (f) 60 s; (g) 150 s; (h) 190 s; (i) 250 s; (j) 1000 s

Fig. 8-6 Vapour velocity vectors at (a) 18 s; (b) 25 s; (c) 40 s; (d) 60 s; (e) 150 s; (f) 190 s; (g) 250 s; (h) 1000 s

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

(a)

(b)

(c)

(d)

(e)

(f)

(g)

(h)

(i)

Fig. 8-7 Temperature and liquid saturation contours at (a) 2 s; (b) 10 s (c) 15 s; (d) 18 s; (e) 40 s; (f) 60 s; (g) 150 s; (h) 190 s; (i) 250 s; (j) 1000 s

8.2.3.3 Discrete Heat Flux at the Upper Wall - the TH Case

The case with discrete heat flux imposed from the upper wall at the same axial location as the BH case is now discussed. Figures 8-8 to 8-10 show the evolutions of the fluid velocity and temperature profiles. At the beginning of the heating, the liquid had a similar flow profile (Fig. 8-8a) as in the BH case. The temperature contours show the reversed “dome” shape (Fig. 8-10a). However, this will change upon further heating. Driven by the buoyancy force, the liquid with high temperature and low density below the heated surface of the upper wall flows upward. The upstream parallel flow from the inlet is counteracted by this buoyancy-induced upward flow of the liquid, resulting in a circulatory flow in the leading edge of the heated surface (Fig. 8-8c).

The onset of boiling occurs at $t = 21$ s (Fig. 8-8d), 3 seconds later than that in the BH case. The buoyancy force drives the high temperature liquid to flow towards the heated surface of the upper wall, diminishing the convection heat transfer between the heated surface and the liquid below, which in turn delays the time for the liquid to be heated to its boiling point. At the early stages of phase change, the generated vapour with lower density accumulates on the top of the heated surface due to the buoyancy force. The quantity of the vapour at this stage is extremely small so that its velocity cannot be seen until $t = 28$ s

(Fig. 8-9a). As time progresses, more liquid vaporises. Since the vapour density is much smaller than the liquid, the volume of the vapour expands, leading to the expansion of the two-phase zone. The velocity vectors of the vapour can be clearly seen in Fig. 8-9(b). Given the strong incoming sub-cooled liquid flow, the temperature contours with reversed “dome” shape incline slightly to the outlet (Fig. 8-10e). As heating continues, the two-phase zone expands downward and migrate downstream (Fig. 8-8f). The main stream, coming from the inlet boundary flows downward to bypass the vortex. Upon reaching the heated surface, it is driven to flow upward towards the condensation front due to the buoyancy force. This is different from the BH case in which the liquid was deflected to bypass the condensation front. Driven by the capillary pressure gradient, it is interesting to note that the liquid in the two-phase zone flows towards the heated surface (Fig. 8-8g), while the vapour in the two-phase zone flows primarily downwards (Fig. 8-9e). Such counter-percolation movement of the liquid and vapour is the main heat transfer mechanism within the two-phase zone.

As more liquid is vaporised, the two-phase zone assumes the shape of a half-ellipse (Figs. 8-8h, 8-9f and 8-10h). The resulting shape is not only affected by gravity, but also by the upward flow of the liquid as it approaches the condensation front. Under steady state conditions, the upper wall, from the leading edge of the heated surface to the outlet is covered by the two-phase zone, as seen from Figs. 8-8(j), 8-9(h) and 8-10(j) at $t = 1000$ s. The two-phase zone is smaller compared to its counterpart in the BH case.

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

(a)

(b)

(c)

(d)

(e)

(f)

(g)

(h)

(i)

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

(j)

→ 0.2 mm/s

Fig. 8-8 Liquid velocity vectors at (a) 2 s; (b) 10 s; (c) 15 s; (d) 21 s; (e) 40 s; (f) 60 s; (g) 150 s; (h) 190 s; (i) 250 s; (j) 1000 s

(a)

(b)

(c)

(d)

(e)

(f)

(g)

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

(h)

→ 20 mm/s

Fig. 8-9 Vapour velocity vectors at (a) 28 s; (b) 30 s; (c) 40 s; (d) 60 s; (e) 150 s; (f) 190 s; (g) 250 s; (h) 1000 s

(a)

(b)

(c)

(d)

(e)

(f)

(g)

Fig. 8-10 Temperature and liquid saturation contours at (a) 2 s; (b) 10 s; (c) 15 s; (d) 21 s; (e) 40 s; (f) 60 s; (g) 150 s; (h) 190 s; (i) 250 s; (j) 1000 s

8.2.3.4 Discrete Heat Flux at Both the Lower and Upper Walls - the BTH Case

In this third case study, both the upper and lower walls are discretely heated. As the liquid flows across these walls, it absorbs heat from the solid portion of the porous medium. Its temperature increases (Figs. 8-13a and 8-13b), leading to a smaller density. The liquid motion is now affected by the buoyancy force (Figs. 8-11a and 8-11b). The liquid across the heated surface of the lower wall, flows downstream in a slightly upward manner. The liquid upstream of the heated surface of the lower wall, which is cooler and therefore denser, tends to flow downward towards the heated surface of the lower wall to replenish the higher temperature liquid leaving the region above the heated surface of the lower wall. This combination of incoming and outgoing flows to the region above the heated surface of the lower wall is favourable for heat removal from the heated surface of the lower wall. Buoyancy assists to draw cooler fluid towards the heated surface and expel hotter liquid from the region above the

heated wall. Heat is convected away to the downstream fluid from the heated surface of the lower wall. This is not the case for the liquid flowing across the heated surface of the upper wall. Buoyancy actually drives the higher temperature liquid upward towards the heated surface of the upper wall. In such a situation, the convection heat transfer between the heated surface of the upper wall and the liquid is reduced. This is not favourable for heat transfer.

The action of the buoyancy force eventually creates two circulatory flows in the channel as shown in Fig. 8-11(c). These circulatory flows are located upstream of the heated surface of the upper wall and downstream of the heated portion of the lower wall. The circulatory flow near the upper wall is much stronger. The temperature distribution at $t = 50$ s (Fig. 8-13d) is obviously not symmetrical about the middle horizontal plane of the channel. The poorer heat transfer performance for the upper wall results in a generally higher liquid temperature in its adjacent. As time progresses, the liquid adjacent to the heated surface of the upper wall reaches its boiling point earlier than that adjacent to the heated surface of the lower wall. Phase change occurs at around $t = 75$ s at the upper wall (Figs. 8-11d and 8-13d) and later at the lower wall. An extremely thin layer of the two-phase zone covers the heated surface of the upper wall at this time. The vapour velocity at the upper wall is not obvious until $t = 90$ s (Fig. 8-12b) where at this time, the vapour velocity at the lower wall cannot even be clearly noticed.

The two-phase zones at both the upper and lower walls expand as heating continues. The two-phase zone at the lower wall is swept downstream to cover the heated surface of the wall partially (Fig. 8-12c). On the contrary, the two-phase zone blankets the heated surface of the upper wall entirely (Fig. 8-12c).

The presence of the circulatory flow upstream of the heated surface of the upper actually hinders cooler liquid from reaching the two-phase zone. This results in poorer heat transfer. Driven by the buoyancy force, the vapour on the upper wall attaches to the heated surface. Therefore, the two-phase zone remains thin and is less affected by the liquid flow in the sub-cooled zone at the upper wall. The vapour in the two-phase zone at both the lower and upper walls flows primarily to the condensation front.

With continued heating, the expanded two-phase zone pushes the vortex at the lower wall further downstream (Fig. 8-11g). The spreading of the heat reduces the temperature difference of the liquid at the downstream of the heated surface (Fig. 8-13g). This suppresses the buoyancy-induced circulatory flow at the rear of the two-phase zone on the lower wall. At $t = 190$ s, this circulatory flow vanishes (Fig. 8-11h) leaving only the circulatory flow at the upstream of the heated surface on the upper wall. The minimum liquid saturation of 0.8 occurred at the upper wall prior to the lower wall (Fig. 8-13h). An implication of this result could be that in thermal systems with heat sources at both the lower and upper walls, a dry-out zone could appear at the upper wall. In the steady state (Figs. 8-11j, 8-12g and 8-13j), the upper wall, from the leading edge of the heated surface to the outlet, is covered by the two-phase zone, while in the lower wall, the two-phase zone starts from the latter part of the heated surface, showing better heat transfer on the lower wall compared with the upper wall.

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

(a)

(b)

(c)

(d)

(e)

(f)

(g)

(h)

(i)

Fig. 8-11 Liquid velocity vectors at (a) 10 s; (b) 15 s; (c) 50 s; (d) 75 s; (e) 85 s; (f) 120 s; (g) 150 s; (h) 190 s; (i) 250 s; (j) 1000 s

Fig. 8-12 Vapour velocity vectors at (a) 85 s; (b) 90 s; (c) 120 s; (d) 150 s; (e) 190 s; (f) 250 s; (g) 1000 s

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

(a)

(b)

(c)

(d)

(e)

(f)

(g)

(h)

(i)

Fig. 8-13 Temperature and liquid saturation contours at (a) 10 s; (b) 15 s; (c) 50 s; (d) 75 s; (e) 85 s; (f) 120 s; (g) 150 s; (h) 190 s; (i) 250 s; (j) 1000 s

8.2.3.5 Variation of the Vapour Volume Fraction

The liquid saturation s is defined as (Kaviany, 1995)

$$s = \frac{\varepsilon_l}{\varepsilon} \quad (8-10)$$

where ε is the porosity of the porous media and ε_l is the fraction of the volume occupied by the liquid. According to the above definition, the volume fraction occupied by the generated vapour can be calculated as

$$s_v = \frac{\int_{V_f} 1 dV_f - \int_{V_f} s dV_f}{V_f} \quad (8-11)$$

where V_f is fluid volume in the porous media. Based on Eq. (8-11), the variations of s_v with time for different cases are shown in Fig. 8-14. The BH and BTH cases achieved steady state in a shorter time compared with the TH case, indicating enhanced cooling of the heated surface. The rate of vapour generation, implied by the slope of the lines, in the BH case is the highest among the three cases. The largest volume fraction of vapour is also found in the BH case. This can be attributed to better heat transfer for the case where heating is from the lower wall. The TH case produces a faster generation of the vapour before $t = 250$ s compared with the BTH case. The rate of vapour

generation decreases after that, resulting in the smallest amount of the vapour among the three studied cases.

Fig. 8-14 Time variation of vapour fraction for different cases

8.3 TRANSIENT TWO-PHASE FLOW AND HEAT TRANSFER IN A 2D VERTICAL POROUS CHANNEL

This problem is chosen to study the transient behaviour of fluid flow and heat transfer with phase change under aiding and opposing flows. The problem is of significant importance to many engineering applications such as nuclear reactors (Schäfer and Lohnert, 2006) and PEM fuel cells (You and Liu, 2002; Karimi *et al.*, 2009; Afshari and Jazayeri, 2009).

8.3.1 Problem Specification

The schematic diagram of the problem considered is shown in Fig. 8-15. A two-dimensional vertical channel ($W \times L$) is filled with a porous medium. A finite heat source with constant heat flux is located on the right side wall which is adiabatic elsewhere. The left side wall is perfectly insulated. The external pressure difference drives the sub-cooled water with low temperature T_{in}

through the channel either in aiding (Fig. 8-15a) or opposing (Fig. 8-15b) flow configurations.

8.3.2 Numerical Details

For all cases studied, the width of the channel W is 20 mm. The length of the heated section on the right side wall is $2W$. The lengths of the unheated sections of the right side wall namely, l_1/W and l_2/W are set to 3. These lengths were chosen because the inlet and exit boundary conditions have no effect on the solution. Water enters the domain with a uniform velocity and a constant temperature of 22°C. Note that the axial coordinate is measured from the inlet to the porous medium. A constant heat flux $q'' = 10 \text{ W/cm}^2$ is imposed at the heated section on the right wall. The porosity of the porous media is 0.75. The effective thermal conductivity for the porous medium was calculated based on the model proposed in Chapter 5. For the sake of brevity, only the most essential features of the results are presented.

A mesh of 60 (transverse) $\times$ 480 (axial) CVs with $\Delta t = 0.5 \text{ s}$ produces a grid-independent solution. All subsequent computations were performed using this time-step and mesh size.

Fig. 8-15 Schematic of the problem and the coordinate systems for (a) aiding flow, and (b) opposing flow

8.3.3 Results and Discussion

The effects of the Rayleigh and Peclet numbers are discussed here. There are two Rayleigh numbers based on different flow regimes (Ramesh and Torrance, 1990; Wang *et al.*, 1994). One is for the sub-cooled liquid zone and is based on the temperature difference. The second Rayleigh number is for the two-phase zone and it involves the density difference between the liquid and vapour phases. Detailed information can be found in the works of Ramesh and Torrance (1990) and Wang *et al.* (1994). As the primary interest in the current work is to study the fluid flow and heat transfer behaviour in the two-phase zone in the porous media, the author has chosen the form of the Rayleigh number based on the two-phase zone. The definitions of Rayleigh and Peclet numbers are

$$Ra \equiv \frac{Kg \rho_l C_{pl} W}{\nu_l k_{eff}} \quad (8-12)$$

$$Pe \equiv \frac{v_{in} W}{\alpha} \quad (8-13)$$

8.3.3.1 Effects of Rayleigh Number in Aiding Flow

To study the effects of Rayleigh number, the Peclet number is set to 0.24. The numbers in the temperature and liquid saturation contours have the same meanings as those discussed in Section 8.2.2. In this section, Figs. 8-16(b), 8-17(b) and 8-18(b) at $Ra = 226$ will be taken as examples to describe the transient behaviour of the fluid flow and heat transfer first and then comparisons will be made for different Ra .

At $t = 0$, the liquid flows into the channel from the bottom with a uniform velocity profile. The discrete heat flux on the right side wall is activated at the same time. The liquid flowing past the heated section on the right side wall absorbs heat and its temperature increases, leading to a smaller density. Acted upon by the buoyancy force, the liquid adjacent to the heated section with higher temperature flows slightly faster than the liquid located away from the heated section of the right side wall (Fig. 8-16b at $t = 0.5$ s). At this moment of time, since the density variation is small due to the small temperature difference (Fig. 8-18b at $t = 0.5$ s), the buoyancy force remains insignificant and its effect is less obvious. At $t = 5$ s, given the large temperature difference of the liquid adjacent to the heated section and the liquid away from the heated section (Fig. 8-18b at $t = 5$ s), the main stream flows to the heated surface. The buoyancy force assists in accelerating this main stream as it approaches the heated section.

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

This helps to convey the heat from the heated section of the right side wall to the downstream. With continued heating, the effect of the buoyancy force on the acceleration of the main stream becomes so significant that the flow structure is changed (Fig. 8-16b at $t = 8$ s). As the main stream leaves the heated section, it redistributes to the channel downstream. This main stream counteracts with the liquid upstream which has much lower velocity and lower temperature, leading to a recirculation flow near the left side wall. Prior to this, the temperature contours have approximately the dome shape covering the heated section of the right side wall (Fig. 8-18b at $t = 0.5, 5$ and 8 s).

The onset of phase change occurs around $t = 15$ s. An extremely thin layer of two-phase zone forms on the back portion of the heated section on the right side wall. This can be seen in the enlarged figure at the relevant location adjacent to the heated section with the coordinates as Fig. 8-17(b) at $t = 15$ s. Both liquid and vapour coexist in the two-phase zone. At this moment, the quantity of generated vapour is sufficiently small so that its effect on the velocity field is not obvious (Fig. 8-17b at $t = 15$ s). The generated vapour inside the two-phase zone covers the heated section and heat transfer from the heated section to the fluid is thus reduced. As more liquid vaporises at $t = 30$ s, the two-phase zone expands (Fig. 8-17b at $t = 30$ s). The vapour flows to the condensation front in a nearly normal direction. The recirculation flow becomes much more pronounced (Fig. 8-16b at $t = 30$ s). The temperature contours in the leading edge of the heated section are slightly inclined to the outlet (Fig. 8-18b at $t = 30$ s) due to the incoming sub-cooled liquid. The two-phase zone expands further as heating continues (Figs. 8-16b, 8-17b and 8-18b at $t = 50$ s). It is swept downstream to cover the heated section of the right side wall partially. Given

the incoming sub-cooled liquid, the two-phase zone facing the inlet upstream remains thin and it gradually grows downstream. The thickness of the two-phase zone reaches a maximum somewhere within the heated section. Thereafter, the thickness of the two-phase zone diminishes. Since heat is being transferred, the temperature difference of the liquid near the two-phase zone is reduced (Fig. 8-18b at $t = 30$ s and 50 s) thus reducing its density difference. This gradually suppressed the expansion of the recirculation zone in the direction of the width although it continues to expand along the flow direction. At $t = 65$ s, the main stream in the sub-cooled zone bypasses the two-phase zone (Fig. 8-16b at $t = 65$ s). The formed vapour in the two-phase zone occupies the pores partially thus reducing the permeability for liquid flow through the two-phase zone. The deflected liquid pushes the recirculated fluid upwards such that it is confined to a narrow space near the left side wall. Driven by the capillary-induced force, the liquid inside the two-phase zone flows to the heated section of the right side wall. At $t = 100$ s, the recirculation flow adjacent to the left wall disappears totally (Fig. 8-16b at $t = 100$ s). The vapour inside the two-phase zone splits clearly into three streams at this time (Fig. 8-17b at $t = 100$ s), with one flowing upstream to the inlet, the second stream flowing downstream to the outlet, and the third stream flowing in a direction normal to the condensation front. The liquid saturation s reduces as the fluid moves to the heated section of the right side wall. At $t = 1000$ s, the steady state condition is achieved. The right side wall is partially covered by the two-phase zone (Figs. 8-16b, 8-17b and 8-18b at $t = 1000$ s). The minimum liquid saturation s occurs somewhere near the end of the heated section.

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

According to the definition of Ra adopted in the present work, to change Ra while keeping the other parameters constant implies a variation in the permeability of the porous media. In the current work, three different Rayleigh numbers, *viz.* three different permeabilities of the porous media are studied. It is obvious that a larger value of Ra will result in larger strength and area of the recirculatory flow as seen from Figs. 8-16(b) and 8-16(c). No obvious recirculatory flow is observed at $Ra = 113$ (Fig. 8-16a). The recirculatory flow under larger Ra is created earlier than the smaller Ra (Fig. 8-16c and 8-16b at $t = 5$ s). However, such recirculation can be suppressed easily at lower Ra (Figs. 8-16c and 8-16b at $t = 100$ s). With the help of the strong recirculatory flow at a larger $Ra = 339$, more liquid from the inlet is drawn to flow towards the heated section of the right side wall thus pushing the leading edge of the two-phase zone downstream (Figs. 8-17a, 8-17b and 8-17c at $t = 15$ s). The convex shape of the interface between the sub-cooled liquid zone and two-phase zone thus becomes flatter, but longer than that at low Ra . The vapour flows in a similar fashion under different Ra (Figs. 8-17a, 8-17b and 8-17c). The temperature and the liquid saturation fields show that the convection heat transfer is enhanced at a position downstream of the heated surface with an increase of the Rayleigh number.

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

Fig. 8-16 Liquid velocity vectors for aiding flow at $Pe = 0.24$ for (a) $Ra = 113$; (b) $Ra = 226$; (c) $Ra = 339$

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

Fig. 8-17 Vapour velocity vectors for aiding flow at $Pe = 0.24$ for (a) $Ra = 113$; (b) $Ra = 226$; (c) $Ra = 339$

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

Fig. 8-18 Temperature and liquid saturation contours for aiding flow at $Pe = 0.24$ for (a) $Ra = 113$; (b) $Ra = 226$; (c) $Ra = 339$

8.3.3.2 Effects of Peclet Number in Aiding Flow

Figures 8-19 to 8-21 demonstrate the effects of Peclet number, Pe on the velocity, temperature and liquid saturation fields for a fixed Rayleigh number, *i.e.* $Ra = 226$. The Peclet number is directly related to the inlet velocity. At low Pe (or low inlet velocity), the recirculation flow is strengthened and its size is increased compared with Fig. 8-16(b) due to the weak incoming flow as shown in Fig. 8-19(a). The eye of the recirculatory flow is moved upstream. Since the main mode of heat transfer in the recirculatory flow is conduction rather than convection, heat transfer in the eye of the recirculation is at its weakest causing the middle of the condensation front to protrude into the sub-cooled liquid zone. The main stream in the sub-cooled liquid zone is deflected as it flows past the middle portion of the two-phase zone. Such a deflected stream finally divides the recirculatory flow into two small cells which is the unique feature in this case (Fig. 8-19a at $t = 80$ s). The two cells finally disappear at $t = 100$ s for $Pe = 0.12$ (Fig. 8-19a). At steady state, the liquid leaves the channel as a liquid-vapour mixture (Fig. 8-19a at $t = 1000$ s). When Pe is sufficiently high, the recirculatory flow caused by the buoyancy force is diminished. The incoming main stream pushes the liquid leaving the heated section of the right side wall downstream directly. This can be seen for $Pe = 0.48$ in Fig. 8-19b. The leading edge of the two-phase zone also moves downstream due to the flow of sub-cooled liquid from the inlet. With the increase of heat transfer from the heated section to the fluid, a small amount of vapour is generated and the two-phase zone shrinks substantially. The vapour flows partially to the condensation front and partially to the outlet. The temperature and liquid saturation fields also

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

change with an increase of Pe . For large Pe , the isotherms are prominently inclined towards the outlet due to the strong incoming flow. Denser temperature contours are observed upstream of the heated section indicating enhanced convection heat transfer. The minimum liquid saturation increases with the increase of Pe .

Fig. 8-19 Liquid velocity vectors for aiding flow at $Ra = 226$ for (a) $Pe = 0.12$; (b) $Pe = 0.48$

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

(a) $Pe = 0.12$

(b)

Fig. 8-20 Vapour velocity vectors for aiding flow at $Ra = 226$ for (a) $Pe = 0.12$; (b) $Pe = 0.48$

(a)

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

Fig. 8-21 Temperature and liquid saturation contours for aiding flow at $Ra = 226$ for (a) $Pe = 0.12$; (b) $Pe = 0.48$

8.3.3.3 Effects of Rayleigh Number in Opposing Flow

Figures 8-22 to 8-24 show the effects of Ra on the velocity, temperature and liquid saturation fields for the case of opposing flow. Comparisons are made between aiding and opposing flows at $Ra = 226$ and $Pe = 0.24$ as seen in Figs. 8-16(b) and 8-22(b).

When the incoming flow moves downward, the buoyancy-induced flow will be in opposing direction to the external pressure-induced flow. As the sub-cooled liquid flows past the heated section of the right side wall, it absorbs heat and its temperature increases and hence, density decreases. Given the buoyancy force, the high-temperature liquid adjacent to the heated section of the right side wall flows upwards (Fig. 8-22b at $t = 2$ s), leading to a recirculatory flow near the heated section. The main stream from the inlet is therefore deflected to the left wall by the occurrence of such recirculatory flow (Fig. 8-22b at $t = 3$ s). This is totally different from what was observed for aiding flow (Fig. 8-16b at $t = 8$ s).

TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT TRANSFER IN POROUS MEDIA

As time progresses, the liquid adjacent to the heated section of the right side wall reaches its boiling point and a two-phase zone is formed (enlarged figure in Fig. 8-23b at $t = 15$ s). Prior to this, the temperature contours (Fig. 8-24b at $t = 0.5, 1-3, 15$ s) are approximately symmetrical about the centre of the heated section. However, this was not observed for the case of aiding flow. This symmetric profile will change upon further heating. With continued heating, the recirculatory flow is strengthened and extended to a larger region which squeezes more liquid to the left wall (Fig. 8-22b at $t = 30$ s). The liquid in the recirculation flow is partially inside the two-phase zone, and partially located in the sub-cooled liquid zone. The presence of such recirculatory flow near the heated section of the right side wall actually hinders the cooler liquid from reaching the heated section. This results in poor heat transfer from the heated section to the surrounding liquid. The vapour velocity at the heated section of the right side wall is not obvious until $t = 30$ s (Fig. 8-23b at $t = 30$ s). It flows in a similar fashion as aiding flow, showing a normal direction to the condensation front. Further heating reduces the temperature difference of the liquid near the heated section of the right side wall and those away from it (Fig. 8-24b at $t = 50$ s). The buoyancy-induced recirculatory flow gradually shrinks (Fig. 8-22b at $t = 50$ s). The main stream pushes the liquid at the back of the recirculation flow downwards (Fig. 8-22b at $t = 80, 100$ and 150 s). When the steady state is reached (Fig. 8-22b at $t = 1000$ s), such recirculatory flow becomes a small cell located at the leading edge of the two-phase zone (Fig. 8-22b at $t = 150$ and 1000 s). The liquid in the small recirculatory flow acts as if it is trapped in the upper corner of the two-phase zone. It is interesting to note that the most of the

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

vapour flows slightly upward to the incoming flow which opposes the incoming flow (Fig. 8-23b at $t = 1000$ s).

Figures 8-22 to 8-24 demonstrate the effects of Ra on the flow and heat transfer fields for $Pe = 0.24$. When Ra is small, the buoyancy force is not significant and the recirculatory flow near the heated section of the right side wall is weak (Fig. 8-22a at $t = 30$ s). Under this circumstance, the recirculatory flow can be easily suppressed (Fig. 8-22a at $t = 80$ s). At steady state, the recirculatory flow is totally diminished. The temperature contours exhibit nearly the symmetric profiles along the centre of the heated section before the occurrence of phase change. The condensation front assumes the shape of a half-ellipse. With an increase of Ra , the recirculatory flow becomes stronger and larger in extent. This is similar to the case of aiding flow. The recirculatory flow at high Ra substantially reduces heat transfer from the heated surface to the adjacent liquid. Therefore, the two-phase zone moves upwards towards the inlet and downwards to the outlet simultaneously, resulting in a long condensation front. Among the three tested Rayleigh numbers, the vapour in the two-phase zone flows in a similar fashion to each other, showing a slightly upward pattern. As the main stream from the inlet flow is squeezed to the left wall due to the recirculation near the heated surface at high Ra , the temperature contours on the left side are significantly inclined to the outlet. The liquid saturation contours among the three Rayleigh numbers indicate that the minimum liquid saturation occurs at low Ra in a location within the heated section of the right side wall. The strong recirculatory flow under large Ra , to some extent, promotes the mixing of the liquid near the heated section and the main stream, resulting in enhanced heat transfer.

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

Fig. 8-22 Liquid velocity vectors for opposing flow at $Pe = 0.24$ for (a) $Ra = 113$; (b) $Ra = 226$; (c) $Ra = 339$

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

(a)

(b)

(c)

– 20 mm/s

Fig. 8-23 Vapour velocity vectors for opposing flow at $Pe = 0.24$ for (a) $Ra = 113$; (b) $Ra = 226$; (c) $Ra = 339$

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

Fig. 8-24 Temperature and liquid saturation contours for aiding flow at $Pe = 0.24$ for (a) $Ra = 113$; (b) $Ra = 226$; (c) $Ra = 339$

8.3.3.4 Effects of Peclet Number in Opposing Flow

Figures 8-25 to 8-27 display the effects of Pe on the flow and heat transfer fields for opposing flow. With the decrease of Pe (Fig. 8-25a at $Pe = 0.12$), the incoming flow becomes weak. The buoyancy force becomes so dominant that it produces a stronger recirculation flow near the heated section of the right side wall compared with the case of $Pe = 0.24$. The recirculation flow is produced in a rather large area causing the upstream sub-cooled liquid to bypass it. The liquid from the inlet is deflected to the left wall and is accelerated significantly. As the sub-cooled liquid is prevented from flowing to the heated section, heat transfer is therefore greatly reduced, shifting the condensation front further upstream. When steady state is achieved (Fig. 8-25a at $t = 1000$ s), the two-phase zone covers a large portion of the right wall beyond the heated section on the right side wall. The recirculatory flow in the leading edge of the two-phase zone at steady state is much stronger than the case for $Pe = 0.24$ (Fig. 8-22b at $t = 1000$ s). The vapour at the edge of the two-phase zone flows directly upward given the buoyancy force (Fig. 8-26a at $t = 1000$ s). As Pe increases (Fig. 8-25b at $Pe = 0.48$), the recirculatory flow is greatly reduced in its strength and extent. The strong incoming flow pushes the recirculation as well as the condensation front downstream. Heat transfer is greatly increased and the two-phase zone is reduced significantly.

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

Fig. 8-25 Liquid velocity vectors for opposing flow at $Ra = 226$ for (a) $Pe = 0.12$; (b) $Pe = 0.48$

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

Fig. 8-26 Vapour velocity vectors for opposing flow at $Ra = 226$ for (a) $Pe = 0.12$; (b) $Pe = 0.48$

Fig. 8-27 Temperature and liquid saturation contours for opposing flow at $Ra = 226$ for (a) $Pe = 0.12$; (b) $Pe = 0.48$

8.3.3.5 Liquid Saturation along the Heat Wall

The variation of the liquid saturation s at steady state along the heated wall for both aiding and opposing flows for different Rayleigh numbers at $Pe = 0.24$ is shown in Fig. 8-28. An increase of Ra results in an increase of s although with different trends for aiding and opposing flows. The liquid saturation s is kept at a constant value of 1 in the entrance for both flow configurations, indicating a sub-cooled liquid in this region. When the external flow is upward, s begins to drop as the liquid approaches the heated section of the right side wall. It reaches a minimum at the end of the heated section and then increases when the fluid leaves the heated section as a liquid-vapour mixture. This is not the case for opposing flow. In opposing flow, the variations of s have the shape of a half-ellipse on the heated section. The minimum s is located approximately at the centre of the heated section for a high Rayleigh number $Ra = 339$. It shifts downstream with the decrease of Ra . The strong recirculatory flow at high Ra moves the two-phase zone upstream significantly. Compared with aiding flow, the two-phase zone in opposing flow covers a wider range on the right wall. This implies that as the buoyancy force is opposed to the incoming flow, heat can be spread to a larger region in the system. Special attention should be paid to such problems.

Figure 8-29 shows the effects of Pe on the liquid saturation s for both aiding and opposing flows at $Ra = 226$. Generally, a smaller Pe suggests a weaker convection effect, *i.e.* a longer residence time of the fluid within the channel. The fluid absorbs more heat, leading to the generation of more vapour. Therefore, for flow with smaller Pe , a smaller s is to be expected. For example,

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

in aiding flow, the liquid saturation reaches a minimum of around 0.46 for $Pe = 0.12$ but only 0.65 for $Pe = 0.48$. A similar behaviour is observed for opposing flow. With increasing Pe , *i.e.* a stronger convection effect, the leading edge of the two-phase zone is swept further downstream. Of particular interest is the effect of buoyancy on opposing flows. As the buoyancy effect works against convection, it moves the leading edge of the two-phase zone upstream. This effect is more significant for small Peclet numbers where the amount of vapour generated is larger. For the same reason, s is generally smaller for the case of opposing flow compared to aiding flow.

Fig. 8-28 The variation of liquid saturation s along the heated wall for aiding and opposing flows under different Rayleigh numbers

Fig. 8-29 Variation of liquid saturation s along the heated wall for aiding and opposing flows under different Peclet numbers

8.4 TWO-PHASE FLOW AND HEAT TRANSFER IN A 3D ASYMMETRICALLY HEATED POROUS CHANNEL

This problem is chosen to test the capability of the developed 3D code. Although the assumption of a 2D setting in the numerical simulation is applicable in some very specific cases, a 3D numerical simulation is more desirable since it is more realistic for comparison with actual experimental data. It is especially important for understanding and predicting the development of complex flow structures and the two-phase zone for cases which cannot be simplified to a 2D setting such as the case studied in this section.

However, before the case study, the developed 3D code is validated against its counterpart of 2D code. With symmetric boundary conditions enforced along the width direction in the 3D domain (fluid flows along length (x) direction and heating along height (z) direction), the results from the present 3D code should reduce to its 2D counter-part with these boundary conditions. This comparison is given in Fig. 8-30. The temperature distributions obtained from the 2D and 3D codes are found to overlap at different locations. The validity of the 3D code is therefore demonstrated.

8.4.1 Problem Specification

The schematic diagram of the problem is shown in Fig. 8-31(a). A 3D channel with dimensions $L \times W \times H$ is filled with a porous medium. A finite heat source with constant heat flux is applied on one side wall. The length of unheated section is chosen based on that the inlet and exit boundary conditions have no effects on the solution. An external pressure difference drives the sub-cooled

water with low temperature T_{in} through the channel. The flowing liquid is heated as it flows past the heated section of the wall.

Fig. 8-30 Comparison of 3D results under symmetric boundary conditions with 2D results (a) $x = 1/3L$; (b) $x = 1/2L$

8.4.2 Numerical Details

The width W and height H of the channel are fixed at 20 mm. The length of the heated section on the side wall is 20 mm. The lengths of the unheated sections of the side wall namely, l_1/H and l_2/H are set to 1. Water enters the domain with a uniform velocity and at a constant temperature of 22°C. A constant heat flux $q'' = 20 \text{ W/cm}^2$ is imposed at the heated section of the side wall.

TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT TRANSFER IN POROUS MEDIA

Solutions are obtained based on meshes of $30 \times 10 \times 10$ CVs with $\Delta t = 2.0$ s, $60 \times 20 \times 20$ CVs with $\Delta t = 1.0$ s and $120 \times 40 \times 40$ CVs with $\Delta t = 0.5$ s. The temperature distribution on the side wall with heated section shows that a mesh of $60 \times 20 \times 20$ CVs with $\Delta t = 1.0$ s produces a grid-independent solution.

Fig. 8-31 Schematic of the problem and the coordinate systems (a) the simulation domain; (b) selected planes

8.4.3 Results and Discussion

8.4.3.1 Effects of Peclet Number on Fluid Flow and Heat Transfer

Figure 8-32 shows the flow fields for $Ra = 226$ and $Pe = 0.06$. As the liquid flows into the channel with a uniform velocity profile, it absorbs heat when it

flows past the heated section of the wall. Its temperature increases, leading to a lower density. Acted upon by the buoyancy force, the heated liquid flows in a slightly upward manner. The liquid upstream of the heated section of the wall with low temperature tends to flow downward to replace the upward flow of the heated liquid. As the buoyancy force becomes important, this combination of downward flow of the cooler liquid from the upstream of the heated section and upward flow of the hotter liquid downstream results in a circulatory flow. This circulation flow is located at the leading edge of the two-phase zone (Fig. 8-32a). The recirculation hinders the liquid upstream from penetrating the circulation cell. Therefore, the liquid in the sub-cooled liquid zone from the inlet flows decisively downward to bypass the recirculation (Fig. 8-32a). When it approaches the condensation front, it shows a slightly upward motion. This slightly upward motion of the liquid is due to a decrease in the permeability which causes the sub-cooled liquid to flow through the two-phase zone. The two-phase zone is occupied by both the liquid and vapour. The presence of the generated vapour in the void space reduces the permeability for the sub-cooled liquid to flow through the two-phase zone. The liquid therefore has difficulty in penetrating the void space in the porous media which also contains vapour. When the incoming flow is weak, the liquid in the sub-cooled liquid zone is blocked as it hits the condensation front. The blocked liquid thus reverses its flow direction to the inlet as the two-phase zone presents a region of high flow resistance. This can be seen in the xy plane with $z/H = 0.5$ (Fig. 8-32b). Further away from the heated section of the wall, less vapour is generated. The flow resistance caused by the presence of the vapour to the flowing liquid is therefore smaller. Hence, the liquid away from the heated section in the sub-

cooled liquid zone flows across the condensation front into the two-phase zone (Fig. 8-32b). In the locations near the heated section of the wall, more liquid vaporizes, leading to a smaller liquid saturation. In contrast, at locations away from the heated section, the liquid saturation is higher. This difference in the liquid saturation causes a liquid saturation gradient in the two-phase zone. Such a liquid saturation gradient gives rise to the capillary-induced force. The liquid is driven by the capillary force to flow from the locations with high volume fraction of the liquid to the locations with low volume fraction of the liquid; *viz.* the capillary force drives the liquid in the two-phase zone towards the heated section of the wall. This can be seen from the motion of the liquid in the vicinity of the heated section of the wall shown in the xy plane with constant $z/H = 0.5$ (Fig. 8-32b). In the zone with low liquid saturation gradient, the liquid flows approximately parallel to the outlet. Under the combined effect of the buoyancy and capillary forces, the liquid from the inlet divides into two streams. While one stream flows backwards toward the inlet in a circulatory manner, the other stream flows to the outlet. In the yz plane with constant $x/H = 0.1$ (Fig. 8-32c), the two-phase zone occupies a small portion of the plane in the upper-left corner. Given the buoyancy force and the weak flow from inlet, the vapour rises and spreads upstream of the heated section. At a certain elevation, the buoyancy force assists the liquid to flow upward while the liquid below such elevation flow directly downward (Fig. 8-32c). Driven by the capillary force, this downward flow was induced to fill the empty space caused by the upward flow in a slightly lateral manner to the heated section. As the fluid flows into the middle of the domain with the heated section at $x/H = 0.3$, an extensive region of the plane is covered by the two-phase zone. The sub-cooled

liquid zone is pushed to a small area located in the lower right corner of the plane (Fig. 8-32d). This two-phase zone grows to become almost symmetrical to the diagonal of the plane. When the liquid leaves the heated section at $x/H = 0.5$ (Fig. 8-32e), the two-phase zone expands. As most of the fluid flows parallel to the outlet of the domain (Fig. 8-32a and 8-32b), the liquid flow in the yz plane becomes insignificant.

Fig. 8-32 Liquid velocity in different planes for $Pe = 0.06$ and $Ra = 226$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$

Figure 8-33 shows the characteristics of the vapour flow in different planes.

The vapour is first generated on the heated section of the wall and then it moves

away from the heated source in different directions. A portion of the vapour is carried downstream in the form of a vapour-liquid mixture while the other portion flows upward due to buoyancy force (Figs. 8-33a and 8-33b). As a result, the upper portion of the channel becomes progressively richer in vapour along the flow direction. On the other hand, the lower portion of the channel is relatively rich in liquid. This phase separation is caused by the large density difference between the liquid and vapour. The quantity of the vapour increases along the flow direction since more liquid vaporises (Fig. 8-33d). It is noted that the vapour flows toward the condensation front where it is condensed.

Fig. 8-33 Vapour velocity in different planes for $Pe = 0.06$ and $Ra = 226$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$

TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT TRANSFER IN POROUS MEDIA

Figures 8-32 to 8-37 demonstrate the effects of Peclet number, Pe on the velocity fields for a fixed Rayleigh number of 226. The Peclet number is directly related to the inlet velocity. At low Pe (or low inlet velocity), the buoyancy force is dominant. The circulatory flow caused by the buoyancy force is therefore strengthened. The eye of the recirculatory flow is shifted upstream. Generally, a smaller Pe suggests a weaker convection effect and a longer residence time for the fluid staying in the channel. The fluid absorbs more heat, leading to the generation of more vapour. Therefore, a larger two-phase zone is developed in the channel. The interface shape between the sub-cooled liquid zone and the two-phase zone is a direct result of the strong upward vapour flow. At $Pe = 0.12$, the increase in the relative strength of the inlet flow reduces the circulatory flow (Fig. 8-34a). The inertial of the liquid increases so that it gradually changes the direction for the back flow of the liquid near the condensation front in the xy plane (Fig. 8-34b). When Pe is sufficiently high ($Pe = 0.24$), the strong incoming liquid pushes the liquid near the condensation front in the xy plane decisively downstream (Fig. 8-36b). With the increase of the inlet velocity, the heat transfer from the heated section of the wall to the fluid is increased. A small amount of vapour is generated and the two-phase zone shrinks substantially. It is observed that the two-phase zone recedes appreciably away from the inlet (Fig. 8-36b). The vapour in the two-phase zone flows partially to the condensation front and partially to the outlet.

Fig. 8-34 Liquid velocity in different planes for $Pe = 0.12$ and $Ra = 226$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$

Fig. 8-35 Vapour velocity in different planes for $Pe = 0.12$ and $Ra = 226$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$

Fig. 8-36 Liquid velocity in different planes for $Pe = 0.24$ and $Ra = 226$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$

Fig. 8-37 Vapour velocity in different planes for $Pe = 0.24$ and $Ra = 226$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$

Figure 8-38 shows the isotherms within the porous medium. The temperature difference between successive isotherms is 15°C . The isotherm of 100°C is not smooth. This is caused by the interpolation error in plotting the data, a limitation of the plotting software. Within the two-phase zone, the liquid-vapour mixture remains at the saturation point of 100°C . The results from 3D simulation are more realistic compared with those from a 2D simulation since the heat transfer field can be clearly seen in the whole domain. For different Peclet numbers, the temperature contours are all inclined towards the outlet due to the incoming sub-cooled liquid flow. The presence of the circulatory flow in

TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT TRANSFER IN POROUS MEDIA

the upper left-hand corner (Figs. 8-32a, 8-34a and 8-36a) of the plane actually hinders cooler liquid from reaching the heated section. This results in poor heat transfer from the heated section to the surrounding liquid. Therefore, denser temperature contours are observed in the region with circulatory flow. This indicates a large temperature gradient. For large Pe , the isotherms are prominently inclined towards the outlet due to the strong incoming flow. The increase of Pe increases the convection heat transfer between the heated section of the wall and the fluid. The region covered by the two-phase zone is therefore reduced greatly.

The variation of the liquid saturation s on the wall with heated section under different Peclet numbers is shown in Fig. 8-39. The liquid saturation s is kept at a constant value of 1 at the entrance for different inlet velocities, indicating a sub-cooled liquid at this region. However, it begins to drop as the liquid approaches the heated section of the wall, reaches a minimum somewhere within the heated section and then increases at the point in which the liquid leaves the heated section as a liquid-vapour mixture. Generally, s reduces along the direction of the height (z direction) as the vapour flows upward; indicating that the upper wall is relatively rich in vapour. Special attention should be paid to such problems in engineering applications. An implication of this result could be that in thermal systems with heat sources located at the side walls, a dry-out zone could appear near the upper wall within the heated section. The temperatures at these locations would increase significantly and local hot spots may be induced. This could be potentially disastrous for sensitive electronic devices and clearly demonstrates the need for detailed 3D studies to accurately capture the locations of minimum liquid saturation. The increase of Pe pushes

TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT TRANSFER IN POROUS MEDIA

the two-phase zone downstream. As a result, the location where s drops from 1 recedes away from the inlet. The minimum liquid saturation also increases with the increase of the inlet velocity.

Fig. 8-38 Temperature distributions in the 3D domain for $Ra = 226$ and (a) $Pe = 0.06$; (b) $Pe = 0.12$; (c) $Pe = 0.24$

(a)

(b)

(c)

Fig. 8-39 Liquid saturation on the wall with heated section for $Ra = 226$ and (a) $Pe = 0.06$; (b) $Pe = 0.12$; (c) $Pe = 0.24$

8.4.3.2 Effects of Rayleigh Number on Fluid Flow and Heat Transfer

Attention is now focused on the effects of Rayleigh number on the velocity, temperature and liquid saturation fields. For all cases, the Peclet number is set to 0.24.

The flow fields for both liquid and vapour under different Rayleigh numbers are shown in Figs. 8-40 to 8-41. Generally, a large Ra suggests significant buoyancy effect. In this case, buoyancy works against convection. At high Rayleigh number, $Ra = 226$, as shown in Fig. 8-32, the buoyancy force produces a strong circulatory flow in the xz plane. The recirculation deflects the liquid from inlet to flow downwards. Such a downward flow is accelerated in a narrow region adjacent to the lower wall. The circulatory flow reduces convection heat transfer near the upper wall while the accelerated liquid increases convection near the lower wall. Therefore, the two-phase zone blankets the upper wall in a larger region compared with that on the lower wall, leading to a more inclined condensation front. This is different from what was observed at low Rayleigh number. With the decrease of Ra , the buoyancy force becomes insignificant. This reduces the circulatory flow. At $Ra = 56$ (Fig. 8-42), the recirculation near the upper wall is almost curtailed by the incoming flow. The leading edge of the two-phase zone on the upper wall is pushed further downstream. The condensation front becomes steeper (Fig. 8-42a). Unlike the case for $Ra = 226$, no two-phase zone is observed in the yz plane with constant $x/H = 0.1$ at $Ra = 113$ and $Ra = 56$ (Figs. 8-40c and 8-42c). The vapour flows in a similar fashion for different Ra , having a direction which is primarily normal to the condensation front.

Fig. 8-40 Liquid velocity in different planes for $Pe = 0.06$ and $Ra = 113$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$

Fig. 8-41 Vapour velocity in different planes for $Pe = 0.06$ and $Ra = 113$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$

Fig. 8-42 Liquid velocity in different planes for $Pe = 0.06$ and $Ra = 56$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$

Fig. 8-43 Vapour velocity in different planes for $Pe = 0.06$ and $Ra = 56$ (a) the xz plane with $y/H = 0.5$; (b) the xy plane with $z/H = 0.5$; the yz plane at (c) $x/H = 0.1$; (d) $x/H = 0.3$; (e) $x/H = 0.5$

The temperature contours under $Ra = 113$ and $Ra = 56$ are shown in Figs. 8-44 (a) and 8-45(b), respectively. Comparisons are made against $Ra = 226$ in Fig. 8-32. As indicated by the temperature contours, a large region of the domain is blanketed by the two-phase zone. The buoyancy force assists the vapour moving from the heated section to expand to a wide region. The heat is not only expelled from the heated source downstream, but also swept upstream. An implication of this result in engineering applications could be that the region which is rich in vapour can be significantly far away from the heat source. Hence, a detailed study of two-phase in a 3D porous system becomes necessary.

**TEST PROBLEMS FOR TWO-PHASE FLOW AND HEAT
TRANSFER IN POROUS MEDIA**

Figure 8-45 shows the distribution of the liquid saturation on the wall with heated section for $Ra = 113$ and $Ra = 56$. A similar profile is observed for different Rayleigh numbers. For flow with large Ra , a larger s is to be expected. For example, s reaches a minimum of about 0.2 for $Ra = 56$, but 0.3 for $Ra = 113$ and 0.4 for $Ra = 226$. Of particular interest is the effect of the buoyancy force, although it helps to spread the heat to a large region of the domain, it also, to some extent, helps to promote mixing of the fluid, reducing the liquid saturation in some local spots. Therefore, the minimum liquid saturation occurs at low Rayleigh number.

Fig. 8-44 Temperature distributions in the 3D domain for $Pe = 0.06$ and (a) $Ra = 113$; (b) $Ra = 56$

Fig. 8-45 Liquid saturation on the wall with heated section for $Pe = 0.06$ and (a) $Ra = 113$; (b) $Ra = 56$

8.5 SUMMARY

In this chapter, the developed 2D and 3D models were applied to a variety of the problems. For these cases, the evolution of the boiling process is discussed. The velocity and the temperature fields as well as the liquid saturation distribution were investigated thoroughly. The results showed that the flow behaviour and heat transfer mode are affected significantly by the occurrence of the vapour in the two-phase zone. The inlet velocity and permeability of the porous media also have strong influence on the velocity and temperature fields. The importance of the 3D study is the prediction of the exact locations of local hot spots. With both the 2D and 3D models, a large class of phase change flow problems can be solved.

CHAPTER 9 EXPERIMENTAL STUDIES OF FLOW BOILING IN POROUS MEDIA

In this chapter, experimental results for flow boiling in porous metal foams are presented. The data obtained from the experiments are compared with the simulation results based on the developed 3D codes illustrated in Chapter 8.

9.1 TEST SPECIMEN AND EXPERIMENTAL SETUP

9.1.1 Test Specimen

The study of air flow in graphite foam reveals that the pressure drop in block graphite foam is rather high. The pressure drop will be higher if a liquid is forced to flow into the graphite foam. A very high head pump will be required if graphite foam were to be chosen for the two-phase experiments. Therefore, in the current phase change experiments, aluminium foam is chosen instead. Figure 9-1(a) shows the test specimen of “DUOCEL” aluminium foams of pore density 40 pores per lineal inch (PPI). This metal foam was manufactured by ERG Materials and Aerospace Corporation, USA using a sintering technique. It has reticulated structures where the pores are interconnected to each other. The ligament diameter of the tested aluminium foam was measured by a SEM as shown in Fig. 9-1(b). Ten ligaments were measured randomly to obtain the average diameter, d_l . For each ligament, three diameters ($d_1 \sim d_3$) at different positions were measured. Other parameters of the metal foam, such as its porosity, inertial coefficient and permeability were determined using the same

method as that pertaining to graphite foams discussed in Chapter 6. The effective thermal conductivity of the metal foam when it is saturated with water was calculated based on the model proposed by Calmidi (2000). The values of these parameters are listed in Table 9-1.

Fig. 9-1 (a) Tested aluminium foams and (b) pores under SEM

Table 9-1 Characteristic properties of metal foams

ε	d_l^* (μm)	K^* (10^{-8} m^2)	C_E^*	k_{se} ($\text{W/m}\cdot\text{K}$)	k_{fe} ($\text{W/m}\cdot\text{K}$)
0.90	112.6	2.8	0.087	7.82	0.57

9.1.2 Experimental Setup

The schematic diagram of the experimental facility is shown in Fig. 9-2(a). A pump manufactured by EHEIM Company with maximum head pressure of 3.1 m was used to drive water from the reservoir to the rectangular channel. The sectional view of the channel is shown in Fig. 9-2(b). The channel is made of Teflon with inner dimensions of 350 mm (L) $\times$ 50 mm (W) $\times$ 10 mm (H). The upper wall of the channel is designed for the placement of thermocouples. An aluminium foam of dimensions 50 mm (L) $\times$ 50 mm (W) $\times$ 10 mm (H) is placed into the channel. The length before the fluid flows into the aluminium foam was determined such that a fully developed velocity profile can be achieved at the

entrance of the foam. A 30-mm-long cubic copper block is positioned at the bottom of the channel to serve as the heat source. Three small heaters from Watlow Company with a total heating capacity of 675 W are embedded in this copper block. The power input was monitored by a heater controller. The inlet velocities were determined from the flow rates measured by a rotameter with an accuracy of $\pm 2.5\%$. Ten type K thermocouple probes with diameter of 1.5 mm are embedded on the bottom of the test foams. These thermocouples are installed at 5, 15, 25, 35, 45 mm from the inlet of the test foam along the flow direction. Among these, five are fixed at the centre while the others are located 20 mm further away from the centreline in the width direction. Three Type K “Omega” miniature thermocouples with wire diameter 0.0127 mm (0.0005”) are attached on the surface of the copper block to measure the surface temperatures of the heater. The accuracy of the thermocouple is within 0.4%. With such accuracy, the temperature of the heater could be reported precisely. These thermocouples are linked to a data acquisition system (Yokogawa MW100) which is directly connected to a personal computer. Thermal grease ($k = 2.3 \text{ W/m}\cdot\text{K}$) was applied as filling material to reduce the thermal contact resistances across the interfaces of the heater, copper plate and test sample. Before the commencement of the experiments, all the thermocouples were calibrated carefully to an accuracy of $\pm 0.5^\circ\text{C}$.

(a)

(b)

Fig. 9-2 (a) Schematic diagram of experimental setup (b) Sectional view of the test channel

9.1.3 Experimental Procedure

Before the commencement of the experiments, the fluid flow rate was first set to the maximum value to flush out the air trapped in the system. The heating power was then activated. The power was varied by controlling the voltage across the heater. With Teflon used as the cover of the test section, the heat loss can be considered negligible. Each experiment was started at low heat flux such that nucleate boiling had not occurred. The voltage of the heat power supply was increased at an interval 5V. At each voltage level, the temperature at the heater surface and the temperature at the substrate of metal foam were

monitored every 3 minutes to ensure that steady state had been achieved. The steady state condition was assumed to be reached if the maximum temperature variation indicated in each of the thermocouple was within 0.5°C . The time to achieve steady state was varied with liquid flow rate and heating power. The same experimental procedure was repeated while changing the test samples. The entire system was cooled to the ambient temperature before another experimental run.

To study the hysteresis effect when fluid flows to metal foam, the heating power was first increased until certain level and then reduced. During this process, the system was left alone for sufficient time to ensure that steady state had been achieved. Such process typically took a few hours.

9.1.4 Uncertainty Analysis

The experimental errors were estimated based on the accuracies in the measurement of the individual quantities. The accuracies of the thermocouples are $\pm 0.5^{\circ}\text{C}$. The power input to the heater was adjusted by a variac and current as well as voltage across the heater. This was calibrated with a digital multimeter. The error in calculating the heating flux is 3%. The accuracy of the flow meter is $\pm 2.5\%$.

9.2 RESULTS AND DISCUSSION

9.2.1 Nucleate Boiling

Figures 9-3 shows the variations of wall superheat under different heat fluxes in 40 PPI aluminium foam. These data were obtained at a flow rate of 0.01 L/min for water. The experiments were started at low heat flux where nucleate boiling

had not occurred. Under these low heat fluxes, the solid ligament of the aluminium foam conducted the heat from the heated surface to the ligaments. Then, the liquid flows across the ligaments and convects the heat downstream. This heat transfer mode will become less dominant after the onset of nucleate boiling. The transition from conduction-convection heat transfer mode to nucleate boiling occurs at a heat flux of 10 W/cm^2 for water with a wall surface superheat of about 10°C . This can be observed from the distinct change in the slopes of the boiling curves in Fig. 9-3. In the boiling process of aluminium foams, nucleate boiling usually starts with the formation of bubbles at the nodes where the ligament of the metal foam is tightly attached on the heater surface. It spreads to the bottom surface quickly with the increase of heat flux. As the heat flux is increased, bubbles would be generated above the heated surface if the ligament is sufficiently superheated. Therefore, the ligaments of the aluminium foam promote bubble incipience. Once nucleate boiling had fully commenced, the liquid and vapour inside the aluminium foam remained saturated. This was verified by the measured temperatures at the centre of the test sample as shown in Fig. 9-4. When the heat flux reaches 10 W/cm^2 , the bottom temperature at the centre of the aluminium foam increases to almost 100°C , indicating the onset of the nucleate boiling. Of particular importance in Fig. 9-4 is the value of heat flux for the onset nucleate boiling predicted by the numerical simulations. The predicted onset of nucleate boiling is at a heat flux of about 9.5 W/cm^2 . This is very close to that obtained from the experiments and demonstrates the capability of the current code in the simulation of two-phase flow and heat transfer in porous media. With this achieved, the current code can be used to predict the heat transfer in porous media with confidence.

Fig. 9-3 Boiling curves in aluminium foam with 40 PPI

Fig. 9-4 Comparisons of the temperatures at the bottom and the centre of the test sample for water

Figure 9-5 (a) shows the local temperature distribution along the centreline for a heat flux $q'' = 10 \text{ W/cm}^2$. The liquid temperature increases slowly at the entrance and then increases sharply when the liquid approaches the heated surface. It reaches the maximum value of the saturated temperature within the heated surface and then begins to drop before the fluid flows out of the heated surface. A comparison between the numerical results and experimental data on the temperature trend reveals strong similarities. Such a temperature trend can be explained by the knowledge of the flow features when the buoyancy force is considered in the problem. As the heated liquid approaches the heated surface,

its temperature increases and hence its density decreases. Driven by the buoyancy force, the higher temperature liquid adjacent to the heated surface flows upward while the lower temperature liquid further away from the heated surface flows downwardly to replenish the vacated fluid. This downward flow stream reduces the temperature henceforth. Figure 9-5(b) shows this phenomena from the simulation results of the velocity distribution near the heat surface. It is noticed that the predicted temperatures on the heated surface are in good agreement when compared against the experimental data. The first and the last points which are located before and after the heated surface show large discrepancies which could be attributed to lateral conduction from the heated surface to the unheated surface. Vapour generated above the heated surface blankets it and greatly reduces heat transfer to the fluid. This could, however, promote lateral conduction heat transfer from the heated surface to the adjacent unheated surface. In the simulations, the unheated surface is assumed to be adiabatic. Therefore, it is not surprising that the measured temperatures are higher than the predicted results.

9.2.2 Hysteresis Effect

Of particular interest in the study of phase change in porous media is the hysteresis effect. This is caused by contact angle hysteresis which is partly responsible for the hysteresis in the phase distributions. To investigate the hysteresis effect, temperatures on the heater surface were measured for both the heating and cooling processes. During the cooling process, the system was left idle for sufficient time to reach the steady state. Figures 9-6 show the temperature variations under these two processes. An obvious temperature

excursion can be observed for water. This clearly demonstrates the effect of hysteresis. Such hysteresis is important to the cooling of the electronic components.

(a)

(b)

Fig. 9-5 (a) The variation of temperature along flow direction at heat flux $q'' = 10 \text{ W/cm}^2$; (b) velocity distribution at xy plane

Fig. 9-6 Hysteresis effects in aluminium foam with 40 PPI

9.2.3 Mass Fraction of Vapour

In the experiments, the liquid and vapour were separated when they flow out of the test section. The separated liquid was cooled to the ambient temperature and then the volume of the liquid was measured by the graduated flask. Such amount of the liquid was subtracted from the inlet flow rate of the liquid, and the result would be the vapour generated in the system. The amount of separated liquid was measured five times and the average value was used to compare with the simulation results. Figure 9-7 shows the comparison between the simulation and the experimental results of the mass fractions of vapour under flow rate of 0.01 L/min. This mass fraction is represented by m_v/m , where m_v is the mass of vapour generated in the system and m is the mass for the liquid flows into the system. Generally, high heat flux results in the conversion of a large quantity of liquid to vapour. The mass fraction of vapour in the system would then increase correspondingly. Simulation can predict the trend of the mass fraction of vapour with the heat flux quantitatively and the comparison between the simulation and experiments reveal strong similarities.

The variation of mass fraction of vapour with the inlet flow rate at heat flux of 13.7 W/cm^2 is depicted in Fig. 9-8. Generally, a smaller inlet velocity suggests a longer residence time during which liquid remains in the test domain. The fluid absorbs more heat, leading to the generation of more vapour. Therefore, a high mass fraction of vapour is expected when the inlet flow rate decreases. Again, the numerical profile has the same trend as the experimental results.

Although the numerical results for both the local temperature distribution and the mass fraction of vapour have similar trends as those obtained from the

experimental results, some relatively large discrepancies are found. These discrepancies between the experimental and simulation results could be due to several reasons. First, some of the empirical relationships, such as the capillary pressure and the relative permeabilities, were obtained from data for packed beds. This may not be suitable for the case of the tested aluminium foam which has a large porosity. Secondly, there are many assumptions in the mathematical formulations such as the homogeneous porous media as well as the adiabatic boundary conditions. Such conditions cannot be realised in the experiments. In addition to these two reasons, there are some other issues that remain to be resolved in order to achieve quantitative agreement between the experiments and simulation. These include the study of the thermal dispersion effects and the modelling of the thermal conductivity of the aluminium foams.

Fig. 9-7 Variation of mass fraction of vapour with heat flux

Fig. 9-8 Variation of mass fraction of vapour with flow rate

9.3 SUMMARY

In this chapter, the experimental results of flow boiling in porous metal foam are presented. The results are compared with the numerical simulation based on the developed 3D code. The experimental results show that the nucleate boiling starts at a heat flux of 10 W/cm^2 which is close to the simulation result of 9.5 W/cm^2 . It was found that hysteresis exists when water flows through the 40-PPI-aluminium foam. The comparison of the mass fraction of vapour between the numerical and experimental results reveals similar trends. Reasonable agreement was achieved. The capability of the current model to simulate flow boiling in porous media is demonstrated.

CHAPTER 10 CONCLUSIONS AND RECOMMENDATIONS FOR FUTURE WORK

In this chapter, the major findings of this thesis are stated. The main contributions of the present work are highlighted. This is followed by some suggestions for future research.

10.1 CONCLUSIONS

In this thesis, numerical procedures for single and two-phase flows in porous media are developed based on the finite volume method (FVM). For both codes, the procedures are formulated for both 2D and 3D systems. These procedures are applied to a variety of problems and more importantly to compare with the present experimental results.

10.1.1 Single-Phase Flow and Heat Transfer

In the single-phase flow, a general numerical procedure is introduced to study the transport phenomena in porous media of different configurations. The introduced procedure is capable of dealing with the interface between the porous and open regions without any special treatment. This has been employed successfully in porous media with the shapes of staggered porous block (2D), zigzag (3D) and baffle (3D). Further validation of the proposed numerical procedure is achieved through experimental studies. In the experimental studies for single-phase flow, the porous graphite foam developed in ORNL, USA (Klett, 2000) is used for the validation purpose. These graphite foams are

CONCLUSIONS AND RECOMMENDATIONS FOR FUTURE WORK

fabricated into zigzag and baffle shapes. Comparisons are made for the pressure drops and the average Nusselt number in the graphite foam between the experimental and numerical results. Reasonable agreement is achieved. Based on the numerical and experimental studies for graphite foam with different shapes, the following conclusions can be drawn:-

- (1) A unit cell model is developed to represent the microscopic geometry of graphite foam. It can be used to calculate the effective thermal conductivity of porous graphite foams. The model can be changed easily to accommodate pores in the graphite foam that have different number of holes. This makes it more flexible than the existing models (Klett *et al.*, 2004; Yu *et al.*, 2006; Tee *et al.*, 2008).
- (2) The study of pressure drops in the different shapes of graphite foams show that the BLK foam has the highest pressure drop compared with the ZZG and BAF foams. The pressure drop in the BLK foam is more than 2 times larger than in the ZZG foam and almost 5 times larger than in the BAF foams at a fixed inlet velocity. The numerical results of the velocity field show that most of the fluid passes through the structured slots in the ZZG and BAF graphite foams and a small portion of the fluid could penetrate the designed thin foam walls. However, due to the interlaced foam walls in the ZZG and BAF foams, the quantity of fluid flow through the pores inside the thin foam walls is greatly increased.
- (3) Both the numerical and experimental studies show that the Nusselt number for different configurations of graphite foams increases with an increase of Reynolds number. The BLK foam offers the best heat transfer performance with the most uniform temperature at the substrate of the foams compared with

CONCLUSIONS AND RECOMMENDATIONS FOR FUTURE WORK

the ZZG and BAF foams under the same Reynolds number. The simulation of temperature distribution inside the graphite foams shows that the temperature difference between the solid and fluid phases is not significant in the BLK foam. However, this temperature difference cannot be neglected in the ZZG and BAF foams. For these two structured foams, the LTNE model should be used.

(4) The guideline of using graphite foams in actual application is that the BLK foam can be used to maximise the enhancement of heat transfer performance in cases where pumping power is not a major concern. The other configurations of graphite foam can be chosen as heat sinks depending on the heat flux, where the system is sensitive to pressure drop. In addition, the appropriate structure of graphite foams can be designed to achieve the corresponding heat removal capacity for the specific application.

(5) The agreement of the experimental and numerical results demonstrates the capability of the developed numerical procedure for simulation fluid flow and heat transfer in different configurations of porous media serving as heat sinks. With this achieved, such a numerical procedure can be used in preliminary designs of heat sinks for practical thermal engineering applications.

10.1.2 Two-Phase Flow and Heat Transfer

In two-phase flow, a numerical procedure is proposed based on the two-phase mixture model for studying the boiling process in porous media in both 2D and 3D systems. The model is tested and validated in a variety of problems, viz. two-phase flow and heat transfer in a 2D horizontal porous channel, a 2D vertical porous channel, and 3D asymmetrically heated porous channel. Furthermore, the current model is also validated with experimental work. An experimental

CONCLUSIONS AND RECOMMENDATIONS FOR FUTURE WORK

study of the flow boiling process through metal foams with 40 PPI is undertaken. Comparisons between the experimental and simulation results are made on the local temperature distribution at the substrate of the metal foam and the mass fraction of vapour. Qualitative agreements are achieved. The prediction of the heat flux for the onset of nucleate boiling is close to the experimental data. Some conclusions can be drawn based on the numerical and experimental studies of the two-phase flow in porous media:-

(1) Through the comparison between the traditional harmonic mean method and the “modified” Kirchhoff method in dealing with the discontinuity in the thermal diffusion coefficient, it is found that the “modified” Kirchhoff method has more advantages over the traditional harmonic mean method. Using the “modified” Kirchhoff method, a very smooth interface between the liquid and vapour phases can be obtained.

(2) The numerical study of two-phase flow in the horizontal porous channel show that the location of the heat flux has a significant effect on fluid flow and heat transfer. Generally, the liquid in the sub-cooled liquid zone bypasses the two-phase zone as the latter presents a large resistance to the liquid flow. The liquid in the two-phase zone flows to the heated surface while the vapour in the two-phase zone flows primarily to the condensation front. The counter-percolation movement of the liquid and vapour within the two-phase zone is the main heat transfer mechanism.

(3) The numerical study of two-phase flow in the vertical porous channel show that the velocity, temperature and liquid saturation fields for both aiding and opposing flows are totally different under various Rayleigh and Peclet numbers. For the case of aiding flow, the liquid velocity field indicates that a

CONCLUSIONS AND RECOMMENDATIONS FOR FUTURE WORK

recirculatory flow exists in a region far away from the heated surface. However, in the case of opposing flow, the recirculatory flow appears near the heated section of the right side wall. The recirculatory flow under both aiding and opposing flows will be gradually suppressed by the expansion of the two phase zone under certain Rayleigh and Peclet numbers. The liquid saturation study indicates that for aiding and opposing flows, the liquid saturation s increases with the increase of Ra and Pe . The minimum s , for the case of aiding flow, is located at the trailing edge of the heated section. However, the minimum s for opposing flow occurred at some location within the heated section and shifts upstream with the increase of Ra and decrease of Pe .

(4) The experimental study of water flow through 40-PPI-aluminium foam shows that the transition from conduction-convection heat transfer mode to nucleate boiling occurs at a heat flux of 10 W/cm^2 with a wall surface superheat of about 10°C . Hysteresis and temperature excursion exist with both heating and cooling processes.

(5) The comparison between the numerical and experimental studies on both the substrate temperature of the aluminium foam and the mass fraction of vapour show reasonable agreements. With these achieved, the capability of the developed code in the prediction of two-phase heat transfer in porous media is demonstrated.

10.2 CONTRIBUTIONS

The present research has resulted in the development of in-house codes for single and two-phase flow and heat transfer in porous media. There are two components in the current work: (1) single-phase flow and heat transfer in a

CONCLUSIONS AND RECOMMENDATIONS FOR FUTURE WORK

porous channel with different configurations and (2) two-phase flow and heat transfer in a porous channel. For each of the component, both 2D and 3D codes based on the Cartesian grid are developed. These codes are validated against related published works and more importantly against the present experimental results. Given below is the list of the main contributions of this thesis.

(1) A unit cell model is proposed to represent the microstructures of the graphite foam (Leong and Li, 2011). The model is flexible and can be used to reflect any particular pore in graphite foam with slight change in its geometry. It is used to calculate the effective thermal conductivity of porous graphite foams. The results on the effective thermal conductivity obtained from this model agree very well with existing data in the literature (Klett *et al.*, 2004; Yu *et al.*, 2006; Tee *et al.*, 2008).

(2) A general simplified numerical procedure based on a 2D setting to simulate fluid flow and heat transfer in a channel with both porous and open regions is presented (Li *et al.*, 2010a). With this achieved, a 3D code was developed to investigate the single-phase transport phenomena in porous media of different configurations. It had been employed successfully to study single-phase flow in porous media with the shapes of zigzag and baffle. This is the first attempt to numerically investigate fluid flow and heat transfer in such configurations (Leong *et al.*, 2010b, Leong *et al.*, 2010c).

(3) Experimental studies are performed based on different porous media configurations made of graphite foam for validation purpose. The physical properties of graphite foams such as the porosity, pore diameter, density, permeability and Forchheimer coefficient are obtained from the experiments. These measured physical properties are particularly useful for engineers who

CONCLUSIONS AND RECOMMENDATIONS FOR FUTURE WORK

are interested in the use of these graphite foams for heat transfer applications. The temperature distributions on the substrate of graphite foam with different configurations are measured and these are compared against the numerical predictions. The comparisons show that the numerical prediction and the experimental results are in good agreement. This demonstrates the capability of the developed code in the prediction of heat and fluid flow in channel with porous media serving as heat sink. With these, the current code can be convincingly used to predict flow and heat transfer in more complex geometries with porous media heat sinks generally found in practical engineering applications.

(4) Apart from the single-phase study, the current work also studied two-phase flow and heat transfer in porous media. Specifically, this part of the study is aimed at phase change heat transfer that is capable of removing a substantially large amount of heat in a cooling system. Of particular importance in the current model is the treatment of the discontinuity in the diffusion coefficient. The “modified” Kirchhoff method is implemented instead of the traditional harmonic mean method. With this, a smoother interface can be achieved between the sub-cooled liquid and two-phase regions (Li *et al.*, 2010b). The current numerical model has been employed successfully for transient studies of the boiling processes in horizontal (Li *et al.*, 2010b) and vertical porous channels (Li *et al.*, 2010c) for a 2D setting. Upon achieving that, a 3D code was developed in the spirit of the 2D code for a more realistic simulation of two-phase flow and heat transfer in porous channel aimed at a large class of phase change flow problems (Li *et al.*, 2010d). This is the first full-fledged 3D study of two-phase flow and heat transfer in porous media.

CONCLUSIONS AND RECOMMENDATIONS FOR FUTURE WORK

(5) Experimental work for flow boiling in metal foam is carried out for validation purpose (Li and Leong, 2011). The onset of nucleate boiling is studied. The local temperature distribution at the bottom of the metal foam is obtained. Comparisons are made between the experimental and numerical results. Reasonable agreement is achieved. The prediction of the heat flux for the onset of nucleate boiling is close to the experimental data. The agreement between the numerical and experimental results demonstrates the capability of the developed code in the prediction of two-phase heat transfer in porous media.

10.3 RECOMMENDATIONS FOR FUTURE WORK

The following is a list of possible extensions of the present work:-

- (1) A unit cell model to predict the effective thermal conductivity of graphite foams is proposed in the current thesis. Although the results from the current model has been compared with existing models as well as the experimental data from other researchers, further experimental validation of the current model in predicting the effective thermal conductivity for graphite foam with different porosities is necessary.
- (2) Although the graphite foam has proven to be a good candidate material for heat sinks compared with traditional heat sinks, further experimental studies are needed to test its mechanical stability when different liquids are used as coolants.
- (3) The 3D numerical model developed for single-phase flow and heat transfer in porous media in this thesis is sophisticated enough to handle complex structures of porous media. Future work can be focused on the applications of this model to predict fluid flow and heat transfer in practical situations. For the

CONCLUSIONS AND RECOMMENDATIONS FOR FUTURE WORK

numerical work, parallel simulation could be implemented in the code to save computational time.

(4) Due to the small capacity of the heater in the current experimental setup, the simulation results on phase change in porous media are validated only against the heat flux corresponding to the onset of nucleate boiling in the experiments. Further validation of the current model to predict the heat flux for film boiling would be necessary by using a larger capacity heater or a coolant with a low saturation temperature.

(5) The present two-phase mixture model for simulation of phase change in porous media is only valid for low fluid velocity. Although it is satisfied in many practical systems, a general model which is valid for both low and high velocities is necessary. Such a model would involve more empirical relationships, such as the relative permeability, the capillary pressure. These relationships can be obtained through further experimental studies.

(6) Recently, the Lattice Boltzmann Method (LBM) has been developed as a new tool for simulation of fluid flow and heat transfer in porous media (Zhao *et al.*, 2010; Yan *et al.*, 2011). This method is a micro and mesoscale modelling method based on particle kinematics. With LBM, fluid flow and heat transfer at the pore level of the porous media can be revealed. This could be a potential research method for porous media instead of the local volume-averaging method used in the current thesis.

REFERENCES

- Afshari, E., and Jazayeri, S.A., Analyses of heat and water transport interactions in a proton exchange membrane fuel cell, *Journal of Power Sources*, Vol. 194, pp. 423-432 (2009)
- Alam, M.K., and Maruyama B., Thermal conductivity of graphitic carbon foams, *Experimental Heat Transfer*, Vol. 17, pp. 227-241 (2004)
- Alam, M.K., Druma, A.M. and Druma, C., Thermal transport in graphitic carbon foams, *Journal of Composite Materials*, Vol. 38, pp. 1993-2006 (2004)
- Alazmi B., and Vafai K., Analysis of fluid flow and heat transfer interfacial conditions between a porous medium and a fluid layer, *International Journal of Heat and Fluid Flow*, Vol. 44, pp. 1735-1749 (2001)
- Alazmi, B., and Vafai, K., Constant wall heat flux boundary conditions in porous media under local thermal non-equilibrium conditions, *International Journal of Heat and Mass Transfer*, Vol. 45, pp. 3071-3087 (2002)
- Alazmi, B., and Vafai, K., Analysis of variable porosity, thermal dispersion, and local thermal nonequilibrium on free surface flows through porous media, *ASME Journal of Heat Transfer*, Vol. 126, pp. 389-399 (2004)
- Albriola, L.M., and Pinder, G.F., A multiphase approach to the modelling of porous media contamination by organic compounds I: equation development, *Water Resources Research*, Vol. 21 pp. 11-18 (1985)
- Amiri, A., and Vafai, K., Analysis of dispersion effects and non-thermal equilibrium, non-Darcian, variable porosity incompressible flow through porous media, *International Journal of Heat and Mass Transfer*, Vol. 37, pp. 939-954 (1994)
- Amiri, A., Vafai, K., and Kuzay, T.M., Effects of boundary conditions on non-Darcian heat transfer through porous media and experimental comparison, *Numerical Heat Transfer, Part A*, Vol. 27, pp. 651-664 (1995)
- Anderson T.M., and Mudawar, I., Microelectronic cooling by enhanced pool boiling of a dielectric fluorocarbon liquid, *ASME Journal of Heat Transfer*, Vol. 111, pp. 752-759 (1989)
- Athreya, B.P., and Mahajan, R.L., Pool boiling of FC-72 over metal foams: effect of foam orientation and geometry, 8th *AIAA/ASME Joint Thermophysics and Heat Transfer Conference*, 24-26 June 2002, St. Louis, Missouri, USA

- Bau, H.H., and Torrance, K.E., Boiling in low-permeability porous material, *International Journal of Heat and Mass Transfer*, Vol. 25, pp. 45-55 (1982a)
- Bau, H.H., and Torrance, K.E., Thermal convection and boiling in a porous medium, *Letters in Heat and Mass Transfer*, Vol. 9, pp. 431-441 (1982b)
- Bau, H.H., and Torrance, K.E., Low Rayleigh number thermal convection in a vertical cylinder filled with porous materials and heated from below, *ASME Journal of Heat Transfer*, Vol. 104, pp. 166-172 (1982c)
- Bear, J., and Bachmat, Y., *Introduction to Modelling of Transport Phenomena in Porous Media*, Kluwer Academic, Dordrecht, (1990)
- Bear, J., and Bensabat, J., Advective fluxes in multiphase porous media under non-isothermal conditions, *Transport in Porous Media*, Vol. 4, pp. 423-448 (1989)
- Beavers, G., and Joseph, D.D., Boundary conditions at a naturally permeable wall, *Journal of Fluid Mechanics*, Vol. 30, pp. 197-207 (1967)
- Betchen, L., Straatman, A.G., and Thompson, B.E., A non-equilibrium finite volume model for the conjugate fluid/porous/solid domain, *Numerical Heat Transfer Part A*, Vol. 49, pp. 543-565 (2006)
- Bhattacharya, A., and Mahajan, R.L., Finned metal foam heat sinks for electronics cooling in forced convection, *ASME Journal of Electronic Packaging*, Vol. 124, pp. 155-163 (2002)
- Boberg, T., *Thermal Methods of Oil Recovery*, John Wiley, New York, 1988
- Boomsma, K., Poulikakos, D., and Ventikos, Y., Simulation of flow through open cell metal foams using an idealized periodic cell structure, *International Journal of Heat and Fluid Flow*, Vol. 24, pp. 825-834 (2003)
- Bubnovich, V., and Toledo M., Analytical modelling of filtration combustion in inert porous media, *Applied Thermal Engineering*, Vol. 27, pp. 1144-1149 (2007)
- Calmidi, V.V., and Mahajan, R.L., The effective thermal conductivity of high porosity fibrous metal foams, *ASME Journal of Heat Transfer*, Vol. 121, pp. 466-471 (1999)
- Calmidi, V.V., and Mahajan, R.L., Forced convection in high porosity metal foams, *ASME Journal of Heat Transfer*, Vol. 122, pp. 557-565 (2000)
- Chapuis, O., Prat, M., Quintard, M., Chane-Kane, E., Guillot, O., and Mayer, N., Two-phase flow and evaporation in model fibrous media: Application to the gas diffusion layer of PEM fuel cells, *Journal of Power Sources*, Vol. 178, pp. 258-268 (2008)

- Chen, Z.Q., Cheng, P., and Zhao, T.S., An experimental study of two phase flow and boiling heat transfer in bi-dispersed porous channels, *International Communications in Heat and Mass Transfer*, Vol. 22, pp. 293-302 (2000)
- Cheng, P., Heat transfer in geothermal systems, *Advances in Heat Transfer*, Vol. 14, pp.1-105 (1978)
- Chikh, S., Boumedien, A., Bouhade, K., and Lauriat, G., Analytical solution of non-Darcian forced convection in an annular duct partially filled with a porous medium, *International Journal of Heat and Mass Transfer*, Vol. 38, pp. 1543-1551 (1995)
- Chung, M., and Catton, I., Post dry-out heat transfer in a multi-dimensional porous bed, *Nuclear Engineering and Design*, Vol. 128, pp. 289-304 (1991)
- Costa, V.A.F., Oliveira L.A., Baliga B.R., and Sousa A.C.M., Simulation of coupled flows in adjacent porous and open domains using a control-volume finite-element method, *Numerical Heat Transfer Part A*, Vol. 45, pp. 675-697 (2004)
- Crank, J., *Free and Moving Boundary Problems*, Clarendon Press, Oxford, (1984)
- Darcy, H.P.G., *Les Fontaines Publiques de la Ville de Dijon* (in French), Victor Dalmont, Paris (1856)
- Docca, A., Reclaim unusable data centre capacity through integrated IT and Facilities, *Uptime Institute Symposium*, May 9-12, Santa Clara, CA (2011)
- Dullien, F.A., *Porous Media: Fluid Transport and Pore Structure*, Academic, New York, 2nd edition (1992)
- Easterday, O.T., Wang, C.Y., and Zheng, P., A numerical and experimental study of two-phase and heat transfer in a porous formation with localized heating from below, *Proceedings of ASME International Mechanical Engineering Congress and Exposition*, San Francisco, USA, pp. 723-732 (1995)
- Ergun, S., Fluid flow through packed column, *Chemical Engineering Progress*, Vol. 48, pp. 89-94 (1952)
- Estes, K.A., and Mudawar, I., Comparison of two-phase electronic cooling using free jets and sprays, *ASME Journal of Electronic Packaging*, Vol. 117, pp. 323-332 (1995)
- Faghri A., *Heat Pipe Science and Technology*, Taylor & Francis, Washington, DC (1995)

- Farsi, H., Joly, J.L., Moiscevic, M., Platel, V., and Mazet, N., An experimental and theoretical investigation of the transient behaviour of two-phase closed thermosyphon, *Applied Thermal Engineering*, Vol. 23, pp. 1895–1912 (2003)
- Fu, H.L., Leong, K.C., Huang, X.Y., and Liu, C.Y., An experimental study of heat transfer of a porous channel subjected to oscillating flow, *ASME Journal of Heat Transfer*, Vol. 123, pp. 162-170 (2001)
- Fu, W.S., Huang, H.C., and Liou, W.Y., Thermal enhancement in laminar channel flow with a porous block, *International Journal of Heat and Mass Transfer*, Vol. 39, pp. 2165-2175 (1996)
- Gallego, N.C., and Klett, J.W., Carbon foams for thermal management, *Carbon*, Vol. 41, pp. 1461-1466 (2003)
- Garimella, S. V., and Singhal, V., Single-phase flow and heat transport and pumping considerations in microchannel heat sinks, *Heat Transfer Engineering*, Vol. 25, pp. 15-25 (2004)
- Garman, R.W., and Elwell, R.J., Thermal performance of a graphite foam material with water flow for cooling power electronics, http://www.ms.ornl.gov/researchgroups/CMT/foam/PCIM_2001_Paper.pdf (2002), accessed on 3 Aug. 2008
- Gartling, D.K, Hickox, C.E., and Givler, R.C., Simulation of coupled viscous and porous flow problems, *International Journal of Computational Fluid Dynamics*, Vol. 7, pp. 23-48 (1996)
- Gima, S., Nagata, T., Zhang, X., and Fuji, M., Experimental study on CPU cooling system of closed-loop two phase thermosyphon, *Journal of Heat Transfer – Asian Research*, Vol. 34, pp. 147-159 (2005)
- Gray, W.A., Leijnse, A., Kolar, R.L., and Blain, C.A., *Mathematical Tools for Changing Spatial Scales in the Analysis of Physical Systems*, CRC press, Boca Raton, Florida (1993)
- Gray, W.G., General conservation equations for multi-phase systems: 4. constitutive theory including phase change, *Advances in Water Resources*, Vol. 6, pp. 130-140 (1983)
- Hsu, C.F., Sparrow, E.M., and Patankar, S.V., Numerical solution of moving boundary problems by boundary immobilization and a control-volume-based finite-difference scheme, *International Journal of Heat and Mass Transfer*, Vol. 24, pp. 1335-1343 (1981)
- Hsu, C.T., and Cheng, P., Thermal dispersion in a porous medium, *International Journal of Heat and Mass Transfer*, Vol. 33, pp. 1587-1597 (1990)

- Hsu, C.T., Cheng, P., and Wong, K.W., A lumped-parameter model for stagnant thermal conductivity of spatially periodic porous media, *ASME Journal of Heat Transfer*, Vol. 117, pp. 264-269 (1995)
- Huang, P.C., and Vafai, K., Analysis of forced convection enhancement in a channel using porous blocks, *Journal of Thermophysics and Heat Transfer*, Vol. 8, pp. 563-573 (1994)
- Hwang, G.J., Wu, C.C., and Chao, C.H., Investigation of non-Darcian forced convection in an asymmetrically heated sintered porous channel, *ASME Journal of Heat Transfer*, Vol. 117, pp. 725-732 (1995)
- Jeng, T.M., Tzeng, S.C., and Hung, Y.H., An analytical study of local thermal equilibrium, *International Journal of Heat and Mass Transfer*, Vol. 49, pp. 1907-1914 (2006)
- Jiang, P.X., and Lu, X.C., Numerical simulation and theoretical analysis of thermal boundary characteristics of convection heat transfer in porous media, *International Journal of Heat and Fluid Flow*, Vol. 28, pp. 1144-1156 (2007)
- Jiang, P.X., and Ren, Z.P., Numerical investigation of forced convection heat transfer in porous media using a thermal non-equilibrium model, *International Journal of Heat and Fluid Flow*, Vol. 22, pp. 102-110 (2001)
- Jiang, P.X., Ren, Z.P., Wang, B.X., and Wang, Z., Numerical simulation of forced convection heat transfer in porous plate channels using thermal equilibrium and non-thermal equilibrium models, *Numerical Heat Transfer Part A*, Vol. 35, pp. 99-113 (1999)
- Jiang, P.X., Xu, Y.J., Lv, J., Shi, R.F., He, S., and Jackson, J.D., Experimental investigation of convection heat transfer CO₂ at super-critical pressures in vertical mini-tubes and in porous media, *Applied Thermal Engineering*, Vol. 24, pp. 1255-1270 (2004)
- Jin, L.W., Leong, K.C., Pranoto, I., Li, H.Y., and Chai, J.C., Experimental study of a two-phase thermosyphon with porous graphite foam insert, *ASME Journal of Thermal Science and Engineering Applications*, Vol. 3, No.2, pp. 024502-1-024502-6 (2011)
- Joseph, D.D., Nield, D.A., and Papanicolaou, G., Nonlinear equation governing flow in a saturated porous medium, *Water Resources Research*, Vol. 18, pp. 1049-1052 (1984)
- Karimi, G., Jafarpour, F., and Li, X., Characterization of flooding and two-phase flow in polymer electrolyte membrane fuel cell stacks, *Journal of Power Sources*, Vol. 187, pp. 156-164 (2009)

- Karimian, S.A.M., and Straatman, A.G., Numerical modelling of multidirectional flow and heat transfer in graphitic foams, *ASME Journal of Heat Transfer*, Vol. 131, pp. 1-11 (2009)
- Kaviany, M., *Principles of Heat Transfer in Porous Media*, 2nd edition, Springer-Verlag, New York (1995)
- Kenneth, G. B., 2005-2010 heat density trends, in data processing, computer systems, and telecommunications equipment: perspectives, implications, and the current reality in many data centres, http://www.msiinet.com/technologies/data-center/white-papers/2005-2010_Heat_Density_Trends.pdf(2006), accessed on 15 Sep. 2010
- Kim, S.Y., Lee, M. H., and Lee, K.S., Heat removal by aluminium foam heat sinks in a multi-air jet impingement, *IEEE Transactions on Components and Packaging Technologies*, Vol. 28, pp. 142- 148 (2005)
- Kim, S.Y., Paek, J.W., and Kang, B.H., Thermal performance of aluminium-foam heat sinks by forced air cooling, *IEEE Transactions on Components and Packaging Technologies*, Vol. 26, pp. 262-267 (2003)
- Klett, J.W., Hardy, R., and Romine, E., High thermal conductivity, mesophase-pitch-derived carbon foam: effect of precursor on structure and properties, *Carbon*, Vol. 38, pp. 953-973 (2000)
- Klett, J.W., McMillan, A. D., and Stinton, D., Modelling geometric effects on heat transfer with graphite foam, *The 26th Annual Conference on Ceramic, Metal, and Carbon Composites, Materials and Structures*, Florida, USA (2002)
- Klett, J.W., Mcmillan, A.D., Gallego, N.C., and Walls, C.A., The role of structure on the thermal properties of graphite foams, *Journal of Materials Science*, Vol. 39, pp. 3659-3676 (2004)
- Klett, J.W., *Process for Making Carbon Foam*, US Patent 6033506 (2000)
- Ko, K.H., and Anand, N.K., Use of porous baffles to enhance heat transfer in a rectangular channel, *International Journal of Heat and Mass Transfer*, Vol. 46, pp. 4191-4199 (2003)
- Koh, J.C.Y., and Stevens, R.L., Enhancement of cooling effectiveness by porous material in coolant passage, *ASME Journal of Heat Transfer*, Vol. 97, pp. 309-311 (1975)
- Kopanidis, A., Theodorakakos, A., Gavaises, E., and Bouris, D., 3D numerical simulation of flow and conjugate heat transfer through a pore scale model of high porosity open cell metal foam, *International Journal of Heat and Mass Transfer*, Vol. 53, pp. 2539-2550 (2010)

- Krishnan, S., Garimella, S.V., and Murthy, J. Y., Simulation of thermal transport in open-cell metal foams: effect of periodic unit-cell structure, *ASME Journal of Heat Transfer*, Vol. 130, pp. 1-5 (2008)
- Krishnan, S., Murthy, J.Y., and Garimella, S.V., Direct simulation of transport in open-cell metal foam, *ASME Journal of Heat Transfer*, Vol. 128, pp. 793-799 (2006)
- Kuznetsov, A.V., Thermal non-equilibrium, non-Darcian forced convection in a channel filled with a fluid saturated porous medium-a perturbation solution, *Applied Scientific Research*, Vol. 57, pp. 119-131 (1996)
- Leblond, G., and Gosselin, L., Effect of non-local equilibrium on minimal thermal resistance porous layered systems. *International Journal of Heat and Fluid Flow*, Vol. 29, pp. 281-291 (2008)
- Lee, S.L., and Tzong, R.Y., An enthalpy formulation for phase change problems with a large thermal diffusivity jump across the interface, *International Journal of Heat and Mass Transfer*, Vol. 34, pp. 1491-1502 (1991)
- Leong, K.C., Jin, L.W., Pranoto, I., Li, H.Y., and Chai, J.C., "Experimental study of a two-phase thermosyphon with graphite foam insert", *14th International Heat Transfer Conference, IHTC-14*, August 8-13, Washington, DC, USA (2010a)
- Leong, K.C., Li, H.Y., Jin, L.W., and Chai, J.C., Convective heat transfer in graphite foam heat sinks with baffle and stagger structures, *ASME Journal of Heat Transfer*, Vol. 133, pp. 060902-1 - 060902-9 (2010b)
- Leong, K.C., Li, H.Y., Jin, L.W., and Chai, J.C., Numerical and experimental study of forced convection in graphite foams of different configurations, *Applied Thermal Engineering*, Vol. 30, No. 5, pp. 520 – 532 (2010c)
- Leong, K.C., Jin, L.W., Pranoto, I., Li, H.Y., and Chai, J.C., Experimental study of enhanced pool boiling heat transfer using graphite foam inserts, *Defect and Diffusion Forum*, Vols. 312-315, pp. 352 – 357, (2011)
- Leong, K.C., and Li, H.Y., Theoretical study of the effective thermal conductivity of graphite foam based on a unit cell model, *International Journal of Heat and Mass Transfer*, Vol. 54, pp.5491-5496 , (2011)
- Levec, J., Saez, A.E., and Carbonell, R.G., The hydrodynamics of trickling flow in packed beds, part II: experimental observations, *American Institute of Chemical Engineers Journal*, Vol. 32, pp. 369-380 (1986)
- Leverett, M.C., Capillary behaviour in porous solids, *Transactions of AIME*, Vol. 142, pp. 152-169 (1941)

- Li, H.Y., Leong, K.C., Jin, L.W., and Chai, J.C., Analysis of fluid flow and heat transfer in a channel with staggered porous blocks, *International Journal of Thermal Sciences*, Vol. 49, pp. 950-962 (2010a)
- Li, H.Y., Leong, K.C., Jin, L.W., and Chai, J.C., Transient two-phase flow and heat transfer with localized heating in graphite foams, *International Journal of Thermal Sciences*, Vol. 49, pp. 1115 – 1127 (2010b)
- Li, H.Y., Leong, K.C., Jin, L.W., and Chai, J.C., Transient behavior of fluid flow and heat transfer with phase change in vertical porous channels, *International Journal of Heat and Mass Transfer*, Vol. 53, pp. 5209–5222 (2010c)
- Li, H.Y., Leong, K.C., Jin, L.W., and Chai, J.C., Three-dimensional numerical simulation of fluid flow with phase change heat transfer in an asymmetrically heated porous channel, *International Journal of Thermal Sciences*, Vol. 49, pp. 2363 – 2375 (2010d)
- Li, H.Y., and Leong, K.C., Experimental and numerical study of single and two-phase flow and heat transfer in aluminium foams, *International Journal of Heat and Mass Transfer*, Vol. 54, pp. 4904 – 4912 (2011)
- Liu, S. and Masliyah, J.H., Dispersion in porous media, *Handbook of Porous Media*, 2nd edition, Taylor and Francis, Boca, Raton, FL, pp. 81-140 (2005)
- Lu, W., and Zhao, C.Y., Numerical modelling of flow boiling heat transfer in horizontal metal-foam tubes, *Advanced Engineering Materials*, Vol. 11, pp. 832-836 (2009).
- Martys, N., Bentz, D.P., and Garboczi, E.J., Computer simulation study of the effective viscosity in Brinkman equation, *Physics of Fluids*, Vol. 6, pp. 1434-1439 (1994)
- Miranda, B.M.D.S., and Anand, N.K., Convection heat transfer in a channel with porous baffles, *Numerical Heat Transfer Part A*, Vol. 46, pp. 425-452 (2004)
- Mualem, D., A new model for predicting the hydraulic conductivity of saturated porous media, *Water Resources Research*, Vol. 12, pp. 513-522 (1976)
- Mudawar, I., and Estes, K.A., Optimizing and predicting CHF in spray cooling of a square surface, *ASME Journal of Heat Transfer*, Vol. 118, pp. 672-679 (1996)
- Mudawar, I., and Valentine, W.S., Determination of the local quench curve for spray-cooled metallic surfaces, *Journal of Heat Treating*, Vol. 7, pp. 107-121 (1989)

- Mudawar, I., and Wadsworth, D.C., Critical heat flux from a simulated electronic chip to a confined rectangular impinging jet of dielectric liquid, *International Journal of Heat and Mass Transfer*, Vol. 34, pp. 1465-1480 (1991)
- Mudawar, I., Assessment of high-heat-flux thermal management schemes, *IEEE Transactions on Components and Packaging Technologies*, Vol. 24, pp. 122-140 (2001)
- Muskat, M., and Meres, M.W., The flow of heterogeneous fluids through porous media, *Journal of Applied Physics*, Vol. 7, pp. 346-363 (1936)
- Najjari, M., and Nasrallah, S.B., Étude numérique de l'ébullition en convection mixte dans une couche poreuse verticale (in French), *International Journal of Thermal Sciences*, Vol. 41, pp. 913-925 (2002)
- Najjari, M., and Nasrallah, S. B., Numerical study of boiling in inclined porous layer, *Journal of Porous Media*, Vol. 6, pp. 71-81 (2003)
- Najjari, M., and Nasrallah, S.B., Effects of latent heat storage on heat transfer in a forced flow in porous layer, *International Journal of Thermal Sciences*, Vol. 47, pp. 825-833 (2008)
- Najjari, M., and Nasrallah, S.B., Heat transfer between a porous layer and a forced flow: influence of layer thickness, *Drying Technology*, Vol. 27, pp. 336-343 (2009)
- Nield, D.A., and Bejan, A., *Convection in Porous Media*, 3rd edition, Springer-Verlag, New York (2006)
- Ochoa-Tapia, J.A., and Whitaker, S., Momentum transfer at the boundary between a porous medium and a homogeneous fluid-I. theoretical development, *International Journal of Heat and Mass Transfer*, Vol. 38, pp. 2635-2646 (1995)
- Ott, R. D., Zaltash, A., and Klett, W. J., Utilization of a graphite foam radiator on a natural gas engine-driven heat pump, *Proceedings of ASME International Mechanical Engineering Conference and Exposition*, 17-22 Nov., Louisiana, USA (2002)
- Palm, B., and Khodabandeh, R., Choosing working fluid for two-phase thermosyphon system for cooling of electronic, *ASME Journal of Electronic Packaging*, Vol. 125, pp. 276-281 (2003)
- Patankar, S.V., *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publishing Corporation (1980)
- Pavone, F., Explicit solution for free-fall gravity drainage including capillary pressure, *American Society of Mechanical Engineers, Fluids Engineering Division*, Vol. 82, pp. 55-62 (1989)

- Poulikakos, D., and Kazmierczak, M., Forced convection in a duct partially filled with a porous material, *ASME Journal of Heat Transfer*, Vol. 109, pp. 653-662 (1987)
- Ramesh, P.S., and Torrance, K.E., Numerical algorithm for problems involving boiling and natural convection in porous materials, *Numerical Heat Transfer Part B*, Vol. 17, pp. 1-24 (1990)
- Ramesh, P.S., and Torrance, K.E., Boiling in a porous layer heated from below: effects of natural convection and a moving liquid/two-phase interface, *Journal of Fluid Mechanics*, Vol. 257, pp. 289-309 (1993)
- Richards, L.A., Capillary conduction of liquids through porous media, *Physics*, Vol. 1, pp. 318-333 (1931)
- Samant, K.R., and Simon, T.W., Heat transfer from a small heated region to R-113 and FC-72, *ASME Journal of Heat Transfer*, Vol. 111, pp. 1053-1059 (1989)
- Schäfer, P., and Lohnert, G., Boiling experiments for the validation of dryout models used in reactor safety, *Nuclear Engineering and Design*, Vol. 236, pp. 1511-1519 (2006)
- Scheidegger, A.E., *The Physics of Flow through Porous Media*, 3rd edition, University of Toronto Press (1974)
- Slattery, J.C., Single-phase flow through porous media, *American Institute of Chemical Engineers Journal*, Vol. 15, pp. 866-872 (1969)
- Slattery, J.C., Two-phase flow through porous media, *American Institute of Chemical Engineers Journal*, Vol. 16, pp. 345-352 (1970)
- Sondergeld, C.H., and Turcotte, D.L., An experimental study of two-phase convection in a porous medium with applications to geological problems, *Journal of Geophysical Research*, Vol. 82, pp. 2045-2053 (1977)
- Sondergeld, C.H., and Turcotte, D.L., Flow visualization studies of two-phase thermal convection in a porous layer, *Pure Applied Geophysics*, Vol. 117, pp. 321-330 (1978)
- Straatman, A.G., Gallego, N.C., Betchen, L., and Thompson, B.E., Forced convection heat transfer and hydraulic losses in graphite foam, *ASME Journal of Heat Transfer*, Vol. 129, pp. 1237-1245 (2007a)
- Straatman, A.G., Gallego, N.C., Thompson, B.E., and Hangan, H., Thermal characterization of porous carbon foam-convection in parallel flow, *International Journal of Heat and Mass Transfer*, Vol. 49, pp. 1991-1998 (2006)

- Straatman, A.G., Gallego, N.C., Yu, Q., and Thompson, B.E., Characterization of porous carbon foam as a material for compact recuperators, *ASME Journal of Engineering for Gas Turbines and Power*, Vol. 129, pp. 326-330 (2007b)
- Sung, H.J., Kim, S.Y., and Hyun, J.M., Forced convection from an isolated heat source in a channel with porous medium, *International Journal of Heat and Fluid Flow*, Vol. 16, pp. 527-535 (1995)
- Targui, N., and Kahalerras, H., Analysis of fluid flow and heat transfer in a double pipe heat exchanger with porous structures, *Energy Conversion and Management*, Vol. 49, pp. 3217-3229 (2008)
- Taylor, J.R., *An Introduction to Error Analysis*, 2nd Edition, University Science Books, California (1997)
- Tee, C.C., Klett, J. W., Stinton, D. P., and Yu, N., Thermal conductivity of porous carbon foam, *Proceedings of the 24th Bienial Conference on Carbon*, 11-16 July, Charleston, USA (1999)
- Tee, C.C., Yu, N., and Li, H., Modelling the overall heat conductive and convective properties of open-cell graphite foam, *Modelling Simulation in Material Sciences and Engineering*, Vol. 16, pp. 1-21 (2008)
- Udell, K.S., Heat transfer in porous media considering phase change and capillarity-the heat pipe effect, *International Journal of Heat and Mass Transfer*, Vol. 28, pp. 485-494 (1985)
- Vadakkan, U., Garimella, S. V., and Murthy, J. Y., Transport in flat heat pipes at high heat fluxes from multiple discrete sources, *ASME Journal of Heat Transfer*, Vol. 126, pp. 347-354 (2004)
- Vadász, P., *Emerging Topics in Heat and Mass Transfer in Porous Media*, Springer, New York (2008)
- Vafai, K., and Kim, S.J., Forced convection in a channel filled with a porous medium: an exact solution, *ASME Journal of Heat Transfer*, Vol. 111, pp. 1103-1106 (1989)
- Vafai, K., and Sozen, M., Analysis of energy and momentum transport for fluid flow through a porous bed, *ASME Journal of Heat Transfer*, Vol. 112, pp. 690-699 (1990)
- Vafai, K., and Tien, C.L., Boundary and inertial effects on flow and heat transfer in porous media, *International Journal of Heat and Mass Transfer*, Vol. 24, pp. 195-203 (1981)
- Vafai, K., and Tien, C.L., Boundary and inertial effects on convection mass transfer in porous media, *International Journal of Heat and Mass Transfer*, Vol. 25, pp. 1183-1190 (1982)

- Vafai, K., *Handbook of Porous Media*, 2nd edition, Taylor & Francis, New York (2005)
- van Genuchten, M., Th., A closed-form equation for predicting the hydraulic conductivity of unsaturated soils, *Soil Science Society of America Journal*, Vol. 44, pp. 892-898 (1980)
- Versteeg, H.K., and Malalasekera, W., *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, Prentice Hall, 2nd edition (2007)
- Voller, V.R., and Swaminathan, C.R., Treatment of discontinuity thermal conductivity in control-volume solutions of phase change problems. *Numerical Heat Transfer Part B*, Vol. 24, pp. 161-180 (1993)
- Waite, M.W., and Amin, M.R., Numerical investigation of two-phase fluid flow and heat transfer in porous media heated from the side, *Numerical Heat Transfer Part A*, Vol. 35, pp. 271-290 (1999)
- Wang, C.Y., A fixed-grid numerical algorithm for two-phase flow and heat transfer in porous media, *Numerical Heat Transfer Part B*, Vol. 32, pp. 85-105 (1997)
- Wang, C.Y., and Beckermann, C., A two-phase mixture model of liquid-gas flow and heat transfer in capillary porous media, I: Formulation, *International Journal of Heat and Mass Transfer*, Vol. 36, pp. 2747-2758 (1993a)
- Wang, C.Y., and Beckermann, C., A two-phase mixture model of liquid-gas flow and heat transfer in capillary porous media-II: application to pressure-driven boiling flow adjacent to a vertical heated plate, *International Journal of Heat and Mass Transfer*, Vol. 36, pp. 2759-2768 (1993b)
- Wang, C.Y., Beckermann, C., and Fan, C., Numerical study of boiling and natural convection in capillary porous media using the two-phase mixture model, *Numerical Heat Transfer Part A*, Vol. 26, pp. 375-398 (1994)
- Whitaker, S., Advances in theory of fluid motion in porous media, *Industrial and Engineering Chemistry*, Vol. 61, pp. 14-28 (1969)
- Whistaker, S., Simultaneous heat, mass, and momentum transfer in porous media: a theory of drying, *Advances in Heat Transfer*, Vol. 13, pp.119-203 (1977)
- Whitaker, S., Flow in porous media I: a theoretical derivation of Darcy's law, *Transport in Porous Media*, Vol. 1, pp. 3-25 (1986a)
- Whitaker, S., Flow in porous media II: the governing equations for immiscible, two-phase flow, *Transport in Porous Media*, Vol. 1, pp. 105-125 (1986b)

Williams, Z.A., and Roux, J.A., Graphite foam thermal management of a high packing density array of power amplifiers, *ASME Journal of Electronic Packaging*, Vol. 128, pp. 456-465 (2006)

Williams, Z.A., and Roux, J.A., Thermal management of a high packing density array of power amplifiers using liquid cooling, *ASME Journal of Electronic Packaging*, Vol. 129, pp. 488-495 (2007)

Willingham, T.C., and Mudawar, I., Channel height effects on forced convection boiling and critical heat flux from a linear array of discrete heat sources, *International Journal of Heat and Mass Transfer*, Vol. 35, pp. 1865-1880 (1992)

Wyllie, M.R.J., Relative permeabilities, *Petroleum Production Handbook*, McGraw-Hill, Vol. 2, Chapter 25, pp.1-4 (1962)

Yan, Y.Y., Zu, Y.Q., and Dong, B., LBM, a useful tool for mesoscale modelling of single-phase and multiphase flow, *Applied Thermal Engineering*, Vol. 31, pp. 649-655 (2011)

You, L., and Liu, H., A two-phase flow and transport model for the cathode of PEM fuel cells, *International Journal of Heat and Mass Transfer*, Vol. 45, pp. 2277-2287 (2002)

Yu, Q.J., Straatman, A.G., and Thompson, B.E., Carbon-foam finned tubes in air-water heat exchangers, *Applied Thermal Engineering*, Vol. 26, pp. 131-143 (2006b)

Yu, Q.J., Thompson, B.E., and Straatman, A.G., A unit cube-based model for heat transfer and fluid flow in porous carbon foam, *ASME Journal of Heat Transfer*, Vol. 128, pp. 352-360 (2006a)

Yuki, K., Abei, J., Hashizume, H., and Toda, S., Numerical investigation of thermofluid flow characteristics with phase change against high heat flux in porous media, *ASME Journal of Heat Transfer*, Vol. 130, pp. 1-12 (2008)

Zehner, P., and Schlunder, E.U., Thermal conductivity of granular materials at moderate temperatures (in German), *Chemie Ingenieur Technik*, Vol. 42, pp. 933-941 (1970)

Zhang, B.L., and Zhao, Y., A numerical method for simulation of forced convection in a composite porous/fluid system, *International Journal of Heat and Fluid Flow*, Vol. 21, pp. 432-441 (2000)

Zhao, C.Y., Dai, L.N., Tang, G.H., and Li, Z.Y., Numerical study of natural convection in porous media (metals) using Lattice Boltzmann Method (LBM), *International Journal of Heat and Fluid Flow*, Vol. 31, pp. 925-934 (2010)

REFERENCES

Zhao, C.Y., Lu, W., and Tassou, S.A., Flow boiling heat transfer in horizontal metal-foam tubes, *ASME Journal of Heat Transfer*, Vol. 131, pp.121002-1-121002-7 (2009)

Zhao, T.S., and Liao, Q., Mixed convective boiling heat transfer in a vertical capillary structure heated asymmetrically, *Journal of Thermophysics and Heat Transfer*, Vol. 13, pp. 302-307 (1999)

Zhao, T.S., Cheng, P., and Wang, C.Y., Buoyancy-induced flows and phase change heat transfer in a vertical capillary structure with symmetric heating, *Chemical Engineering Science*, Vol. 55, pp. 2653-2661 (2000)

Zhao, T.S., Liao, Q., and Cheng, P., An experimental study on variations of buoyancy-induced mass flux from single-phase to two-phase flow in a vertical porous tube, *ASME Journal of Heat Transfer*, Vol. 121, pp. 646-652 (1998)

Zukauskas, A.A., Convective Heat transfer in cross-flow, *Handbook of Single-Phase Heat Transfer*, Wiley, New York (1987)

APPENDIX: LIBRARY OF COMPUTER PROGRAMS

The programs developed in this thesis are written in FORTRAN 77. There are several modules needed to run the program. These modules are listed in Tables A-1 and A-2 for single and two-phase flow and heat transfer, respectively. Soft copies of the codes are available on CDROM from the author.

Table A-1 List of FORTRAN programs for single-phase flow and heat transfer

Module name	Function
COMMON.FOR	Transfer all the required information through all the modules.
PARAM.FOR	Define the maximum control volumes and zones.
MAIN.FOR	Construct the grid. Define the coefficients in the solved equations.
SOLVE.FOR	Solve the governing equations.
TCPLT.FOR	Output the necessary calculation results.
FDIFF.FOR	Define the coefficients of the flow flux.
FOAM-CAVITY.FOR	Define the 2D simulation domain. Define the number of control volumes. Specify the physical properties, such as density, viscosity <i>et al.</i> . Specify the boundary conditions.
BLOCK_3D	Define the 3D simulation domain of BLK foam. Define the number of control volumes. Specify the physical properties, such as density, viscosity <i>et al.</i> . Specify the boundary conditions.
ZIGZAG_3D	Define the 3D simulation domain of ZZG foam. Define the number of control volumes. Specify the physical properties, such as density, viscosity <i>et al.</i> . Specify the boundary conditions.
BAFFLE_3D	Define the 3D simulation domain of BAF foam. Define the number of control volumes. Specify the physical properties, such as density, viscosity <i>et al.</i> . Specify the boundary conditions.

Table A-2 List of FORTRAN programs for two-phase flow and heat transfer

Module name	Function
COMMON.FOR	Transfer all the required information through all the modules.
PARAM.FOR	Define the maximum control volumes and zones.
MAIN.FOR	Construct the grid. Define the coefficients in the solved equations.
MAIN-PM.FOR	Define the coefficients in the momentum equation.
SOLVE.FOR	Solve the governing equations.
MIS.FOR	Output the necessary calculation results.
FDIFF.FOR	Define the coefficients of the flow flux.
USER.FOR	Define the 2D and 3D simulation domain, respectively. Define the number of control volumes. Specify the physical properties, such as density, viscosity <i>et al.</i> . Specify the boundary conditions.