CHT-12 PROGRAMME

Time	Sunday 1 July	Monday 2 July	Tuesday 3 July	Wednesday 4 July	Thursday 5 July	Friday 6 July	Time
8.30-9.00		Registration (all day)	Poster 2 Heat Equipment HE (10) Energy EN (4)	KN 6 Catton	KN 8 Dhir	KN 11 Bennacer	8.30-9.00
9.00-9.30							9.00-9.30
9.30-10.00		Opening session		Poster 5 Natural Convection NC (11) Verification & Valid VV (3)	Poster 6 Combustion CF (6) Turbulence Modelling TM (8)	Poster 8: Bio BH (1) Natural Conv NC (7) Forced Conv FC (5) Solid'n etc. SM (2)	9.30-10.00
10.00-10.30		KN1 Timchenko					10.00-10.30
10.30-11.00			Free: change posters				10.30-11.00
11.00-11.30		Tea/Coffee break*	Poster 3 Multiphase MP (11)				11.00-11.30
11.30-12.00		KN 2 Spalding		KN 7 Runchal	KN 9 Dombrovsky	Closing session	11.30-12.00
12.00-12.30							12.00-12.30
12.30-1.00		Lunch		Lunch	Lunch		12.30-1.00
1.00-1.30			Lunch				1.00-1.30
1.30-2.00							1.30-2.00
2.00-2.30	Registration	KN 3 Sazhin		Free for Tours	KN 10 Zhang		2.00-2.30
2.30-3.00			Guest lecture Bar-Cohen				2.30-3.00
3.00-3.30		Poster 1 Computational Methods CM (11) BL flow (3)			Poster 7 Micro & Nano MN (8) Radiation RD (5)		3.00-3.30
3.30-4.00			Poster 4 Mixed modes MX (9) Materials & Manuf. MM (4)				3.30-4.00
4.00-4.30							4.00-4.30
4.30-5.00							4.30-5.00
5.00-5.30		KN 4 Tucker			Free		5.00-5.30
5.30-6.00			KN 5 Chai				5.30-6.00
6.00-6.30							6.00-6.30
6.30-7.00							6.30-7.00
7.00-	Welcome Reception (Guildhall) To 9.00 pm			ICHMT-EC/ AIHTC dinner	Gala Dinner (to 10 pm)		7.00-

***Note: Tea/Coffee will be available during all Poster Sessions.**

MONDAY, July 2nd

Time	Session No	ID	Title	Authors
10:00 - 11:00		KN 1	LASER INDUCED HYPERTHERMIA OF SUPERFICIAL TUMORS: A TRANSIENT THERMAL MODEL FOR INDIRECT HEATING STRATEGY	**V. Timchenko,** Leonid Dombrovsky
11:30 - 12:30		KN 2	A ROLE FOR COMPUTATIONAL HEAT TRANSFER IN ENGINEERING EDUCATION	**B. Spalding**
14:00 - 15:00		KN 3	DROPLET HEATING AND EVAPORATION - RECENT RESULTS AND UNSOLVED PROBLEMS	**Sergei S. Sazhin,** Morgan R. Heikal
15:00 - 17:00	1	BL01	Linear Instability To Longitudinal Rolls Of The Darcy-Hadley Flow In A Weakly Heterogeneous Porous Medium	D.A. Nield, A. Barletta
	1	BL02	Convective instability in an inclined porous layer subject to linearly varying boundary temperatures	D.A.S. Rees, A. Barletta
	1	BL03	Effect Of The Width Of A Vertical Parallel Plate Channel On The Transition Of The Developing Thermal Boundary Layer	Ali S. Alzwayi, M.C. Paul
	1	CM01	Multi-Scale Modeling Of PEMFC Cathode By Coupling Finite Volume Method And Lattice Boltzmann Method	Wen-Quan Tao, Ya-Ling He, Yong-Liang Feng, Chen-Xi Song, Li Chen
	1	CM02	The quantitative research on ray tracing number of Monte Carlo method in total radiant exchange area computation	Guo-Jun Li, Jiming Sun, Xiating Liu, Haigeng Chen
	1	CM05	Modeling and Simulation of Thermal Heating in 3D Integrated Circuits	A. El Boukili
	1	CM06	Molecular Dynamics Simulations of Mass Transfer due to a Temperature Gradient	Gülru Babaç, Jason M. Reese, Konstantinos Ritos
	1	CM08	UPWIND DIFFERENCING SCHEME IN EIGENFUNCTION EXPANSION SOLUTION OF CONVECTIVE HEAT TRANSFER PROBLEMS	L. A. Sphaier, D.J.N.M. Chalhub, L. S. de B. Alves
	1	CM09	An immersed boundary method to solve flow and heat transfer problems involving a moving object	Ming-Jyh Chern, Dedy Zulhidayat Noor, Tzyy-Leng Horng
	1	CM10	Combined numerical and analytical method for fin array heat transfer	Kaj Lampio, Reijo Karvinen
	1	CM11	A Comparison Of Momentum, Eulerian Fractions And d-Approximation Method For Modelling Of Two-Phase Turbulent Flows With Phase Transitions	Gennady A. Philippov, Artur R. Avetissian
	1	CM12	Advection and Diffusion Simulations Using Lagrangian Blocks	Vincent H. Chu, Wihel Altai
	1	CM13	A Predictor-Corrector Split Projection Method for Turbulent Reactive Flow	Darrell W. Pepper, Xiuling Wang, David B. Carrington
	1	CM14	Purely Elastic Flow Asymmetries in Cross-slot Geometry- Are They Always Present?	F. T. Pinho, L. Khezzar, A. Filali, M. A. Alves
	1	OF03	FALSE DIFFUSION IN MANAGING SPURIOUS OSCILLATIONS BY FLUX LIMITERS	Vincent H. Chu and Congwei Gao
17:00 - 18:00		KN 4	ZONAL RANS-LES MODELLING FOR TURBINE AEROENGINES	**P. Tucker**, R. Jefferson-Loveday, J. Tyacke and N. Vadlamani

Tuesday, July 3rd

Time	Session No	ID	Title	Authors
8:30 - 10:30	2	EN01	Separation of carbon dioxide and hydrogen from syngas by pressure swing adsorption	Cheng-tung Chou, Chih-Hsiang Huang, Hong-sung Yang, Yu-xuan He, Fei-hong Chen
	2	EN02	Piston Effect Characteristic Time Dependence On Equation Of State Model Choice	L. S. de B. Alves, P. C. Teixeira
	2	EN03	INFLUENCE OF THERMAL BEHAVIOUR OF A PHOTOVOLTAIC MODULE ON ITS ELECTRICAL PERFORMANCE	C. Menezo, L. Weiss
	2	EN04	PRESSURE WORK AND VISCOUS DISSIPATION INFUENCES ON FLOW ENERGY AND ENTROPY BALANCES	Stuart Norris, Gordon Mallinson
	2	HE02	A THREE DIMENSIONAL NUMERICAL STUDY OF THE HYDRODYNAMICS AND HEAT TRANSFER CHARACTERISTICS OF NOVEL HEAT EXCHANGERS	Li-Ting Tian, Ya-Ling He, Wen-Quan Tao, Yu Wang
	2	HE03	A MICRO FLOWMETER BASED ON THE MEASUREMENT OF A DIFFUSION TEMPERATURE RISE OF A LOCALLY HEATED THERMAL FLOW IN A HAGEN-POISEUILLE FLOW	H. Koizumi, Y. Kato, T. Kimura
	2	HE04	PRESSURE DROP AND HEAT TRANSFER IN SPIRALLY CORRUGATED TUBE FOR A COUNTER-FLOW HEAT EXCHANGER	Amarin Tongkratoke, Anchasa Pramuanjaroenkij, Apichart Chaengbamrung
	2	HE05	PARAMETRIC STUDIES ON MEMBRANE-BASED TOTAL HEAT EXCHANGER PERFORMANCE	Xiaomin Wu, Jingchun Min, Ming Su
	2	HE06	Effect of operating conditions on performance of proton exchange membrane fuel cell (PEMFC)	Yasmina Ziari Kerboua, Youcef Kerkoub, Ahmed Benzaoui
	2	HE07	Numerical Simulation Of A Phase Change Material (PCM) In A Domestic Refrigerator Powered By Photovoltaic Power	B. Abbad, F. Yahi, M. Laidi, M. Ouali, M. Berdja
	2	HE08	Numerical Investigation of Transpiration Cooling of a Liquid Rocket Thrust Chamber Wall	Mehdi M. Faridani, Hikmet S. Aybar, Mehmet Sozen
	2	HE09	Optimization of fluidized horizontal heat exchanger with lengthwise dispersion	Artur Poświata
	2	HE10	Model-based application for leak testing of buried fuel lines	Timothy Smith, Michael O'Donnell, Michael R. Maixner
	2	HE12	Conjugate Heat Transfer Modelling Of Internal Combustion Engine Structures And Coolant Flows	Nick Tiney, Richard Penning, Vlado Pržulj
	2	OF02	THE EFFECT OF NON-NEUTRAL STABILITY IN WIND FARM SIMULATIONS	C A Montavon and I P Jones

Tuesday, July 3rd

11:00 - 13:00	3	MP01	Computational heat transfer modeling of rice – water suspension in tube.	A. K. Datta, Kanishka Bhunia
	3	MP02	Kinetic and molecular dynamics simulations of n-dodecane droplet heating and evaporation	Bing-Yang Cao, Irina Shishkova, Sergei S. Sazhin, Jian-Fei Xie
	3	MP03	Simulation of interphase heat transfer during bulk condensation in the flow of vapor-gas mixture	A.K. Yastrebov, N.M. Kortsenshteyn
	3	MP04	Exponential Euler time integrator for isothermal incompressible two-phase flow in heterogeneous porous media	Antoine Tambue
	3	MP05	Modelling of flow patterns and convective heat transfer in an intermittent impinging spray	Viktor Terekhov, Maksim Pakhomov
	3	MP06	Numerical analysis and control of two-phase flow instabilities in a vertical tube during evaporation	Ghazali Mébarki, Samir. Rahal
	3	MP09	Advancement in turbulent spray modelling: the effect of internal temperature gradient in droplets	S.S.. Sazhin, V.A. Talalov, C. Crua, A.S. Tsoi, A. Yu. Snegirev
	3	MP10	Three-Dimensional Simulation Of Bubble Growth On Microstructured Surfaces	Gihun Son, Woorim Lee
	3	MP12	Numerical Investigation Of Particle Temperature Change In Supersonic Flows	Masaya Suzuki, Makoto Yamamoto, Ryouhei Sakamaki
	3	MP13	Modeling Of Phase Transition Of Partially Miscible Solvent Systems: Hydrodynamics And Heat Transfer Phenomena	Amos Ullmann, Neima Brauner, Vered Segal
	3	MP14	Heat transfer in shear-driven thin liquid film flows	J. R. Marati, M. Budakli, T. Gambaryan-Roisman, P. Stephan
14:30 - 15:30		Guest Lecture	THERMOFLUID/ELECTRICAL CO-DESIGN OF EMERGING ELECTRONIC COMPONENTS – CHALLENGES AND OPPORTUNITIES	**Avram Bar-Cohen**

Tuesday, July 3rd

15:30 - 17:30	4	MM01	Complex Heat Transfer At Directed Crystallization Of Semitransparent Materials	V. Deshko, A. Karvatskii, I. Lokhmanets
	4	MM02	SHELL FORMATION IN DRY SPINNING	J. I. Ramos
	4	MM03	COMPUTATIONAL PREDICTION OF RADIATIVE PROPERTIES OF POLYMER CLOSED-CELL FOAMS WITH RANDOM STRUCTURE	Rémi Coquard, Dominique Baillis, Jaona Randrianalisoa
	4	MM04	Thermal Conductivity Of Cellular Foams: Influence Of Cell Randomness	Rémi Coquard, Dominique Baillis, Jaona Randrianalisoa
	4	MX01	Radiation Effects on Participating Electrical Conductive Fluid in a Square Cavity	Ben-Wen Li, Zhang-Mao Hu, Jing-Kui Zhang
	4	MX02	FLOW REVERSAL IN MIXED CONVECTION IN VERTICAL CONCENTRIC ANNULI	Esmail M. A. Mokheimer
	4	MX04	Numerical Study Of Coupled Molecular Gas Radiation And Natural Convection In A Differentially Heated Cubical Cavity	Patrick Le Quéré, Philippe Rivière, Shihe Xin, Anouar Soufiani, Laurent Soucasse
	4	MX06	Numerical study of unsteady airflow phenomena in a ventilated room	Kana Horikiri, Yufeng Yao, Jun Yao
	4	MX07	Interaction Effects Between Surface Radiation And Double-Diffusive Turbulent Natural Convection In An Enclosed Cavity Filled With Solid Obstacles	Draco Aluya Iyi, Reaz Hasan, Roger Penlington
	4	MX10	Marangoni effect on the longitudinal rolls of a mixed convection flow in a horizontal open channel	B. Benhamou, K. El Omari, S. Blancher, Y. Le Guer, L. Bammou
	4	MX11	Numerical simulation of coupled heat transfer through building facades in the arid zone	A. Missoum, B. Draoui, A. Slimani, M. Elmir
	4	MX12	A new process for species separation in a binary mixture using mixed convection	Abdelkader Mojtabi, Marie C. C. Mojtabi, Ouattara Bafetigue, Khouzam Ali
	4	MX14	Hybrid Experimental-Numerical Approach To Solve Inverse Convection Problems	Yogesh Jaluria, Joseph VanderVeer
17:30 - 18:30		KN 5	LEVEL-SET METHOD FOR MULTIPHASE FLOWS	Y.F. Yap, **J. C. Chai**

Wednesday, July 4th

Time	Session No	ID	Title	Authors
8:30 - 9:30		KN 6	THE USE OF VOLUME AVERAGING THEORY TO ADDRESS HEAT TRANSFER WITHIN ENGINEERED HETEROGENEOUS HIERARCHICAL STRUCTURES	**Ivan Catton**
9:30 - 11:30	5	NC01	Cooling Electronic Components Mounted On A Vertical Wall By Natural Convection	Rachid Bessaih, Karim Lahmer
	5	NC02	Numerical Study Of Heat Transfer By Natural Convection Along A Wavy Vertical Plate With Variable Wall Temperature	Mayouf Si Abdallah, Belkacem Zeghmati
	5	NC03	Numerical Investigation Of Free convection And Heat Transfer Between Vertical Parallel Plates With Different Temperatures	Ali Ekaid, Viktor Terekhov
	5	NC05	BOUNDARY CONDITION EFFECTS ON NATURAL CONVECTION OF BINGHAM FLUIDS IN A SQUARE ENCLOSURE WITH DIFFERENTIALLY HEATED HORIZONTAL WALLS	R. J. Poole, O. Turan
	5	NC06	Natural convection and surface radiation between a horizontal heat generating solid cylinder and a thick outer cylindrical shell	G. S. V. L. Narasimham, Vinay Senve
	5	NC07	Insight into Rayleigh number effects on turbulence characteristics of a thermally driven flow adjacent to upward-facing horizontal heated round plate by large-eddy simulation	Hitoshi Suto, Shuji Ishihara, Yuzuru Eguchi, Yasuo Hattori
	5	NC08	Transient Natural Convection In Enclosures Of Circular Cross Section Filled With Humid Air, Including Wall Phase Change	Vítor A.F. Costa
	5	NC09	Large-Eddy Simulation Of A Buoyant Plume Past A Bluff Body - Effects Of Flow Structures On Entrainment Characteristics	Yasuo Hattori, Hitoshi Suto
	5	NC10	Rayleigh–Taylor Instability In Two-Fluid And Stratified Media	Sergey N. Yakovenko
	5	NC11	Natural Convective Heat Transfer from a Vertical Isothermal High Aspect Ratio Rectangular Cylinder with an Exposed Upper Surface Mounted on a Flat Adiabatic Base	Abdulrahim Kalendar, Almounir Alkhazmi, Patrick H. Oosthuizen
	5	NC12	Three-Dimensional Numerical Simulation Of Turbulent Natural Convection In An Enclosure Having Finite Thickness Heat-Conducting Walls	Mikhail A. Sheremet
	5	VV01	COMPARISON BETWEEN PIV RESULTS AND CFD SIMULATIONS OF AIR FLOWS IN A THIN ELECTRONICS CASING MODEL	Masaru Ishizuka, Tomoyuki Hatakeyama, Risako Kibushi, Shinji Nakagawa
	5	VV02	Experimental and numerical modelling of enhanced thermal diffusion in a structured packed bed	PG Rousseau, TL Kgame, CG du du Toit, ACN Preller
	5	VV03	Exact analytical solutions for verification of numerical codes in transient heat conduction	James V. Beck, Filippo de Monte
11:30 - 12:30		KN 7	THE FUTURE OF CFD AND THE CFD OF THE FUTURE	**Akshai K. Runchal**

Thursday, July 5th

Time	Session No	ID	Title	Authors
8:30 - 9:30		KN 8	BUBBLE DYNAMICS DURING POOL BOILING UNDER MICROGRAVITY CONDITIONS	**V. K. Dhir**
9:30 - 11:30	6	CF01	Large Eddy Simulation of Fire in a Large Test Hall	G.H. Yeoh, J. Tang, A.C.Y. Yuen
	6	CF02	CFD Modelling Of Thermo-Chemical Process In Catalytic Pyrolysis Of Sawdust In Bubbling Fluidized Beds	L.-M. Armstrong, S. Gu, K. Papadikis, N. Dong , K. Luo
	6	CF03	Maldistribution Of Fuel/Air Flows In A Stack Of Planar SOFC Caused By Temperature Non-Uniformity	Daisuke Hayashi, Hideo Yoshida, Motohiro Saito, Hiroshi Iwai
	6	CF04	Development Of Advanced Models Of Energetic Materials Combustion By Means Of Data Mining Tools: Future Of Knowledge Base Of Energetic Materials World	V.S. Abrukov
	6	CF05	Reaction-Diffusion Phenomena With Relaxation	J. I. Ramos
	6	CF06	Numerical study of the effect of heat loss ontriple flame propagation in a porous channel	Faisal Al-Malki
	6	TM01	Numerical study of closure models applied to turbine blade film cooling.	A. Berkache, Rabah Dizene
	6	TM03	Numerical simulation of turbulent two-phase free surface flows	H. Mhiri, Philippe Bournot, N. Khaldi
	6	TM05	Conjugate Heat Transfer Analyses Of A Turbine Vane With Different RANS Turbulence Models	Sinan Inanli, Ilhan Gorgulu, I. Sinan Akmandor
	6	TM06	Numerical Modeling Of Thermal Turbulent Wall Flows With Different Codes	Rafik Absi, Ahmed Benzaoui, Najla El Gharbi, Mohammed El Ganaoui
	6	TM07	The Effect Of Dynamics And Thermal Prehistory On Turbulent Separated Flow At Abrupt Tube Expansion	Viktor Terekhov, Tatyana Bogatko
	6	TM08	Large Eddy Simulations Of Turbulent Heated Jets	Pedro J. Coelho, Carlos B. da Silva, Maxime Roger
	6	TM09	RANS And LES Simulation Of A Swirling Flow With Different Swirl Intensities	Hadef Redjem, Lyes Khezzar, Nabil Kharoua, Li Zhuowei
	6	TM11	LES OF TURBULENT THERMAL MIXING IN CIRCULAR AND SQUARE T-JUNCTION CONFIGURATIONS	D. Lakehal, M. Labois
	6	OF01	STUDIES OF A TURBULENT PATCH ARISING FROM INTERNAL WAVE BREAKING AT DIFFEREN	Sergey N. Yakovenko, T. Glyn Thomas, Ian P. Castro
11:30 - 12:30		KN 9	THE USE OF TRANSPORT APPROXIMATION AND DIFFUSION-BASED MODELS IN RADIATIVE TRANSFER CALCULATIONS	**Leonid A. Dombrovsky**
14:00 - 15:00		KN 10	AIR HUMIDIFICATION WITH HOLLOW FIBER MEMBRANES: NEW IMPACTS ON CONJUGATED HEAT AND MASS TRANSFER	Si-Min Huang, **Li-Zhi Zhang**

Thursday, July 5th

15:00 - 17:00	7	MN01	Numerical investigation of nanofluid flow and heat transfer in a plate heat exchanger	Nicolas Galanis, Iulian Gherasim, Cong Tam Nguyen
	7	MN02	Numerical Study of Nanofluid Heat Transfer Enhancement with Mixing Thermal conductivity models	Anchasa Pramuanjaroenkij, Apichart Chaengbamrung, Amarin Tongkratoke, Sadik Kakaç
	7	MN03	Heat transfer enhancement using CuO/water nanofluid	Noureddine Zeraibi, Amar Maouassi, Meriem Amoura
	7	MN04	Enhancement of the heat transfer due to the laminar forced convection of a nanofluid in a channel	A. Barletta, M. Celli, E. Rossi di Schio
	7	MN05	Effect Of Channel Pressure Difference In Heat Transfer Enhancement In Micro-Channel With Synthetic Jet	G.H. Yeoh, J. A. Reizes, V. Timchenko, A. Lee
	7	MN06	Mixed Convection Flow On A Vertical Permeable Surface In A Porous Medium Saturated By A Nanofluid With Internal Heat Generation	Fudziah Ismail, Ioan Pop, Ioan Pop, Norihan Md Arifin, Norihan Md Arifin, Roslinda Nazar, Mohd Hafizi Mat Yasin
	7	MN07	CONJUGATED CONVECTIVE-CONDUCTIVE HEAT TRANSFER IN MICRO CHANNELS WITH UPSTREAM REGION PARTICIPATION	Carolina P. Naveira-Cotta, Renato M. Cotta, Diego C. Knupp
	7	MN08	Mixed Convection Flow OVER A Horizontal Circular Cylinder With A Constant Surface Heat Flux In A Nanofluid	Ioan Pop, Roslinda Nazar, Leony Tham
	7	RD01	OUR RECENT WORKS ON THE COLLOCATION SPECTRAL METHOD FOR THERMAL RADIATION IN PARTICIPATING MEDIA	Ya-Song Sun, Ben-Wen Li, Shuai Tian, Jing Ma, Hang-Mao Hu
	7	RD02	Prediction of radiative heat transfer in 2D and 3D irregular geometries using the collocation spectral method based on body-fitted coordinates	Ben-Wen Li, Ya-Song Sun
	7	RD03	Collocation Spectral Method to Solve Radiative Transfer Equation for Three-dimensional Emitting-absorbing and Scattering Media Bounded by Gray Walls	Ben-Wen Li, Zhangmao Hu
	7	RD04	Combined two-flux approximation and Monte Carlo model for identification of radiative properties of highly scattering dispersed materials	Leonid Dombrovsky, Krithiga Ganesan, Wojciech Lipinski
	7	RD05	A Robust Monte Carlo Based Ray-Tracing Approach For The Calculation Of View Factors In Arbitrary 3d Geometries	G.W. Barton, S.-C. Xue, T. Walker

Friday, July 6th

Time	Session No	ID	Title	Authors
8:30 - 9:30		KN 11	UNSTABLE ANISOTHERMAL MULTICOMPONENT CONVECTIVE FLOW: FROM SMALL TO LARGE SCALES	**Rachid Bennacer**
9:30 - 11:30	8	BH01	New computational method for the short-pulsed NIR light propagation in biological tissue	F. Asllanaj, S. Fumeron
	8	FC01	Investigation of turbulent forced convection in helically grooved tubes	Gergely Kristóf, Gábor Varga, Loránd Romvári, Zoltán Hernádi
	8	FC05	The Influence Of Spanwise Wavelength Of Görtler Vortices In The Heat Transfer	Joseph T.C. Liu, Leandro F. de Souza, Vinicius Malatesta
	8	FC06	NUMERICAL SIMULATION OF HEAT TRANSFER OF A SPHERICAL PARTICLE IN AN AIR STREAM	Nafiseh Talebanfard, Bendiks Jan Boersma
	8	FC07	Numerical Study Of Heat And Fluid Flow Past A Cuboidal Particle At Subcritical Reynolds Numbers	Andreas Richter, Petr A. Nikrityuk, Kay Wittig
	8	FC08	Influence Of Three-Dimensional Perturbations On The HeatTransfer At Hypersonic Flow	Vladimir Shvedchenko, Ivan Egorov
	8	NC13	CFD simulation of heat transfer in a Two-Dimensional vertical Conical Partially Annular Space	Noreddine Retiel, Belkacem Ould Said, Modamed Aichouni
	8	NC14	NATURAL CONVECTION IN A RECTANGULAR ENCLOSURE WITH AN ARRAY OF DISCRETE HEAT SOURCES	P. Kandaswamy, V.P.M. Senthil Nayaki, A. Purusothaman, S. Saravanan
	8	NC15	Geometric Optimization For The Maximum Heat Transfer Density Rate From Cylinders Rotating In Natural Convection	J. Meyer, T. Bello-Ochende, L.G. Page
	8	NC16	A Numerical Study of Natural Convective Heat Transfer from an Inclined Isothermal Plate with a "Sinusoidally Wavy" Surface	Jane T. Paul, Patrick H. Oosthuizen
	8	NC17	NUMERICAL SIMULATION OF THERMAL CONVECTION IN A CLOSED CAVITY IN THE PRESENCE OF A THIN HORIZONTAL HEATED PLATE	A. Bendou, D. R. Rousse, H. Hamdi, L.Boukhattem Laurent
	8	NC18	NUMERICAL AND EXPERIMENTAL INVESTIGATION OF UNSTEADY NATURAL CONVECTION IN AN OPEN CHANNEL	C. Ménézo, E. Sanvicente, G.H. Yeoh, J. Reizes, M. Fossa, S. Giroux-Julien, V. Timchenko, G.E. Lau
	8	NC19	Numerical investigation for natural convection in a vertical open-ended channel: Comparison with experimental data	Christophe Daverat, Christophe Ménézo, Shihe Xin, Stéphanie Giroux-Julien, Hervé Pabiou, Zoubir Amine
	8	SM03	Heat Transfer, Phase Change and Coalescence of Particles during Selective Laser Sintering of Metal Powders	Eberhard Abele, Tatiana Gambaryan-Roisman, Ram Dayal
	8	SM04	Fixed-Grid Simulations Of Steady, Two-Dimensional, Ice-Water Systems With Laminar Natural Convection In The Liquid	B. Rabi Baliga, Sylvain Bories, Nabil Elkouh
11:30 - 12:30			**CLOSING SESSION**	

KEYNOTE - 1

LASER INDUCED HYPERTHERMIA OF SUPERFICIAL TUMORS: A TRANSIENT THERMAL MODEL FOR INDIRECT HEATING STRATEGY

Victoria Timchenko[*,§] and Leonid Dombrovsky[**]

[*]School of Mechanical and Manufacturing Engineering,
The University of New South Wales, Sydney 2052, Australia
[**]Joint Institute for High Temperatures, NCHMT, Moscow 111116, Russia
[§]Correspondence author. Fax: +61 2 9663 1222 Email: V.Timchenko@unsw.edu.au

An indirect heating strategy based on laser irradiation of surrounding tissues as an alternative to a direct irradiation of superficial tumors is presented in this paper. The computational analysis is based on two-dimensional axisymmetric models for both radiative transfer and transient heat transfer in a human body. Coupled transient energy equations and kinetic equations for composite human tissue take into account the metabolic heat generation and heat conduction, blood perfusion through capillaries, the volumetric heat transfer between arterial blood and ambient tissue, the thermal conversions in blood and tumor tissue, the periodic laser heating, and also heat transfer between the human body and ambient medium. An example problem for a superficial human cancer has been solved numerically to illustrate the relative role of the problem parameters on the transient temperature field and degree of thermal conversions in human tissues during hyperthermia treatment. In particular, the effect of embedded gold nanoshells which strongly absorb the laser radiation is analyzed. It is shown that required parameters of tumor hyperthermia can be also reached without gold nanoshells.

KEYNOTE - 2

A ROLE FOR COMPUTATIONAL HEAT TRANSFER IN ENGINEERING EDUCATION

Brian Spalding
Concentration Heat and Momentum Limited, UK
Email: brianspalding@cham.co.uk

Traditional engineering education, like its underlying sciences, has two main aspects: theoretical and experimental; and the first of these also has two parts:

- quantitative formulation of the relevant general laws of science; and
- deduction of their implications in particular practical circumstances.

The deductions are conducted by mathematical methods in which differential calculus plays a large part. Students lacking proficiency in such methods are not admitted to engineering schools. Observers who remark that few **practising** engineers ever exercise that proficiency have long doubted the wisdom of the disbarment; and of the excessive attention to functional analysis in engineering curricula. Now that the digital computer makes all the deductions which are needed in engineering practice, those doubts must be seriously addressed. Differential calculus applies the laws of science to **infinitesimal** volumes; and it expresses its deductions in terms of a handful of time-honoured pre- tabulated functions: exponential, logarithmic, trigonometric, *etc*. Only rarely do experiments show that reality conforms to them well enough for use in equipment design, without the application of large safety factors. Computer-based analysis applies the laws to **finite** volumes; and suffers **no restraint** on the form of its tabulated results. Experiments show that reality conforms closely to its deductions very often. Safety factors can therefore be much nearer to unity; with great economic advantage. The present lecturer argues that these facts should be reflected in both the admission procedures and the teaching methods of engineering education. In respect of the second, detailed suggestions are offered as to what should be done. The suggestions are applicable generally across the whole of engineering education; but, being presented at a conference on Computational Heat Transfer (CHT), they are here exemplified by application to heat-exchanger theory.

KEYNOTE -3

DROPLET HEATING AND EVAPORATION - RECENT RESULTS AND UNSOLVED PROBLEMS

Sergei S Sazhin[1] and Morgan R Heikal[1,*]
[1]Sir Harry Ricardo Laboratories, School of Computing, Engineering and Mathematics,
University of Brighton, Brighton BN2 4GJ, UK, e-mail: S.Sazhin@brighton.ac.uk
[*]Current address: Department of Mechanical Engineering, Universiti of Teknologies PETRONAS, Bandar Sri Iskandar 31750 Tronoh, Perak Darul Ridzuan, Malaysia

Recently developed approaches to the hydrodynamic, kinetic and molecular dynamic modelling of fuel droplet heating and evaporation are reviewed. Two new solutions to the heat conduction equation, taking into account the effect of the moving boundary during transient heating of an evaporating droplet, are discussed. The first solution is the explicit analytical solution to this equation, while the second one reduces the solution of the differential transient heat conduction equation to the solution of the Volterra integral equation of the second kind. The new approach predicts lower droplet surface temperatures and slower evaporation rates compared with the traditional approach. An alternative approach to the same problem has been based on the assumption that the time evolution of a droplet's radius $R_d(t)$ is known. For sufficiently small time steps, the time evolutions of droplet surface temperatures and radii predicted by both approaches coincide. A simplified model for multi-component droplet heating and evaporation, based on the analytical solution to the species diffusion equation inside droplets, is discussed. Two new solutions to the equation, describing the diffusion of species during multi-component droplet evaporation taking into account the effects of the moving boundary, are presented. A quasi-discrete model for heating and evaporation of complex multi-component hydrocarbon fuel droplets is described. The predictions of the model, taking into account the effects of the moving boundary during the time steps on the solutions to the heat transfer and species diffusion equations, are discussed. A new algorithm, based on simple approximations of the kinetic results, suitable for engineering applications, is discussed. The results of kinetic modelling, taking into account the effects of inelastic collisions, and applications of molecular dynamics simulations to study the evaporation of n-dodecane droplets are briefly summarised. The most challenging and practically important unsolved problems with regard to the modelling of droplet heating and evaporation are summarised and discussed.

KEYNOTE - 4

ZONAL RANS-LES MODELLING FOR TURBINE AEROENGINES

P. Tucker*, R. Jefferson-Loveday, J. Tyacke and N. Vadlamani
Whittle laboratory, University of Cambridge, UK
*Correspondence author. Email: pgt23@cam.ac.uk

The cost of LES (Large Eddy Simulation) modelling in various zones of gas turbine aeroengines is outlined. This high cost clearly demonstrates the need to perform hybrid LES-RANS (Reynolds Averaged Navier-Stokes) over the majority of engine zones - the Reynolds number being too high for pure LES. The RANS layer is used to cover over the fine streaks found in the inner part of the boundary layer. The hybrid strategy is applied to various engine zones. It is shown to typically give much greater predictive accuracy than pure RANS simulations. However, the cost estimates show that the RANS layer should be disposed with in the low-pressure turbine zone. Also, the nature of the flow physics in this zone makes LES most sensible.

KEYNOTE – 5

LEVEL-SET METHOD FOR MULTIPHASE FLOWS

Y.F. Yap and J. C. Chai*
Department of Mechanical Engineering, The Petroleum Institute, Abu Dhabi, UAE
*Corresponding author. Email: jchai@pi.ac.ae

This article presents a single-fluid formulation for modeling multiphase flows using the level-set method. The formulation is generic in the sense that additional physics involving heat and mass transfer can be incorporated easily. Numerical solutions of the governing equations are performed under the framework of the finite volume method. Three examples are presented to showcase the present approach. The present approach can readily be extended to three-dimensions.

KEYNOTE – 6

THE USE OF VOLUME AVERAGING THEORY TO ADDRESS HEAT TRANSFER WITHIN ENGINEERED HETEROGENEOUS HIERARCHICAL STRUCTURES

Ivan Catton
Department of Mechanical and Aerospace Engineering
School of Engineering and Applied Science
University of California, Los Angeles
Los Angeles, California, 90095 USA
Fax: 1 310 206 4830 Email: catton@ucla.edu

Optimization of Heat Sinks (HS) and Heat Exchangers (HE) by design of their basic structure is the focus of this work. Consistant models are developed to describe transport phenomena within the porous structure that take into account the scales and other characteristics of the medium morphology. Equation sets allowing for turbulence and two-temperature or two-concentration diffusion are obtained for non-isotropic porous media with interface exchange. The equations differ from known equations and were developed using a rigorous averaging technique, hierarchical modeling methodology, and fully turbulent models with Reynolds stresses and fluxes in the space of every pore. The transport equations are shown to have additional integral and differential terms. The description of the structural morphology determines the importance of these terms and the range of application of the closure schemes. A natural way to transfer from transport equations in a porous media with integral terms to differential equations with coefficients that can be experimentally or numerically evaluated and determined is described. The relationship between CFD, experiment and closure needed for the volume averaged equations is discussed. Mathematical models for modeling momentum and heat transport based on well established averaging theorems are developed. Use of a 'porous media' length scale is shown to be very beneficial in collapsing complex data onto a single curve yielding simple heat transfer and friction factor correlations. It was also found that properly defining and using the closure expressions leads to a heat transfer coefficient that is independant of the mode of heating and is constant even within the thermal development region.

KEYNOTE – 7

THE FUTURE OF CFD AND THE CFD OF THE FUTURE

Akshai K. Runchal
Analytic & Computational Research, Inc.
1931 Stradella Road, Bel Air California
runchal@ACRiCFD.com www.ACRiCFD.com

CFD is undergoing a rapid evolution. The distinction between CFD and the so called Structural FE codes is disappearing. The solids and plastics are already being viewed as special subsets of fluids. In the next few years the distinction between the structural and fluid codes will all but disappear. The algorithmic advancements will have to include much stronger emphasis on rheology, fluid structure interaction and physics that includes the complete gamut of solids, plastics, liquids, gases and other phases in-between.

A second evolution is occurring in the computer architecture. The codes of today, with a few exceptions, rely heavily on iterative matrix solvers. The algorithmic core of most of the CFD codes of today was developed when the paradigm was a single CPU with limited memory or a parallel system with multiple CPUs –at the most numbered in 100s - in a MIMD or SIMD architecture. Hence the methodology used is that adapted to such architectures. The current trends indicate that the architecture of the future will be multi-core CPUs, GPUs or ASICs in a cloud or grid-computing architecture. The use of matrix solvers for such architecture will present bottlenecks associated with communication and management software. This will necessitate a new look at how to solve the governing equations and how to do so effectively in the paradigm of cloud computing.

A third evolution will occur in how to implement and view CFD as a design tool. Increasingly the emphasis will shift to embedded applications and CFD as a stand-alone tool will practically disappear. The CFD as an embedded application will merge with virtual reality tools and be included in intelligent AI type interfaces where the emphasis is on the design function of interest rather than on CFD. CFD will then be part of an interactive tool such as the one for x-ray tomography or the performance analysis of an aircraft engine. Further with increasing task specific embedded applications, the day is perhaps not far when specific ASIC chips may implement CFD for such applications. This will give rise to EVR (Engineering Virtual Reality) Design Tools.

In summary, CFD will surely become ubiquitous but so buried that most often it will not be obvious that the CFD tool exists. Everyone knows there is an engine in a car yet hardly anyone cares to ask what that engine is.

KEYNOTE – 8

BUBBLE DYNAMICS DURING POOL BOILING UNDER MICROGRAVITY CONDITIONS

V. K. Dhir
Henry Samueli School of Engineering and Applied Science, UCLA
Los Angeles, California

A numerical tool has been developed over the last decade to study bubble dynamics and associated heat transfer during nucleate pool boiling. The numerical model divides the domain of interest into micro and macro regions. The micro region is the ultra-thin liquid layer that forms between the advancing or receding vapor-liquid interface and the solid wall. The macro region is the vapor-liquid occupied region away from the heated wall and excluding the micro region. Lubrication theory is used for the solution of the microlayer. Complete conservation equations of mass, momentum and energy are solved in the macro region. A level set function is used to capture the evolving, merging and breaking interfaces. Gravity is an important variable of the problem. Experiments at earth normal gravity, reduced gravity in the parabolic flights and microgravity conditions on the International Space Station (ISS) are used to validate the numerical results. The reduced gravity is shown to increase the length and time scale of the process. Although bubble dynamics and vapor removal process (except the bubble size) remain the same up to one-hundredth of earth normal gravity, there is a significant change in the vapor removal pattern under microgravity conditions.

KEYNOTE – 9

THE USE OF TRANSPORT APPROXIMATION AND DIFFUSION-BASED MODELS IN RADIATIVE TRANSFER CALCULATIONS

Leonid A. Dombrovsky
Joint Institute for High Temperatures, NCHMT, Krasnokazarmennaya 17A, 111116, Moscow, Russia
E-mail: ldombr@yandex.ru

The paper presents a discussion of the use of both transport approximation for scattering phase function and diffusion-based models for radiative transfer in absorbing and anisotropically scattering media like many disperse systems in nature and engineering. The main attention is paid to heat transfer problems and traditional methods of identification of spectral radiative properties of dispersed materials when the details of angular distribution of the radiation intensity are not so important. The latter makes reasonable the use of the above mentioned approximations. In more complex applied problems, the diffusion approximation appears to be a good approach at the first step of a combined two-step solution. Some example problems solved recently by the author and his colleagues are used to illustrate the approach considered in the paper.

KEYNOTE – 10

AIR HUMIDIFICATION WITH HOLLOW FIBER MEMBRANES: NEW IMPACTS ON CONJUGATED HEAT AND MASS TRANSFER

Si-Min Huang[*], Li-Zhi Zhang[*,§]
[*] Key Laboratory of Enhanced Heat Transfer and Energy Conservation of Education Ministry, School of Chemistry and Chemical Engineering, South China University of Technology, Guangzhou 510640, China
[§]Correspondence author. Fax: +86 20 87114268 Email: lzzhang@scut.edu.cn

Fluid flow and conjugate heat and mass transfer in a hollow fiber membrane contactor used for air humidification are investigated. The contactor is like a shell-and-tube heat exchanger where the water stream flows in the tube side, while the air stream flows in the shell side in a counter flow arrangement. To overcome the difficulties in a direct modeling of the whole contactor, a representative cell comprising of a single fiber, the water flowing inside the fiber and the air stream flowing outside the fiber, is considered. The air stream outside the fiber has an outer free surface. Further, the equations governing the fluid flow and heat and mass transfer in the two streams are combined together with the heat and mass diffusion equations in membranes. The conjugate problem is then solved to obtain the velocity, temperature and concentration distributions in the two fluids and in the membrane. The local and mean Nusselt and Sherwood numbers in the cell are then obtained and experimentally validated.

The results found that (1) The boundary conditions are neither uniform temperature/concentration nor uniform heat/mass flux boundary conditions. They are developed shortly after inlet. The fully developed Nusselt numbers for air stream $Nu_{C,a}$ is between Nu_H and Nu_T. $Nu_{C,a}$ is closer to Nu_T. The Sherwood numbers ($Sh_{C,a}$) are somewhat higher than the air side $Nu_{C,a}$. (2) The boundary conditions are non-uniform value boundary conditions as well. Mass transfer resistance can be neglected in the water side. Water side $Nu_{C,s}$ is nearly equal to Nu_H. $Nu_{C,s}$ is almost unchanged when the packing fraction is varied. The obtained data can be used for future system design.

KEYNOTE – 11

UNSTABLE ANISOTHERMAL MULTICOMPONENT CONVECTIVE FLOW: FROM SMALL TO LARGE SCALES

Rachid Bennacer
LMT-ENS Cachan, 61 av. du president Wilson
F-94235 Cachan Cedex, France
§Correspondence author. Tel: +33 1 47407478 Email: rachid.bennacer@ens-cachan.fr

The present document covers fundamental, academical, and practical topics. Indeed, the mixed convection in channels heated and cooled differentially has been studied in relation to several practical applications. Most interest in these flows are encountered in several domains to explain certain geological phenomena and atmospheric flows, and intervene in many industrial applications such as cooling of electronic appliances, plastics manufacturing, building sciences, moisture transfer or in the chemical vapour deposition (CVD) and other crystal growth techniques. It either permits simulation of pollution problems or storage of different fluid mixtures either in natural storage (traps structures) or in industrial tanks. Being bound by any such applications, the subject is currently open for exploration, in order to enrich our knowledge of this complicated problem.

We consider the 3D thermosolutal mixed convection (TSMC) with flow confined between two parallel and horizontal planes where the lower and upper surfaces are hot and cold respectively. Such configuration of convective flows is referred to as Poiseuille-Rayleigh-Bénard (***PRB***) which is based upon the famous Rayleigh-Bénard (***RB***) problem. In such ***PRB*** configuration, the flow results from superposition of two convective sources: horizontal pressure gradient that causes the main flow within the duct, and a vertical temperature and/or concentration gradient, which are the cause of thermoconvective structures formation. The stability diagram and the effect of the entrance domain on the exchanges are presented.

We classify the different behaviours and we complete the classical stability analysis by linear stability on a bounded domain. The numerical results demonstrate explicitly the important effect of the entrance domain on the obtained solution and also on the resulting exchange (heat and mass).

GUEST LECTURE

THERMOFLUID/ELECTRICAL CO-DESIGN OF EMERGING ELECTRONIC COMPONENTS – CHALLENGES AND OPPORTUNITIES*

Avram Bar-Cohen[§],
MTO, DARPA
Ankur Srivastava,
ECE, Univ of Maryland
[§]Email: abc@darpa.mil

The continued miniaturization of high performance solid-state electronics and the emergence of 3D chip stack technology have pushed package volumetric heat generation rates to unprecedented levels. While recent DARPA-funded efforts and R&D throughout the electronic industry have aggressively reduced individual resistances, as well as the overall junction-to ambient resistance, of the “thermal stack,” the performance enhancements afforded by the traditional “remote cooling” paradigm have reached a plateau. The thermal conduction and spreading resistances in the commonly used substrates and across multiple material interfaces constrain the ability of remotely located heat rejection surfaces to control the temperature rise of critical devices. Thus, many electronic systems are thermally-limited, performing well below the anticipated physical limits of the device technology.

A paradigm change – bringing aggressive thermal management into the chip and package – is needed to “close the gap” between package-level heat generation rates and system-level heat removal rates. The emergence of an integrated, “embedded cooling” paradigm, designing thermal management into advanced electronic devices in analogous fashion to the way on-chip power distribution is incorporated into the chip and package design, will place thermal management on an equal footing with functional design and power delivery. This “embedded cooling” paradigm will require thermal and electrical co-design to balance the use of resources (e.g., layout area) and identify novel device architectures that tailor the functional layout to the thermal attributes and capabilities.

This lecture will open with a brief history of thermal packaging technology, starting with the vacuum tube ENIAC in 1946, and discuss the “triple threat” posed by current nanoelectronics (high power, hot spots, and 3D architectures). Next, the state-of-the-art in heat sinks, spreaders, thermal interface materials, and near-junction cooling technology will be discussed, along with the need for, and promise of, embedded microfluidic cooling for high performance computing. Attention will then turn to a description of the computational heat transfer challenges associated with bridging the divide between thermofluid analysis and functional design, implemented with the tools and the design flow used in EDA (or ECAD) to lay out high transistor count semiconductor chips. The lecture will close with a discussion of the challenges, developments, and opportunities in thermal, fluid and electrical co-modeling and co-design.

*Full text not available

POSTER PRESENTATIONS

POSTER SESSION 1

CHT12-CM01

MULTI-SCALE MODELLING OF PEMFC CATHODE BY COUPLING FINITE VOLUME METHOD AND LATTICE BOLTZMANN METHOD

Li Chen , Chen-Xi Song, Yong-Liang Feng, Ya-Ling He and Wen-Quan Tao[§]
Key Laboratory of Thermo-Fluid Science and E ngineering of MOE,
School of Energy and Power Engineering, Xi'an Jiaotong University, Xi'an, Shaanxi 710049, China
[§]Correspondence author. Fax: +86 2982669106 Email: wqtao@mail.xjtu.edu.cn

A multi-scale modelling strategy combining finite volume method (FVM) and lattice Boltzmann method (LBM), called coupling simulation strategy, is developed to predict the transport phenomenon and electrochemical reaction in PEMFC cathode with parallel gas channel (GC), a gas diffusion layer (GDL) with porous structures and a catalyst layer (CL) with idealized microstructures. In the strategy, PEMFC cathode side is divided into two sub-domains, one is composed of GC and the other includes the GDL and CL. FVM is used for fluid flow and mass transport in GC sub-domain, while LBM is employed for pore-scale flow and mass transport in GDL and CL and proton conduction in CL in the other sub-domain. Two reconstruction operators transferring macro density and velocities in FVM to density distribution function and concentration distribution functions are developed to transfer information at the interface between the two sub-domains. Besides, electrochemical reactions taking place in the vicinity of the triple-phase contact interfaces in CL are directly considered using LBM in the strategy. The simulation results show that the coupling simulation strategy developed are competent to predict transport phenomena in GDL as well as capture the pore-scale transport process as well as electrochemical reactions in porous GDL and CL.

CHT12-CM02

THE QUANTITATIVE RESEARCH ON RAY TRACING NUMBER OF MONTE CARLO METHOD IN TOTAL RADIANT EXCHANGE AREA COMPUTATION

Li Guojun[*,§], Sun Jiming[**], Liu Xiaoting[*] , Chen Haigeng[*]
[*]School of Materials & Metallurgy, Northeastern University, Shenyang 110004, China
[**]Pangang Group Company Ltd.
[§]ligj@smm.neu.edu.cn

Due to the advantages of flexibility, high accuracy, and easy treatment of complicated boundary conditions, the Monte Carlo method has been widely used in computational physics. One of its important applications is to evaluate the total radiation exchange areas in the closed enclosure. As a kind of statistical method, the Monte Carlo can give rise to certain statistical error and its accuracy is proportional to the tracing number n, while the computation time will increase exponentially with respect to n. In order to reduce the computation time, an analysis on the determination of optimal tracing number both qualitatively and quantitatively is conducted for Monte Carlo method, which is adopted to compute the total radiation exchange areas in a two dimensional closed enclosure. Based on the probability theory, the expressions of the average absolute error and reasonable number n_r

of rays are deduced in this paper; further the relationship between the tracing number n and the accuracy is derived. To avoid the repeat computation, the resulted n_r are given in tabular form for the computation of total exchange areas between the gas and surface zones and they can be referred with ease.

CHT12-CM05

MODELING AND SIMULATION OF THERMAL HEATING IN VERTICAL INTEGRATED CIRCUITS

Abderrazzak El Boukili
School of Science and Engineering, Al Akhawayn University, Ifrane, Morocco
Correspondence author. Fax: +21 2 535 86 20 00 Email: a.elboukili@aui.ma

Interconnect is one of the main performance determinant of modern integrated circuits (ICs). The new technology of vertical ICs places circuit blocks in the vertical dimension in addition to the conventional horizontal plane. Compared to the planar ICs, vertical ICs have shorter latencies as well as lower power consumption due to shorter wires. This also increases speed and adds to ICs density. The benefits of vertical ICs increase as we stack more die, due to successive reductions in wire lengths. However, as we stack more dies, the lattice self-heating becomes a challenging and critical issue due to the difficulty in cooling down the layers away from the heat sink. In this paper, we provide a quantitative electro-thermal analysis of the amount of temperature rise due to stacking. We use mathematical models based on steady state non-isothermal drift-diffusion transport equations coupled to heat flow equation that accounts for spatially varying lattice temperature. These physically based mathematical models and the different heat sources in semiconductor devices will be presented and discussed. Three dimensional numerical results did show that, compared to the planar ICs, the vertical ICs with 2-die technology increase the maximum temperature by 17 Kelvin in the die away from the heat sink. These numerical results will also be presented and analyzed for a typical 2-die structure of complementary metal oxide semiconductor (CMOS) transistors.

CHT12-CM06

MOLECULAR DYNAMICS SIMULATIONS OF MASS TRANSFER DUE TO A TEMPERATURE GRADIENT

Konstantinos Ritos[*,§], Gülru Babaç[*,**] and Jason M Reese[*]
* Dept. of Mechanical & Aerospace Engineering, University of Strathclyde, Glasgow, UK.
** Institute of Energy, Istanbul Technical University, Istanbul, Turkey.
§Corresponding author. Fax: +44 (0)141 552 5105 Email: konstantinos.ritos@strath.ac.uk

The molecular dynamics (MD) technique simulates atomistic or molecular interactions and movements directly through Newton's laws. While to date MD has been mainly applied to study biological systems and chemical processes, there are certain micro and nanoscale engineering applications and technologies that require an understanding of molecular phenomena in order to determine the macroscopic system behaviour. In this paper we demonstrate the application of MD to the benchmark case of the flow of a gas inside a nanochannel connecting two reservoirs with different temperatures. A mass flow is generated between the reservoirs by the thermal

gradient — this phenomenon, known as the "Thermal Creep Effect", is not captured by conventional fluid dynamics with the no-slip boundary condition, and leads to unexpected macroscopic observations. We study the effect of the temperature gradient in cases with different densities and we also report the importance of the wall boundary conditions. Detailed and accurate measurements of temperature, density and pressure that are difficult to obtain through experiments are presented. MD simulations can emulate the realistic molecular conditions and flows, and yield new insight into diffusive transport in non-equilibrium gas flows. This paper demonstrates that the engineer interested in studying and designing new nanotechnologies can deploy molecular dynamics as an effective flow simulation tool.

CHT12-CM08

UPWIND DIFFERENCING SCHEME IN EIGENFUNCTION EXPANSION SOLUTION OF CONVECTIVE HEAT TRANSFER PROBLEMS

D. J. N. M. Chalhub[*], L. A. Spahaier[*,§] and L. S. de B. Alves[**]
[*]Dept. of Mech. Engineering, PGMEC, Universidade Federal Fluminense - UFF, RJ, Brazil.
[**] Dept. de Eng. Mecânica e de Materiais, Instituto Militar de Engenharia - IME, RJ, Brazil
[§]Correspondence author. Email: lasphaier@id.uff.br

A new methodology for solving convective heat transfer problems has been developed, and is herein presented. The proposed solution scheme is based on writing the unknown potential in term of eigenfunction expansions, as traditionally carried out in the Generalized Integral Transform Technique (GITT). However, a different approach is used for handling advective derivatives. Rather than transforming the advection terms as done in traditional GITT solutions, upwind discretization approximations are used prior to the integral transformation. With the introduction of upwind approximations, numerical diffusion is introduced, which can be used to reduce unwanted oscillations that arise at higher Péclet values. The solution methodology is illustrated by employing it for solving a two dimensional Burgers' equation, arising from the analysis of transient thermally-developing flow between parallel plates, with the presence of axial diffusion. The flow is dynamically developed and a robin boundary condition is prescribed at the solid wall. The simulation results show cases for which the dissipative error and the associated numerical diffusion can actually improve the GITT solution. It is seen that a proper usage of the upwind approximation parameter can effectively reduce solution oscillations for higher Péclet values.

CHT12-CM09

AN IMMERSED BOUNDARY METHOD TO SOLVE FLOW AND HEAT TRANSFER PROBLEMS INVOLVING A MOVING OBJECT

Ming-Jyh Chern[*,§], Dedy Zulhidayat Noor[*] and Tzyy-Leng Horng[**]
[*] Department of Mechanical Engineering, National Taiwan University of Science and Technology, Taipei, Taiwan
[**] Department of Applied Mathematics, Feng Chia University, Taichung, Taiwan
[§]Correspondence author. Fax: +886-2-27376460 Email: mjchern@mail.ntust.edu.tw

A direct-forcing immerse boundary method with both virtual force and heat source is developed here to solve Navier-Stokes and the associated energy transport equations to study some thermal flow problems caused by a moving rigid solid object within. The key point of this novel numerical method is that the solid object, stationary or moving, is first treated as

fluid governed by Navier-Stokes equations for velocity and pressure, and by energy transport equation for temperature in every time step. An additional virtual force term is then compensated to the right hand side of momentum equations at the solid object region to make it acting mechanically like a solid rigid body immersed in fluid exactly. Likewise, an additional virtual heat source term is applied to the right hand side of energy equation at the solid object region to maintain the solid object at prescribed temperature all the time. The current method was validated by some benchmark forced and natural convection problems such as a uniform flow past a heated circular cylinder, and a heated circular cylinder inside a square enclosure. We further demonstrated this method by studying a mixed convection problem involving a heated circular cylinder moving inside a square enclosure. Our current method avoids the otherwise requested dynamic grid generation in traditional method and shows great efficiency in the computation of thermal and flow fields caused by fluid-structure interaction.

CHT12-CM10

COMBINED NUMERICAL AND ANALYTICAL METHOD FOR FIN ARRAY HEAT TRANSFER

Reijo Karvinen, Kaj Lampio§
Tampere University of Technology, Finland
Department of Energy and Process Engineering, P.O. Box 589, 33101 Tampere, Finland
§Correspondence author. Email: kaj.lampio@tut.fi

Fins and fin arrays are used for cooling heat generating components in electronics. Heat transfer composed of simultaneous convection and conduction can be solved numerically using commercial CFD codes but it takes plenty of time especially if the goal is to optimize the geometry. A faster method is obtained by adopting analytical expressions for convection and solving only conduction numerically. In the paper this type of approach and equations are presented, and the validity of the method is checked by comparing results to measured data and to purely numerically obtained results. It was observed that the calculation time using the presented method is much shorter compared to that if velocity and temperature fields of the flow and solid are solved numerically. Thus, the method is a very suitable tool for instance in a multiobjective optimization where hundreds of solutions are required.

CHT12-CM11

ON COMPARISON OF MOMENTUM, EULERIAN FRACTIONS AND δ -APPROXIMATION METHODS FOR MODELLING OF TWO-PHASE TURBULENT FLOWS WITH PHASE TRANSITIONS

Artur R. Avetissian[*,§], Gennady A. Philippov[**]
[*]Nuclear Safety Institute of the Russian Academy of Sciences, Moscow, Russia
[**]Russian Academy of Sciences, Moscow, Russia
[§]Correspondence author. Email: avetis@ibrae.ac.ru

In the paper we present the comparison of three Eulerian approaches (fractions, moments and δ -approximationton methods) to solve the kinetic equation for the probability density function (PDF) of dispersed phase mass distribution for modeling of two-phase turbulent flows with phase transitions. The results of two-dimensional computations are presented and compared with experimental data for transonic flows in flat and round nozzles in the case of both the absence and the presence of initial wetness supplied to the nozzle inlet. Methods under consideration compared in view of some exact solutions of kinetic equation. The results are depicted. The advantages and disadvantages of each method are discussed.

CHT12-CM12

ADVECTION AND DIFFUSION SIMULATIONS USING LAGRANGIAN BLOCKS

Vincent H. Chu[*,§] and Wihel Altai[*]
[*]Dept. of Civil Engineering and Applied Mechanics, McGill University, Canada.
[§]Correspondence author. Fax: +514 398 7361 Email: vincent.chu@mcgill.ca

The Lagrangian block advection and diffusion is offered as an alternative numerical procedure to the solution of the transport equation. The blocks are computational elements that are defined by the zero moment, the first moments and the second moments of the blocks. The mass centres move with the advection velocity. The second moments of the blocks increase at a rate proportional to the diffusivity. The accuracy of the simulations by the Lagrangian block method is assessed by comparing the block simulation with the exact solution of the advection-and-diffusion equation. Unlike most numerical methods, the error associated with the Lagrangian block method is small and is not cumulative even when a very coarse block size is employed for the computation. Artificial numerical diffusion error is totally avoidable when Lagrangian blocks are used to do the computation.

CHT12-CM13

A PREDICTOR-CORRECTOR SPLIT PROJECTION METHOD FOR TURBULENT REACTIVE FLOW

David B. Carrington[*,§], Xiuling Wang[**] and Darrell W. Pepper[***]
[*]T-3 Fluid Dynamics and Solid Mechanics, Los Alamos National Laboratory, NM, USA
[**]Department of Mechanical Engineering, Purdue University Calumet, Hammond, IN,USA
[***]Department of Mechanical Engineering, University of Nevada, Las Vegas, NV, USA
[§]Correspondence author. Fax: 505 667 3568 Email: dcarring@lanl.gov

A Predictor-Corrector Split (PCS) projection method based on a fractional step Finite Element Method (FEM) is used for modeling turbulent combustion. The PCS system advances the accuracy and range of applicability of the KIVA combustion model. The algorithm, combined with KIVA's spray and chemistry models and a moving mesh capability, is being implemented into a new generation of KIVA software, KIVA-hpFE to increase modeling predictability. The FEM formulation uses an *h*-adaptive procedure to model turbulent reactive flow over a wide range of velocities of various fluids. A *k-ω* turbulent closure scheme is used in conjunction with the turbulent form of the Navier-Stokes equations.

The method is applicable to Newtonian and non-Newtonian flows and engineering problems involving fluid structure interactions, porous media, and solidification. The model is designed to produce a minimal amount of computational effort when compared to fully resolved grids at comparable accuracy.

CHT12-CM14

PURELY ELASTIC FLOW ASYMMETRIES IN CROSS-SLOT GEOMETRY - ARE THEY ALWAYS PRESENT?

A. Filali[1], L. Khezzar[1(*)], F. T. Pinho[2] and M. A. Alves[3]
[1]Department of Mechanical Engineering, Petroleum Institute, Abu Dhabi, United Arab Emirates.
[2]CEFT, Department of Mechanical Engineering, Faculty of Engineering,
[3]Department of Chemical Engineering, CEFT, Faculty of Engineering, University of Porto, 4200-465, Porto, Portugal.
[(*)]Corresponding author. Fax: 0097126075200 Email: lkhezzar@pi.ac.ae

Numerical investigations of purely-elastic instabilities occurring in creeping flows ($Re \approx 0$) are reported in planar cross-slot geometries with both Sharp and Round corners. The fluid is described by the Upper-Convected Maxwell (UCM) model, and the governing equations are solved using the finite element technique in the POLYFLOW code. The calculations performed show that the onset of flow asymmetries above a critical Deborah number is extremely sensitive to the numerical characteristics. Specifically, extensive simulations were carried out on symmetric and distorted meshes for a wide range of rheological parameters showing that for asymmetric meshes and above a critical Deborah number De_{cr}, the flow becomes asymmetric, but remains steady. The effect of rounding the corners is also addressed. When the flow becomes asymmetric, the numerical results obtained are found to be in good quantitative agreement with previously published numerical results that include velocity profiles, stream function, variation of Weissenberg number with De and flow asymmetry parameter.

CHT12-BL01

LINEAR INSTABILITY TO LONGITUDINAL ROLLS OF THE DARCY-HADLEY FLOW IN A WEAKLY HETEROGENEOUS POROUS MEDIUM

Antonio Barletta[*,§] and Donald A. Nield[**]
[*]DIENCA, Alma Mater Studiorum – Università di Bologna, Viale Risorgimento 2, Italy
[**]Department of Engineering Science, University of Auckland, New Zealand
[§]Correspondence author. Fax: +39 051 209 3296 Email: antonio.barletta@unibo.it

The aim of this contribution is to investigate the effects of a weak vertical heterogeneity of the porous medium on the stability of the Darcy-Hadley flow. We will assume that the permeability and the thermal conductivity of the porous medium undergo a weak linear change in the vertical direction. The stability analysis will be carried out by considering linear perturbations of the basic state in the form of longitudinal rolls, under the hypothesis that the weak heterogeneity

does not alter the result that these modes are the most unstable. The neutral stability conditions and the critical values of the vertical Darcy-Rayleigh number will be obtained on varying the horizontal Darcy-Rayleigh number as well as the parameters of heterogeneity relative both to the permeability and to the conductivity. The solution of the linear disturbance equations will be obtained by employing Galerkin's method of weighted residuals.

CHT12-BL02

CONVECTIVE INSTABILITY IN AN INCLINED POROUS LAYER SUBJECT TO LINEARLY VARYING BOUNDARY TEMPERATURES

Antonio Barletta[*,§] and Andrew Rees[**]
[*]DIENCA, Alma Mater Studiorum – Università di Bologna, Viale Risorgimento 2, Italy
[**]Dept. of Mechanical Engineering, University of Bath, UK.
[§]Correspondence author. Fax: +39 051 209 3296 Email: antonio.barletta@unibo.it

The stability of the parallel Hadley flow in an inclined plane porous layer where the boundaries are subject to a linearly varying temperature is studied. The inclination of the layer modifies the nature of the basic single-cell Hadley flow. In fact, the Hadley flow in an inclined layer displays a nonlinear velocity profile, with the possible existence of singularities for special inclination angles of the layer to the horizontal. The stability of the basic flow in the inclined porous layer is investigated by a linear analysis of the longitudinal rolls. The stability analysis provides the neutral stability curves for the onset of convection, and the critical values of the Rayleigh number for different inclinations of the layer. The linearized disturbance equations are solved by means of the collocation method of weighted residuals.

CHT12-BL03

EFFECT OF WIDTH OF A VERTICAL PARALLEL PLATE CHANNEL ON THE TRANSITION OF THE DEVELOPING THERMAL BOUNDARY LAYER

Ali S. Alzwayi and Manosh C. Paul[§]
Systems, Power & Energy Research Division, School of Engineering,
University of Glasgow, Glasgow, G12 8QQ, UK
[§]Correspondence author. Fax: +44(0)141 330 4343 E-mail: Manosh.Paul@glasgow.ac.uk

Numerical simulations were performed to study the transition of the development of the thermal boundary layer of air along an isothermal heated plate in a large channel which is bounded by an adiabatic plate. In particular, the aim is to investigate the effects of the channel width, *b*, on the transition of the flow under various ambient air and plate temperatures. Three different RANS based turbulent k-ε models, namely standard, RNG and Realizable with an enhanced wall function were employed in the simulations. The channel width was varied from 0.04 m to 0.45 m and the numerical results of the maximum values of the flow velocity, turbulent kinetic energy were recorded along the vertical axis to examine the critical distance of the developing flow. The results show that the transition delays when the width is increased from 0.04 m to 0.08 m, and particularly, the critical distance at b = 0.08 m reaches its maximum with the Grashof number of 2.8×10^{10}. However, when b is increased further from 0.08 m to 0.45 m, the critical distance drops, indicating an early transition of the flow. Comparisons of the selected numerical results are made with available experimental data of turbulent flow, and a satisfied agreement is received.

FALSE DIFFUSION IN MANAGING SPURIOUS OSCILLATIONS BY FLUX LIMITERS

Vincent H. Chu[*,§] and Congwei Gao[*]
[*]Dept. of Civil Engineering and Applied Mechanics, McGill University, Canada.
[§]Correspondence author. Fax: +514 398 7361 Email: vincent.chu@mcgill.ca

ABSTRACT The error in estimating fluxes on the faces of the finite volume using a truncated series produces spurious numerical oscillations. Occasional switching to the diffusive upwind scheme is the strategy of most flux limiters to gain computational stability. An extensive series of numerical simulations has been carried out to evaluate the performance of six well-known flux limiters: MINMOD, SUPERBEE, SMART, MUSCL, Ultra-QUICK and Ultra-CD. False diffusion coefficients and over- and under-shoots associated with the simulations are determined for wide range of Courant number and grid size. The diffusion error produced by the simple advection using the flux limiters MINMOD and Ultra-QUICK on a coarse grid are delineated in Fig. 1 for one Courant number. Flux limiter such as MINMOD produces computationally stable solution but the numerical solution is significantly compromised by the false diffusion. Ultra-QUICK is less diffusive. However, it is most unstable. The MINMOD may be selected for computational stability. The Ultra-QUICK on the hand could be selected for its low diffusivity. The paper is a comprehensive study of false diffusion. The result is presented in the form of a guide for judicious selection of a flux limiter.

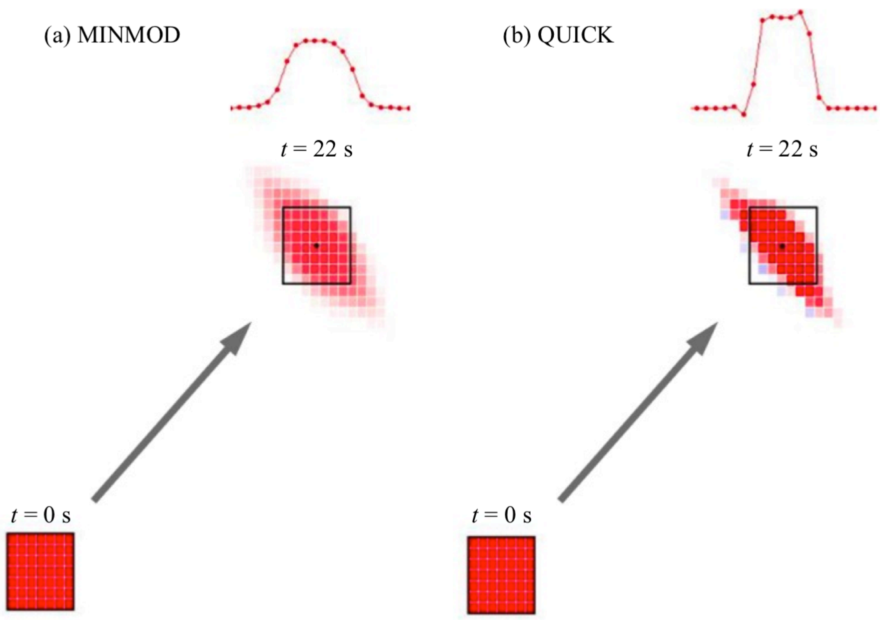

Figure 1. Advection of a 0.5-m x 0.5-m block of tracer in 45-degree direction using the flux limiter (a) MINMOD and (b) Ultra-QUICK; Courant number Co = 0.2, $\Delta x = \Delta y = 1/7$ m, $U = V = 0.1$ m/s. The black frame outlines the exact solution. The profile at the top-right corner of the figure shows the tracer concentration distribution along the cross-section in the middle.

POSTER SESSION 2

CHT12-HE02

THREE DIMENSIONAL NUMERICAL STUDY OF FLUID FLOW AND HEAT TRANSFER CHARACTERISTICS ON NOVEL HEAT EXCHANGERS

Yu Wang, Ya-Ling He[§], Li-Ting Tian, Wen-Quan Tao
Key Laboratory of Thermo-Fluid Science and E ngineering of MOE,
School of Energy and Power Eng., Xi'an Jiaotong University, Xi'an, Shaanxi 710049, China
[§]Correspondence author. Fax: +86 2982665930 Email: yalinghe@mail.xjtu.edu.cn

Three-dimensional numerical simulations are performed study the air-side heat transfer and fluid flow characteristics of the fin-and-tube heat exchangers with delta winglets. The Reynolds number based on the tube outside diameter ranges from 204 to 816. The effects of three different arrangements of delta winglets are studied, and the overall and local performance comparisons among the fin with delta winglets, slit fin, slit-wavy complex fin and wavy fin are performed. The numerical results show that the delta winglet applied in the fin is a very promising heat transfer enhancement technology for the fin-and-tube heat exchangers. The delta winglet obviously enhances the heat transfer of the fin while the pressure drop penalty induced by the delta winglets is relatively small. Compared with the wavy fin, the pressure drops for the fin with delta winglets decrease by 6.0%~19.9%, 0.4%~10.7% and 0.2%~6.5% for the fin with delta winglets in NCFD, CFD and CFU, respectively. The slit fin not only enhances the heat transfer of the fin but also increases the pressure drop. Based on the comparisons of the performance evaluation criteria for the heat exchanger, the overall performance of the fin with delta winglets in NCFD is better than that of the others. The configuration of the delta winglets has great effectiveness on the overall and local performance.

CHT12-HE03

A MICRO FLOWMETER BASED ON THE MEASUREMENT OF A DIFFUSION TEMPERATURE RISE OF A LOCALLY HEATED THERMAL FLOW IN A HAGEN-POISEUILLE FLOW

Hiroyoshi Koizumi*, Yoshihiro Kato and Taichi Kimura
Dept. of Mechanical Engineering and Intelligent Systems,
The University of Electro-Communications, Tokyo, Japan
*Correspondence author. Fax: +81 42 443 5395 Email: koizumi@mce.uec.ac.jp

A simple measurement method for micro volumetric flow rate with high precision and low cost is proposed for industrial use. A locally heated thermal flow is produced by small electric heater that is placed in the center part of an upwardly directed Poiseuille flow, and by a thermocouple that is set in the downward position. Furthermore, the diffuser-reducer is set in a halfway position within the pipe with an inner diameter of 3 mm for advancing the mixing of the locally heated fluid. It was found that the volumetric flow rate Q_v in a pipe is directly proportional to the maximum diffusion temperature rise of the heated flow ΔT_{max} between the heater and the sensor. Q_v can be obtained by using the linear relationship between ΔT_{max} and Q_v. The linear relationship between ΔT_{max} and Q_v was confirmed for

water flow rate below 1 mL/min experimentally and numerically. Furthermore, this flowmeter numerically confirmed that the change of water temperature at the pipe inlet could not affect the relationship between ΔT_{max} and Q_v. Transient numerical simulations are performed using Storm/CFD2000 software. The unsteady three-dimensional Boussinesq set of equations was used in order to validate the measurement principle of this new micro flowmeter, and also to determine the optimized flowmeter design. Furthermore, this flowmeter could be applied as a leak detector, because it is possible to measure the flow rate down to Q_v =0 mL/min.

CHT12-HE04

PRESSURE DROP AND HEAT TRANSFER IN SPIRALLY CORRUGATED TUBE FOR A COUNTER-FLOW HEAT EXCHANGER

Apichart Chaengbamrung*,§, Amarin Tongkratoke* and Anchasa Pramuanjaroenkij**
[*]Department of Mechanical Engineering, Kasetsart University, Bangkok, 10900, Thailand.
[***] Department of Mechanical and Manufacturing Engineering, Kasetsart University, Chalermphrakiat Sakon Nakhon Province Campus, Sakon Nakhon, 47000, Thailand.
[§]Correspondence author. Fax: +66 2 5794576 Email: fengacc@ku.ac.th

This research studied heat transfer enhancement of a counter-flow heat exchanger by replacing a smooth inner tube with a spirally corrugated tube inside the heat exchanger. Results are obtained from the computational fluid dynamics (CFD) program. Performance of heat exchanger can be enhanced by increasing heat transfer area and increasing the turbulent intensity of fluid flow. A three-dimensional, steady, forced turbulent, and convectional flow model of fluid were developed in this research, water flowing inside smaller tube with an inner diameter of 20 mm., oil flowing in the annulus with the outer diameter of 76 mm., and domain length of 900 mm. The flow and physical parameters of the spirally corrugated tube; the corrugated pitch (*p*) and Reynolds number (Re) which affect friction factor (*f*) and Nusselt number (*Nu*) of the tube were studied in order to provide equations in which, later, was used to predict solutions numerically by calculating friction factor and Nusselt number of the tube. From the numerical results, we found that the friction factor depended on the corrugated pitch and Re as shown in the equation; $f = -0.0091\text{Re} + 0.0715$, and $f = (3\times10^{-5})p^3 - 0.002p^2 - 0.004\,p^2 + 0.0043$. However, the Nusselt number is expressed numerically as $Nu = 89.93\text{Re} + 3.4578$, and $f = -1.5674p^3 - 20.217p^2 - 84.042\ p^2 + 232.34$. This simulation is valid for following ranges; $10000 < \text{Re} < 40000$ and the pitch distances between 20 mm. and 120 mm

PARAMETRIC STUDIES ON MEMBRANE-BASED ENERGY RECOVERY VENTILATOR PERFORMANCE

Jingchun Min[*,§], Xiaomin Wu[**], Ming Su[***]
[*] Dept. of Engineering Mechanics, Tsinghua University, Beijing 100084, China
[**] Dept. of Thermal Engineering, Tsinghua University, Beijing 100084, China
[***] Energy Research Inst., National Development and Reform Commission, Beijing 100038, China
[§]Correspondence author. Fax: +86 10 6279 5832 Email: minjc@tsinghua.edu.cn

A mathematical model was presented to predict the performance of an energy recovery ventilator with a vapour-permeable membrane core. The model considers combined heat and mass transfer as well as pressure drop in the core. The model provides analytical expressions for the heat and moisture transfer resistances through the membranes, both of which are operating condition dependent. Numerical calculations are carried out to investigate the effects of the membrane core geometry, the membrane physical property and the outdoor air state on the ventilator performance. The core geometry includes the membrane spacing (channel height) and thickness, the membrane property includes the membrane transport and adsorption parameters, and the outdoor air state includes the outdoor air temperature and humidity. The calculation results show that under equal fan power condition, as the channel height increases, the total heat transfer rate initially increases, after attaining to a maximum, turns to decrease, whereas the enthalpy effectiveness monotonously decreases. As the moisture diffusivity in membrane increases, the total heat transfer rate increases; as the membrane adsorption constant increases, the total heat transfer rate initially increases, after reaching a maximum, turns to decrease. Similar changes are observed for the enthalpy effectiveness. As the outdoor air temperature or humidity increases, the total heat transfer rate increases. For fixed outdoor temperature, as the outdoor humidity increases, the enthalpy effectiveness initially decreases to a minimum and then increases. The enthalpy effectiveness decreases with increasing outdoor temperature for low outdoor humidities but shows little change with the outdoor temperature for higher humidities.

EFFECT OF OPERATING CONDITIONS ON PERFORMANCE OF PROTON EXCHANGE MEMBRANE FUEL CELL (PEMFC)

Kerboua ZiariYasmina[*,§] Kerkoub Youcef[*] , Benzaoui Ahmed[*]
[*]Laboratory of Thermodynamics and Energy Systems, Faculty of Physics, University of Science and Technology Houari Boumediene (USTHB), BP 32 El-Alia Bab Ezzouar, Algiers,
[§]Correspondence author. Email: yasminaziari @yahoo.fr, ykerboua@usthb.dz

A three dimensional, non isothermal and steady state model is presented for all elements of single channel of proton exchange membrane fuel cell (PEMFC). In this model are taken into account all the elements of the cell, the solid collectors, the two flow channels, membrane diffusers and catalyst layers. The catalyst layers are considered as volumes with finite thicknesses rather than interfaces to predict all transport phenomena with high accuracy. The purpose of this work is modelling and simulation of transport of reactants, electrochemical

reactions, heat, charge species and investigate the effect of operating conditions such as pressure, temperature~~,~~ and humidity of inlet gases over the performance of PEM fuel cell and predict how these operating conditions influence the water management in the cell and the photonic conductivity of the member, and consequently the performance of the cell. This paper especially focuses on the effect of pressure gradients of reactant inlet gases on the efficiency of the cell. The model is implemented into computational fluid dynamics (CFD) for solving all coupled equations. The results derived using the proposed model compare with known experimental results for certain conditions, described in published research work.

CHT12-HE07

NUMERICAL SIMULATION OF A PHASE CHANGE MATERIAL (PCM) IN A DOMESTIC REFRIGERATOR POWERED BY PHOTOVOLTAIC POWER

M. Berdja[*,§]. Abbad[*] , M. Laidi[*], F. Yahi[*] and M. Ouali[*].
[*]Unité de développement des équipements solaires, (UDES), N°11,
BP386, Bou-Ismail, 42415, Tipaza, Algeria
[§]Correspondence author. Tel. / Fax : 00 213 24 41 02 00 /01 33 berdjamohand@yahoo.fr

We present in this work the results of a numerical simulation of a phase change material (PCM) used as a storage device in a domestic vapor compression refrigerator powered by photovoltaic energy. The phase change material is a eutectic solution. The PCM permits to keep the refrigerator compartment temperature quasi constant during the extended shutdown of the compressor. We use the finite volume method (FVM) as a numerical approach to address the conservation equations. The Phase change is simulated using the enthalpy method.

Our goal is to simulate - through a mathematical modeling of the phase change Kinetics phenomena in the PCM - the operating regime of the refrigerator. The model validity will be provided by the consistency of the numerical solution of the solidification phenomena, with the experimental results obtained in the laboratory. This comparison is made according to the start and the switch off sequences of the compressor, controlled by the temperature at the thermostat, which is located on the lateral side of the aluminum shell containing the PCM. The use of a PCM is a way of decreasing the cost of the photovoltaic systems by opting for a thermal storage cooling instead of electric storage batteries. It is also a way of minimizing the electrical consumption by reducing the operating time of the compressor.

CHT12-HE08

NUMERICAL INVESTIGATION OF TRANSPIRATION COOLING OF A LIQUID ROCKET THRUST CHAMBER WALL

Hikmet S. Aybar[1], Mehdi M. Faridani[1,§], Mehmet Sözen[2]
[1]Dept. of Mechanical Eng., Eastern Mediterranean University, G. Magosa, Mersin10, Turkey
[2]College of Eng. & Comp., Grand Valley State University, Grand Rapids, MI 49504
[§]Correspondence Author. Fax: +90 392 365 3715 Email: mehdi.faridani@cc.emu.edu.tr

The purpose of this research was to develop and compare different numerical schemes modeling one-dimensional transpiration cooling in order to expand the model to a more realistic two-dimensional model. In the current research, mathematical models and computer codes were developed to simulate transpiration cooling process in a liquid rocket thrust chamber wall for different situations. Explicit and implicit schemes were applied to the incompressible model. A comparison was made between explicit and implicit incompressible model.

CHT12-HE09

OPTIMIZATION OF FLUIDIZED HORIZONTAL HEAT EXCHANGER WITH LENGTHWISE DISPERSION

Artur Poświata
Faculty of Chemical and Process Engineering, Warsaw University of Technology, Poland
Correspondence author. Fax: +48 22 825 14 40 Email: poswiata@ichip.pw.edu.pl

In this study authors considered optimization (minimization of process cost) of solid particles heating in fluidized bed. The solid particles flows along the bed and are heated by hot gas provided from the bottom. Due to temperature profile, the dispersion of solid particles is present. Therefore the influence of the dispersion on optimal run of process is investigated in this work. The hydrodynamics of fluidized bed is described by two-phase Kunii – Levenspiel's model. Moreover for horizontal fluidized heat exchanger we assumed that the gas flows only vertically, the solid particles are ideally mixed in cross-section of the bed whereas the dispersion flow along the bed length is present.

The profile of inlet temperature of heating gas and total gas flow rate, which minimized the total process cost, are searched during optimization. Because economic values are subject of local and time fluctuation, the accepted objective function describes cost of process expressed in exergy units. The continuous optimization algorithm, developed by Pontryagin, is used to calculation. The adaptation of model equations to can be used in the algorithm and determination the boundary conditions for optimization are presented. The results of the optimization, as optimal trajectories for temperatures of inlet gas and solid are presented. The influence of heat transfer kinetics and dispersion coefficients on optimal runs of the heating process is discussed. Our investigations indicated that for processes with dispersion the optimal profiles of temperature for normalized time also are independent on process kinetics and hydrodynamics if the Peclet Number is constant. The process conditions influent only on the total gas flow rate.

CHT12-HE10

MODEL-BASED APPLICATION FOR LEAK TESTING OF BURIED FUEL LINES

Michael R. Maixner[*,§], Michael O'Donnell[*] and Timothy Smith[*]
[*]Department of Engineering Mechanics, United States Air Force Academy, Colorado, USA
[§]Correspondence author. Fax: +1 719 333 2744 Email: Michael.Maixner@usafa.edu

Isobaric and isometric test procedures are typically used in lieu of more conventional hydrostatic tests on buried airport fuel hydrant systems; using currently installed equipment, the isometric test is implemented in a spreadsheet-style application, designed for ease-of-use by operating and maintenance personnel on-site who may not have a knowledge of fluid flow, heat transfer, and thermodynamics. Visual Basic for Application™ (VBA) program execution is initiated through buttons provided on data worksheets. Final analysis results in a recommendation that the test may be considered satisfactory (i.e., the likelihood of a leak is small), or that a leak (or leaks) exist(s) with an estimate of the magnitude of the leak. Model design was conducted using the REFPROP program from the National Institute of Standards; REFPROP calculations were validated against a simple one-dimensional heat transfer simulation. On-site testing and validation of the leak testing program will be performed during summer 2012.

CHT12-HE12

CONJUGATE HEAT TRANSFER MODELLING OF INTERNAL COMBUSTION ENGINE STRUCTURES AND COOLANT FLOWS

Vlado Pržulj[*,†], Richard Penning[*] and Nick Tiney[*]
[*]Ricardo Software, Shoreham–by–Sea, West Sussex, BN43 5FG, UK
† Correspondence author. Email: vlado.przulj@ricardo.com

This paper reports on the application of a pressure–based finite volume method for conjugate heat transfer analysis of an automotive engine cooling process. The method provides the implicit (simultaneous) solution of the energy equation throughout any number of fluid and solid domains. In this study, the coolant flow domain and solid components of the engine structure are considered. Close attention is paid to the discretization of the energy equation in terms of temperature, which is well suited for the conjugate heat transfer solution. Within the coolant domain, the RANS equations in conjunction with a variant of the k – Ú model and with enhanced wall functions are solved by using the SIMPLE–like algorithm. Two buoyancy–driven cavity flow benchmarks are used to validate the present solution method. As it is not practical to perform simultaneous computations of the coolant flow, engine structure and the complex unsteady in–cylinder flow, the effects of in–cylinder (gas side) phenomena are modelled by empirically derived technique. This technique estimates the three-dimensional heat flux distribution at combustion chamber walls. Using the empirical gas side thermal conditions, CHT simulations are performed for an in-line four-cylinder Diesel engine, for two steady–state running conditions at 2000 rpm and 4000 rpm. In order to assess the reliability of these simulations, a comparison of the predicted and measured temperature profiles at three locations along the inner and outer part of the cylinder liner is carried out. Agreement between the predicted and measured profiles is found to be satisfactory for the engine speed of 2000 rpm. The thermal analysis has shown that

temperature and coolant velocity fields are within the expected limits for the engine used for this study.

CHT12-EN01

SEPARATION OF CARBON DIOXIDE AND HYDROGEN FROM SYNGAS BY PRESSURE SWING ADSORPTION

Fei-hong Chen[*], Chih-Hsiang Huang[*], Yu-xuan He[*], Hong-sung Yang[**] and Cheng-tung Chou[*,§]
[*]Department of Chemical and Materials Eng., National Central University, Jhong-Li, Taiwan
[**]Department of Chemical Eng., Hwa-Hsia Institute of Technology, Chung-Ho District, New Taipei City, Taiwan
[§]Correspondence author. Fax: +886 3 425 2296 Email: t310030@ncu.edu.tw

The CO composition in syngas is reacted with steam to generate CO_2 and H_2 via water-gas-shift reaction. In this study, pressure swing adsorption (PSA) is utilized to separate H_2 and CO_2 from outlet stream of water-gas-shift reactor at room temperature. The purified H_2 can be sent to gas turbine for generating electrical power or can be used for other energy source, and CO_2 can be recovered to reduce the green-house-gas effect. Pressure swing adsorption is a cyclic process to separate gas mixtures based on the difference of adsorption capacity of each component on adsorbent. This technology consists of gas adsorption at high pressure and desorption at low pressure to produce high-purity product. In this simulation study, adsorbent AC5-KS and a dual-bed eight-step PSA process are utilized to separate H_2 and CO_2 from the outlet stream of water-gas-shift reactor at 303K. Non-moisture inlet is studied for PSA process, i.e., water is removed before entering PSA. After that, the feed entering PSA process includes 1.3% CO, 41.4% CO_2 and 57.3% H_2. The optimal operating condition is obtained by varying the operating variables, such as feed pressure, bed length and step time. This process could achieve 99.98% purity and 78.55% recovery of H_2 for top product, and about 75.87% purity and 100% recovery of CO_2 for bottom product.

CHT12-EN02

PISTON EFFECT CHARACTERISTIC TIME DEPENDENCE ON EQUATION OF STATE MODEL CHOICE

P. C. Teixeira[*] and L. S. de B. Alves[*,§]
[*]Dept. of Mechanical Engr. and Material Sci., Military Institute of Engr., Rio de Janeiro, Brazil
[§]Correspondence author. Fax: +55 21 25467045 Email: leonardo_alves@ime.eb.br

The present paper proposes a novel equation of state based on the compressibility factor definition that is as simple as the van der Waals equation of state but more accurate for arbitrary fluids near the thermodynamic critical point. A reference simulation, that utilizes a polynomial correlation as equation of state based on empirical data from NIST, is used here as the base for comparisons. All three equations of state are compared through the classical Piston Effect problem describing the heat transfer inside a one-dimensional cavity filled with CO_2 at its critical pressure. The novel formula leads to accurate results with less than 5% relative error for temperatures as close as 1K from the critical one, whereas the threshold is approximately 10K for the van der Waals equation of state.

CHT12-EN03

INFLUENCE OF THERMAL BEHAVIOUR OF A PHOTOVOLTAIC MODULE ON ITS ELECTRICAL PERFORMANCE

Lucas Weiss[1, 2,§], Mohamed Amara[1] and Christophe Menezo[1,3]
[1]Centre de Thermique de Lyon (CETHIL, CNRS-INSA Lyon-UCBL), Villeurbanne, France
[2]Voltec Solar, Dinsheim, France
[3]Chaire INSA of Lyon/EDF "Habitats et Innovations Energétiques" ,Villeurbanne, France
([§] corresponding author: lucas.weiss@insa-lyon.fr)

Photovoltaic (PV) energy is considered as one of the most promising ways to reduce environmental impact of electricity production. The success of an energy source depends on its global cost. Recent years have seen a huge reduction of the cost/performance ratio of PV due to political, economical and research efforts. For the latter case, which represents the scientific level, efforts lead to improvements of devices' efficiency.

Research on the PV modules can bring new solutions for the improvement of PV performance. However, different effects also lead to a reduction of PV performance: first optical effects due to encapsulation, second electrical effects due to inter-connection of the cells into the module, third thermal effects when the module is functioning, also additional consumable costs can be considered a fourth effect since the global cost is the final measurement value of performance. A complete understanding of the influence degree of these different effects is needed in order to increase.

This work is intended to evaluate the effect of heat transfer on PV performance. This approach allows a desired optimization of all parameters. Within this frame, we developed a physical and numerical model, which resolves optical, electrical and heat transfer problems involved in the PV module. The module is composed of a solar cell encapsulated into Ethyl Vinyl Acetate (EVA) with a protecting glass as frontsheet and a protecting polymer as backsheet. Input elements are environmental boundary conditions (illumination, wind speed, ambient temperature) and output factors are temperature of the solar module and power of the module. This approach has the advantage of providing information in order to optimize materials and strategy for PV modules' construction. Through a simple optimisation of material parameters, an increase of performance of 1.8% was obtained compared to standard technology.

CHT12-EN04

PRESSURE WORK AND VISCOUS DISSIPATION INFUENCES ON FLOW ENERGY AND ENTROPY BALANCES

Gordon Mallinson[§] and Stuart Norris
Department of Mechanical Engineering, The University of Auckland, New Zealand
[§]Correspondence author. Fax: +64 9 373 7549 Email: g.mallinson@auckland.ac.nz

The contributions of pressure work and viscous dissipation to heat transfer by natural convection in a closed cavity that may have a sliding lid have been estimated numerically to determine how they interact with the flow and heat transfer processes. By using a computational method that has been carefully constructed to guarantee algebraic conservation it has been shown that the pressure work and viscous dissipation contributions, although very small, do satisfy a first law analysis of the problem in that their difference is equal to the work done by the moving lid. In order to achieve this balance and to correctly represent the fluid dynamics and heat transfer, both pressure work and viscous dissipation must be included in numerical models.

OF-02

THE EFFECT OF NON-NEUTRAL STABILITY IN WIND FARM SIMULATIONS

C A Montavon and I P Jones
ANSYS UK Ltd., 97 Milton Park, Abingdon OX14 4RY

Introduction

Varying atmospheric stability conditions have a significant impact on the performance of wind farms. In this light, understanding the effect of atmospheric stability becomes very important.

From a physical perspective, stability conditions affect the development of atmospheric turbulence, promoting turbulence in unstable conditions, and damping the turbulent fluctuations in stable conditions. This can affect the flow in complex terrain, and also wake recovery downstream of a wind turbine. While additional turbulence is not a desirable feature from a perspective of structural integrity, it has the effect to increase turbulent mixing, thereby allowing a faster wake recovery. Reduced turbulence levels in stable conditions inhibit wake recovery and are also expected to slow down the boundary layer regeneration process downstream of the wind farm.

Modelling approach

In the current contribution we present some work in progress using the WindModeller front end for ANSYS CFX. Following Montavon [1] atmospheric stability is accounted for via an additional equation for the potential temperature. The effect of buoyancy in the momentum and turbulence equations are explicitly modelled with additional source terms, expressed as a function of the potential temperature. Wakes are modelled via an actuator disk approach, assuming uniform loading across the rotor disks, with the loading calculated from the wind turbine thrust coefficient.

Results

The method is applied to several wind farm sites both onshore and offshore, where data is available. The results indicate that

- For several onshore wind farm sites in relatively complex terrain where data is available from several met masts, taking account of atmospheric stability using a US standard atmosphere can give reduced errors, compared to assuming neutral stability.

- For offshore wind farms, where wake interaction effects are very important, the results are in general agreement with measured data, as demonstrated in Figure 1.

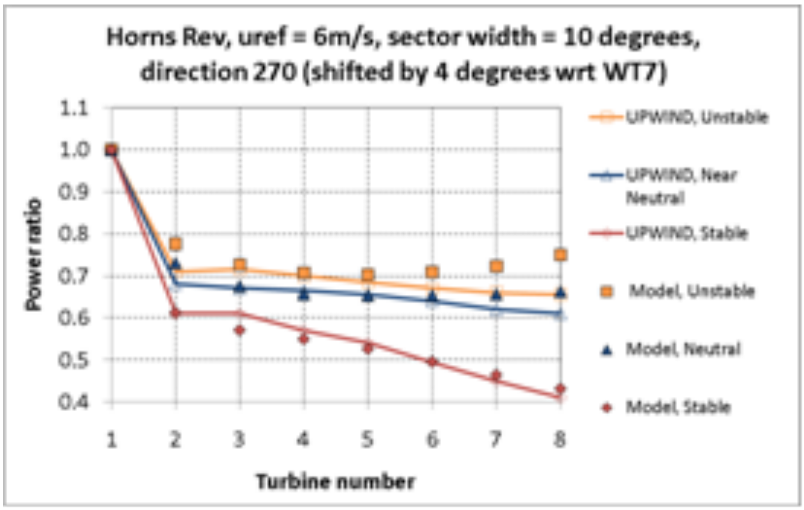

Figure 1 Normalised power down a line of turbine at HornsRev for varying surface stability conditions. Simulations with free stream stability conditions typical of the standard atmosphere, using varying ground heating and cooling conditions.

The poster will also discuss the development of longitudinal roll vortices for cases with temperature inversions and ground heating, and compare results with those from simpler models and observations.

References

1. Montavon, Simulation of atmospheric flows over complex terrain for wind power potential assessment, Ph D thesis EPPF, , http://library.epfl.ch/theses/?nr=1855,1998.

POSTER SESSION 3

CHT12-MP01

COMPUTATIONAL HEAT TRANSFER MODELING OF RICE - WATER SUSPENSION IN TUBE

Kanishka Bhunia and A. K. Datta
Agricultural and Food Engineering Department
Indian Institute of Technology Kharagpur
Paschimbanga, India – 721 302
akd@agfe.iitkgp.ernet.in

In this study of solid-liquid flow, rice was used as the dispersed medium and water was used as the carrier fluid. Experiments were carried out on slurries with solid concentrations of 5%, 10%, and 15% w/w which flowed in a 13 mm ID and 3 m long tube- in- tube heat exchanger. Steam was used as the heating medium. The effects of flow rate, particle surface area and particle concentration were investigated. Calculated convective film to particle heat transfer coefficient (h_{fp}) values ranged from 11 to 32 kW m $^{-2}$ K $^{-1}$ for rice with uncertainty of $\pm$ 2 kW m $^{-2}$ K $^{-1}$. A decrease in heat transfer coefficient values was found as a result of short residence time at the higher flow rates. To investigate the solid-liquid two-phase flow Eulerian $k - \epsilon$ multiphase model was adopted in simple axisymmetric geometry. Velocity profiles of the liquid and solid phases with different particle fractions were estimated from the simulated results. The respective velocities of both phases were higher in the upper part of the tube than in the lower portion because of settling caused by gravity. The slip velocity of the particles was estimated from the simulations and it ranged from 13.83 cm s^{-1} to 19.38 cm s^{-1} for the rice particles. The rice grains always lagged the liquid phase. The particle volume concentration profile was also investigated and it was observed that a high particle concentration formed a core around tube centre line.

Dimensionless correlations were developed to predict liquid-to-particle heat transfer coefficients from combining experimental results and simulations. The combinations were able to predict h_{fp} values within particular range of slip Reynolds number (Re_s) obtained from slip velocities as predicted by the simulations obtained from FLUENT. The exponents of Re_s and the solid volume fraction were negative and it was positive for the Prandtl number. The correlations predicted h_{fp} and Nusselt number values with a maximum error value of 10%. The correlations developed in this work gave better fitting lines with R^2 value of 0.98 and a R_d (relative deviation) value of 1.08 for rice particulate flow. Further relative to experimental values a correlation was able to represent the relationship between dependent and independent parameters at more than 95% level of significance.

CHT12-MP02

KINETIC AND MOLECULAR DYNAMICS SIMULATIONS OF N-DODECANE DROPLET HEATING AND EVAPORATION

Jian-Fei Xie[*], Sergei S Sazhin[*,§], Irina Shishkova[**] and Bing-Yang Cao[***]
[*] Sir Harry Ricardo Laboratories, School of Computing, Engineering and Mathematics,
University of Brighton, Cockcroft Building, Brighton BN2 4GJ, UK
[**] Low Temperature Department, Moscow Power Engineering Institute, Krasnokazarmennaya 14, Moscow 111250, Russia
[***] Department of Engineering Mechanics, Tsinghua University, Beijing 100084, China
[§]Correspondence author. Fax: +44 1273 642677 Email: S.Sazhin@brighton.ac.uk

Results of recent developments in kinetic and molecular dynamics simulations of n-dodecane droplet heating and evaporation are summarised. The effect of inelastic collisions between two molecules on the solution of the Boltzmann equation is taken into account by presenting the change of state of molecules after collisions as a random movement along the surface of an *N*-dimensional sphere, the squared radius of which is equal to the total energy of the molecules before and after the collision in the reference system of the centre of mass. The kinetic energies of two molecules are described by the first six dimensions of the system, and the remaining (*N*-6) dimensions describe the internal energies. This analysis is complemented by molecular dynamics simulations of the evaporation and condensation of liquid n-dodecane ($C_{12}H_{26}$), the closest approximation to Diesel fuel. The interactions within a molecule and between molecules are calculated using an optimised potential for liquid simulation (OPLS). The evaporation/condensation coefficient is estimated and the results are shown to be compatible with the estimates based on the previous molecular dynamics (MD) simulations and the transition state theory. The velocity distribution functions of molecules at the liquid-vapour equilibrium state are found in the liquid phase, the interface, and the vapour phase. The functions in these three phases are found to be close to isotropic Maxwellian for velocity components parallel to the interface. The function in the vapour phase is found to be close to bi-Maxwellian with the temperature for the velocity component normal to the interface being larger than that parallel to the interface. Both kinetic and molecular dynamics models, described above, are recommended to be used for the analysis of heating and evaporation of n-dodecane droplets in conditions relevant to Diesel engines.

CHT12-MP03

SIMULATION OF INTERPHASE HEAT TRANSFER DURING BULK CONDENSATION IN THE FLOW OF VAPOR-GAS MIXTURE

N.M. Kortsenshteyn[*,§], A.K. Yastrebov[**]
[*]Krzhizhanovsky Power Engineering Institute, Moscow, Russia
[**]National Research University "MPEI", Moscow, Russia
[§]Correspondence author. Email: naumkor@yandex.ru

A solution is presented for the problem of growth of a single droplet and the heat transfer between it and a vapor - gas mixture. The proposed algorithm can be used for an arbitrary regime of the droplets growth but in this paper we consider only the free molecular one. The effect of interphase heat transfer on the dynamics of the macroparameters and the droplet size

distribution function was studied for bulk condensation during the flow of the vapor - gas mixture in a nozzle with the use of the results obtained. We compared results obtained for general formulation and for certain simplifying assumptions on the droplets' temperature for different concentrations in a non-condensable gas. The kinetic equation for the droplet size distribution function was used for the mathematical description of the bulk condensation process and our method of direct numerical solution was used in order to solve this equation. The solution of the condensation problem was obtained by taking into account the decrease of the nucleation rate due to the heating of the droplets. As a result, taking into account the finite rate of interphase heat transfer can result in noticeable quantitative changes in the distribution function and the integral characteristics in comparison with the data obtained using the limiting values of droplets' temperature. It was shown that possibility of use of simplifying assumptions depends on gas concentration. The accuracy of the one-temperature model increases with the dilution of vapor by a non-condensing gas. Otherwise, the use of the assumption about the equality of the droplet temperature and the saturation temperature is justified, first of all, for the description of the condensation process in a pure vapor. The decrease of nucleation rate due to heat release on the surface of the droplets has a stronger effect on the kinetics of the bulk condensation at a low partial pressure of the non-condensing gas, i.e. at the low intensity of interphase heat transfer.

CHT12-MP04

EXPONENTIAL EULER TIME INTEGRATOR FOR ISOTHERMAL INCOMPRESSIBLE TWO-PHASE FLOW IN HETEROGENEOUS POROUS MEDIA

Antoine Tambue
Department of Mathematics, University of Bergen, Johannes Bruns. Gate 12
5020 Bergen, Norway
Email: Antoine.Tambue@math.uib.no

Accurate reservoir modelling requires stable and efficient time integrators. For quite a long time, standard time integrators (full implicit Euler, Crank–Nicolson, explicit Euler and implicit-explicit schemes) were mostly used for time integration. These standard time integrators only provide first or second order accuracy in time for simple linear problems, beside explicit Euler scheme suffers for time step constraints. In this paper, we present the exponential Euler time integrator combined with the finite volume (two-point or mult-point flux approximations) space discretization for simulating isothermal incompressible two-phase flow in heterogeneous porous media. This method linearizes the saturation equation at each time step and makes use of a matrix exponential function of the Jacobian, then solves the corresponding stiff linear ODEs exactly in time up to the given tolerance in the computation of a matrix exponential function of the Jacobian from the space discretization. Using a Krylov subspace technique makes this computation efficient. Beside this computation can be done using the free-Jacobian technique. All our numerical examples demonstrate that our method can compete in terms of efficiency and accuracy with the standard time integrators for reservoir simulation in highly anisotropic and heterogeneous porous media.

CHT12-MP05

MODELLING OF FLOW FIELD AND CONVECTIVE HEAT TRANSFER IN AN INTERMITTENT IMPINGING SPRAY

Maksim Pakhomov, Viktor Terekhov§
Lab. of Thermal and Gas Dynamics, Kutateladze Institute of Thermophysics Siberian Branch of Russian Academy of Sciences, Novosibirsk, Russia
E-mail addresses: terekhov@itp.nsc.ru; pakhomov@ngs.ru
§Correspondence author. Fax: +7 383 330 84 80 Email: terekhov@itp.nsc.ru

The flow structure and heat transfer of an intermittent impinging mist jet with low mass concentration of droplets (not more than 1 %) was studied numerically. In the range of small distances between the tube edge and obstacle $H/(2R) \leq 6$ in the pulsed jet heat transfer at stagnation point increases with a rise of pulse frequency, whereas at high distances $H/(2R) > 8$ frequency rise causes heat transfer reduction. Heat transfer intensity during flow pulse action increases and exceeds significantly the corresponding value for the stationary case. At time moment, when there is no flow, the value of Nusselt number decreases considerably. Results obtained were compared with available data of other authors, and satisfactory agreement was obtained for the influence of pulse frequency on heat transfer of the gas jet with impinging surface.

CHT12-MP06

NUMERICAL ANALYSIS AND CONTROL OF TWO-PHASE FLOW INSTABILITIES IN A VERTICAL TUBE DURING EVAPORATION

Ghazali Mébarki§ and Samir Rahal.
LESEI Laboratory, Department of Mechanical engineering,
College of technology, University of Batna, Algeria.
§Correspondence author. Email: g.mebarki@yahoo.fr or ghazali.mebarki@univ-batna.dz

A better understanding of two-phase flows with evaporation allows leading to an optimal design of heat exchangers particularly evaporators. For that purpose, numerical simulations are very useful. In this paper, a numerical study using Fluent has been carried out in order to model and simulate the combination of a two-phase flow with evaporation in a vertical tube. For that purpose, the VOF multiphase flow model and a phase-change model for the mass transfer have been used. For an accurate modelling, the effect of axial conduction has been also taken into account using a conjugate heat transfer model. Our numerical simulation procedure has been validated by comparing our results with those obtained by other authors. Temperature and void fraction fields as well as the heat transfer rate have been calculated for various conditions. Indeed a parametric study has been carried out for various conditions (Reynolds number, imposed lateral heat flux, position along the tube, etc…). Since thermal oscillations are undesirable as they can lead to the failure of the tube, flow instabilities have also been analyzed, using FFT (Fast Fourier transforms), in order to comprehend their behaviour and influence. A control study of the flow instabilities in the tube is also presented. For that purpose tube inlet temperature has been varied using a gain control parameter.

CHT12-MP09

ADVANCEMENT IN TURBULENT SPRAY MODELLING: THE EFFECT OF INTERNAL TEMPERATURE GRADIENT IN DROPLETS

A.Yu. Snegirev[*,§], V.A. Talalov[*], A.S. Tsoi[*], S.S. Sazhin[**], C. Crua[**]
[*]Department of Thermal Physics, Saint-Petersburg State Polytechnic University, Polytechnicheskaya, 29, Saint-Petersburg 195251, Russia
[**] Sir Harry Ricardo Laboratories, School of Computing, Engineering and Mathematics, University of Brighton, Brighton BN2 4GJ, U.K.
[§]Correspondence author. Email: a.snegirev@phmf.spbstu.ru

Two simple and yet sufficiently accurate approaches to predict surface temperature of a vaporizing droplet (higher order polynomial approximation and the heat balance integral method) are proposed. Being computationally inexpensive these approaches are tested as candidates for use in high-resolution LES spray modelling. Robust and efficient numerical algorithm for solving inherently stiff equations of droplet heating and evaporation has been developed. Robustness and computational efficiency of the proposed algorithm is achieved by use of unconditionally stable strongly implicit integration scheme and appropriate adaptation of the time step. The above methodology has been implemented in CFD spray model included in the in-house Fire3D code. The spray model has been applied to replicate three essentially different experimental scenarios in which turbulent sprays of water, acetone, and diesel fuel were investigated. Reasonable agreement has been demonstrated for predicted and measured droplet sizes and velocities as well as for the spray tip penetration dynamics. New numerical algorithms used to calculate surface temperatures of evaporating droplets with non-uniform internal temperature did not incur observable increase of CPU time in turbulent spray simulations.

CHT12-MP10

THREE-DIMENSIONAL SIMULATION OF BUBBLE GROWTH ON MICROSTRUCTURED SURFACES

Woorim Lee[*] and Gihun Son[*,§]
[*]Dept. of Mechanical Engineering, Sogang University, Republic of Korea
[§]Correspondence author. Fax: +82 2 712 0799 Email: gihun@sogang.ac.kr

With an increasing demand for higher cooling capacity, surface modification by micro machined or MEMS fabricated structures has been used as an effective way to enhance nucleate boiling. In this paper, three-dimensional numerical simulations of the bubble motion and boiling heat transfer on a surface with microstructures such as fins are performed to further clarify the boiling process and to find the better conditions for boiling enhancement. The phase interfaces are determined by a sharp-interface level-set method which is modified to include the effect of phase change at the liquid-vapor interface and to treat the no-slip and contact-angle conditions on immersed solid surface of microstructures. The computational results show that finned surface augments boiling heat transfer compared to a plain surface. The effect of fin height on bubble growth and heat transfer are investigated.

CHT12-MP12

NUMERICAL INVESTIGATION OF PARTICLE TEMPERATURE CHANGE IN SUPERSONIC FLOWS

Ryohei Sakamaki[*], Masaya Suzuki[**] and Makoto Yamamoto[**,§]
[*]Graduate School of Tokyo University of Science, Tokyo, Japan
[**]Department of Mechanical Engineering, Tokyo University of Science, Tokyo, Japan
[§]Correspondence author. Fax: +81 3 5213 0977 Email: yamamoto@rs.kagu.tus.ac.jp

Gas-particle two-phase flow is very important and also a key issue, in order to design various machines. A great number of investigations have been carried out by means of theoretical, experimental, and numerical works. However, particle motion in a supersonic flow has not been clarified sufficiently. Hence, in order to clarify the interactions between flow and particles, the authors cast a spotlight on the characteristics of particle motion, especially the velocity and temperature. In the present study, a conventional converging-diverging supersonic nozzle is employed as our target. For the gas phase, the turbulent flow in the nozzle is computed with the finite difference and RANS methods. For the particle phase, the particle motion is simulated in a Lagrangian manner. In addition, taking into account the light particle loading, a weak coupling method is employed. Through this investigation, we show that the particle velocity monotonically increase from the nozzle throat to the nozzle outlet, the working gas of helium can accelerate the particle more than that of nitrogen, and the smaller particle tend to obtain larger speed and lower static temperature.

CHT12-MP13

MODELING OF PHASE TRANSITION OF PARTIALLY MISCIBLE SOLVENT SYSTEMS: HYDRODYNAMICS AND HEAT TRANSFER PHENOMENA

Vered Segal, Amos Ullmann and Neima Brauner[§]
School of Mechanical Engineering, Tel Aviv University, Israel.
[§]Correspondence author. Fax: +972 3 640 7617 Email: brauner@eng.tau.ac.il

A numerical model for critical quench of binary mixtures in a 2D geometry is developed, whereby two opposite walls are cooled below the critical temperature. The model equations for the conservation of mass, momentum and energy are derived according to the diffuse interface approach. The energy equation has been re-formulated to identify the heat source term which is associated with liquid-liquid phase separation. The numerical tool is used for simulating the separation process and to obtain the velocity, concentration and temperature fields. The 2D simulation enables the analysis of the evolving velocity field induced by the non-equilibrium Korteweg force. The numerical model developed can be further used for the analysis of the convective heat transfer phenomena. This convective motion is believed to be responsible for the heat transfer rate enhancement observed in the experiments during non-isothermal phase separation.

HEAT TRANSFER IN SHEAR-DRIVEN THIN LIQUID FILM FLOWS

J. R. Marati*§, M. Budakli*, T. Gambaryan-Roisman*, **, P. Stephan*, **
* Technische Thermodynamik, Technische Universität Darmstadt,
** Center of Smart Interfaces, Technische Universität Darmstadt,
Petersenstrasse 32, D-64287 Darmstadt, Germany
§Correspondence author. Email: jmarati@ttd.tu-darmstadt.de

The objective of the study is to investigate hydrodynamics and heat transfer in a shear-driven liquid film flow. This process is relevant to fuel flow inside lean pre-mixed pre-vaporization (LPP) chambers. A combined numerical and experimental study has been performed to determine the heat transfer in gas-driven thin liquid films on the outer surface of vertical heated tubes. Numerical simulations have been performed using the volume of fluid (VOF) method implemented in an open source computational fluid dynamics (CFD) code OpenFOAM for turbulent air/water flow conditions. The code has been extended for simulation of two-phase flows with heat transfer. The Reynolds averaged Navier–Stokes equations (RANS) with the k - ε turbulence model for gas–liquid two-phase flows have been solved using the finite volume method. The results on wall temperature distribution and average film thickness have been compared with experimental data. A good agreement between the simulations and experiment has been found. The results indicate that the heat transfer is enhanced with increasing gas Reynolds number due to the film thinning and intensification of convection.

POSTER SESSION 4

CHT12-MX01

RADIATION EFFECTS ON A PARTICIPATING ELECTRICALLY CONDUCTIVE FLUID IN A SQUARE CAVITY

Zhang Jing-Kui[*,**] , Li Ben-Wen[*,§] and Hu Zhang-Mao[*]
[*] Key Laboratory of Electromagnetic Processing of Materials (Ministry of Education), Northeastern University, Shenyang 110004, China
[**] Key Laboratory for Ferrous Metallurgy and Resources Utilization of Ministry of Education, Wuhan University of Science and Technology, Wuhan 430081, China
[§]Correspondence author. Fax: +86 24 8368 1756 Email: heatli@hotmail.com

A numerical investigation is presented for fluid flow and heat transfer of an electrically conductive liquid in a square cavity. In this work, the right wall of the cavity is kept at a high constant temperature, and the others are at low uniform temperature. The cool walls and the fluid are heated by both convection and radiation. The motion of the fluid is described by continuity equation and momentum equation which are solved by artificial compressibility algorithm based on finite volume method (FVM). The heat transfer of the fluid along with the walls is described by energy equation and radiative transfer equation (RTE) which are solved by FVM and discrete ordinates method (DOM) respectively. New dimensionless number A and B are obtained by dimensional analysis of governing equations. Investigations are performed with different A number and Ha number. The results show that radiation has remarkable effects on fluid flow and heat transfer under different values of parameters.

CHT12-MX02

NUMERICAL AND ANALYTICAL INVESTIGATION ON THE FLOW REVERSAL DUE TO MIXED CONVECTION IN VERTICAL CONCENTRIC ANNULI: WHY IT OCCURS IN BUOYANCY-AIDED FLOWS

Esmail M. A. Mokheimer
Mechanical Engineering Department
King Fahd University of Petroleum and Minerals, Dhahran 31261, Saudi Arabia
esmailm@kfupm.edu.sa

The main purpose of this article is to shed more light on the cause of flow reversal in vertical concentric annular channels for buoyancy-aided flows under isothermal boundary conditions. The concept used to quantify the critical values of the modified buoyancy parameter $(Gr/Re)_{critical}$ at which the adverse pressure gradient onset is outlined and applied to analytically estimate these critical values. The conditions for flow reversal are also analytically obtained for buoyancy aided and buoyancy-opposed flows. The analytical solutions showed that pressure buildup takes place only for buoyancy-aided flows while flow reversal occurs for both buoyancy-aided and buoyancy-opposed flows. The analytical solutions revealed also that values of the buoyancy-parameter required to initiate the pressure buildup for buoyancy-aided flows in the vertical annuli are smaller than those required to initiate the flow reversal. Thus, for buoyancy aided flows, pressure buildup would precede

the flow reversal. Numerical scheme is developed and used to simulate the development of pressure and pressure gradient in the entrance region of the vertical concentric annuli. The numerical scheme is validated by the analytical solution as well as the previously published pertinent results. Numerical results that show the pressure buildup in the developing entry region of the annuli for values of the modified buoyancy parameter (Gr/Re) greater than its critical values $(Gr/Re)_{critical}$ along with other flow and heat transfer parameters of importance are also presented and discussed. The locations of pressure build up as well as those of flow reversal have been obtained and reported. The numerical solutions show clearly that pressure buildup for buoyancy-aided flows takes place prior to the flow reversal onset.

CHT12-MX04

NUMERICAL STUDY OF COUPLED MOLECULAR GAS RADIATION AND NATURAL CONVECTION IN A DIFFERENTIALLY HEATED CUBICAL CAVITY

Laurent Soucasse[*], Philippe Rivière[*], Shihe Xin[**], Patrick Le Quéré[***] and Anouar Soufiani[*,§]
[*]CNRS, UPR 288, Laboratoire EM2C, Châtenay-Malabry, France
[*]Ecole Centrale Paris, Châtenay-Malabry, France
[**]CNRS/INSA de Lyon, UMR5008, CETHIL, Villeurbanne, France
[***]CNRS, UPR 3251, LIMSI, Orsay, France
§Correspondence author. Fax: +33 1 4702 8035. Email: Anouar.Soufiani@em2c.ecp.fr

The coupling between natural convection and gas and wall radiation is studied numerically in a differentially heated cubical cavity filled with an air/CO_2/H_2O mixture. In order to solve coupled flow, heat transfer and radiation equations, we develop a 3D radiative transfer model based on the deterministic ray tracing method, coupled with a pseudo-spectral Chebyshev method for natural convection under Boussinesq approximation. Absorption Distribution Function (ADF) model is used to describe gas radiative properties. Coupled simulations are performed at $Ra=10^5$, 10^6 and 3×10^7, considering wall and/or gas radiation. Steady solutions were obtained except at the highest Rayleigh number in the case of radiating walls. Results show a strong influence of radiative transfer on temperature and velocity fields. The global homogenization of the temperature field induced by radiation leads to a decrease of the thermal stratification parameter. Two different mechanisms leading to this behaviour, involving either wall/wall or gas radiative exchanges, are identified. In addition, we observe a thickening of the vertical boundary layers and an increase of the global circulation in the cavity. The influence of the Rayleigh number and 3D effects are also discussed.

CHT12-MX06

NUMERICAL STUDY OF UNSTEADY AIRFLOW PHENOMENA IN A VENTILATED ROOM

Kana Horikiri[1], Yufeng Yao[1§] and Jun Yao[2]
[1] Faculty of Science, Engineering and Computing, Kingston University, London SW15 3DW, UK
[2] School of Engineering, University of Lincoln, Brayford Pool, Lincoln LN6 7TS, UK
[§]Correspondence author. Tel: +44 (0)208 417 4822 Email: Y.Yao@kingston.ac.uk

Numerical simulation of airflow in an indoor environment has been carried out for forced, natural and mixed convection modes respectively, by using computational fluid dynamics (CFD) approach of solving the Reynolds-averaged Navier-Stokes equations. Three empty model rooms in two-dimensional configuration were studied first; focusing on the effects of grid refinement, mesh topology, and turbulence model. It was found that structured mesh results were in better agreement with available experimental measurements for all three convection scenarios, while the re-normalized group (RNG) k-ε turbulence model produced better results for both forced and mixed convections and the shear stress transport (SST) turbulence model for the natural convection prediction. Further studies of air velocity and temperature distributions in a three-dimensional cubic model room with and without an obstacle have shown reasonably good agreements with available test data at the measuring points. Interestingly, CFD results exhibited some unsteady flow phenomena that have not yet been observed and reported in previous experimental studies for the same problem. After analyzing the time history of velocity and temperature data using fast Fourier transformation (FFT), it was found that both air velocity and temperature field oscillated at low frequencies up to 0.4Hz and the most significant velocity oscillations were occurred at a vertical height of an ankle level (0.1m) from the floor, where temperature oscillation was insignificant. The reasons for this flow unsteadiness were possibly due to a higher Grashof number, estimated 0.5×10^6 based inflow conditions, and thus strong buoyancy driven effects caused the oscillations in the flow field. The appearance of an obstacle in the room induced flow separation at its sharp edges and this would further enhance the oscillations due to the unsteady nature of detached shear-layer flow.

CHT12-MX07

INTERACTION EFFECTS BETWEEN SURFACE RADIATION AND DOUBLE-DIFFUSIVE TURBULENT NATURAL CONVECTION IN AN ENCLOSED CAVITY FILLED WITH SOLID OBSTACLES

Draco Aluya Iyi, Reaz Hasan and Roger Penlington
School of Computing, Engineering & Information Sciences, Northumbria University, Newcastle upon Tyne, NE1 8ST, United Kingdom
Email: Draco.iyi@unn.ac.uk

The work reported here is a 2D numerical study on the buoyancy-driven low speed flow of humid air inside a rectangular cavity partially filled with solid cylindrical objects and whose vertical walls are maintained at 1.2 and 21 ^{o}C. This is a case of double diffusion where both temperature and concentration gradients are significant. Detailed calculations were carried out and results compared with reliable data, with the aim of investigating the influence of surface

emissivity on heat and moisture transport. The Rayleigh number of the fluid mixture (air and water vapour) based on the height of the vertical wall is found to be 1.45 x 10^9.

In the computations, turbulent fluxes of the momentum, heat and mass were modelled by low-*Re* (Launder-Sharma) *k-ε* eddy viscosity model. The effect of radiation has been found to be significant even for the moderate temperature difference of 19.8 ^{o}C between the hot and the cold walls with the humid air participating in the radiation heat transfer. Variations of average Nusselt number and buoyancy flux are analysed and profiles of turbulent quantities are studied in order to observe the net effect of the intensity of turbulence. It has been found that a change in surface emissivity influences the humidity distribution and heat transfer within the cavity. It was also observed that during natural convection process the air/water vapour combination results in an increase in the heat transfer as compared to pure natural convection. An increase in heat transfer is observed using thermo-physical materials of higher surface emissivity. It can thus be implied that with the appropriate choice of components, the fluid flow, heat and mass transfer due to natural convection can be increased passively.

CHT12-MX10

MARANGONI EFFECT ON THE LONGITUDINAL ROLLS OF A MIXED CONVECTION FLOW IN A HORIZONTAL OPEN CHANNEL

L. Bammou[1,2], K. El Omari[1§], Y. Le Guer[1], S. Blancher[1], B. Benhamou[3]
[1]Laboratoire des sciences de l'ingénieur Appliquées à la Mécanique
et au génie Electrique (SIAME),
Université de Pau et des Pays de l'Adour (UPPA), IFR, rue Jules Ferry,
BP 7511, 64075 Pau Cedex, France.
[2]Laboratoire de Mécanique, Procédés de l'Energie et de l'Environnement(LMP2E),
ENSA, B.P. 1136, 80000 Agadir, Morocco.
[3]Laboratoire de Mécanique des Fluides et d'Énergétique (CNRST-URAC27),
Université Cadi Ayyad, Dépt. de Physique, Faculté des Sciences Semlalia,
40001 Marrakech, Morocco.
[§]Correspondence author. Email: kamal.elomari@univ-pau.fr

This paper concerns a numerical study of three-dimensional laminar mixed convection within a liquid flowing in a horizontal channel heated uniformly from below. The upper surface is free and supposed flat. The coupled Navier-Stokes and energy equations are solved numerically by the finite volume method taking into account the thermocapillarity effect (Marangoni effect). When the forces induced by the currents of natural and forced convection are of the same order of magnitude (for Richardson number $Ri=Ra/(Pr.Re^2)=O(1)$), the results show the development of instabilities in the form of steady longitudinal convective rolls similar to those encountered in the Poiseuille-Rayleigh-Bénard flow. The number and spatial distribution of these rolls along the channel depend on the flow conditions. The objective of this work is to study the influence of parameters such as the Biot number and the aspect ratio of the channel on the flow patterns and heat transfer characteristics. The effects of the variation of the surface tension with temperature gradients (Marangoni effect) are also considered.

CHT12-MX11

NUMERICAL SIMULATION OF COUPLED HEAT TRANSFER THROUGH BUILDING FACADES IN THE ARID ZONE

A.Missoum[*,§], B.Draoui[*],A.Slimani[*], M.Elmir[*],
R.Mehdaoui[*],R.Khelfaoui[*], M.Bouanini[*], R.Belarbi[**]
[*] Laboratory of Energetique in Arid Zone ENERGARID, Bechar University Algeria
[**] Laboratory LEPTIAB La Rochelle University, French
[§]Corresponding author. Fax: +0021349815244 Email: missoum101@yahoo.fr

This paper presents a numerical study of two-dimensional heat transfer through the walls in the construction of building envelopes in the arid zone. We study the influence of external stress (temperature) on the flow of air inside the building through a solid interface (facade). The system considered consists of two milieus, an internal milieu that represents the local and external that represents the environment of the building separated by a wall. Equations governing the natural convection in the two milieus and the heat conduction in the wall interface are solved using by finite difference technique based on the control volume approach and the SIMPLER algorithm. Results are presented for wide ranges of dimensionless parameters governing the problem. These ranges correspond to the practical value of the excitation temperature, the wall thickness and aspect ratio of the external environment.

CHT12-MX12

A NEW PROCESS FOR SPECIES SEPARATION IN A BINARY MIXTURE USING MIXED CONVECTION

Khouzam Ali[*,§] , Marie Catherine Charrier Mojtabi[**], Abdelkader Mojtabi[*] and Ouattara Bafétigué[*]
* IMFT, UMR CNRS/INP/UPS N°5502, UFR MIG, Université Paul Sabatier,
118 route de Narbonne, 31062, Toulouse cedex, France
** PHASE, EA 3028, UFR PCA, Université Paul Sabatier, 118 route de Narbonne,
31062, Toulouse cedex, France
[§]Correspondence author. Fax: +33 534 322899, Email: akhouzam@imft.fr

In this paper, a numerical and analytical analysis is performed in order to improve the species separation process in a binary fluid mixture by decoupling the thermal gradient from the convective velocity. The configuration considered is a horizontal rectangular cavity, of large aspect ratio, filled with a binary fluid. A constant tangential velocity is applied to the upper horizontal wall. The two horizontal impermeable walls are maintained at different and uniform temperatures T_1 and T_2 with $\Delta T = T_1 - T_2$. Species separation is governed by two control parameters, the temperature difference ΔT and the velocity of the upper plate $U\vec{e}_x$. The intensity of the thermodiffusion is controlled by the temperature, while the velocity $U\vec{e}_x$ controls the convective flow. This problem depends on six dimensionless parameters, namely, the separation ratio, ψ, the Lewis number, Le, the Prandlt number Pr, the aspect ratio of the cell, A and two control parameters: the thermal Rayleigh number, Ra and the Péclet number Pe. In this study, the formulation of the separation (mass fraction difference

between the two ends of the cell) as a function of the Péclet number and the Rayleigh number is obtained analytically. For a cell heated from below, the optimal separation $m=\sqrt{42}/15$ is obtained for $Pe=\sqrt{42}/Le$ and $Ra=540/Le\,\psi$. Two dimensional numerical results, obtained by solving the full governing equations, are in good agreement with the analytical results based on a parallel flow approach.

CHT12-MX14

HYBRID EXPERIMENTAL-NUMERICAL APPROACH TO SOLVE INVERSE CONVECTION PROBLEMS

Joseph VanderVeer and Yogesh Jaluria§
Dept. of Mech. and Aero. Eng., Rutgers University Piscataway, NJ 08854, USA
§Corresponding author. Fax: +1 732 445 3124 Email: jaluria@jove.rutgers.edu

A methodology is developed to utilize both experimental and numerical information in solving inverse convection problems. The method used combines an empirical relationship with a regularization scheme. The method is applied to a plume generated by an electrically heated copper block set within a small wind tunnel to provide cross flow. This approach attempts to solve for, within acceptable error, the source location and source temperature, which are not known a priori. A key factor in practicality of the approach is limited experimental sampling. Results show typical methodology errors of less than 1% for source temperature and 5% for source location. Results of combined experimental, experimental-numerical, and methodology errors were found to be typically less than 3% for source temperature and 6% for source location. The paper presents the basic methodology, typical results obtained, and the accuracy of the predictions. Practical problems, where this approach may be useful, are outlined.

CHT12-MM01

COMPLEX HEAT TRANSFER AT DIRECTED CRYSTALLIZATION OF SEMITRANSPARENT MATERIALS

Iurii Lokhmanets§, Valeriy Deshko and Anton Karvatskii
Heat Engineering Dept., Kiev Polytechnic Institute, Kiev, Ukraine
§Corresponding author. Email: lokhmanets@gmail.com

The present paper deals with use of numerical simulation of complex heat transfer at semitransparent crystal growth. The sensibility of thermal regimes at crystal-melt system to a number of inner of outer process parameters was explored. This allows justification of several possible simplifying approaches at development of numerical models of crystal growth furnaces, including on-line models for operative control of growth process.

For this research a numerical model of radiation-convective and radiation-conductive heat transfer was developed with commercially available CFD software. Several advanced features of the model, such as dynamic evolution of the semitransparent crystallization front, were realized by implementation of user-defined functions. The 2D axisymmetric model is limited geometrically to cylindrical crystal-melt system since heat regimes and temperature gradients in the area near crystallization front are the most important.

We examine the combined effect of radiation, convective and conductive heat transfer mechanisms on the formation of temperature fields and heat flows, position and shape of the crystallization front, and distribution of temperature gradients in the crystal-melt system. Numerical simulations are carried out for the oxide and alkali-halide classes of semitransparent materials at different growth conditions, considering selectivity of their absorptivity. Analysis of the results allowed developing the recommendations for approximation the effects of radiation and convection heat transfer and their interaction.

CHT12-MM02

SHELL FORMATION DUE TO HEAT AND MASS TRANSFER

J. I. Ramos[§] and Francisco J. Blanco-Rodríguez
Escuela de Ingenierías, Universidad de Málaga.
Doctor Ortiz Ramos, s/n; 29071 Málaga; Spain
[§]Correspondence author. Fax: +34 951952522 Email: jirs@lcc.uma.es

A two-dimensional model of dry-spinning that employs a Newtonian rheology which depends on the local temperature and solvent concentration and accounts for the axial and radial distributions of temperature and polymer concentration, is presented. The model employs the leading-order equations for the fiber's geometry and axial and radial velocity components derived from an asymptotic analysis of slender fibers at low Reynolds, Nusselt and Sherwood numbers, and two-dimensional equations for the temperature and solvent concentration fields. The diffusion of the solvent is assumed to depend on both the temperature and polymer concentration. The two-dimensional model presented here is an integro-differential boundary-value problem and is solved iteratively using as initial guess the solution corresponding to the leading-order equations of a non-isothermal fiber whose dynamic viscosity is constant. It is shown that a shell is formed at the fiber's outer interface on account of the large increase in viscosity due to cooling and solvent evaporation. It is also shown that the fiber's cooling may hinder the solvent diffusion but favor the polymer orientation.

CHT12-MM03

COMPUTATIONAL PREDICTION OF RADIATIVE PROPERTIES OF POLYMER CLOSED-CELL FOAMS WITH RANDOM STRUCTURE

, Rémi Coquard[*,§], Jaona Randrianalisoa[**] and Dominique Baillis[***]
[*] EC2-MODELISATION, 66 Boulevard Niels Bohr, F69603 Villeurbanne, France
[**] GRESPI, Université Reims, EA 4301, Moulin de la Housse, F51687 Reims, France
[***]LaMCoS, INSA-Lyon, CNRS UMR 5259, 18-20 Rue des Sciences,
F69621 Villeurbanne, France
[§]Correspondence author. Fax: +33 4 37 48 84 05 Email: remi.coquard@ec2-modelisation.fr

Polymer foams with closed-cells are widely used as insulating material in many applications from building to space launchers. In these applications, the heat transfer within the foam is governed mainly by conduction and radiation. To enhance the insulating system capability, the knowledge of the thermal properties of these materials is indispensable.

From the thermal radiation point of view, polymer foams behave as semitransparent materials. Current knowledge shows that the radiative properties of the foam depend on the optical and surface properties of the solid phase, on the cell size and shape, and on the wall thickness (or relative density). Among these parameters, the influences of the cell morphology are not well understood. This contribution aims to investigate numerically the influences of the cell architecture on their radiative properties.

At first, three-dimensional samples are modeled through the Voronoï tessellation method. Different foam morphologies are considered ranging from the periodic assembly of the famous Kelvin'cells to assemblies of totally random cells.

Secondly, the equivalent radiative properties are determined from a Ray-Tracing (RT) method performed inside the REV [Randrianalisoa 2010, Coquard 2011]. This method consists in tracking the path of a large number of energetic rays (or photon bundles) from their possible emission location to their extinction location. The absorption and scattering coefficients are determined from the history of extinction paths while the scattering phase function is determined from the history of scattering direction distribution.

The radiative properties of the samples are compared with analytical models of the literature. The evolution of these properties with the cell randomness is analysed and discussed. An analysis of the anisotropy of the radiative behaviour is also conducted. For validation purpose, the calculated hemispherical transmittances and reflectances of plan parallel foam slices are compared with the corresponding measurements conducted on XPS foam slices. The agreement between experimental and computed transmittances and reflectance is quite satisfactory, demonstrating the suitability of the numerical approach.

CHT12-MM04

THERMAL CONDUCTIVITY OF OPEN- AND CLOSED-CELL FOAMS: INFLUENCES OF CELL RANDOMNESS

Jaona Randrianalisoa[*,§], Rémi Coquard[**] and Dominique Baillis[***]
[*]GRESPI, Université de Reims, EA 4301, Moulin de la Housse, F-51687 Reims, France
[**]EC2-MODELISATION, 66 Boulevard Niels Bohr, F-69603 Villeurbanne, France
[***]LaMCoS, INSA-Lyon, UMR CNRS 5259, 18-20 Rue des Sciences, F-69621 Villeurbanne, France
[§]Correspondence author. Fax: +33 3 26 91 32 51 Email: jaona.randrianalisoa@univ-reims.fr

The effects of cell randomness and sample size on thermal conductivity of cellular foams were investigated through finite-element method (FEM) applied to numerical samples generated by the perturbed Voronoï diagram. The 3D aspect of foam materials, the nature of the cells (open or closed), the distribution of cell size, and the randomness of the cell shape and location were taken into account. For model validation, a comparative study with experimental and existing numerical results of open-cell (metallic and ceramic) and closed-cell (polymer) foams was carried out. The limited size of samples leads to an under or overprediction of the thermal conductivities of cellular foams depending on the cell nature. The size effects are notably due to the non-uniformity of small and large volume porosity (or relative density) having identical cell wall thickness or strut cross-section area. These effects become negligible when samples, the representative elementary volume (REV), contain tens of cells along each volume side. The cell randomness (in term of shape and location), captured here through the cell size standard deviation of Gaussian-normal distribution, results in an increasing of the material turtuosity and consequently a decreasing of the foam thermal conductivity. The effects are more important for open-cell structures than for closed-cell structures. The proposed approach predicts perfectly the numerical results of volume-finite method applied to tomographied foams. Its suitability for modeling thermal conductivity of both open- and closed cell foams is confirmed by the agreement between calculations and measurements in wide temperature and porosity ranges.

POSTER SESSION 5

CHT12-NC01

COOLING ELECTRONIC COMPONENTS MOUNTED ON A VERTICAL WALL BY NATURAL CONVECTION

Karim Lahmer[§,*], Rachid Bessaih[**]
[*,**]Laboratory of Applied Energy and Pollution, Dept. of Mechanical Engineering, University of Mentouri - Constantine
25000, ALGERIA
[§]Correspondence author: Fax: +21331912807 - Email: lahmer.karim@sonelgaz.dz

In this paper we determine the effects of some parameters like Grashof number (Gr), the distance (S) between the heat sources and the outlet distance (Le). The purpose is an optimal management of the heat flux by laminar free convection in a vertical open 2D domain, containing two heated electronic components. The results show that a regular and uniform heat sources distribution in the inlet is very important in order to take advantage from the starting boundary layer. The impact of parameters on the heat dissipation, characterized by the Nusselt number, has a different importance. For a Prandtl number Pr = 0.71, the (Gr) increases the heat exchange that is reflected by a growing Nusselt number, and also participates to the formation of recirculation zones. The total Nusselt number *Nu* is increased when (Gr) is multiplied by 10^2. With regards to the distance (S) between the heat sources, the results show that *Nu* increases about 7% if the distance (S) doubles, and becomes approximately 17.8% when (S) quadruples. At the end, the heat transfer increases when increasing the distance (L_e) of the channel exit length, mainly on the second component. The Total Nusselt number increases by 1.6% when (L_e) increases by about 67%.

CHT12-NC02

NUMERICAL STUDY OF HEAT TRANSFER BY NATURAL CONVECTION ALONG A WAVY VERTICAL PLATE WITH VARIABLE WALL TEMPERATURE

Mayouf Si Abdallah [*,§], Belkacem Zeghmati
[*]Physics Department, Faculty of Sciences, University of M'Sila, Algeria
[**]Laboratoire de Mathématiques et Physique,
Université de Perpignan Via Domitia, France
[§]Correspondence author. Fax: +213 35554479 Email: s_maayouf@yahoo.fr

The effects of wavy geometry on natural convection heat transfer and boundary layer flow along a vertical plate with variable wall temperature are presented. The homotopic transformation is employed to transform the physical domain into a flat plate. The boundary layer equations and the boundary conditions are discretized by the finite difference scheme and solved numerically using the Gauss-Seidel algorithm with relaxation coefficient. Effects of the wavy geometry and the variable wall temperature on the velocity, temperature profiles, local Nusselt number and the developments of the skin-friction coefficient are presented and discussed in detail.

Results show that increasing of the variable wall temperature leads to increase the heat transfer rate, while, in general, high amplitude reduce the heat transfer.

CHT12-NC03

NUMERICAL INVESTIGATION OF FREE CONVECTION AND HEAT TRANSFER BETWEEN VERTICAL PARALLEL PLATES WITH DIFFERENT TEMPERATURES

Viktor I. Terekhov[§] and Ali L. Ekaid
Lab. of Thermal and Gas Dynamics, Kutateladze Institute of Thermophysics Siberian Branch of Russian Academy of Sciences, 630090, 1, Acad. Lavrent'ev Avenue, Novosibirsk, Russia
§Correspondence author. Fax: +7 383 3308480, Email: terekhov@itp.nsc.ru

Results are presented of a numerical investigation of the flow and heat transfer in laminar free convection between vertical parallel isothermal plates with different temperatures above and below the ambient. The Prandtl number was constant Pr = 0.71, and the Rayleigh number changed within (Ra = 10^3 – 10^5). The ratio of geometrical sizes of plates and distances between them was constant $A = L/w$ =10. The Navier-Stokes and energy equations could be solved completely by the method of finite volumes on staggered meshes. The effect of temperature ratio on the flow structure at the channel inlet and outlet was analyzed. Data on velocity and temperature distributions between the plates and local and integral heat transfer are presented; this allows better understanding for the mechanism of transfer processes between the parallel plates with asymmetric heating.

CHT12-NC05

BOUNDARY CONDITION EFFECTS ON NATURAL CONVECTION OF BINGHAM FLUIDS IN A SQUARE ENCLOSURE WITH DIFFERENTIALLY HEATED HORIZONTAL WALLS

O. Turan[*], R. J. Poole[**] and N. Chakraborty[***,§]
[*]Dept. of Mechanical Engineering, Karadeniz Technical University, Turkey
[**] School of Engineering, University of Liverpool, UK.
[***]School of Mechanical and Systems Engineering, Newcastle University, UK
[§]Correspondence author. Fax: +44 191 222 8600 Email: nilanjan.chakraborty@newcastle.ac.uk

Natural convection of Bingham fluids in square enclosures with differentially heated horizontal walls has been numerically analysed for both constant wall temperature (CWT) and constant wall heat flux (CWHF) boundary conditions for different values of Bingham number Bn (i.e. non-dimensional yield stress) for nominal Rayleigh and Prandtl numbers ranging from 10^3-10^5 and 0.1-100 respectively. A semi-implicit pressure-based algorithm is used to solve the steady-state governing equations in the context of the finite-volume methodology in two-dimensions. It has been found that the mean Nusselt number $\overline{Nu}$ increases with increasing Rayleigh number but $\overline{Nu}$ is found to be smaller in Bingham fluids than in Newtonian fluids (for the same nominal values of Rayleigh and Prandtl numbers) due to augmented flow resistance in Bingham fluids. Moreover, $\overline{Nu}$ monotonically decreases with increasing Bingham number irrespective of the boundary condition. Bingham fluids exhibit non-

monotonic Prandtl number Pr dependence on $\overline{Nu}$ and a detailed physical explanation has been provided for this behaviour. Although variation of $\overline{Nu}$ in response to changes in Rayleigh, Prandtl and Bingham numbers remain qualitatively similar for both CWT and CWHF boundary conditions, $\overline{Nu}$ for the CWHF boundary condition for high values of Rayleigh number is found to be smaller than the value obtained for the corresponding CWT configuration for a given set of values of Prandtl and Bingham numbers. The physical reasons for the weaker convective effects in the CWHF boundary condition than in the CWT boundary condition especially for high values of Rayleigh number have been explained through a detailed scaling analysis. The scaling relations are used to propose correlations for $\overline{Nu}$ for both CWT and CWHF boundary conditions and the correlations are shown to capture $\overline{Nu}$ satisfactorily for the range of Rayleigh, Prandtl and Bingham numbers considered in this analysis.

CHT12-NC06

NATURAL CONVECTION AND SURFACE RADIATION BETWEEN A HORIZONTAL HEAT GENERATING SOLID CYLINDER AND A THICK OUTER CYLINDRICAL SHELL

Vinay Senve, G. S. V. L. Narasimham*
Department of Mechanical Engineering
Indian Institute of Science, Bangalore 560012, India
*Corresponding author, Telephone: +91-80-22932971, Fax: +91-80-23600648
Email: mecgsvln@mecheng.iisc.ernet.in

A computational study of two-dimensional conjugate natural convection and its interaction with surface radiation in the horizontal annulus between a heat generating solid cylinder and a thick outer cylindrical shell is performed. The natural convection in the annulus is driven by the uniform volumetric heat generation inside the solid cylinder. Numerical solutions of the Boussinesq equations and the solid energy equation in primitive variables are obtained on a staggered mesh with a pressure correction method, with a custom code. The surface radiation is coupled to the conduction and convection at the solid-fluid interfaces. Steady state is obtained as long time solution of time-dependent formulation. Results for various quantities of interest, namely, the flow and temperature distributions, local and average Nusselt numbers, dimensionless local and average interface temperatures, are presented. The Grashof number based on the volumetric heat generation and gap width is varied from 10^5 to 5×10^9. The solid-to-fluid thermal conductivity ratio for the inner cylinder ranges from 5 to 20. The metallic outer wall has a conductivity ratio 1358 corresponding to lead. The dimensionless thickness (with respect to gap width) of the outer shell is in the range 0.0825-1, while the inner cylinder dimensionless radius is 0.2. Air is the working medium for which the Prandtl number is 0.71. As the Grashof number increases, the strength of the natural convection and the tendency for temperature stratification increases. Radiation, while weakening the natural convection, dissipates a considerable fraction of the heat generated inside the system.

CHT12-NC07

INSIGHT INTO RAYLEIGH NUMBER EFFECTS ON TURBULENCE CHARACTERISTICS OF A THERMALLY DRIVEN FLOW ADJACENT TO UPWARD-FACING HORIZONTAL HEATED ROUND PLATE BY LARGE-EDDY SIMULATION

Yasuo Hattori[*,§], Hitoshi Suto[*], Shuji Ishihara[**] and Yuzuru Eguchi[*]
[*]Central Research Institute of Electric Power Industry, Chiba, Japan
[**]Denryoku Computing Center, Ltd., Tokyo, Japan
[§]Correspondence author. Fax: +81 4 784 7142 Email: yhattori@criepi.denken.or.jp

The Rayleigh number effects on the thermally-driven flow adjacent to an upward-facing isothermally-heated round plate, which is horizontally placed at a floor in air, are investigated by using a well-resolved large-eddy simulation. The special attention is paid to the turbulence characteristics, including developing process, in the vicinity of the heated plate. The simulation is performed with an open-source code, OpenFOAM; the Cartesian coordinate grid system with a large computational domain are employed to avoid the decrease in a grid resolution near the edge of heated plate and to reduce the disturbance of ambient fluid. The Rayleigh number based on the radius of the heated plate is set to 2.1×10^{7} or 4.2×10^{7}; both yield transition of the boundary layer from the laminar to turbulence. The visualized structures and statistics of flow and thermal fields reveal that the developing process with the transition includes vertical wavy motions. The vertical wavy motions might be related to the vortex rings periodically generated by the discontinuity of thermal conditions at the edge of heated plate. The increase in Rayleigh number results in weakening of the coherency of the vortex ring, but this impact is limited to the near the fringe of the heated plate. Thus, the turbulence characteristics of fully developing region is maintained regardless of Rayleigh number: of both the simulations performed, the cell-like structures are observed and the vertical profiles of turbulence intensity of temperature and vertical velocity fluctuations are expressed by the similarity of λ-layer.

CHT12-NC08

TRANSIENT NATURAL CONVECTION IN ENCLOSURES OF CIRCULAR CROSS SECTION FILLED WITH HUMID AIR, INCLUDING WALL PHASE CHANGE

Vítor A. F. Costa
Departamento de Engenharia Mecânica, Universidade de Aveiro, 3810-193 Aveiro, PORTUGAL, Fax: +351 234370953 Email: v.costa@ua.pt

Containers of circular cross section filled with humid air and walls subjected to varying temperatures are commonly used in practice. Circular ducts containing humid air can be also considered as containers when out of service. Gradients of density, induced by gradients of temperature and water vapor concentration, lead to double-diffusive natural convection in the container. Under such conditions, condensed water could exist at the wall, evaporation of condensed water could occur for increasing wall temperatures, or additional condensation could occur for decreasing wall temperatures. Due to heating (and possible evaporation) or cooling (and possible condensation), significant changes could occur on the pressure level and also on density, and the Boussinesq approximation cannot be used. Mass, momentum and energy conservation equations, including the water mass conservation equation and the possible

phase change at the wall are considered in the physical model. Possible fog formation is not considered, but integral mass and energy conservations are guaranteed. Very specific issues when the water mass conservation equation shifts from or to the saturation condition at the wall are discussed in detail. An equal order control volume based finite element method for two dimensional combined heat and vapor transfer by natural convection, including phase change at the wall, is proposed and used to solve the problem. Solutions for the cooling (and condensation) and heating (and evaporation) situations are obtained. Results show how the stream function, temperature, vapor concentration and relative humidity change with space and time, how absolute pressure level and the overall Nusselt and Sherwood numbers vary with time, and how the thickness of condensate varies over the wall for the cooling (and condensation) situation.

CHT12-NC09

LARGE-EDDY SIMULATION OF A BUOYANT PLUME PAST A BLUFF BODY: EFFECTS OF FLOW STRUCTURES ON ENTRAINMENT CHARACTERISTICS

Hitoshi Suto[*,§] and Yasuo Hattori[*]
[*]Central Research Institute of Electric Power Industry, Chiba, Japan
[§]Correspondence author. Fax: +81 4 7184 7142 Email: suto@criepi.denken.or.jp

A large-eddy simulation (LES) of a buoyant plume past a bluff body (BB) is performed. The modified Rayleigh number based on the total heat input and the diameter of a BB is set at 1.2×10^{10}. Distributions of basic statistics showed that the existence of a BB greatly varies the spatial structures of momentum and energy transport, although they gradually approached distributions corresponding to the similarity law for a fully developed plume when z/D>2.5 (z: vertical position from a heat source, D: BB diameter) since the influence of a BB becomes weaker there. Then, the relationship between entrainment and flow structures was investigated and the entrainment coefficient was found to be locally large right above a BB owing to stationary horizontal flows and azimuthal vortices. Moreover, in the case of an insulated BB, some azimuthal vortices were formed in the process of developing a plume, 1.0<z/D<2.0, owing to both weak buoyancy and BB effects, and it is suggested that they are linked to the increase of entrainment there.

CHT12-NC10

RAYLEIGH–TAYLOR INSTABILITY IN TWO-FLUID AND STRATIFIED MEDIA

Sergey N. Yakovenko[*,**,§]
[*] Khristianovich Institute of Theoretical and Applied Mechanics SB RAS, Novosibirsk, Russia
[**] Dept. of Physics, Novosibirsk State University, Novosibirsk, Russia
[§]Correspondence author. Fax: +7 383 330 72 68 Email: yakovenk@itam.nsc.ru

Two-fluid interface evolution is studied by direct simulations of Navier–Stokes and volume-fraction equations. To capture surface tension, the continuum surface force model is used where the mollified volume-fraction function varies smoothly across the interface due to convolving the original function with the eight-order polynomial smooth kernel. Tests of Rayleigh–Taylor instability show that the average of spike and bubble amplitudes has an initial exponential growth, corresponding to a linear instability stage with the constant growth

rate. Evolution of this rate shows that both viscosity and surface tension effects damp the instability development in agreement with theory and measurements data. For real fluids (water-air) good prediction is obtained for both linear instability and nonlinear stages. If density difference across the interface is not so large, the nonlinear stage shows Kelvin–Helmholtz instability effects leading to typical mushroom-like structures. For large density difference (e.g. for water-air interface), the heavier fluid penetrates deeply into the lighter one and forms high columns. Surface tension omission gives spurious distortion of the interface, then its fragmentation. The vortex-sheet method yields under-estimation of the growth rate. Application of the continuum surface force model leads to the correct interface evolution within the experiments data scatter. The similar convective structures as for two-fluid flows with moderate density difference are observed in direct numerical simulations of a single-phase stably stratified flow above the obstacle (when overturning/breaking internal gravity waves produce unstable layers with strong density gradients) and provide a source of quasi-steady turbulent patches. To define the buoyancy terms in the Navier–Stokes equations with the Boussinesq approximation, the density-deviation equation is used. The overall behavior and the wavelength of dominant structures evaluated from the simulation results correspond to the second (non-linear) stage of the scenarios for the Rayleigh–Taylor instability evolution when memory of initial conditions is lost.

CHT12-NC11

NATURAL CONVECTIVE HEAT TRANSFER FROM A VERTICAL ISOTHERMAL HIGH ASPECT RATIO RECTANGULAR CYLINDER WITH AN EXPOSED UPPER SURFACE MOUNTED ON A FLAT ADIABATIC BASE

Patrick H. Oosthuizen[§], Abdulrahim Kalendar and Almounir Alkhazmi
Department of Mech. and Materials Eng., Queen's University, Kingston, ON Canada K7L 3N6
[§]Correspondence author. Fax: +01 613 533-6489 Email: oosthuiz@me.queensu.ca

Some electrical component cooling problems can be approximately modeled as involving natural convective heat transfer from a vertical isothermal cylinder with a rectangular cross-section mounted on a flat horizontal adiabatic base plate. The cylinder may be pointing vertically upwards or vertically downwards. The heat transfer from the surface of the cylinder for this situation has been numerically and experimentally investigated here. Because of the applications being considered, the length-to-width ratios of the rectangular cross-section of the cylinders considered are relatively large. The cylinder is mounted on a flat horizontal adiabatic base plate and both the case where the cylinder points vertically upwards and where it points vertically downwards have been considered. In the numerical study the flow has been assumed to be steady and it has been assumed that the fluid properties are constant except for the density change with temperature which gives rise to the buoyancy forces, this having been treated by using the Boussinesq approach. The solution has been obtained by numerically solving the three-dimensional governing equations using the commercial CFD code FLUENT ©. Because of the applications that motivated this study, results have been obtained only for a Prandtl number of 0.74, i.e. essentially the value for air. A relatively wide range of the other governing parameters has been considered. A supporting experimental study has also been undertaken. In this study solid aluminum cylinders were used and the mean heat transfer rate from the cylinder was determined using the transient method in which the heat transfer rate is deduced from the rate at which a uniformly preheated cylinder cools. Experimental results were also obtained for the case where the cylinder points vertically

upward and where it points vertically downward. Good agreement between the experimental and numerical results was obtained.

CHT12-NC12

THREE-DIMENSIONAL NUMERICAL SIMULATION OF UNSTEADY TURBULENT NATURAL CONVECTION IN AN ENCLOSURE HAVING FINITE THICKNESS HEAT-CONDUCTING WALLS

Mikhail A. Sheremet
Dept. of Mechanics and Mathematics, Tomsk State University, Russia;
Institute of Power Engineering, Tomsk Polytechnic University, Russia
Fax: +7 3822 529740 Email: Michael-sher@yandex.ru

Three-dimensional unsteady natural convection in a cubic enclosure having finite thickness walls subject to opposing and horizontal temperature gradient has been investigated by a finite volume method. The flow is driven by conditions of constant temperature imposed along the external surfaces of two vertical side walls while the remaining walls are adiabatic from outside. The turbulent flow considered into the volume is described mathematically by the 3D Reynolds averaged Navier-Stokes equations, including the time averaged energy equation for the mean temperature field. The Reynolds stresses appearing in the Reynolds equations have been calculated on the basis of the standard k-ε model with wall functions. Staggered grid procedure was used with a power law differencing scheme for the convection terms and central differencing scheme for the diffusion terms. The pressure-velocity coupling is achieved using the SIMPLER method. The velocity and temperature distributions were calculated at fixed Rayleigh and Prandtl numbers, $Ra = 10^7$, $Pr = 0.7$ and different values of the dimensionless time $0 \le \tau \le 500$.

CHT12-VV01

COMPARISON BETWEEN PIV RESULTS AND CFD SIMULATIONS OF AIR FLOWS IN A THIN ELECTRONICS CASING MODEL

Masaru Ishizuka*, Tomoyuki Hatakeyama, Risako Kibushi and Shinji Nakagawa
*Dept. of Mechanical Systems Engineering, Toyama Prefectural University, Japan.
*Corresponding author. Fax: +81 766 56 6131 Email: ishizuka@pu-toyama.ac.jp

The aim of this study was to acquire benchmark test data for simulating computational fluid dynamics in thin electronic equipment. Flow in the model of thin electronic equipment was measured by using particle image velocimetry PIV). Dummy components were placed in the model and their configurations altered. The temperature rise of a heat source in the model was also measured and the cooling performance examined. The PIV measurement results revealed the changes in flow with changes in the configuration of the components. Comparison of the experimental results with numerical results showed good agreement in terms of the overall velocity field.

CHT12-VV02

EXPERIMENTAL AND NUMERICAL MODELLING OF ENHANCED THERMAL DIFFUSION IN A STRUCTURED PACKED BED.

CG du Toit[§], PG Rousseau, TL Kgame and CNA Preller
School of Mech. & Nuc. Eng., NWU, Potchefstroom, South Africa
[§]Correspondence author. Fax: +27 18 299 1322 Email: jat.dutoit@nwu.ac.za

The effective thermal conductivity in high temperature packed bed reactors is usually derived by lumping all the relevant heat transfer mechanisms into a single representative value. It can be split into two or more contributing components. Here the focus is on the effective fluid thermal conductivity which characterises the enhanced thermal diffusion in the fluid due to the turbulent mixing that occurs due to the porous structure of the packed bed. A brief overview is given of the Braiding Effect Test Sections (BETS) with homogeneous porosities of 0.36, 0.39 and 0.45 that were constructed to investigate this phenomenon. To determine the thermal diffusion, sets of thermocouples were installed at two levels in the packed beds contained in the BETS. The BETS was mounted in the pressure vessel of the High Pressure Test Unit (HPTU) and the experiments were performed under suitable quality assurance certification. A thorough uncertainty analysis was performed on all measured variables and error propagation was used to determine the uncertainty associated with derived variables. Four test runs were performed to ensure repeatability. The measured temperature profiles were normalised to account for the varying ambient conditions between the test runs. A numerical model was generated of a quarter of the cross section and the lower half of the BETS36 test section. The grid was generated according to the findings of a thorough grid dependence study and LES was found to be the best to model the turbulent nature of the flow. The relevant data from one test case was used as the boundary conditions for the simulation. Good agreement was obtained between the simulated and the corresponding measured temperatures. Based on the uncertainty associated with the positions of the thermocouples and the temperature gradients in the bed, the simulation could explain the scatter in the measured temperatures.

CHT12-VV03

EXACT ANALYTICAL SOLUTIONS FOR VERIFICATION OF NUMERICAL CODES IN TRANSIENT HEAT CONDUCTION

Filippo de Monte[*,§] and James V. Beck[**]
[*]Dept. of Mechanical Engineering, University of L'Aquila, Italy
[**]Dept. of Mechanical Engineering, Michigan State University, USA
[§]Correspondence author. Fax: +39 0862 434303 Email: filippo.demonte@univaq.it

This paper is intended to provide very accurate analytical solutions modeling transient heat conduction processes in 2D Cartesian finite bodies for small values of the time. Analysis of diffusion of thermal deviation effects indicates that, when the space and time coordinates satisfy a certain criterion, the simple transient 1D semi-infinite solutions may be "used" for generating extremely accurate values for temperature and heat flux at any point of a finite rectangle. Also, they may be "used" with excellent accuracy as short-time solutions when the time-partitioning method is applied (so avoiding the usually difficult integration of the short-cotime Green's functions). A complex 2D semi-infinite problem is solved explicitly and evaluated numerically as part of the analysis. The proposed criterion is based on an accuracy

of one part in 10^n ($n = 1, 2, ..., 10, ...$), where n = 2 is for engineering insight and visual comparison while n = 10 is for verification purposes of large numerical codes.

POSTER SESSION 6

CHT12-CF01

LARGE EDDY SIMULATION OF FIRE IN A LARGE TEST HALL

A.C.Y. Yuen[*], G.H. Yeoh[*,**,§], R.K.K. Yuen[***] and J. Tang[*]

[*]School of Mechanical and Manufacturing Engineering,
University of New South Wales, Sydney 2052, Australia
[**]Australian Nuclear Science and Technology Organisation (ANSTO),
PMB 1, Menai, NSW 2234, Australia
[***]Department of Civil and Architectural Engineering, City University of Hong Kong,
Tat Chee Avenue, Kowloon Tong, Hong Kong, PRC
[§]Corresponding Author: Email: g.yeoh@unsw.edu.au

A fully-coupled Large Eddy Simulation (LES) model which incorporates all essential subgrid-scale (SGS) turbulence, combustion and radiation has been developed to simulate the temporal and fluid dynamical effects of the burning of methanol in a square pan with a heat release rate of 0.9 MW in a large test hall. A scalar dissipation conditioned SGS combustion model is introduced to account for the non-equilibrating combustion caused by microscopic mixing processes. Numerical results are obtained through the two-step predictor and corrector explicit marching scheme. Predicted transient temperatures are compared against experimental measured data at different spatial locations. Reasonable agreement has been achieved. Effects of different turbulent Prandtl numbers (and Schmidt numbers which they are interrelated) on the transient temperature development are assessed.

CHT12-CF02

CFD MODELLING OF THERMO-CHEMICAL PROCESS IN CATALYTIC PYROLYSIS OF SAWDUST IN BUBBLING FLUIDIZED BEDS

Nanhang Dong[1], Sai Gu[2,*], Lindsay-Marie Armstrong[1], Konstantinos Papadikis[1],
and Kaihong Luo[1]
[1]Faculty of Engineering & The Environment, University of Southampton, UK
[2]School of Engineering, Cranfield University, UK
[*]Correspondence author. Fax: +44 1234 755232 Email: s.gu@cranfield.ac.uk

Simulation of catalytic cracking of sawdust samples has been carried out in bubbling fluidized beds with in situ catalyst. The techniques of bio-oil upgrading are one of the critical issues in industrial applications. Computational modelling of thermochemical processing of biomass can be quite complex, since it involves the interaction of multiphase flow dynamics coupled with chemical reaction. The current work presents a numerical model that investigates the online catalytic upgrading of fast pyrolysis derived tars. The Eulerian-Eulerian approach is employed to model the multiphase flow in fluidized beds, coupled with the Kinetic Theory of Granular Flows. The kinetic scheme for catalytic pyrolysis is incorporated into the simulation to represent the decomposition of biomass. The simulation results show that the convection processes dominate the heating up of the sawdust particles while the space time, the ratio of catalyst mass to tar flow rate, can significantly influence the final product yield distribution.

CHT12-CF03

MALDISTRIBUTION OF FUEL/AIR FLOWS IN A STACK OF PLANAR SOFC CAUSED BY TEMPERATURE NON-UNIFORMITY

Hiroshi Iwai[*,§], Daisuke Hayashi[*], Motohiro Saito[*] and Hideo Yoshida[*]
[*]Dept. of Aeronautics and Astronautics, Kyoto University, Japan
[§]Correspondence author. Email: iwai.hiroshi.4x@kyoto-u.ac.jp

A numerical model for an anode-supported direct-internal-reforming planar solid oxide fuel cell (SOFC) short stack was developed. In this model, the volume-averaging method is applied to the flow passages by assuming that a porous metal is inserted in the passages as current collectors. This treatment reduces the computational time and cost by avoiding a full three-dimensional simulation while maintaining the ability to solve the flow and pressure fields in the streamwise and spanwise directions. Quasi-three-dimensional multicomponent gas flow fields, the temperature field, and the electric potential/current fields were solved. The steam-reforming reaction of methane, the water-gas shift reaction, and the electrochemical reactions of hydrogen were taken into account. The model was applied to a short stack consisting of 10 cells. A heat-loss model based on radiative heat transfer was applied to the top wall surface of the stack. The temperature gradient was found to have eased owing to the heat-loss effect; however, the decrease in the average cell temperature increased the internal loss of the cell and lowered the cell efficiency. The difference in the average cell temperature affected the gas viscosity and led to the maldistribution of both the fuel and air flows in the stack. The fuel utilization of one cell in the stack was shown to be very different from the overall fuel utilization of the entire stack. The cooling air tends to flow to the colder side of the stack, worsening the temperature non-uniformity. The results show the importance of careful thermal management of SOFCs, particularly in a small system.

CHT12-CF05

REACTION–DIFFUSION PHENOMENA WITH RELAXATION

J. I. Ramos
Escuela de Ingenierías, Universidad de Málaga.
Doctor Ortiz Ramos, s/n; 29071 Málaga; Spain
Fax: +34 951952542 Email: jirs@lcc.uma.es

Relaxation phenomena in two-dimensional nonlinear activator-inhibitor processes governed by reaction-diffusion equations are studied by means of a time-linearized implicit method based on the time-linearization and discretization of the time derivatives that result in linear elliptic equations at each time level. These two-dimensional elliptic equations are then factorized in terms of one-dimensional operators which are discretized by means of second-order accurate finite differences, and the second-order approximate factorization errors are neglected. The numerical method uses the dependent variables and its first-order derivative as unknowns, is self-starting and, for relaxation times equal to zero, reduces to the well-known second-order accurate, time-linearized Crank-Nicholson procedure; therefore, the method may be used to study conventional multi-dimensional reaction-diffusion equations as well as relaxation phenomena in reaction-diffusion characterized by hyperbolic equations. Applications of the method to activator-inhibitor problems governed by reaction-diffusion equations with relaxation

and in the presence of a solenoidal velocity field that corresponds to a modified Lamb-Oseen vortex indicate that, for small vortex strengths, the spiral wave is almost unaffected by the flow except near its tip, whereas for strong vortices and core radii on the order of the drift of the spiral wave, substantial changes in the spiral wave morphology are observed depending on whether the vortex rotates in a clockwise or counter-clockwise direction. For counter-clockwise rotations, it is shown that spiral wave arm may exhibit a bulge in the activator concentration on the wave's inner side, while the inhibitor concentration may exhibit a bulge in the inner side and a valley in the outer side. For clockwise vortex motion, it is shown that the spiral wave may thicken on account of the opposing flow field and its tip may acquire a pointed feature. It is also shown that, for the conditions considered in this work, the spiral wave does not break up; neither does it form tiles and stripes as it has been found with other velocity fields.

CHT12-CF06

NUMERICAL STUDY OF THE EFFECT OF HEAT LOSS ON TRIPLE FLAME PROPAGATION IN A POROUS CHANNEL

Faisal Al-Malki [*,§],
[*]Department of Mathematics, Taif University, Taif P.O.Box 888, Saudi Arabia
[§]Correspondence author. Fax: +9662727200 Email: falmalki@tu.edu.sa

We numerically examine in this study the effect of heat-loss on triple flame propagation in porous channels. Unlike previous studies that were based on a counterflow configuration, we consider a porous channel in which a non-strained two-dimensional mixing layer is created as a result of the constant concentrations of the reactants -fuel and oxidizer- supplied at the walls. The problem is formulated mathematically within the framework of thermo-diffusive model and a single step chemistry along with a volumetric heat-loss, and then solved numerically using finite elements. The results has identified the effect of two main non-dimensional parameters on flame propagation, namely the heat loss κ and the flame-front thickness ε. Such parameters are found to be crucial for the existence of multiple solutions and hysteresis phenomena characterizing distinct combustion regimes. The number and the domain of existence of multiple solutions are identified in the $\kappa-\varepsilon$ plane. The overall impact of heat-loss on triple flames is to large extent found to be qualitatively similar to that reported for flame edges propagating in counterflow. The study has also described the transition of the flame structure across the combustion regimes, between triple flames, edge flames and flame tubes.

CHT12-TM01

NUMERICAL STUDY OF CLOSURE MODELS APPLIED TO TURBINE BLADE FILM COOLING.

Amar Berkache [*,§], Rabah Dizene [**]
[*)] Department. Mechanical Engineering University Mohamed Boudiaf Msila ALGERIA
[**)] LMA Laboratory University Houari Boumediene Bab Ezzouar Algiers ALGERIA
[§] Amar Berkache. Email: omar_berkache@yahoo.fr

A numerical study of the interaction of a row of jets of a secondary coolant fluid (R<1) in transverse flow is performed. Discrete jets are arranged across a surface exposed to a wall

boundary layer of parallel compressible stream (M_∞=0,72), as occurs in certain discrete-hole cooling systems for turbine blades . The simulation is performed by solving the governing equations numerically, with the effects of turbulence modeled in a way which allows for the anisotropies existing in the real situation and the effects of stream curvature. Comparisons between the results of three turbulence models obtained for injection angle of 45 deg and blowing rate less than unity show discrepancies observed in the flow near the wall. The causes of these differences are identified and discussed.

CHT12-TM03

NUMERICAL SIMULATION OF TURBULENT TWO-PHASE FREE SURFACE FLOWS.

Nawel.Khaldi[*, §], Hatem.Mhiri[**] and Philippe Bournot[***]
[*]Unité de thermique et de thermodynamique des procédés industriels, Monastir, Tunisia.
[**] Ecole Nationale d'Ingénieurs de Monastir, Monastir, Tunisia.
[***] Equipe IMFT, Institut de Mécanique de Marseille, Marseille, France.
[§]Correspondence author. Email: Khaldi.nawel@yahoo.fr

In this work, we propose a numerical study for a turbulent two-phase free surface flow involving an incompressible liquid and a compressible gas and taking into account the surface tension effects. The simulations were carried out using Fluent 6.3 with three dimensional meshes. The aim was to produce a predictive hydrodynamic model for a turbulent two-phase free surface flow, which may become the basis for convection heat transfer analysis. The air/water interface was modeled with the Volume of Fluid (VOF) method. Prediction of the free surface is heavily influenced by the quality of the mesh. The numerical results prove that better prediction of the free surface is obtained with grid adaptation and a sharper free surface position is obtained with time step adaptation. The k-epsilon turbulence model gives an improved velocity profile prediction. Fluid heights predicted by the CFD agree well with those measured experimentally.

CHT12-TM05

CONJUGATE HEAT TRANSFER ANALYSES OF A TURBINE VANE WITH DIFFERENT RANS TURBULENCE MODELS

Ilhan Gorgulu[*,§], Sinan Inanli[*], I. Sinan Akmandor[**]
[*]Tusas Engine Industries, Inc., Eskisehir, Turkey
[**]Middle East Technical University, Ankara, Turkey
[§]Correspondence author. Fax: +90 222 211 21 00 Email: ilhan.gorgulu@tei.com.tr

Conjugate heat transfer analyses which use popular RANS equations do not provide desired accuracy for turbine cooling applications. The main reason of this inaccuracy is stemming from assumption of isotropic turbulence (excluding Reynolds Stress Model) causing insensitivity to curvatures. Turbine blades' suction side and bends in the cooling channels causes high curvatures for flow to follow. New applications to solve these inaccuracy problems are being introduced and one of them is V2-f turbulence model. In this research, V2-f model together with different RANS models (k-e Realizable, k-w SST, RSM) are used to simulate the conjugate heat transfer for internally-cooled turbine vane. The experimental studies conducted by Hylton et. al. [1983] are used as test cases to validate the analyses. Two cases (one subsonic and one

supersonic) are solved and compared with the experimental data. Findings are presented in the form of surface heat transfer coefficient and surface temperature.

CHT12-TM06

SOME COMPARISONS ON SIMULATING TURBULENT ANISOTHERM FLOWS CLOSE TO THE WALL

Najla El Gharbi[*,***§], Rafik Absi[**], Ahmed Benzaoui[***] and Mohammed El Ganaoui[*]
[*]Université de Lorraine, IUT Henri Poincaré de Longwy, 186 rue de Lorraine, 54400 Longwy - Cosnes et Romain, France
[**]EBI, Inst. Polytech. St-Louis, 32 Boulevard du Port, 95094, Cergy-Pontoise Cedex, France
[***]USTHB, PB 32 El Alia Bab Ezzouar 16111 Alger, Algérie
[§]Correspondence author. Fax: +33 3 82 39 62 93 Email: najla.el-gharbi@univ-lorraine.fr

Turbulent channel flow with forced convection, where channel walls are uniformly heated, is simulated by an in-house code using a low Reynolds number k-ε turbulence model. The model predictions are validated by available DNS data and compared to the commercial code Fluent. The objective of this study is to investigate the performance of our CFD code and to simulate accurately the turbulent flow in near wall region.

CHT12-TM07

THE EFFECT OF DYNAMICS AND THERMAL PREHISTORY ON TURBULENT SEPARATED FLOW AT ABRUPT TUBE EXPANSION

Tatyana Bogatko, Viktor Terekhov
Kutateladze Institute of Thermophysics SB RAS, 630090,
Lavrent'ev Av., 1, Novosibirsk, Russia
Email: terekhov@itp.nsc.ru; bogatko1@mail.ru

Results of numerical investigation the effect of aerodynamics and heat boundary layer thickness in front of an abrupt expansion of a round tube on turbulent transfer in the zone of flow separation, attachment and relaxation are presented. Studying of this problem is very important for understanding of physics heat and mass transfer processes in separated flows. Before separation the flow was hydrodynamically stable, and the heat layer in front of expansion could change its thickness in maximally possible limits: from zero to a half of tube diameter. In the other case thickness of an aerodynamic layer could change, and thermal layer before a separation is absent.

The Reynolds number varied from $6.7 \cdot 10^3$ to $1.33 \cdot 10^5$. Expansion ratio of the tube is constant $ER = (D_2/D_1)^2 = 1.78$. The RANS method, comprising the Fluent 6.2 software, was used for numerical calculations. According to the experience of this package application for calculation of separated flows, the model of shear stress transfer (k–ω SST) gives the best coincidence with experimental data because its description of transfer features in the flows with recirculation is the most adequate.

It was found out that the growth of dynamics and heat layer thickness leads to reduction of heat transfer intensity in the separation area and moving away of the coordinate of maximal

heat transfer from the place of tube expansion. Generalizing dependence for the maximal Nusselt number is given for variation of the heat layer thickness. Comparison with experimental data proved the main behavior tendencies of heat and mass transfer processes in separation flows behind a back-facing step with different dynamics and thermal prehistory.

CHT12-TM08

LARGE EDDY SIMULATIONS OF TURBULENT HEATED JETS

Maxime Roger[*,§], Pedro J. Coelho[*] and Carlos B. da Silva[*]
* Dept. of Mechanical Engineering, Instituto Superior Técnico/IDMEC,
Technical University of Lisbon, Av. Rovisco Pais, 1049-001 Lisboa, Portugal.
[§]Correspondence author. Fax: +351 218475545 Email: roger.maxime@ist.utl.pt

Large eddy simulations of turbulent heated jets have been carried out and validated against experimental data. The code employed in this work is based on the OpenFOAM platform. Heated jets with Reynolds number varying from 3800 to 8500 and with exit Froude numbers varying from 3500 to 13200 are simulated. The numerical results are validated against the experimental data provided by Anderson and Bremhorst [2002], and compared with the self-similarity laws derived by Chen and Rodi [1980] for the temperature and velocity. The validation tests are satisfactory and allow us to use the code with confidence for numerical investigations. A parametric study is presented in order to analyse the temperature influence on turbulence. It is observed that an increase of the temperature gradient decreases the turbulence intensity.

CHT12-TM09

RANS AND LES SIMULATION OF A SWIRLING FLOW IN A COMBUSTION CHAMBER WITH DIFFERENT SWIRL INTENSITIES

Li Zhuowei[*], Nabil Kharoua[*§], Hadef Redjem[**], and Lyes Khezzar[*]
[*] Mechanical Engineering Department, The Petroleum Institute,
P.O. Box 2533, Abu Dhabi, United Arab Emirates
[**] Faculté des Sciences et de la Technologie, Université Larbi Ben M'Hidi,
Oum El Bouaghi, Algeria
[§]Correspondence author. Fax: +971 (0)2 6075200 Email: nkharoua@pi.ac.ae

A numerical study of the effect of the swirl intensity on the turbulent flow inside an isothermal model combustor is conducted. The study started with a non-reacting swirling flow, in a combustion chamber, generated by a radial-type swirl generator with a Reynolds number, based on the outer radius of the annular inlet and the bulk inlet velocity, equal to 61090. Similarly to what is found in the literature, a lack of accuracy was observed starting at a distance equal to 120 mm from the inlet for the mean flow field and the situation was even worse for the turbulent field. The results showed remarkable improvement when the LES turbulence model was used. The mean and turbulent flow field exhibited remarkable differences between the low and high swirl flows. The axial velocity component decays rapidly from the inlet, for the high swirl case whereas the situation inverts for the tangential velocity component. The low swirl generates a conical inner recirculation zone and a large outer one while the high swirl generates an annular and conical inner recirculation zones and a smaller

outer one. For the high swirl case, inner and outer shear layers were generated immediately downward of the inlet annular slot. The low swirl case exhibited disordered turbulent structures. In both cases a precessing vortex core with high frequency oscillations is noticed for the low swirl, close to the inlet.

CHT12-TM11

LES OF TURBULENT THERMAL MIXING IN CIRCULAR AND SQUARE T-JUNCTION CONFIGURATIONS

D. Caviezel, M. Labois and D. Lakehal[§]
ASCOMP GmbH, Zurich Switzerland
[§]Corresponding author. Email: Lakehal@ascomp.ch

Large Eddy Simulation (LES) results of thermal mixing in both circular and square T-junction configurations at high Reynolds number (~10^5) are reported in this work. The circular configuration was studied at the Älvkarleby Laboratory, Vattenfall R&D. The test rig consists of a horizontal pipe for the cold water flow and a vertically oriented pipe for the hot water flow. The square T-junction configuration with different channel sizes was studied experimentally at the Department of Mechanical Engineering of Mie University, Japan. The Vattenfall case was selected as a benchmark by OECD held in Washington DC in 2010; the second case was selected as a benchmark for thermal mixing in the ERCOFTAC Workshop held in EDF Chatou, France, 2011. Large-Eddy and URANS simulations were performed with the code TransAT©. The comparison shows excellent agreement between LES and the data.

CHT-12-OF01

STUDIES OF A TURBULENT PATCH ARISING FROM INTERNAL WAVE BREAKING AT DIFFERENT PRANDTL/SCHMIDT NUMBERS

Sergey N. Yakovenko[1,2], T. Glyn Thomas[2], Ian P. Castro[2]

[1] Khristianovich Institute of Theoretical and Applied Mechanics SB RAS, Novosibirsk, Russia
[2] Faculty of Engineering and the Environment, University of Southampton, Southampton, UK

Turbulent patches observed in geophysical flows and laboratory experiments can arise when internal waves overturn and break. Proper DNS studies of wave-breaking turbulence at high Prandtl/Schmidt numbers Sc would require very fine grids [1]. Estimations for the finest grid used in [1] show that it is still insufficient to capture all turbulent scales of the density field, and the scalar dissipation microscale is much smaller that both the mesh size and the Kolmogorov microscale of the velocity field. Preliminary runs at Sc = 700 (using "DNS" with no subgrid-scale (SGS) models) indeed have inadequate resolution of the dissipation range as illustrated by both spatial and temporal spectra (Fig.1). Moreover, the density field quickly generates the significant noise which remains not only in the wave-breaking turbulent patch itself, but also in the surrounding fluid (Fig.2) which would expected to have very low noise levels due to strongly stable stratification. Similar difficulties may arise at Reynolds numbers of engineering and environmental problems, which are much higher than Re = 4000 in [1].
Therefore, it is a great challenge to resolve adequately as many details of the flow as possible at high Reynolds or Prandtl/Schmidt numbers. To overcome the problems of insufficient grid

resolution, we apply the SGS models of Smagorinsky type for both velocity and density equations, with the standard value of Smagorinsky constant, C_s = 0.1, and demonstrate that the SGS Prandtl/Schmidt number, Sc_{sgs} = 0.3 (as in buoyant jet studies [2]), is appropriate. For Re = 4000 and at the fine grid applied in [1], the SGS eddy viscosity is almost everywhere below the molecular one (except separate points during the late stage of transition to the turbulence), however the SGS eddy diffusivity at high Prandtl/Schmidt numbers is much larger than the molecular one, so has a considerable effect (Figs.1,2). Nevertheless, we can still capture transition details and obtain the velocity field statistics in the resulting turbulent patch at Sc = 700 which is expected to be nearly the same as for Sc = 1.

1. Yakovenko, S. N., Thomas, T. G., and Castro, I. P. A turbulent patch arising from a breaking internal wave, *J. Fluid Mech.*, 2011, Vol. 677, pp. 103-133.
2. Zhou, X., Luo, K. H., and Williams, J. J. R. Study of density effects in turbulent buoyant jets using large-eddy simulation, *Theoret. Comput. Fluid Dynamics*, 2001, Vol. 15, pp. 95-120.

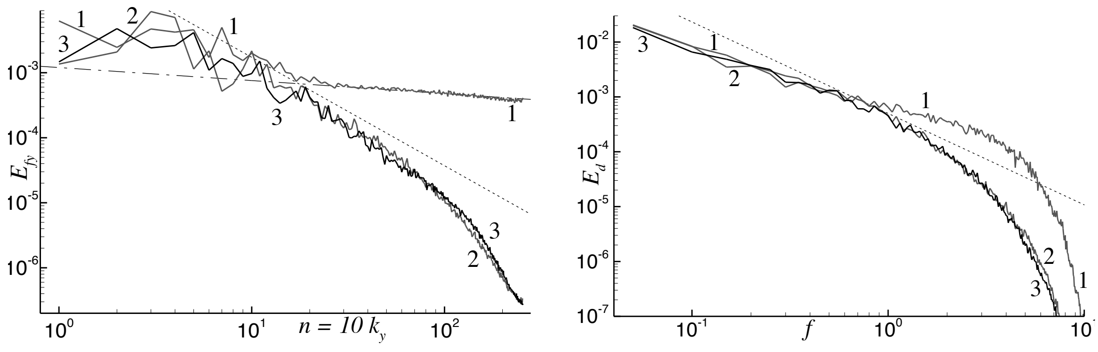

Fig. 1. Scalar (density) variance spectra (1 – "DNS" at Sc = 700, 2 – DNS at Sc = 1, 3 – "LES" at Sc = 700, dashed and dashed-dotted lines show the '–5/3' and '–1' power laws, respectively): spanwise spectra for t = 42.5 averaged at $1.8 \le x \le 2.3$ and $2.4 \le z \le 2.9$ (left); temporal spectra at x = 2.19 and z = 2.97 averaged over 64 nodes along the span (right).

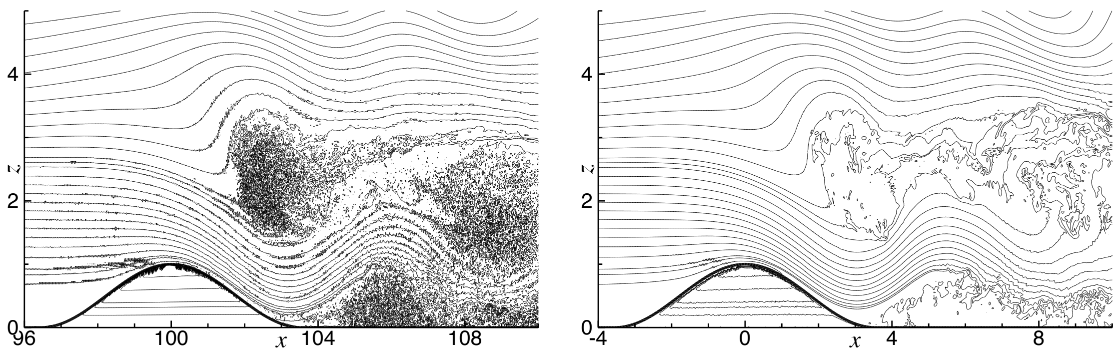

Fig. 2. Density contours for t = 35 and y = 0 (at the central vertical plane) in "DNS" (left) and "LES" (right) at Sc = 700.

POSTER SESSION 7

CHT12-MN01

NUMERICAL INVESTIGATION OF NANOFLUID FLOW AND HEAT TRANSFER IN A PLATE HEAT EXCHANGER

Iulian Gherasim [1], Nicolas Galanis*[,2] and Cong Tam Nguyen[3]
[1] Faculty of Civil Engineering and Building Services, Dept of Building Services, Technical University "Gheorghe Asachi", Iasi, Romania, 700050
[2]Faculty of Engineering, Université de Sherbrooke, Qc., Canada J1R 2R1
[3]Faculty of Engineering, Université de Moncton, Moncton, NB, Canada E1A 3E9
*Corresponding author: Fax: 1 819 821 7163, E-mail: nicolas.galanis@usherbrooke.ca

This paper presents a numerical investigation of the flow and heat transfer behavior of two nanofluids, namely CuO-water and Al2O3-water, inside a plate heat exchanger. Both laminar and turbulent flows are studied under steady state conditions. In the turbulent regime the RANS-based Realizable κ-ε turbulence model was used. The homogeneous single-phase fluid model was employed to characterize the nanofluids. All fluid properties were considered temperature dependant. The adopted unstructured mesh possesses approximately 9.63x106 elements and was used for both laminar and turbulent flows. Results show that a considerable heat transfer enhancement was achieved using these nanofluids and the energy-based performance comparisons indicate that some of them do represent a more efficient heat transfer medium for this type of application. In general, all nanofluids cause higher pressure losses due to friction compared to that of water.

CHT12-MN02

NUMERICAL STUDY OF NANOFLUID HEAT TRANSFER ENHANCEMENT WITH MIXING THERMAL CONDUCTIVITY MODELS

Amarin Tongkratoke*,§, Anchasa Pramuanjaroenkij**, Apichart Chaengbamrung* and Sadik Kakaç***
[*]Department of Mechanical Engineering, Kasetsart University, Bangkok, 10900, Thailand.
[**]Department of Mechanical and Manufacturing Engineering*, Kasetsart University, Chalermphrakiat Sakon Nakhon Province Campus, Sakon Nakhon, 47000, Thailand.*
[***]Department of Mechanical Engineering, TOBB University of Economics and Technology, Ankara, 06560, Turkey.
[§]Correspondence author. Fax: +66 42 725034 Email: kaset.amarin@gmail.com

Nanofluids has shown its possibility in enhancing heat transfer performance above its base fluids. This work presents a numerical study to analyze the nanofluid heat transfer enhancement using different theoretical models; the effective viscosity and the effective thermal conductivity models. The Maxwell, Brownian motion, and Yu and Choi models are considered as thermal conductivity models and the models are used in the simulation domain alternately, named the mixing models. The Al_2O_3-water nanofluids is chosen in this study and assumed to flow under laminar fully developed flow condition through a rectangular pipe as in a circuit application. The

governing equations written in terms of the primitive variables are solved through an in-house program using the finite volume method and the SIMPLE algorithm. The results showed that different effective viscosity and thermal conductivity models play important roles, especially at wall surfaces where the convective heat transfer is enhanced effectively. Moreover, the small volume fractions, the nanofluid volume fractions from 0.01 to 0.03 can increase the heat transfer enhancement. Therefore, the volume fraction, the effective viscosity and the effective thermal conductivity at the wall region can be increased by increasing nanoparticle amount. This work can strongly support the literatures that the volume fraction, the effective viscosity and the effective thermal conductivity can enhance the heat transfer performance in the nanofluid flows not only with the single-phase model considered but also with the mixing models examined.

CHT12-MN03

HEAT TRANSFER ENHANCEMENT USING CuO/WATER NANOFLUID

Meriem AMOURA[*,§], Amar MAOUASSI[**] and Noureddine Zeraibi[**]
[*]Université des Sciences et de la Technologie Houari Boumedienne, Faculté de Physique, Dépt. Energétique. B.P. 32 El-Alia, 16111 Bab-Ezzouar, Alger, Algeria
[**]Université de Boumerdes, Faculté des hydrocarbures, Dépt. Transport et Equipements, Avenue de l'indépendance, 35000 Boumerdes, Algeria
[§]Correspondence author. Fax: +(213) 21 247 344 Email: am_louni@yahoo.fr

Buoyancy induced flow and heat transfer is an important phenomenon in engineering systems due to its wide applications in electronic cooling, heat exchangers, double pane windows etc.. The enhancement of heat transfer in these systems is an essential topic from an energy saving perspective. The lower heat transfer performance of conventional fluids, such as water, engine oil and ethylene glycol obstructs the performance and the compactness of such systems. The use of solid particles as an additive suspended into the base fluid is a technique for heat transfer enhancement.

Therefore, the aim of this work is to study heat transfer enhancement taking into account the variation of both thermal conductivity and viscosity in the governing equations when using the nanofluid CuO/water in a horizontal circular tube with a finite length and a diameter D. The cylinder is subjected to a uniform heat flux on the upper half of its circumference.

CHT12-MN04

ENHANCEMENT OF THE HEAT TRANSFER DUE TO THE LAMINAR FORCED CONVECTION OF A NANOFLUID IN A CHANNEL

Eugenia Rossi di Schio[§], Michele Celli, Antonio Barletta
DIENCA, Alma Mater Studiorum – Università di Bologna, Viale Risorgimento 2, Italy
[§]Correspondence author. Fax: +39 051 209 3296 Email: eugenia.rossidischio@unibo.it

A steady laminar forced convection in a parallel-plane channel using nanofluids is studied. The flow is assumed to be fully developed, and described through the Hagen-Poiseuille profile. A boundary temperature varying with the longitudinal coordinate in the thermal entrance region is prescribed. A study of the thermal behaviour of the nanofluid is performed by solving numerically the fully elliptic coupled equations. The numerical solution is obtained by

Galerkin's finite element method implemented through the software package Comsol Multiphysics (Comsol, Inc.). The paper shows that, for physically interesting values of the Péclet number and for physically interesting boundary conditions, the concentration field depends very weakly on the temperature distribution.

CHT12-MN05

EFFECT OF CHANNEL PRESSURE DIFFERENCE IN HEAT TRANSFER ENHANCEMENT IN MICRO-CHANNEL WITH SYNTHETIC JET

A. Lee [1,*], J. A. Reizes [1], V. Timchenko [1] and G.H. Yeoh[1,2]

[1]School of Mechanical and Manufacturing Engineering, University of New South Wales, Sydney, NSW 2052, Australia
[2]Australian Nuclear Science and Technology Organisation (ANSTO), PMB 1, Menai, NSW 2234, Australia
*Corresponding author. Fax +61(2) 9663 1222 Email: ann.lee@unsw.edu.au

A three-dimensional computational model has been developed to investigate the cooling of a microchip by micro-channels in which water is flowing and in which a synthetic jet generator has been installed at the mid-point of the channels. The synthetic jet operates at a fixed frequency and amplitude of the diaphragms while the pressure difference between the ends of the channels is varied. For a microchip with a constant thermal load, when the synthetic jet is not operating, because the flow rate in the channels increases as the pressure difference is raised, as would be expected, the maximum temperature in the wafer is reduced as the pressure difference is increased. As the result in this steady flow, the maximum temperature in the silicon wafer with a thermal load of 1 MWm^{-2} is decreased by 11.3 K when the channel pressure difference is raised from 500 Pa to 1500 Pa. It is shown that when the synthetic jet is activated at a constant frequency and amplitude, the flow patterns in the channels are quite different at the two pressure differences. As a consequence, whilst the maximum temperature in the silicon wafer is lowered in each case below that prevailing when the flow is steady, after the 40^{th} cycle of membrane oscillation the reductions in the maximum temperature in the silicon wafer are 9 K and 7.5 K below those in steady flow for with channel pressure differences of 500 Pa and 1500 Pa respectively. It follows that the reduction in temperature caused by a synthetic jet is lessened if the pressure difference between the ends of the channels is significantly increased. Therefore, the channel pressure difference and the synthetic jet parameters are equally important in determining the optimal operating condition.

CHT12-MN06

MIXED CONVECTION FLOW ON A VERTICAL PERMEABLE SURFACE IN A POROUS MEDIUM SATURATED BY A NANOFLUID WITH INTERNAL HEAT GENERATION

Mohd Hafizi Mat Yasin[*], Norihan Md Arifin[*,§], Roslinda Nazar[**], Fudziah Ismail[*] and Ioan Pop[***]

[*]Department of Mathematics and Institute for Mathematical Research,
Universiti Putra Malaysia, Malaysia.
[**]School of Mathematical Sciences, Faculty of Science and Technology,
Universiti Kebangsaan Malaysia, Malaysia.
[***]Faculty of Mathematics, University of Cluj, Romania.
[§]Correspondence author. Fax: +60389466997 Email: norihan@math.upm.edu.my

An analysis of the mixed convection boundary layer flow on a vertical surface in a porous medium saturated by a nanofluid with internal heat generation is performed in this paper. The similarity equations are solved numerically for three types of metallic or non-metallic nanoparticles, namely copper (Cu), alumina (Al_2O_3) and titania (TiO_2), in a water-based fluid, to investigate the effect of the heat generation coefficient and nanoparticle volume fraction parameter of the nanofluid on the flow and heat transfer characteristics. The numerical results on the skin friction coefficient are presented in the forms of tables and figures along with the velocity profiles.

CHT12-MN07

CONJUGATED CONVECTIVE-CONDUCTIVE HEAT TRANSFER IN MICRO CHANNELS WITH UPSTREAM REGION PARTICIPATION

Diego C. Knupp, Renato M. Cotta[*], and Carolina P. Naveira-Cotta
Universidade Federal do Rio de Janeiro - COPPE & POLI, UFRJ
Mechanical Engineering Department - Laboratory of Transmission and Technology of Heat
Cx. Postal 68503, Rio de Janeiro, RJ, 21945-970, Brazil
[*]Correspondence author. Email: cotta@mecanica.coppe.ufrj.br

Conjugated heat transfer in laminar micro-channel flow is analyzed, taking into account the axial diffusion effects which are often of relevance in micro-channels, including pre-heating or cooling of the region upstream of the heat transfer section. The methodology is based on the hybrid numerical-analytical approach known as the Generalized Integral Transform Technique (GIIT), applied to a single domain formulation, which is proposed for modelling the heat transfer phenomena at both the fluid stream and the channel walls regions. By making use of coefficients represented as space dependent functions, with abrupt transitions occurring at the fluid-wall interface, the mathematical model thus carries the information concerning the transition of the two domains, unifying the model into a single domain formulation with variable coefficients. Convergence of the proposed eigenfunction expansions is illustrated and emphasis is placed on the effects of heat transfer to the upstream flow region.

CHT12-MN08

MIXED CONVECTION FLOW OVER A HORIZONTAL CIRCULAR CYLINDER WITH A CONSTANT SURFACE HEAT FLUX IN A NANOFLUID

Leony Tham[*,§], Roslinda Nazar[**] and Ioan Pop[***]
[*]Faculty of Agro Industry and Natural Resources, Universiti Malaysia Kelantan, Malaysia.
[**]School of Mathematical Sciences, Faculty of Science and Technology,
Universiti Kebangsaan Malaysia, Malaysia.
[***]Faculty of Mathematics, University of Cluj, Romania.
[§]Correspondence author. Fax: +603 8925 4519 Email: leonytham@umk.edu.my

The laminar mixed convection boundary layer flow from a horizontal circular cylinder in a nanofluid, which is maintained at a constant surface heat flux, has been studied for both cases of a heated and cooled cylinder. The resulting system of nonlinear partial differential equations is solved numerically using an implicit finite-difference scheme. Three different types of nanoparticles are considered, namely Cu, Al_2O_3 and TiO_2. The solutions for the flow and heat transfer characteristics are evaluated numerically and studied for various values of the governing parameters, namely the nanoparticle volume fraction φ and the mixed convection parameter λ. It is worth mentioning that heating the cylinder ($\lambda>0$) delays the separation of the boundary layer and if the cylinder is hot enough (large values of $\lambda>0$), then it is suppressed completely. On the other hand, cooling the cylinder ($\lambda< 0$) brings the boundary layer separation point nearer to the lower stagnation point and for a sufficiently cold cylinder (large values of $\lambda< 0$), the flow completely detaches from the cylinder.

CHT12-RD01

OUR RECENT WORKS ON THE COLLOCATION SPECTRAL METHOD FOR THERMAL RADIATION IN PARTICIPATING MEDIA

Li Ben-Wen[*,§], Sun Ya-Song[*,**], Tian Shuai[***], Ma Jing[*], Hu Hang-Mao[*]
[*] Key Laboratory of Electromagnetic Processing of Materials (Ministry of Education), Northeastern University, P.O.Box 314, Shenyang 110004, China
[**] Beijing Key Laboratory of Multiphase Flow and Heat Transfer in Low-grade Energy Utilization Systems, North China Electric Power University, Beijing, 102206, China
[***] Department of Chemical Engineering, University of Birmingham, Edgbaston, Birmingham, B15 2TT, UK
[§]Correspondence author. Fax: +86 24 83681758 Email: heatli@hotmail.com

Due to the motivation of numerical simulation on radiative magnetohydrodynamics, as the first step, in recent few years, the authors and co-workers have adopted Chebyshev collocation spectral method (CSM) for the solutions of radiation transfer equation (RTE) in different situations and obtained many achievements. The solution cases include 1D thermal radiation with strong fluctuations and complex boundary conditions, coupled radiation and conduction in concentric spherical participating medium, pure radiation in graded index media, the steady and transient combination of radiation and conduction, etc. One most valuable work is the direct 3D Schur-decomposition for the 3D matrix equations after the discretization of RTE. The process of the applications of collocation spectral method to thermal radiation is introduced beginning from 3D case. The most favourite parts of CSM for RTE are the direct

solution, the very high spatial accuracy, and the good potential of incorporation into computational fluid dynamics (CFD) and magnetohydrodynamics (MHD). As to irregular multi-dimensional geometric systems, CSM can also be adopted to solve the RTE together with the body fitted coordinates (BFC). Compared with discrete ordinates method (DOM), the accuracy of CSM is more sensitive to the number of discretized directions. Some obstacles, say, the direct solution of RTE under the cases of non-homogeneous radiative properties, the inconsistence between the spectral accuracy in space and the only second-order accuracy in angular discretization, are stated. Finally, some possible futures are mentioned.

CHT12-RD02

PREDICTION OF RADIATIVE HEAT TRANSFER IN 2D AND 3D IRREGULAR GEOMETRIES USING THE COLLOCATION SPECTRAL METHOD BASED ON BODY-FITTED COORDINATES

Sun Yasong[*,**], Li Benwen[**,§]
[*]Beijing Key Laboratory of Multiphase Flow and Heat Transfer in Low-grade Energy Utilization Systems, North China Electric Power University, Beijing, 102206, China
[**]Key Laboratory of Electromagnetic Processing of Materials (Ministry of Education), Northeastern University, Shenyang 110004, China
[§] heatli@hotmail.com

In this work, a collocation spectral method (CSM) based on body-fitted coordinates (BFC) is employed to simulate thermal radiation heat transfer problems in 2D and 3D irregular geometries. Due to the exponential convergence of the CSM, a very high accuracy can be obtained even using a small number of grid points. This numerical method simultaneously makes use of the merits of both the CSM and BFC. In the numerical approach, the discrete ordinates form of radiative transfer equation (RTE) in orthogonal Cartesian coordinates is transformed into an equation written in general body-fitted coordinates. In order to test the efficiency of the method, several 2D and 3D complex irregular enclosures with curved boundaries are examined including a 2D quadrilateral enclosure, 3D hexahedral enclosure, and 3D elliptical enclosure in which absorbing, emitting and scattering media may be present. The accuracy of the results obtained by the CSM are assessed by comparing the them with those in literature. These comparisons indicate that the CSM based on BFC can be recommended as a good option for solving thermal radiation heat transfer problems in irregular geometries.

CHT12-RD03

COLLOCATION SPECTRAL METHOD TO SOLVE RADIATIVE TRANSFER EQUATION FOR THREE-DIMENSIONAL EMITTING-ABSORBING AND SCATTERING MEDIA BOUNDED BY GRAY WALLS

Hu Zhangmao[*], Li Benwen[*,§]
[*] Key Laboratory of Electromagnetic Processing of Materials (Ministry of Education), Northeastern University, P.O.Box 314, Shenyang 110004, China
[§]heatli@hotmail.com

A collocation spectral method (CSM) based on discrete-ordinates equation is employed to solve three dimensional radiative transfer equation (RTE) in an absorbing, emitting and scattering medium. Numerical results by the CSM are compared with those of discrete ordinates method (DOM) and benchmarks. These comparisons indicate that the CSM is more accurate than DOM.

CHT12-RD04

COMBINED TWO-FLUX APPROXIMATION AND MONTE CARLO MODEL FOR IDENTIFICATION OF RADIATIVE PROPERTIES OF HIGHLY SCATTERING DISPERSED MATERIALS

Leonid Dombrovsky[*,§], Krithiga Ganesan[**], and Wojciech Lipiński[**]
[*]Joint Institute for High Temperatures, Moscow, Russia
[**]Dept. of Mechanical Engineering, University of Minnesota, Minneapolis, USA
[§]Correspondence author. Fax: +7 495 362 5590 Email: ldombr@yandex.ru

An identification procedure is developed for obtaining spectral radiative properties of highly scattering dispersed materials such as porous ceramics. Traditional techniques based on measurements of the directional-hemispherical reflectance and transmittance are of limited use because of difficulties in fabricating sufficiently thin and mechanically stable samples to obtain reliable values of directional-hemispherical transmittance. However, one can use the directional-hemispherical reflectance measurements for optically thick samples to obtain the transport scattering albedo. A one-dimensional analytical solution employs the modified two-flux approximation for the identification of transport scattering albedo. An additional transmittance measurement is required to identify the transport extinction coefficient. Bi-normal narrow cone transmittance is measured for this purpose. Because the one-dimensional analytical solution is not applicable to model the bi-normal narrow cone transmittance, Monte Carlo ray-tracing technique is used to identify the transport extinction coefficient. The identification procedure is applied to obtain near-infrared radiative properties of porous ceria ceramics used in solar thermochemical reactors. The identified transport scattering coefficient is shown to be in good agreement with theoretical estimates based on the Mie theory for polydisperse pores and grains. This verifies the applicability of a model based on independent scattering and Mie theory for theoretical predictions of radiative properties of two types of ceria ceramics with porosity of 0.08 and 0.72, and for extrapolating the properties of both ceramics in a limited near-infrared range to the range of significant absorption.

A ROBUST MONTE CARLO BASED RAY-TRACING APPROACH FOR THE CALCULATION OF VIEW FACTORS IN ARBITRARY 3D GEOMETRIES

T. Walker, S.-C. Xue and G.W. Barton*
School of Chemical and Biomolecular Engineering
University of Sydney, NSW 2006, Australia
(*) Tel: +61 2 9351 3780; Fax: +61 2 9351 2854
Email: geoff.barton@sydney.edu.au

Drawing on ideas from computer-based graphical representations, the conventional use of finite element based approaches to represent three-dimensional (3D) geometries of interest is challenged in this work by the use of a modest suite of geometric ‘primitives’ (i.e. generic shapes such as a sphere, a cone, a flat surface) that in combination via a set of affine transformations can provide a realistic approximation to almost any conceivable 3D body. Initially, a robust C^{++} program using the latest CPU vectorisation technologies (*eg* OpenMP and Streaming SIMD Extensions) was developed and validated against a broad range (around a dozen) of known analytical view factor solutions. The impact of ray density level, random number generators, and ‘fast’ numerical approximations for widely used trigonometric functions were all extensively examined at this stage in terms of solution accuracy and required run-time. Extensive use was made at this stage of the in-built program profiling capabilities within the XCode 4.2 IDE to identify ‘choke points’ within the evolving computer code.

The program was subsequently interfaced to the ANSYS Polyflow package to develop a fully conjugate heat transfer model of an operational furnace used to draw specialised polymer optical fibres. The Monte Carlo Ray-Tracing (MC-RT) calculated view factors for all surfaces within the drawing furnace were found to be in excellent agreement with those calculated by numerical solution of the integral equations used to define the various view factors, while a good fit was obtained between the heat transfer model and measured experimental temperature profiles for the case of a non-deforming preform. A wide range of preform drawing cases was then examined, with rapid convergence (within 3-4 iterations) obtained between the furnace heat transfer calculations and the updating of the various view factor estimates.

POSTER SESSION 8

CHT12-BH01

NEW COMPUTATIONAL METHOD FOR THE SHORT-PULSED NIR LIGHT PROPAGATION IN BIOLOGICAL TISSUES

F. Asllanaj[§] and S. Fumeron
LEMTA-INPL, Université de Lorraine
2 Avenue de la forêt de Haye, BP 160, 54504 Vandoeuvre cedex, France
[§]Correspondence author - Phone : +33 3 83 59 55 26 ; Fax : +33 3 83 59 55 51
E-mail: Fatmir.Asllanaj@univ-lorraine.fr

Optical tomography is a medical imaging technique based on light propagation in the NIR (Near Infra-Red) part of the spectrum. This article presents a new way of solving the short-pulsed NIR light propagation using a time-dependent two-dimensional radiative transfer equation in absorbing and strongly anisotropically scattering medium. A cell-vertex finite-volume method is proposed for the discretization of the spatial domain. The closure relation based on the exponential scheme and linear interpolations was applied for the first time in the context of time-dependent radiative heat transfer problems. Details are given about the application of the original method on unstructured triangular meshes. The angular space (4π Sr) is uniformly subdivided into discrete directions and a finite-differences discretization of the time domain is used. Numerical simulations for media with physical properties analog to healthy and metastatic human liver subjected to a collimated short-pulsed NIR light are presented and discussed. As expected, discrepancies between the two kinds of tissues were found. At each time step, the level of light flux was found to be weaker (inside the medium and at boundaries) in the healthy medium than in the metastatic one.

CHT12-NC13

CFD SIMULATION OF HEAT TRANSFER IN A TWO-DIMENSIONAL VERTICAL CONICAL PARTIALLY ANNULAR SPACE

Belkacem Ould said[*, §], Noureddine Retiel[*], Mohamed Aichouni[**]
[*]Laboratory of Numerical and Experimental Modeling of Mechanical phenomena, Mechanical Engineering Department, Mostaganem University, B.P.300, Route BelhacelMosraganem, Algeria.
[**] Associate Professor, Engineering College, Hail University, S.A. Saudi BinLaden Research Chair in Quality & Productivity Improvement in Construction Industry.
[§]Correspondence author. Fax: +21345201891 Email: kazemou@yahoo.fr

In this paper, a numerical study, of two-dimensional steady flow analysis has been made on natural convection in a differentially heated vertical conical partially annular space. The heat transfer is assumed to take place by natural convection. Where the inner and outer surfaces of annulus are maintained at uniform wall temperature. The annulus is filled with air. The CFD FLUENT12.0 code is used to solve the governing equations of mass, momentum, energy quantities using constant properties and Boussinesq approximation for density variation.The streamlines and the isotherms of the fluid are presented for different annulus with different boundary conditions and Rayleigh number. Emphasis is placed on the influences of the height of the inner vertical cone on the flow and the temperature fields. In addition, the numerical results of the heat transfer are discussed for various values of physical parameters of the fluid

and geometric parameters of the annulus on the heat transfer. The heat transfers on hot walls of annulus are also calculated in order to make comparisons between the cylinder annulus for boundary conditions and several Rayleigh numbers. A good agreement of Nusselt number has been found between the present predictions and reference from the literature data.

CHT12-NC14

NATURAL CONVECTION IN A RECTANGULAR ENCLOSURE WITH AN ARRAY OF DISCRETE HEAT SOURCES

P Kandaswamy[*, §], V P M Senthil Nayaki[**], A Purusothaman[**] and S Saravanan[**]
[*]Department of Mathematics, Gandhigram Rural University, Dindigul 624 302, India
[**]UGC DRS Centre for Fluid Dynamics, Dept. of Mathematics, Bharathiar University, Coimbatore 641 046, India
[§]Correspondence author. Email: pgkswamy@yahoo.co.in

A study of natural convection from a 3×3 array of isoflux discrete heat sources mounted on one of the vertical walls in a three dimensional rectangular enclosure is made. Two different boundary conditions are considered, one in which the two side walls are cooled and the other in which the top and bottom walls are cooled. Numerical solutions are obtained using a finite volume method. Flow and heat transfer characteristics are investigated as a function of Rayleigh number Ra, aspect ratio A and Prandtl number Pr. The results show that the heat transfer rate increases with an increase of Ra. It also increases against A in the range of 1-3, however attains a maximum around A=3 and decreases beyond that. The entire enclosure above the bottom heaters is found to be more thermally active when the top and bottom walls are kept cold. However an effective cooling is possible only in the case of side cold walls.

CHT12-NC15

GEOMETRIC OPTIMIZATION FOR MAXIMUM HEAT TRANSFER DENSITY RATE FROM CYLINDERS ROTATING IN NATURAL CONVECTION

Logan Page[*], Tunde Bello-Ochende[*§] and Josua Meyer[*]
[*]Department of Mechanical and Aeronautical Engineering, University of Pretoria, South Africa
[§]Corresponding author. Fax: +27(0) 12 420 6632 Email: Tunde.Bello-Ochende@up.ac.za

In this study we investigate the thermal behaviour of an assembly of consecutive cylinders in a counter-rotating configuration cooled by natural convection with the objective of maximizing the heat transfer density rate (heat transfer rate per unit volume). A numerical model was used to solve the governing equations that describe the temperature and flow fields and an optimisation algorithm was used to find the optimal structure for flow configurations with two or more degrees of freedom. The geometric structure of the consecutive cylinders was optimized for each flow regime (Rayleigh number) and cylinder rotation speed for one and two degrees of freedom. Smaller cylinders were placed at the entrance to the assembly, in the wedge-shaped flow regions occupied by fluid that had not yet been used for heat transfer, to create additional length scales to the flow configuration. It was found that the optimized spacing decreases and the heat transfer density rate increases as the Rayleigh number increases, for the optimized structure.

For the single scale configuration it was also found that the optimized spacing decreases and the maximum heat transfer density rate increases, as the cylinder rotation speed was increased at each Rayleigh number. Results further showed that there was an increase in the heat transfer density rate of the rotating cylinders over stationary cylinders for a single scale configuration. For a multi scale configuration it was found that there was almost no effect of cylinder rotation on the maximum heat transfer density rate, when compared to stationary cylinders, at each Rayleigh number; with the exception of high cylinder rotation speeds, which serve to suppress the heat transfer density rate. It was, however, found that the optimized spacing decreases as the cylinder rotation speed was increased at each Rayleigh number. Results further showed that the maximum heat transfer density rate for a multi scale configuration (with stationary cylinders) was higher than a single scale configuration (with rotating cylinders) with an exception at very low Rayleigh numbers.

CHT12-NC16

A NUMERICAL STUDY OF NATURAL CONVECTIVE HEAT TRANSFER FROM AN INCLINED ISOTHERMAL PLATE WITH A "SINUSOIDALLY WAVY" SURFACE

Patrick H. Oosthuizen§ and Jane T. Paul
Dept. of Mech. & Mat. Eng., Queen's University, Kingston, ON, Canada K7L 3N6
§Correspondence author. Fax: +01 613 533 6489 Email: oosthuiz@me.queensu.ca

Natural convective heat transfer from a wide isothermal plate which has a "wavy" surface, i.e. a surface which periodically rises and falls, has been numerically studied. The surface waves run in the horizontal direction, i.e. are normal to the direction of flow over the surface. Attention has been restricted to the case where the waves have a sinusoidal cross-sectional shape. The plate is, in general, inclined to the vertical with consideration being given both to inclination angles at which the heated plate is facing upwards and to inclination angles at which the heated plate is facing downwards. The range of Rayleigh numbers considered extends from values that for a non-wavy vertical plate would be associated with laminar flow to values that would be associated with fully turbulent flow. The flow has been assumed to be steady and fluid properties have been assumed constant except for the density change with temperature that gives rise to the buoyancy forces, this being treated by means of the Boussinesq approximation. The Reynolds averaged governing equations in conjunction with a standard *k*-epsilon turbulence model with buoyancy force effects accounted for have been used in obtaining the solution. The governing equations have been solved using the commercial cfd code FLUENT©. The solution has the following parameters: i) the Rayleigh number based on the height of the heated plate, ii) the Prandtl number, iii) the ratios of the amplitude of the surface waviness and of the pitch of the surface waves to the height of the plate, and iv) the angle of inclination of the plate to the vertical. Results have been obtained for a Prandtl number of 0.74 and for amplitude and pitch ratios of 0.1. The effects of Rayleigh number and angle of inclination on the mean and local surface Nusselt numbers have been numerically studied.

NUMERICAL SIMULATION OF THERMAL CONVECTION IN A CLOSED CAVITY IN THE PRESENCE OF A THIN HORIZONTAL HEATED PLATE

L.Boukhattem [*,§], H. Hamdi[**], A. Bendou[***], D. R. Rousse[****]
[*] EnR2E, CNEREE, University Cady Ayyad Marrakech, Morocco
[**] LMFE URAC27-CNRST, FS, University Cady Ayyad Marrakech, Morocco
[***] ENCG, University Ibn Zohr, Agadir, Morocco
[****] Centre de Technologie Thermique, ETS Montréal Canada
[§]Correspondence author. Fax:(+212) 5 24 66 80 12 Email: boukhattem.lah@gmail.com

In this work, we present a numerical study of heat transfer by natural convection in a two-dimensional closed cavity, containing air, in the presence of a thin heater plate. The vertical walls are kept adiabatic, while the horizontal ones are isothermal. The equations governing the natural convection in the cavity are solved using a finite difference technique based on the control volume approach and the SIMPLEC (Semi-Implicit-Method for Pressure-Linked Equations Corrected) algorithm developed by Patankar [1980]. A non-uniform mesh in both directions, constructed by using a geometric progression, is adopted. The square cavity contains a thin heated plate located at the cavity center with an aspect ratio equal to 0.5. The heater plate is positioned horizontally and has a higher temperature than the isothermal walls. The simulation results are obtained in terms of velocity vectors and isotherms for different Rayleigh numbers values ranging from 10^4 to 10^6.

The symmetric boundary conditions produce a symmetric behaviour of temperature and velocity fields according to the central vertical plan. The increase of Rayleigh number leads to increasing importance of convection heat transfer relative to the conduction heat transfer. The fact is more marked for the regions above the heater plate. It is shown that for high Rayleigh numbers, heat transfer from the heater plate to the isothermal horizontal walls is mainly directed towards the top wall.

CHT12-NC18

NUMERICAL AND EXPERIMENTAL INVESTIGATION OF UNSTEADY NATURAL CONVECTION IN AN OPEN CHANNEL

G.E. Lau[*,§], V. Timchenko[*], C. Ménézo[**,***], S. Giroux-Julien[**], M. Fossa[****], E. Sanvicente[**], J. Reizes[*], G.H. Yeoh[*,*****]
[*] School of Mech. & Manuf. Eng., UNSW, Sydney, Australia
[**] Centre de Thermique de Lyon (CETHIL, CNRS-INSA Lyon, Université Lyon 1), Bât. Sadi Carnot, INSA de Lyon, 20 av. A. Einstein, 69 621 Villeurbanne Cedex, France
[***] Chaire INSA of Lyon / EDF « Habitats et Innovations Energétiques », CETHIL UMR 5008, Bât. Sadi Carnot, INSA de Lyon, 20 av. A. Einstein, 69 621 Villeurbanne Cedex, France
[****]Dime, Universita di Genova, Via Opera Pia 15a, 16145 Genova, Italy
[*****]Australian Nuclear Science and Technology Organisation (ANSTO), PMB 1, Menai, NSW 2234, Australia
[§]Correspondence author. Fax: +61 2 9663 1222 Email: ghar.lau@unsw.edu.au

Numerical and experimental investigations have been carried out to study the transition to turbulence of natural convective flows in a vertical conduit heated from one side. It is shown that the typical dynamics of large-scale structures of the flow and thermal fields of natural convection in open vertical channels are successfully modelled numerically by the use of the large-eddy algorithm. Furthermore, the fluctuations in both the velocity and temperature fields are also captured, providing valuable insights into the instantaneous flow dynamics occurring within the channel. A comparison between numerical and experimental results indicates that external disturbances play a significant role in the evolution of the dynamic behaviour of natural convective flows within the channel. Additionally, it is also demonstrated that a proper characterisation of the disturbance level in the ambient is necessary to bring the numerical and experimental results into close agreement.

CHT12-NC19

NUMERICAL INVESTIGATION FOR NATURAL CONVECTION IN A VERTICAL OPEN-ENDED CHANNEL: COMPARISON WITH EXPERIMENTAL DATA

Zoubir Aminea,b,* , Christophe Daverat a,b, Shihe Xin a,b, Stéphanie Giroux-Julien a,c, Hervé Pabiou a,b, Christophe Ménézo a,d
aUniversité de Lyon, CNRS, France
b INSA-Lyon, CETHIL, UMR5008, F-69621, Villeurbanne, France
c Univ. Lyon 1, CETHIL, UMR5008, F-69621, Villeurbanne, France
d Chaire INSA-EDF Habitats et Innovations Energétiques, CETHIL, UMR5008, F-69621, Villeurbanne, France
∗ Corresponding author: amine.zoubir@insa-lyon.fr

The present study deals with natural convection flow in a vertical open-ended channel with wall constant heat flux. The experimental and numerical investigations are both conducted using water as the working fluid in order to get rid of radiation heat transfer and to have a pure convective flow. The numerical code is developed using Finite Differences scheme of second order in time and space to solve the elliptic two-dimensional Navier-Stokes equations

under the Boussinesq assumption. Concerning the experimental apparatus, it consists of two heated walls immersed in water. Temperature and velocity measurements are provided (by thermocouple and Laser Doppler Velocimetry system) for different modified Rayleigh numbers based on the walls spacing b : $Ra_b^* = 1.67 \times 10^6, 3.6 \times 10^6, 8.97 \times 10^6, 1.69 \times 10^7, 4.29 \times 10^7$. The numerical code is first validated with a numerical benchmark for the case of natural convection in air between asymmetrically heated walls under uniform heat flux without considering radiative effects. Then, the numerical simulations are performed for the case of natural convection of water between two symmetrically heated walls by uniform heat flux and compared to experimental data. The code provides a satisfactory prediction of main quantities compared to the experimental results but only for the lowest Rayleigh numbers $Ra_b^* = 1.67 \times 10^6, 3.6 \times 10^6$. For higher modified Rayleigh numbers $Ra_b^* = 8.97 \times 10^7, 1.69 \times 10^7, 4.29 \times 10^7$, the flow becomes three-dimensional and turbulent. Therefore, 2D numerical simulations fail to predict flow and heat transfer for this range of modified Rayleigh number.

CHT12-FC01

INVESTIGATION OF TURBULENT FORCED CONVECTION IN HELICALLY GROOVED TUBES

Zoltán Hernádi[*,§], Loránd Romvári[**], Gábor Varga[***], Gergely Kristóf[*]
[*]Dept. of Fluid Mechanics, Budapest University of Technology and Economics, Hungary
[**]Furukawa Electric Institute of Technology, Budapest, Hungary
[***]Dept. of Physics, Budapest University of Technology and Economics, Hungary
[§]Correspondence author. Fax: +36 1 463 3464 Email: hernadi@ara.bme.hu

Tubes with helically grooved inner surfaces are widely used in heat exchangers of refrigerators and air conditioners for their heat transfer performance and relatively low pressure drop compared to other heat transfer enhancement techniques. Although there are great databases of measurement data for some groove geometries, the empirical correlations based on these databases are often inaccurate for new groove patterns. Since manufacturing tubes with future groove geometries is too expensive, an accurate numerical model for the performance of non-existing groove geometries is of great importance. A CFD model based on Large Eddy Simulation of turbulent flow with heat transfer is proposed. No empirically tuned parameters are used in the simulations. Forced convection is considered as a single-phase incompressible flow with uniform outside wall heat flux. High temperature variations are beyond the scope of this investigation, therefore thermophysical parameters are assumed to be constants. In the simulation, fully developed conditions are assumed, therefore the describing equations are solved for streamwise periodic variables with consistent source terms in the heat and momentum transport equations. Geometry specification and meshing are carried out with own developed software utilizing parametric description of helically grooved tubes. Simulation results developed in OpenFOAM for Re = 6000 to 10000 are presented for smooth pipe and helically grooved tubes. The simulation methodology is validated by comparing the results with literature data for smooth pipes and with in-house measurements for helically grooved tubes. Difference from literature data is within 5% for friction factors and Nusselt numbers of smooth pipes. Difference from measured data is within 15% for the friction factors and 7% for the Nusselt numbers for helically grooved tubes. The proposed model is concluded to be accurately predictive.

CHT12-FC05

THE INFLUENCE OF SPANWISE WAVELENGTH OF GÖRTLER VORTICES IN HEAT TRANSFER

Vinicius Malatesta [*,§], Leandro F. de Souza[*] and Joseph T. C. Liu[**]
[*]Dept. Applied Math. and Statistics, University of Sao Paulo, Sao Carlos, Brazil
[**]Dept. of Mechanical Engineering, Brown University, USA.
[§]Correspondence author. Fax: +55 16 3373 9650 Email: malatest@icmc.usp.br

The increase in heat transfer rates aiming efficient systems is one of the goals in convection heat transfer exchangers. The boundary layer over concave surfaces can be unstable to centrifugal forces, giving rise to Görtler vortices. These Vortices create two regions in the spanwise direction, the upwash and downwash regions. The downwash region is responsible to compress the boundary layer in the direction of the wall, increasing the heat transfer rate. The upwash region does the opposite. In the nonlinear development of the Görtler vortices it can be observed that the upwash region becomes narrow, and the average heat transfer rate is higher than that for a Blasius boundary layer. In the present paper it is analysed influence of the Görtler vortices spanwise wavelength in the heat transfer. The paper is carried out by a Spatial Direct Numerical Simulation. The Navier-Stokes equation is written in vorticity-velocity formulation. The time integration is done via a classical 4th order Runge-Kutta method. The spatial derivatives are calculated using high-order compact finite difference and spectral methods. Three different wavelengths were analysed. The results shows that steady Görtler flow can increase the heat transfer rates to values above the turbulent values.

CHT12-FC06

DIRECT NUMERICAL SIMULATION OF HEAT TRANSFER OF A SPHERICAL PARTICLE IN AIR STREAM

Nafiseh Talebanfard[§] and Bendiks Jan Boersma
Dept. of 3ME, TU Delft, Netherlands
[§]Correspondence author. Fax: +31 1 5278 2460 Email: n.talebanfard@tudelft.nl

In this paper flow and heat transfer over a spherical particle is modeled by means of Direct Numerical Simulation (DNS) for Reynolds numbers up to 400, and consequently different flow regimes. A three-dimensional model is applied which integrates the Navier stokes equations with a second order Adams-Bashforth Scheme. Temperature distribution inside the sphere and the consequent effect on heat transfer coefficient is also considered in this study. The Bi number is changed over the range of 0.005-20 to study the relation between Bi and Nu numbers. It is seen that for higher Reynolds the rate of heat transfer increases. For higher Bi numbers the surface temperature changes more rapidly, therefore the temperature gradient at the surface of the particle becomes smaller and hence the local Nu number representing the rate of heat transfer at the surface of the particle will be lower.

CHT12-FC07

NUMERICAL STUDY OF HEAT AND FLUID FLOW PAST A CUBICAL PARTICLE AT SUB-CRITICAL REYNOLDS NUMBERS

Kay Wittig*, Andreas Richter and Petr A. Nikrityuk

CIC Virtuhcon, Department for Energy Process Engineering and Chemical Engineering, Technische Universität Bergakademie Freiberg, Fuchsmühlenweg 9, 09596 Freiberg, Germany

*Correspondence author: Fax: +49 3731 39 4555 Email: Kay.Wittig@vtc.tu-freiberg.de,

This work is devoted to a numerical investigation into heat and fluid flow past a particle of a cubical shape. Recently several publications appeared, where numerical calculations of drag forces and heat transfer coefficients for non-spherical particles in laminar flows with an angle of attack of zero were performed. In this work we focus on the influence of the angle of attack on flow past a cubical particle. Due to the asymmetric flow, three-dimensional calculations were carried out based on the immersed boundary method (IBM) in continuous forcing mode. In order to substantiate the results, additional calculations were made based on a so-called conventional CFD solver using body-fitted meshes.

Based on the present analysis of numerical results obtained for a cubical particle, new correlations for both the drag coefficient and the Nusselt number were developed. In addition to the Prandtl and Reynolds numbers both correlations incorporate the angle of attack. A significant influence of the particle orientation on the characteristics c_D and Nu was observed. The accuracy of the closures developed for c_D and Nu is discussed, comparing the relations developed with published models.

CHT12-FC08

INFLUENCE OF THREE-DIMENSIONAL PERTURBATIONS ON HEAT TRANSFER AT HYPERSONIC FLOW

Ivan Egorov[*,§] and Vladimir Shvedchenko[*]

[*]Central Aerohydrodynamic Institute, Zhukovsky, Moscow region, Russia

[§]Correspondence author. Fax: +7 4955564369 Email: ivan.egorov@tsagi.ru

The present paper deals with the investigation of the three-dimensional flow in a shock layer at supersonic transverse flow conditions over the front surface of a cylinder under small, spatially-periodic perturbations along the transverse coordinate. Based on the numerical solution of the unsteady three-dimensional Navier-Stokes equations, it is shown that the small imposed perturbations of the free stream velocity (0.5 - 3 %) along the transverse coordinate lead to shock front curvature, to the formation of vortex structures in the shock wave area and to the formation of significant perturbations of the heat flux on the surface.

CHT12-SM03

HEAT TRANSFER, PHASE CHANGE AND COALESCENCE OF PARTICLES DURING SELECTIVE LASER SINTERING OF METAL POWDERS

Ram Dayal[*], Tatiana Gambaryan-Roisman[**,§] and Eberhard Abele[***]
[*]Center of Smart Interfaces, TU Darmstadt, Petersenstr. 32, Darmstadt, Germany.
[**]Institute of Technical Thermodynamics, TU Darmstadt, Petersenstr. 32, Darmstadt, Germany.
[***]Institute of Production Management, Technology and Machine Tools, TU Darmstadt, Petersenstr. 30, Darmstadt, Germany
[§]Correspondence author. Fax: +49 (0)6151/166561 Email: gtatiana@ttd.tu-darmstadt.de

Flow and heat transfer during selective laser sintering of metal powder is investigated numerically. Heat transfer, melting and resolidification, which are induced by laser irradiance and coupled with coalescence of partially melted particles, are dominant processes governing laser sintering of metal powders. The present model accounts for phase change during the laser heating and cooling cycle and subsequent melt flow due to surface tension forces. The study is aimed to gain understanding of mechanisms governing microstructure formation during selective laser sintering. A two-dimensional boundary element (BEM) model is used to solve the governing equations. The solid-liquid interface during phase change is tracked using Stefan condition at the interface. The liquid melt formed due to melting is assumed to be viscous. The influence of process parameters on temperature field evolution, shape evolution and densification rate is quantified and discussed.

CHT12-SM04

FIXED-GRID SIMULATIONS OF STEADY, TWO-DIMENSIONAL, ICE-WATER SYSTEMS WITH LAMINAR NATURAL CONVECTION IN THE LIQUID

Sylvain Bories[1], Nabil Elkouh[2], and B. Rabi Baliga[1,*]
[1]Dept. of Mech. Eng., McGill Univ., 817 Sherbrooke St. W., Montreal, QC H3A 2K6, Canada
[2]Erigo Technologies LLC, Enfield, NH 03748-0899, U.S.A.
[*]Correspondence author. Fax: +514 398 7365 Email: rabi.baliga@mcgill.ca

A numerical investigation of steady, two-dimensional, ice-water systems with laminar buoyancy-driven natural convection in the water and conduction in the ice is presented. The calculation domains were rectangular enclosures, cooled and heated on opposite vertical side walls, and insulated (adiabatic) on the top and bottom walls. The long-term goal is to contribute to the development of mathematical models and numerical solution methods suitable for use as cost-effective computational tools in the design of enhanced ice-water seasonal cold-storage (IWSCS) systems. A fixed-grid, co-located, finite volume method (FVM) was adapted for use in this work. Predictions were obtained using a variable-property model (VPM) and also a constant-property model (CPM). In simulations with the CPM, all properties of liquid water, except its density, were evaluated at several different reference temperatures and assumed constant, and the thermal conductivity of ice was pegged to its value at the melting temperature. The reference temperature that leads to the lowest differences between the results yielded by the VPM and CPM was determined, and it is the main contribution of this work. The reasons for seeking such a reference temperature are two-fold: 1) the CPM facilitates cost-effective simulations for designing IWSCS systems optimized for specific applications; and 2) porous metal foams are often embedded in IWSCS

systems to improve their performance, and practical volume-averaged approaches to the modeling of fluid flow and heat transfer in such composite systems are usually based on a CPM. The computed streamlines, water-ice interface positions, and values of the average Nusselt number on the hot wall are also presented for the cases considered.