14th UK Heat Transfer Conference

7-8 Sept, 2015

Pollock Halls, The University of Edinburgh, Edinburgh, UK

Agenda and Book of Abstracts

Acknowledgements

EPSRC Pioneering research and skills

THERMACORE EUROPE Thermal Management Solutions

OXFORD UNIVERSITY PRESS

European Commission

Blank Page

The 14th UKHTC Organising Committee

UK Heat Transfer Committee Chairman

Prof. Geoffrey Hewitt
Imperial College London

Conference Chairs

Prof. Khellil Sefiane
The University of Edinburgh

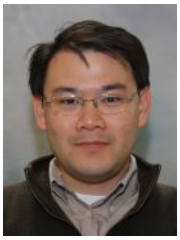
Dr. David McNeil
Heriot-Watt University

Local Organising Committee

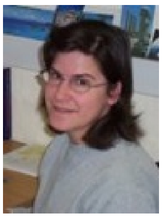
Dr. Prashant Valluri
The University of Edinburgh

Dr. Tadhg O'Donovan
Heriot-Watt University

Dr. John Christy
The University of Edinburgh



Dr. Yeaw Chu Lee
Heriot-Watt University



Dr. Gail Duursma
The University of Edinburgh

Ms. Sue Simpson
Conference Secretariat, The University of Edinburgh

Programme Schedule

Sunday, 6th Sept 2015					
	18:00			***Onsite registration and Drinks reception at 19:00***	
Day 1: Monday, 7th Sept 2015					
	07:30			***Poster presenters set up their posters***	
	08:00 onwards			***Onsite registration***	
Welcome	**08:30**	**-**	**09:00**	**Welcome**	
Keynote1	**09:00**	**-**	**09:30**	**Keynote 1**	**Prof. Tassos Karayiannis (*Brunel University London, UK*): Flow boiling in microchannels**
9:30-10:30 Poster Presentations *Phase Change* *Session Chair: D. A. McNeil*	09:30	-	09:32	PC1	CFD modelling of low pressure evaporation. By *Ahmad et al.*
	09:33	-	09:35	PC2	Novel CFD methods to predict evaporation of water under a vacuum. By *Panesar et al.*
	09:36	-	09:38	PC3	Shell-side boiling of a glycerol-water mixture at low subatmospheric pressures. By *McNeil et al.*
	09:39	-	09:41	PC4	Implementing a multi-disciplinary strategy to understand heat transfer in a reduced pressure highly active evaporator. By *Baker et al.*
	09:42	-	09:44	PC7	Ambient pressure effect on nanofluid sessile droplet evaporation. By *Askounis et al.*
	09:45	-	09:47	PC8	Leidenfrost vitrification of droplets with liquid nitrogen. By *Duursma et al.*
	09:48	-	09:50	PC9	Condensation of R134A at low mass fluxes in a smooth tube at different inclination angles. By *Ewim et al.*
	09:51	-	09:53	PC10	Chemically treated micropillars for enhanced condensation heat transfer. By *Orejon et al.*
	09:54	-	09:56	PC11	Effect of channel orientation on the flow boiling heat transfer and pressure drop in a 1.1 mm diameter channel. By *Pike-Wilson et al.*
	09:57	-	09:59	PC12	The effect of brine temperature and salinity on the rate of electromagnetic attenuation within it. By *Hales et al.*
	10:00	-	10:02	PC13	Thermocapillary convection for an evaporating meniscus with changing contact angle. By *Buffone*
	10:03	-	10:05	PC14	Ice pigging coolant jackets: heat transfer into the ice pig body. By *McBryde et al.*
	10:06	-	10:08	PC15	Ice formation and production in subcooled environments. By *Yun et al.*
	10:09	-	10:11	PC16	Effect of extreme wetting scenarios on pool boiling. By *Valente et al.*
	10:12	-	10:14	PC17	Experimental study on the thermal performance of two-phase closed-loop thermosyphon with liquid heat transfer agent at high heat flux. By *Wei et al.*
	10:15	-	10:17	PC18	Effect of substrate temperature on deposition pattern from nanofluid droplets. By *Zhong et al.*
	10:18	-	10:20	PC19	Numerical simulation of condensation in mini horizontal tubes with different cross-section shapes. By *Zhang et al.*
	10:21	-	10:23	PC20	On the use of phase change materials in low-temperature Fischer-Tropsch (LTFT) reactors. By *Odunsi et al.*
	10:24	-	10:26	PC21	An experimental investigation of the effect of control algorithm on the energy consumption and temperature distribution of a household refrigerator. By *Tolga et al.*
	10:27	-	10:29	PC22	Experimental investigation of bubble behaviours in a heat pump water heating system. By *Qin et al.*
Break	**10:30**	**-**	**11:00**	**Coffee Break + Poster Viewing**	
Keynote2	**11:00**	**-**	**11:30**	**Keynote 2**	**Prof. Ping Cheng (*Shanghai Jiaotong University, China*): Recent numerical and analytical studies on effects of surface characteristics in phase-change heat transfer**
11:30-12:30 *Phase Change*	11:30	-	11:32	PC23	Numerical Simulation of flow and heat transfer around vertical cylinder submerged in water. By *Ramadan et al.*
	11:33	-	11:35	PC24	Saturated Boiling of Water on Biphilic Surfaces under Sub-atmospheric pressure. By *Yamada et al.*
	11:36	-	11:38	PC25	On the evaporation of droplets with related initial and receding contact angles.By *Stauber et al.*

↦ * These are poster-only contributions

Programme Schedule

Phase Change and Applications Sessions

11:30-12:30 **Poster Presentations** ***Phase Change*** ***Session Chair:*** ***T. O'Donovan***	11:39	-	11:41	PC26	Generation and Metastability of Interfacial Nanobubbles. By *Takahashi et al.*
	11:42	-	11:44	PC27	Optimization of Heating and Condensation System of a Water Condensed Type Washer Dryer Regarding Water Consumption. By *Top et al.*
	11:45	-	11:47	PC28/29*	Thermal Response of a Pulsating Heat Pipe on Board the Rexus 18 Sounding Rocket: PHOS Experiment Chronicles. By *Creatini et al.*
	11:48	-	11:50	PC30	Flow Boiling Heat Transfer in a Shallow Metallic Microchannel. By *Ozdemir et al.*
	11:51	-	11:53	PC31/32*	Vapour generation of perfectly wetting liquid by cyclone evaporator under variable gravity level. By *Glushchuk et al.*
	11:54	-	11:56	PC33	Flow Boiling Heat Transfer and Pressure Drop of R134a in a Multi Microchannel Metallic Evaporator. By *Mohammed et al.*
	11:57	-	11:59	PC34	Thermal Energy Storage Using Composite Phase Change Materials: Linking Materials Properties to Device Performance. By *Li et al.*
	12:00	-	12:02	PC35/36*	A Novel Type of Multi-Evaporator Closed Loop Two Phase Thermo-syphon: Thermal Performance Analysis and Fluid Flow Visualization. By *Mameli et al.*
	12:03	-	12:05	PC37	Numerical simulation of water-vapour condensation by means of a flow oriented scheme. By *Karadimou et al.*
	12:06	-	12:08	PC38	Surface tension of n-Butanol and steam mixture on metal surface. By *Jivani et al.*
	12:09	-	12:11	PC39	Effects of Vapour Velocity and Pressure on Marangoni Condensation of Steam-Butanol Mixtures on a Horizontal Tube. By *Jivani et al*
	12:12	-	12:14	PC40	Study of a New Wick Material for Capillary-Driven Heat Pipes. By *De Schampheleire et al.*
	12:15	-	12:17	PC41/42*	Numerical Simulation of Flow Boiling in Micro-channels: Bubble Growth, Detachment and Coalescence. By *Georgoulas et al.*
	12:18	-	12:20	PC43	Evaporation/boiling heat transfer characteristics in an artery porous structure. By *Bai et al.*
	12:21	-	12:23	PC44	Experimental study on direct solar energy absorption of Au-Cu hybrid nanofluids. By *Bai et al.*
	12:24	-	12:26	PC45	Modelling of a tank containing paraffin as phase-change material for cold storage applications. By *Biosca-Taronger et al.*
	12:27	-	12:29	PC46	Thermal Analysis of a Novel Solar-biogas Hybrid System Integrated with PCM Insulation Closure. By *Lu et al.*
Lunch Break	**12:30**	**-**	**13:45**	**Lunch Break + Poster Viewing until 13:15 + Poster Switchover**	
Keynote3	**13:45**	**-**	**14:15**	**Keynote 3**	**Prof. Yasuyuki Takata (*Kyushu University, Japan*): Pool boling on superhydrophobic/philic surfaces**
14:00-15:15 **Poster Presentations** ***Applications*** ***Session Chair:*** ***J. Christy***	14:16	-	14:18	AP1	The Effect of Secondary Flow on Developing Flow in the Transitional Flow Regime. By *Everts et al.*
	14:19	-	14:21	AP2	Thermal conductivities of annular packed beds in axial fluid flow. By *Glass et al.*
	14:22	-	14:24	AP3	Heat Transfer Mechanisms for Single Rising Taylor Bubbles. By *Scammell et al.*
	14:25	-	14:27	AP4	Thermal characterisation of compact heat exchangers for automotive air conditioning. By *Torregrosa-Jaime et al.*
	14:28	-	14:30	AP5	Improving thermocouple measurement accuracy for Solid Oxide Fuel Cell application. By *Barari et al.*
	14:31	-	14:33	AP6	Investigation on thermal efficiency and cost-effective mode of a solid thermal package by utilizing off-peak power. By *Duan et al.*
	14:34	-	14:36	AP7	Development of an optical thermal history sensor based on the oxidation of divalent rare earth ion phosphor. By *Yanez-Gonzalez et al.*
	14:37	-	14:39	AP8	Measurement and Simulation of Low Temperature Packed Bed Regenerators. By *Pike-Wilson et al.*
	14:40	-	14:42	AP9	Variable Conductance Heat Pipes for Managing Thermal Stores By *Hislop et al.*
	14:43	-	14:45	AP10	Nanofluids for Heat Transfer Applications. By *Rudyak et al.*
	14:46	-	14:48	AP11	Flooding behaviour in countercurrent gas-liquid flow in vertical tubes with turbulence promoters. By *Spindler*
	14:49	-	14:51	AP12	Manipulating Phonon Heat Conduction by High Pressure Torsion in Silicon Based Thermoelectrics. By *Tabara et al.*
	14:52	-	14:54	AP13	Experimental Study of Unsteady and Conjugate Heat Transfer in Wavy Film Flows over an Inclined Heated Foil. By *Charogiannis et al.*

* These are poster-only contributions

Programme Schedule

Applications ***Session Chair: J. Christy***	14:55	-	14:57	AP14	An experimental study of rotational pressure loss in rotor ducts. By *Chong et al.*
	14:58	-	15:00	AP15	The Effect of the Inclination Angle on Heat Transfer Performance in Back-ward Facing Step Utilizing Nanofluid. By *Etaig et al.*
	15:01	-	15:03	AP16/17*	Characterization of the Corona Discharge for Ionic Wind Heat Transfer Enhancement in Internal Flow Channels. By *Gallandat et al.*
	15:04	-	15:06	AP18	Novel Heat Transfer using Solid Phase Transport Medium. By *Alexander*.
	15:07	-	15:09	AP19	Dynamic Testing and Modelling of Solar Collectors. By *Guarracino et al.*
	15:10	-	15:12	AP20	A Self-Pumped Heat-Exchanger for Wave-Powered Desalination. By *Hellenschmidt et al.*
	15:13	-	15:15	AP21	Analysis and Experiment on Forced Convection Heat Transfer Coefficient and Pressure Drop of Diamond-Shaped Fin-Array. By *Hirasawa et al.*
Break	**15:15**	**-**	**15:45**	**Coffee Break + Poster Viewing**	
Keynote4	**15:45**	**-**	**16:15**	**Keynote 4**	**Prof. Joe Quarini (*University of Bristol, UK*): Ice slurries, the cool, benevolent bringers of sustainable clean living**
16:15-17:15 Poster Presentations *Applications* ***Session Chair: K. Sefiane***	16:16	-	16:18	AP22	The imitation of the surface temperature variation characteristics of concrete road under periodical ambient conditions. By *Ye et al.*
	16:19	-	16:21	AP23	Improving the operation of a geothermal district heating network through the use of a heat storage tank. By *Kyriakis*
	16:22	-	16:24	AP24/25*	Sensitivity Analysis of a Capillary Pulsating Heat Pipe: Influence of the Tube Characteristics. By *Manzoni et al*.
	16:25	-	16:27	AP26	Effect of Hydraulic Diameter and Aspect Ratio on Single Phase Flow and Heat Transfer in a Rectangular Microchannel. By *Sahar et al.*
	16:28	-	16:30	AP27	Influence of the microstructure on the transport phenomena on horizontal tubes. By *Tomforde et al.*
	16:31	-	16:33	AP28	Development of a Solar Cooling System Based on a Fluid Piston Convertor. By *Mahkamov et al.*
	16:34	-	16:36	AP29	Performance Evaluation of Room Temperature Magnetic Refrigerator Using Corrugated Plate Regenerator. By *Kamran et al.*
	16:37	-	16:39	AP30	Heat Exchanger Analysis of Azeotropes in Organic Rankine Cycles. By *Kirmse et al.*
	16:40	-	16:42	AP31/32*	Parametric Design Study of Vacuum Glazed Windows. *By Ali et al.*
	16:43	-	16:45	AP33	Turbulent heat transfer between two horizontal planes under inherently stable and unstable conditions. By *Jackson et al.*
	16:46	-	16:48	AP34	Measuring the Heat Transfer Coefficient in a Direct Oil-Cooled Electrical Machine with Segmented Stator. By *Camilleri et al.*
	16:49	-	16:51	AP35	Run-around Coils for Energy Efficiency. By *Bentham et al.*
	16:52	-	16:54	AP36	Temperature Dependence of Energy Band Gaps In Triple Junction Solar Cell. By *Maka et al.*
	16:55	-	16:57	AP37	Penetrative convection in a unit aspect ratio enclosure heated by absorption of radiation. By *Amber et al.*
	16:58	-	17:00	AP38/39*	Exp and num. investigation on Natural Convection in Horizontal Channels Partially Filled with Aluminium Foam and Heated from Below. By *Buonomo et al.*
	17:01	-	17:03	AP40	Comparison of temperature and acoustic monitoring of ice pig passage. By *Lucas et al.*
	17:04	-	17:06	AP41	Feasibility Analysis of Molten-Salt Direct Reactor Auxiliary Cooling System. By *Le Brun et al*.
	17:07	-	17:09	AP42	III-V multi-junction cell temperature prediction under concentration and realistic atmospheric conditions. By *Theristis et al.*
	17:10	-	17:12	AP43	Identifying Thermal Performance of Two Heat Exchangers for Thermoelectric Generators with CFD. By *Li et al.*
	17:13	-	17:15	AP44	Heat Transfer Enhancement Using Partly Porous Channelsby. By *Nebbali et al.*
	17:16	**-**	**18:00**	**Poster Viewing and Day 1 Announcements (Poster removal at 18:00)**	
	19:30			***Dinner***	

* These are poster-only contributions

Applications Session

Poster-Only Contributions, Monday 7th Sept 2015

Day 1: Monday, 7th Sept 2015 (POSTER-ONLY CONTRIBUTIONS)		
9:30-12:30 Phase Change Session Poster-Only Contributions	PC5*	CFD Modelling Of Low Pressure Evaporation. By *Ahmad et al.*
	PC6*	The effect of solid deposits on the wall temperature of a heated vessel boiling water at low pressure. By *Elsaye et al.*
	PC29*	Effect of Dead Volumes on Single Closed Loop Pulsating Heat Pipes. By *Creatini et al.*
	PC32*	Experimental study on in-tube convective condensation of inclined tubes in a single minichannel with HFE-7100 at low mass fluxes. By *Gallo et al.*
	PC36*	Numerical Simulation of a Sodium Thermosyphon. By *Manzoni et al.*
	PC42*	Numerical Simulation of Pool Boiling: The Effects of Initial Thermal Boundary Layer, Contact Angle and Wall Superheat. By *Georgoulas et al.*
9:30-12:30 Phase Change Session Poster-Only Contributions	AP17*	Experimental Study of Ionic Wind Heat Transfer Enhancement in Rectangular, Vertical Channels. By *Gallandat et al*.
	AP25*	Numerical Simulation of a Capillary Pulsating Heat Pipe in Various Gravity Conditions. By *Manzoni et al*.
	AP32*	Experimental and numerical study on heat transfer and pressure loss in concentric tube heat exchanger with inserts placed on coolant side. *By Abbasi et al.*
	AP39*	Numerical Investigation on a Latent Thermal Energy Storage with Aluminum Foam. By *Buonomo et al.*

Next Page for Tuesday's (8th Sept 2015) Programme

* These are poster-only contributions

Programme Schedule

Day 2: Tuesday, 8th Sept 2015					
	07:30			*Poster presenters set up their posters*	
Keynote5	**09:00**	**-**	**09:30**	**Keynote 5**	**Dr. Prashant Valluri (*The University of Edinburgh, UK*): Ultra-high resolution 3D direct numerical simulations for phase change applications**
9:30-10:30 Poster Presentations *Modelling* *Session Chair: Y. C. Lee*	09:30	-	09:32	M1	Modelling of Polymer Plate Heat Exchangers. By *Wadekar*
	09:33	-	09:35	M2	A reduced numerical model for counter-current two-layer flows. By *Lavalle et al.*
	09:36	-	09:38	M3	On the generation of nonlinear 3D interfacial waves in gas-liquid flows. By *Naraigh et al.*
	09:39	-	09:41	M4	Heat transfer in falling liquid films at moderate Reynolds and high Peclet numbers. By *Ruyer-Quil et al.*
	09:42	-	09:44	M5	Numercial investigation of a two stage travelling-wave thermoacoustic engine driven heat pump with a hybrid configuration. By *Al-Kayiem.*
	09:45	-	09:47	M6	Fluid Flow & Heat Transfer Modelling of Adjacent Synthetic Jets. By *Alimohammadi et al.*
	09:48	-	09:50	M7	Numerical simulation of Heat Transfer Investigation in New Cooling Schemes of a Stationary Blade Trailing Edge. By *Beniaiche et al.*
	09:51	-	09:53	M8	Radiation Effect on Thermal Boundary Layer Flow past a Stretching Plate with Variable Thermal Conductivity. By *Ravins et al.*
	09:54	-	09:56	M9	SPRS – A Passively Cooled Sellafield Store. By *Moorcroft et al.*
	09:57	-	09:59	M10	Mixed Convection Boundary-Layer Flow near a Stagnation Point towards a Stretching/Shrinking surface in a Nanofluid. By *Yacob et al.*
	10:00	-	10:02	M11	Numerical Study on Heat Transfer in Wavy Annular Gas-Liquid Flow. By *Yang et al.*
	10:03	-	10:05	M12	Developing a scalable and flexible high-resolution DNS code for two-phase flows. By *Bethune et al.*
	10:06	-	10:08	M13	ALE-FEM for two-phase flows with heat and mass transfer. By *Anjos et al.*
	10:09	-	10:11	M14	Large Eddy & Interface Simulation (LEIS) of Disturbance Waves and Heat Transfer in Annular Flows. By *Yang et al.*
	10:12	-	10:14	M15	Impact of Fouling on Thermodynamics Performance of Heat Exchanger: A Computational Fluid Dynamics Study. By *Yang et al.*
	10:15	-	10:17	M16	Flow analysis in three-dimensional double-diffusive convection in an elongated porous enclosure. By *Mimouni et al.*
	10:18	-	10:20	M17	Identification of a Position and Time Dependent Heat Flux Using the Unscented Kalman Filter in 3D Nonlinear Heat Conduction. By *Pacheco et al.*
	10:21	-	10:23	M18	LSA of fingering in convective dissolution in porous media. By *Lucena et al.*
	10:24	-	10:26	M19	Application of Adomian Decomposition Method for a Stepped Fin Space Radiator with Internal Heat Generation. By *Singla et al.*
	10:27	-	10:29	M20	Heat Transfer Rate from Hydrogen to Tank Wall during Fast Refuelling Process. By *Monde et al.*
Break	**10:30**	**-**	**11:00**	**Coffee Break + Poster Viewing**	
Keynote6	**11:00**	**-**	**11:30**	**Keynote 6**	**Dr. Francesco Coletti (*Hexxcell Ltd, UK*): Fouling – Is it Still the Major Unresolved Problem in Heat Transfer?**
11:30-12:30 Poster Presentations *Modelling* *Session Chair: P. Valluri*	11:30	-	11:32	M21	Simulation of Droplet Heated by Laser for PCR Application. By *Wang et al.*
	11:33	-	11:35	M22	Three dimensional simulation of a focused infrared laser heated droplet in microchannels. By *Chen et al.*
	11:36	-	11:38	M23	A Computational Study of Thermal Losses in a Reciprocating Piston-Cylinder System. By *Taleb et al.*
	11:39	-	11:41	M24	Numerical and Experimental Investigation of Pulsating Flow for Fabric Drying Application. By *Zhao et al.*
	11:42	-	11:44	M25	Numerical Investigation on an Inclined Ventilated Roof with Different Exit Section. By *Buonomo et al.*
	11:45	-	11:47	M26	A Comment on Modelling and Analysis of Plasma Gasification as an Emerging Technology for Waste to Energy. By *Carpinlioglu et al.*
	11:48	-	11:50	M27	Preliminary CFD Analysis of Natural Convection Fuel Tubes in Molten Salt Nuclear Reactors. By *Cioncolini et al.*

* These are poster-only contributions

Programme Schedule

Modelling Session

11:30-12:30 Poster Presentations *Modelling* ***Session Chair: P. Valluri***	11:51	-	11:53	M28	A three-dimensional numerical model of free convection in a tilted porous cavity. By *Guerrero et al.*
	11:54	-	11:56	M29	Simulating Heat and Mass Transfer in an Aggregate Dryer Using Coupled CFD and DEM. By *Hobbs*
	11:57	-	11:59	M30	CFD Modelling of Thermal Management in Downhole Tools. By *Hughes et al.*
	12:00	-	12:02	M31	Solving Direct and Inverse Nonlinear Heat Conduction Problems by Means of Trefftz Functions and Kirchhoff Transformation. By *Maciag*
	12:03	-	12:05	M32	Modelling the Free-Surface Turbulent Flow and Heat Transfer in an Unbaffled Vessel Agitated by a Pitched Three-Blade Turbine. By *Mahmud et al.*
	12:06	-	12:08	M33	Modeling the effect of thermotherapy on the inner layer of the bladder. By *Sadee et al.*
	12:09	-	12:11	M34	Decoupling of thermo-physical properties of glycol-water mixtures: insight from nano-scale simulation. By *Cannon et al.*
UKHTC Ends	**12:12**	**-**	**12:30**	**Closing Remarks**	
	12:30	**-**	**13:45**	**Lunch Break + Posters Switchover for invitation-only Thermapower event**	

UKHTC Conference Close & (Invitation-Only) THERMAPOWER (EU Funded) Session Begins

* These are poster-only contributions

14th UK Heat Transfer Conference 2015
Edinburgh (Sept 7 – Sept 8, 2015)

Keynote 1

Flow Boiling in Microchannels

T. G. Karayiannis[1*] and M. M. Mahmoud[1,2]

[1] *Brunel University London, Kingston Lane, Uxbridge, Middlesex, UB8 3PH, UK,*
[2] *Zagazig University, Zagazig, Sharqia, 44519, Egypt,*

Extended Abstract

The rapid advances in performance and miniaturization of electronics and high power devices resulted in huge heat flux values that need to be dissipated effectively. Flow boiling in microchannels is one of the most promising cooling methods for these devices due to the capability of achieving very high heat transfer rates with small variations in the surface temperature. However, several fundamental issues are still not understood and this hinders the transition from the laboratory scale to commercial applications. The present paper starts with a discussion on the possible applications of flow boiling in microchannels in order to highlight the challenges in the thermal management for each application. Then, the paper presents experimental research on flow boiling in single tubes and rectangular multichannels to discuss the following fundamental issues: (1) the definition of microchannel, (2) flow patterns and heat transfer mechanisms, (3) flow instability, (4) effect of channel surface characteristics, (5) prediction of critical heat flux. In addition, a recommendation for the prediction of the flow pattern transition boundaries and heat transfer coefficients in small to mini/micro diameter tubes is also presented. The major concluding points are summarized at the end of the paper.

CFD Modelling of Low Pressure Evaporation

W. Ahmad[1], D.A. McNeil[2], T. Wylie[3]

[1] *National Nuclear Laboratory, 5th Floor Chadwick House, Warrington, WA3 6AE, United Kingdom, waqas.ahmad@nnl.co.uk*
[2] *Room 2.31 James Naysmth Building Heriot-Watt University, Edinburgh, EH14 4AS, United Kingdom, d.a.mcneil@hw.ac.uk*
[3] *Sellafield Limited,WEDD Technical & Strategy, B582 GFN, United Kingdom, tina.l.wylie@sellafieldsites.com*

Extended Abstract

The Heriot-Watt Evaporator Rig (HWER) has been designed, built and commissioned at Heriot-Watt University in Edinburgh. The aim of the rig is to represent key physical phenomena inside a steam heated, low pressure (100mbar), kettle type evaporator. Phenomena such as boiling heat transfer, natural convection heat transfer, solids behaviour, flow patterns under low pressures and unconfined flows.

The HWER consists of a test section (1m height) which is analogous in terms of geometry to a scaled slice through a kettle type evaporator and two upper sections which extend the height of the rig to 2.2m. The test section contains two banks of brass tubes (5mm thick), arranged in a three by six grid on each side of the centre line. These tubes represent helical coils inside the evaporator. Heating is provided by electrical rod heaters inside the tubes and strip heaters along the stainless steel side-wall (5mm thick). The side-wall heating represents the jacket around the outside of the evaporator. The rig is fitted with a 57 thermocouples located within the bulk fluid, near the walls, within the tubes and side-wall.

A number of experimental programmes have been completed to examine the behaviour of the rig under a range of heating conditions and at low operating pressures (50 mbar at the free surface) with 2m head of water. Experiments were also carried out with 0.5mm diameter Ballotini added to the flow to study the behaviour of the rig with representative solids found in kettle type evaporators in the nuclear sector. Data analysis and comparisons are made between the runs carried out with the fluid only and those with the solids.

A range of correlations exist for the calculation of nucleate boiling heat transfer coefficients on heated surfaces e.g. Coopers, Forster-Zuber and Gorenflo. This paper also presents a comparison between these existing correlations and measured boiling heat transfer coefficients from the experiments and evaluates them for suitably representing nucleate boiling heat transfer from walls inside a kettle type evaporator under low pressure conditions in the nuclear sector.

14th UK Heat Transfer Conference 2015
Edinburgh (Sept 7 – Sept 8, 2015)

Novel CFD Methods to Predict Evaporation of Water under a Vacuum

J. S. Panesar[1,*], A. D. Burns[1], and P. J. Heggs[1]

[1]School of Chemical and Process Engineering, faculty of Engineering, University of Leeds, UK, LS2 9JT
*pm11jsp@leeds.ac.uk

Relatively little literature is available for evaporating liquids at sub-atmospheric pressures compared to evaporation at atmospheric pressures and above. There is a demand to better understand sub-atmospheric evaporation for use in industrial processes. Recently experimental work has been performed by McNeil *et al.*, (2015) of boiling water at 50 mbar pressure in an experimental test rig representing a one-quarter scale industrial evaporator.

The commercial CFD code Ansys CFX 15 is used to perform Eulerian-Eulerian phase change simulations of the experimental test rig, and using the experimental data to validate novel CFD models to predict thermal phase change. Two experimental test cases are simulated: a pool height at approximately 0.8 m, and another at approximately 2 m. In both cases the pressure above the pool is set to 50 mbar. The computational domain is 0.75 m wide and 98 mm deep. Heat is supplied by 36 heated tubes set to a heat flux of 65 kW/m^2. During evaporation water is replenished via a feed pipe above the free surface to maintain continuity.

Water and vapour are both treated as continuous phases, separated by a free surface. The interfacial transfer of momentum, heat and mass is directly dependent on the contact surface area between the two phases per unit volume, known as the interfacial area density. The CFX mixture model is used to predict the interfacial area density. A user defined length scale is required as an input parameter into the formulation of the mixture model. In the literature no such length scale exists for evaporating liquids at any pressure. In this work an expression to predict the interfacial length scale has been derived analytically, and is highly dependent on an evaporating rate constant, β which has units of s^{-1}. Determining the optimum value of β is crucial to correctly model evaporation as seen in the experimental data.

A range of values are tested for β (10^{-4} to 10^4 s^{-1}) where the simulation data are compared to the experimental data. All values of β are able to predict the experimental stream temperatures within the correct order of magnitude. Values of β tending to 10^{-4} s^{-1} has the effect of limiting the interfacial heat and mass transfer rates, giving rise to unphysical areas of superheat in the liquid, and large and small sporadic evaporation rates. Values of β tending to 10^4 s^{-1} provides a more physically realistic solution, predicting stream temperatures close to the free surface saturation temperature which is also reported in the experimental investigation. The evaporation rates are also within the expected order of magnitude. There is confidence that the ongoing investigation will identify the optimum value of β which will provide the correct evaporation rates and stream temperatures.

References

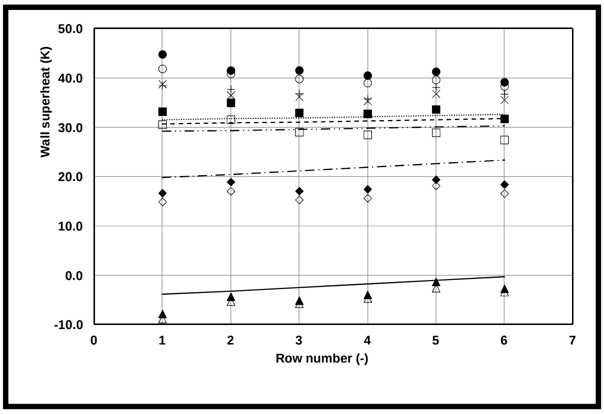
McNeil, D., Burnside, B., Rylatt, D., Elsaye, E., & Baker, S. (2015). Shell-side boiling of water at sub-atmospheric pressures. *International Journal of Heat and Mass Transfer, 85*, 488–504.

Shell-side boiling of a glycerol-water mixture at low sub-atmospheric pressures

D A McNeil[1], B M Burnside[1], E A Elsaye[1], S M Salem[1] and S Baker[2]

[1] *School of Engineering and Physical Science, Heriot-Watt University*
Riccarton, Edinburgh EH14 4AS, UK, D.A.McNeil@hw.ac.uk
[2] *National Nuclear Laboratory, 5th Floor, Chadwick House, Risley, Warrington, WA3 6AE, UK, stephen.r.baker@nnl.co.uk*

Extended Abstract



Experimental data are reported for boiling water and a glycerol-water mixture at a free surface pressure of 50 mbar absolute on the shell-side of a thin slice model of an industrial boiler. The boiler test section was 1 m high, 0.75 m wide, 98 mm long and contained 36 electrically heated, horizontal tubes that were 28.5 mm in diameter. The design of the boiler ensured that the tubes were submerged in a liquid pool. The height of the liquid pool was set to 2 m, submerging the top of the tube bundle by 1.6 m of liquid. The heat flux was varied within the range 10-65 kW/m^2. A near-symmetrical half of the tube bundle contained wall thermocouples. An additional 29 thermocouples were located throughout the liquid pool.

For both fluids, the liquid temperature in the pool was found to be reasonably uniform and controlled by the pressure at the free surface. This led to subcoolings of up to 31 K at the tube surfaces. The reasonably uniform pool temperature suggests that the liquid re-circulates within it.

For water, boiling was initiated at a heat flux within the range 25-40 kW/m^2, whereas boiled was initiated for glycerol-water mixture at a heat flux in the range 10-25 kW/m^2. Below these heat flux ranges, both fluids were in natural convection, with the measured wall superheats in reasonable agreement with predictions from a correlation available in the open literature[1]. The difference in the fluids' boiling onset heat fluxes resulted from the natural convection, heat-transfer coefficients of the glycerol-water mixture being lower than the water values. The boiling wall superheats for water were reasonably well predicted from a correlation available in the open literature[2]. The mixture boiling superheats were reasonably well predicted by a more recent form of this correlation, provided that the mixture effects were accounted for appropriately[3]. Typical mixture results are shown in the figure.

References

CHURCHILL, S.W. & CHU, H.H.S. 1975, *Correlating equations for laminar and turbulent free convection from a vertical plate*, Int. J. Heat Mass Transfer, Vol. 18, pp. 1323-1329.

GORENFLO, D. 1993, *Pool boiling*, VDI-Heat Atlas, VDI-Verlag, Dusseldorf.

UNAL, H.C. 1986, *Prediction of nucleate pool boiling heat transfer coefficients for binary mixtures*, Int. J. Heat Mass Transfer, Vol. 29, pp. 637-640.

Implementing A Multi-Disciplinary Strategy To Understand Heat Transfer in a Reduced Pressure Highly Active Evaporator.

Richard Jarvis[1], Stephen R Baker[1] , Waqas Ahmad[1]

and Stuart Beresford-Kelly[2]

[1] National Nuclear Laboratory, 5th Floor Chadwick House, Warrington, WA3 6AE, United Kingdom,
[2] Sellafield Limited

Extended Abstract

In the UK Nuclear industry kettle-type evaporators concentrate the radioactivity removed from nuclear fuel as part of fuel recycling. The resulting concentrate is then converted into long-lived glass blocks for storage and subsequent disposal.

The liquor processed in these evaporators is corrosive to the stainless steel components it makes contact with. The corrosion rate is strongly dependent on temperature. Therefore to extend the operating life of the evaporators, they are operated at low pressure to reduce the temperature of the heated surfaces. However even at these reduced temperatures the heated surfaces slowly thin.

The evaporators have two types of heating surfaces. Internal coils, which are inspected annually to ensure acceptable thickness remains and an external jacket, which has not been inspected to date. The jacket thickness is therefore calculated. In order to ensure safe operation, these calculations are pessimistic. As such the jacket will be thicker than estimated. The jacket heating profile is known to vary with:

- **Depth in the evaporator** – lower parts of the shell will boil at higher temperatures because of the increased pressure exerted by the liquor
- **Settled solids** – solids are formed during evaporation and, if they settle, lead to local heating and so increased temperatures under the solids
- **Time through a batch** – as the liquor becomes more concentrated its physical properties such as density, heat capacity and viscosity change and so affect metal surface temperatures
- **How heat is transferred from the jacket to the liquor** – boiling can be reduced, or suppressed altogether, by recirculation of liquor over the heated surfaces

Understanding all of these allows surface temperatures, and thus jacket thickness, to be calculated.

To extend evaporator life an integrated strategy using a variety of rigs, modelling, data analysis and engineering calculations has been developed. This strategy has been developed over a number of years to identify actions which will be used to fully understand the temperatures and hence corrosion in an evaporator jacket. The work undertaken to improve estimates of jacket thickness comprises:

- Using a scaled low pressure water rig at Heriot-Watt University to visualize and understand how flow suppresses boiling in the evaporator
- Using an offscale rig processing chemical simulants of the corrosive liquor
- Carrying out modelling and engineering calculations to link the rigs together and to a full scale evaporator.
- Interpreting measured coil data to infer liquor corrosivity

The work has integrated the results to understand a very complex system and is now supporting an improved estimate of evaporator corrosion based on understanding of the processes that influence metal temperature in the evaporator.

CFD Modelling Of Low Pressure Evaporation

W. Ahmad[1], A. Anderson[2], D.A. McNeil[3], T. Wylie[4], S.R. Baker[5], M.J. Moorcroft[6]

[1] National Nuclear Laboratory, 5th Floor Chadwick House, Warrington, WA3 6AE, United Kingdom, waqas.ahmad@nnl.co.uk
[2] ANSYS UK ltd, Sheffield Business Park, 6 Europa View, Shellfield, S9 1XH, United Kingdom, adam.anderson@ansys.com
[3]Room 2.31 James Naysmth Building Heriot-Watt University, Edinburgh, EH14 4AS, United Kingdom, d.a.mcneil@hw.ac.uk
[4]Sellafield Limited,WEDD Technical & Strategy, B582 GFN, United Kingdom, tina.l.wylie@sellafieldsites.com
[5] National Nuclear Laboratory, 5th Floor Chadwick House, Warrington, WA3 6AE, United Kingdom, stephen.r.baker@nnl.co.uk,
[6] National Nuclear Laboratory, 5th Floor Chadwick House, Warrington, WA3 6AE, United Kingdom, matthew.moorcroft@nnl.co.uk

Extended Abstract

The Heriot-Watt Evaporator Rig (HWER) has been designed, built and commissioned at Heriot-Watt University in Edinburgh. The aim of the rig is to represent key physical phenomena inside a steam heated, low pressure (100mbar), kettle type evaporator, on a larger scale than previously possible. Phenomena such as boiling heat transfer, natural convection heat transfer and flow patterns under low pressures and mostly unconfined flows.

The HWER consists of a test section (1m height) which is analogous in terms of geometry to a scaled slice through a kettle type evaporator and two upper sections which extend the height of the rig to 2.2m. The test section contains two banks of brass tubes (each 5mm thick), arranged in a three by six grid on each side of the centre line. These tubes represent helical coils inside the evaporator. Heating is provided by electrical rod heaters inside the tubes and strip heaters along the stainless steel side-wall (5mm thick). The side-wall heating represents the jacket around the outside of the evaporator. The operating fluid was de-ionized water for all work reported here. The rig is fitted with a 57 thermocouples located within the bulk fluid and near the walls and within the tubes and side-wall.

CFD simulations of the rig under different heating arrangements and at two different fluid levels ‘low’ (0.8m head) and ‘high’ (2m head) have been carried out. A ‘surface boiling model’ has been specially developed to replicate the heat transfer and fluid flow behaviour within low pressure evaporators, based on established boiling theory and observations from tests on the HWER. The Cooper correlation is used to model heat transfer due to nucleate boiling from the walls.

Results from the simulations (CFD) have been compared against experimental results (Rig Data). These comparisons show that although the surface boiling model works well in representing the overall flow behavior in the HWER, specific aspects of the model need further development. At the ‘low’ fluid level, examining the tubes, Fig. 1, comparing the Rig Data and CFD results reveals that the Cooper correlation under predicts tube surface temperatures. The differences are smallest at the top of the tube bank (1, 2 and 3) and increase to a maximum near the bottom rows of tubes (16, 17 and 18). Examining the side-wall surface temperatures at various locations, Fig. 2, indicates that the Cooper correlation significantly over predicts metal surface temperatures.

Fluid temperatures in and around the tube banks and near the side-wall compare well. This indicates that the surface boiling model itself is performing well in representing the overall behavior of the evaporator. However further work is needed to better represent the heat transfer near the walls.

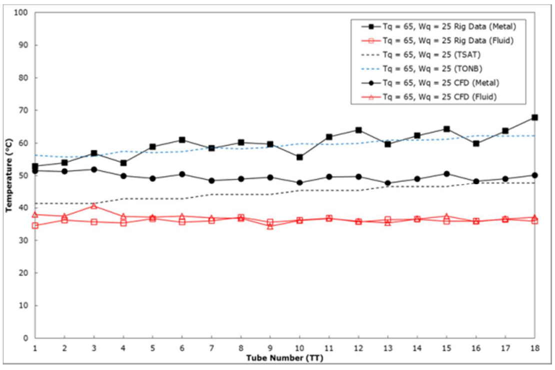


Fig 1: Comparison of tube(s) temperatures between measured and analysis results (low level)

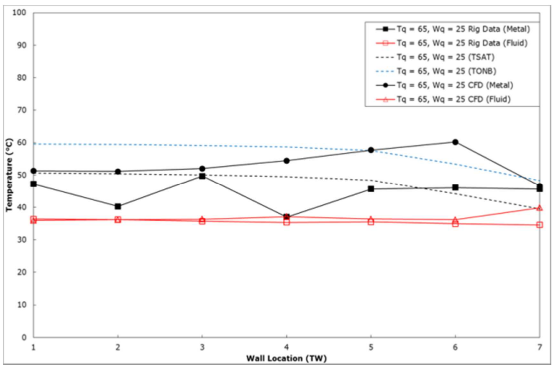


Fig 2: Comparison of wall surface temperatures between measured and analysis results (low level)

The effect of solid deposits on the wall temperature of a heated vessel boiling water at low pressure

I. ELSAYE[1], D. A. McNeil[1], D. I. Rylatt[1] and S. Baker[2]

[1] *School of Engineering and Physical Sciences, Edinburgh, UK, eae30@hw.ac.uk*
[2] *National Nuclear Laboratory,5th Floor Chaswick House, Risley, Warrington, UK, stephen.r.baker@nnl.co.uk*

Abstract

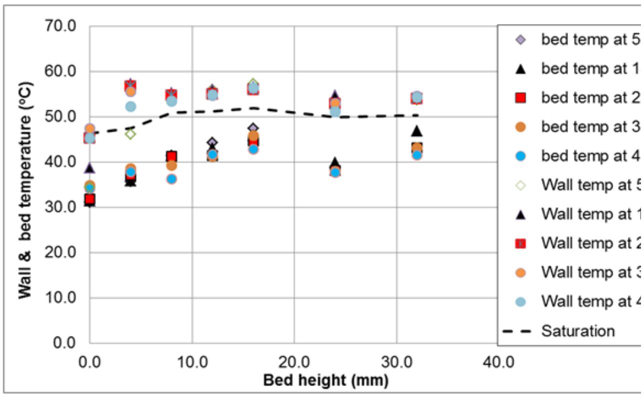


Fig 1: Variation of temperature with bed height

Solids can come out of solution when some process fluids are evaporated. These solids can form a bed of particles on the heated base of the evaporator vessel. The effect of increasing the bed depth on base temperature is experimentaly investigated for water boiling at a pressure of 50 mbar absolute. The bed depth is varied from 0-32 mm using glass particles 500-600 µm in diameter. The evaporator used was a model industrial boiler slice. The boiler test section was 1 m high, 0.75 m wide and 98 mm long. The evaporator contained 36 electrically heated tubes to simulate the presence of the heated coils. The tubes were 28.5 mm in diameter. The design of the boiler ensured that the tubes were submerged in the liquid pool. The tube heat flux was maintained at 65 kW/m^2 and the wall heat flux varied within the range 0-45 kW/m^2.

Outwith the solid bed, the liquid temperature in the liquid pool is shown to be reasonably constant and close to the free surface saturation temperature. This indicates that fluid recirculation is taking place, with fluid flashing to the saturation temperature at the free surface before returning to the depths of the pool. The liquid temperature within to the solid bed is shown to be greater than that in the pool and to decrease with increasing wall heat flux.

The wall temperature of the base is shown to be subcooled in the absence of a solids. The presence of the bed induces boiling at most conditions, indicating that a strong convection current normally cools the base and that the base is insulated from this cooling by the bed. The bed wall temperatures remain reasonably constant as the heat flux increases. However, the bed wall superheat increases with increasing bed depth until a bed depth of 16 mm, decreases at 24 mm and increase again at 32 mm, possibly as a results of the bed partially fluidising[1].

Bubble sizes are estimated form photograophic images. These show that the bubbles formed within the bed increase in size with increasing bed depth and heat flux.

References

SHI, M., ZHAO, Y., & LIU, Z. 2003 *Study on boiling heat transfer in liquid saturated particle bed and fluidized bed*. I. Journal of Heat and Mass Transfer. **46**, 4695-4702.

Ambient pressure effect on nanofluid sessile droplet evaporation

A. Askounis[1,2], Y. Takata[1,2], K. Sefiane[2,3], and M.E.R. Shanahan[4]

[1]*International Institute for Carbon-Neutral Energy Research (WPI-I2CNER), Kyushu University, 744 Motooka, Nishi-ku, Fukuoka 819-0395, Japan, a.askounis@i2cner.kyushu-u.ac.jp*
[2]*Department of Mechanical Engineering, Thermofluid Physics Laboratory, Kyushu University, 744 Motooka, Nishi-ku, Fukuoka 819-0395, Japan, takata@mech.kyushu-u.ac.jp*
[3]*Institute for Materials and Processes, School of Engineering, The University of Edinburgh, King's Buildings, Mayfield Road, Edinburgh, EH9 3JL, United Kingdom, k.sefiane@ed.ac.uk*
[4]*I2M, Univ. Bordeaux, UMR 5295, F-33400 Talence, France, martin.shanahan@u-bordeaux.fr*

Extended Abstract

Addition of nanoparticles in base fluids, e.g. water, ethanol, greatly enhances the heat transfer properties of the fluids, leading to potential applications in various fields [1]. However, the exact evaporation mechanism of these "nanofluids" remains elusive [2,3]. Experimental investigation of the evaporation of nanofluid droplets under different ambient pressures yielded interesting information. Free evaporation of such a droplet followed the well-known stick-slip regime. Further investigations with decreasing ambient pressure, resulted in the first, to the best of our knowledge, report of the exponential relationship between ambient pressure and evaporation rate for nanofluids (Fig. 1). Additionally, lowering ambient pressure resulted in a transition from the stick-slip evaporative regime to the constant pinning one. Complementing these experimental observations with theoretical arguments allowed quantification of the exact effect of ambient pressure on the convective, outwards fluid flow, which, in turn, deposits more particles to the contact line and hence allows stronger droplet pinning (Fig. 2). Last but not least, the hysteretic energy barrier pinning the droplet in each case was calcualted.

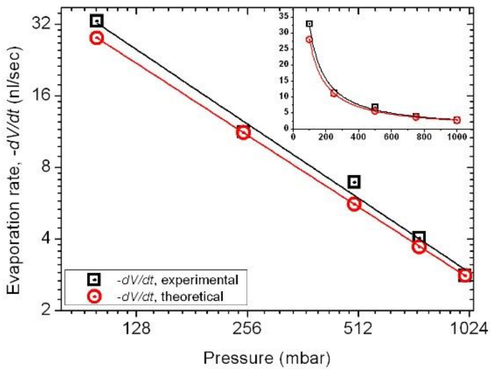


Fig 1: Evaporation rate and ambient pressure relationship.

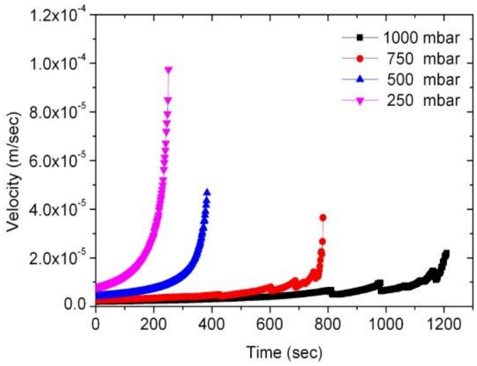


Fig 2: Ambient pressure effect on liquid flow velocity.

References

[1] Wong, V.K., and Leon O. De 2010 *Applications of nanofluids: current and future*, Advances in Mechanical Engineering, **2010**, 519659

[2] Sefiane, K. 2006 *On the role of structural disjoining pressure and contact line pinning in critical heat flux enhancement during boiling of nanofluids*, Appl. Phys. Lett., **89**, 044106

[3] Deegan, R.D., Bakajin, O., Dupont, T. F., Huber, G., Nagel, S. R. and T. A. Witten 1997 *Capillary flow as the cause of ring stains from dried liquid drops*, Nature, **389**, 827–829

Leidenfrost vitrification of droplets with liquid nitrogen

G. Duursma[1],V. Müller[2] and K. Sefiane[3]

[1] *School of Engineering, The University of Edinburgh, UK, Gail.Duursma@ed.ac.uk*
[2] *School of Engineering, The University of Edinburgh, UK, viona.mueller@gmx.de*
[3] *School of Engineering, The University of Edinburgh, UK, K.Sefiane@ed.ac.uk*

Extended Abstract

Many biofluids require precision cooling and freezing to avoid cell damage. This so-called vitrification process needs to be well controlled. Droplets of fluids may be released above liquid nitrogen in order to be vitrified. Film boiling of nitrogen then occurs because of the large excess temperature (temperature of biofluid less that of nitrogen) and this causes the droplet to levitate according to the well-known Leidenfrost effect, whilst being cooled rapidly. Since the bulk nitrogen is liquid, there is a free surface which is deformed near the droplet. This free boundary heat transfer problem is a complicated one to model and to study experimentally. Song *et al.* (2010) were the first to work on vitrification of droplets released directly into nitrogen. They performed vitrification experiments on droplets of propanediol and modelled a spherically symmetric droplet with convective heat transfer boundary condition at the droplet surface, as a first approximation to the real scenario.

Figure 1 Vitrified 1%EG droplet

In our work droplets of fluid (water, mixtures of water-ethylene glycol, and aqueous nanofluids) were vitrified above liquid nitrogen and the process was digitally video-recorded. Ethylene glycol (EG) is sometimes used as a cyroprotectant and hence is important to study. In Fig 1 a vitrified droplet of a water-EG mixture is shown. Droplet sizes were also varied in the experimental programme.

The vitrification process is not always smooth. Some drops shatter in freezing and a reliable protocol needs to be developed and followed.

Levitation times were also measured from the recordings. For water-EG mixtures, the addition of ethylene glycol reduced the levitaion times, with the reduction clearly increasing with increasing EG concentration. Results for nanofluids were not as clear as those for water-EG mixtures, except for larger droplets.

References

SONG, Y., ADLER, D., XU, F., KAYAALP, E., NUREDDIN, A., ANCHAN, R., MAAS, R., & DEMIRCI, U. 2010 *Vitrification and levitation of a liquid droplet on liquid nitrogen.* PNAS **107(10)**, 4596-4600.

CONDENSATION OF R134A AT LOW MASS FLUXES IN A SMOOTH TUBE AT DIFFERENT INCLINATION ANGLES

Daniel. R. E. Ewim[1] and Josua. P. Meyer[2]

Department of Mechanical and Aeronautical Engineering,
University of Pretoria, Pretoria, 0002,
South Africa.
Josua.Meyer@up.ac.za
Daniel.Ewim@up.ac.za

Extended Abstract

This paper presents the heat transfer coefficients and flow pattern results during the condensation of R134a in a smooth tube at different inclination angles. The test section was 8.38 mm in diameter with a length of 1.49 m. Condensation experiments were conducted at different inclination angles from -90° (vertical downwards) to +90° (vertical upwards) with low mass fluxes between 20 kg/m^{2}s and 100 kg/m^{2}s at various mean vapour qualities. The average saturation temperature was 40°C. The flow patterns were recorded with a high speed video camera at the inlet and outlet of the test section through sight glasses. The results show that the flow was stratified and that inclination angles and vapour qualities strongly influenced the heat transfer coefficient. Finally, it was found that the heat transfer coefficient highly dependent on the temperature difference.

References

Cavallini, A., Censi, G., Del Col, D., Doretti, L., Longo, G. A., Rossetto, L., and Zilio, C., "Condensation inside and outside smooth and enhanced tubes—A review of recent research," *Int. J. Refrig.*, **26**, pp. 373–392, (2003)..

Lips, S., Meyer, P.J., "Experimental study of convective condensation in an inclined smooth tube. Part 1: Inclination effect on flow pattern and heat transfer coefficient," *Int. J. Heat Mass Transf.*, **55**, pp. 395 - 404, (2012a).

Lips, S., Meyer, P.J., "Experimental study of convective condensation in an inclined smooth tube. Part II: Inclination effect on pressure drops and void fractions," *Int. J. Heat Mass Transf.*, **55**, pp. 405 - 412, (2012b).

Lips, S., Meyer, P.J., "Stratified flow model for convective condensation in an inclined tube," *Int. J. Heat Fluid Flow,* **36**, pp. 83 - 91, (2012c).

Adelaja, A.O., Dirker, J., Meyer, J.P., "Condensing heat transfer coefficients for R134a at different saturation temperatures in inclined tubes," *Proceedings of the ASME2013 Summer Heat Transfer Conference* (HT 2013 - 17375), Minneapolis, MN, USA, 2013, pp. 1- 9, (2013).

Meyer, J.P., Dirker, J., Adelaja, A.O. "Condensation heat transfer in smooth inclined tubes for R134a at different saturation temperatures," *Int. J. Heat Mass Transf.*, **70**, pp. 515-525, (2014).

Schlager, L. M., Pate, M. B., and Bergles, A. E., "Heat transfer and pressure drop during evaporation and condensation of R22 in horizontal micro-fin tubes," *Int. J. Refrig.*, **12**, pp. 6 – 14, (1989)

Chemically Treated Micropillars for Enhanced Condensation Heat Transfer

D. Orejon*[1,2], O. Shardt[3,4], P. R. Waghmare[3], N. S. K. Gunda[5], S. K. Mitra[5] and Y. Takata[1,2]

[1] *International Institute for Carbon-Neutral Energy Research (WPI-I²CNER), 744 Motooka, Nishi-ku, Fukuoka 819-0385, Japan,* orejon.daniel@heat.mech.kyushu-u.ac.jp & takata@mech.kyushu-u.ac.jp
[2] *Kyushu University, 744 Motooka, Fukuoka 819-0385, Japan,* orejon.daniel@heat.mech.kyushu-u.ac.jp & takata@mech.kyushu-u.ac.jp
[3] *University of Alberta, Edmonton, Alberta, T6G 2G8, Canada,* waghmare@ualberta.ca & oshardt@princeton.edu
[4] *Princeton University, E-Quad, Olden St., Princeton, NJ 08544, USA,* oshardt@princeton.edu
[5] *York University, 4700 Keele Street, Toronto, ON M3J 1P3, Canada* nagasiva@yorku.ca & mitras@yourku.ca

Extended Abstract

Cooling via dropwise condensation (DwC) has received special attention in recent years due to its enhanced performance when compared to filmiwise condensation (FC) [1]. For high efficiency heat transfer, condensation must occur with quick and continuous nucleation, growth and deparure of small droplets, on the order of tenths of micrometers. Challenges in the last step of the process, the self-removal of droplets remain to be addressed [2]. One of the major factors hindering droplet detachment is the adhesion of the condensate to the substrate, which is mainly influenced by surface's wettability and roughness.

In this work we present experimental observation of droplet condensation on silicion micropillar arrays using Environmental Scanning Electron Microscopy (ESEM). These pillared surfaces offer up to 250% more surface area for droplet nucleation than flat substrates. However nucleation and growth within the pillars is undesired since such droplets are difficult to detach. Different geometries, height, and spacing between pillars have been fabricated and studied aiming for high DwC efficiency. Furthremore we have also modified the wettability of the arrays by chemical treatments. Figure 1 shows important qualitative differences in condensation behaviour between non treated (hydrophilic) and chemically treated (hydrophobic) micropillars.

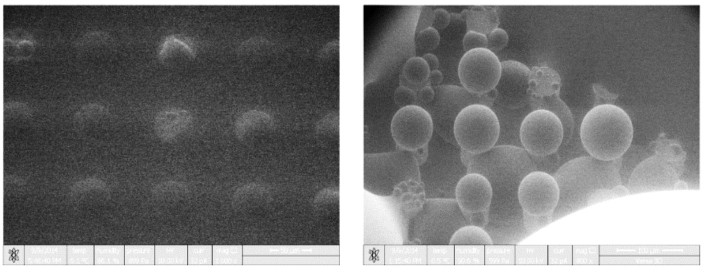


Fig 1: Condensation on (left) hydrophilic (FC) vs. (right) hydrophobic (DwC) micropillars.

A 1500% increase in condensate is reported for completely hydrophobic micropillars when compared to hydrophilic ones. Moreover an interesting phenomenon, the migration of droplets onto the pillar's top, is observed for chemically patterened micropillars, i.e. pillars with a hydrophilic top and hydrophobic sides on a hydrophobic substrate. These surfaces appear to be promissing for DwC due to the low energy of adhesion present between pillars.

References

[1] UMUR, A., & GRIFFITH, P. 1963 *Mechanism of Dropwise Condensation.* Department of Mechanical Engineering Massachusets Institute of Technology, Report No.l 9041-25.

[2]ATTINGER, D., BETZ, A. R., SCHUTZUIS, T. M., GANGULY, R., DAS, A., KIM, C. J., MEGARIDIS, C. M. 2014 *Surface Ingeneering for phase change heat transfer: A review.* MRS Energy & Sustainability **1**, 1.

Effect of channel orientation on the flow boiling heat transfer and pressure drop in a 1.1 mm diameter channel

E. A. Pike-Wilson[1] and T. G. Karayiannis[2]

[1] School of Computing, Engineering and Mathematics, University of Brighton, Brighton BN2 4GJ, United Kingdom, e.pike-wilson@brighton.ac.uk
[2] School of Engineering and Design, Brunel University London, Uxbridge, UB8 3P, United Kingdon, Tassos.Karayiannis@brunel.ac.uk

Extended Abstract

Flow patterns were seen to vary with channel orientation in larger scale channels (Gersey et al. 1995). However the effect of channel orientation was considered negliable in microscale channels (Kandlikar and Balasubramanian, 2004). This was due to the surface tension domintating over gravity as the channel diameter diminishes. Studies which have investigated the effect of channel diameter are commonly focused on a 90 º change in orientation, i.e. vertical to horizontal. Piasecka (2015) investigated the heat transfer coefficient at 0, 90 and 180 º and found a difference in the magnitude of the heat trasfner coffiecent between 0 and 180 °. A difference in the heat transfer coefficient and pressure drop with a 180 º change in orientation are likely to be the result of the effect of surface characteristics, which have already been seen to play an important role in flow boiling (Pike-Wilson et al., 2014).

Experiments were conducted using R245fa in a vertical 1.1 mm diameter stainless steel channel. The pressure and temperature of the fluid is measured at the inlet and outlet of the test section and the wall temperature at fourteen equidistant locations. Experiments were conducted at an inlet pressure of 1.85 bar and mass flux range of 100 – 400 kg/m^{2}s. Tests were conducted at orientations of 0 º and 180 º, effectively reversing the channel flow. A distinct peak was detected in the heat transfer coefficient, attributed to a suface defect at the 0 ° oreintation. At 180 º, a spike in the heat transfer coefficient was also evident at the location of the defect. The measured presssure drop was also seen to vary between the two channel orientations. These differences suggest that the location of the surface characteristics and how they interact with the flow are an important factor in both the heat transfer and pressure drop.

References

GERSEY, C. O., MUDAWAR, I. 1995 *Effects of heater length and orientation on the trigger mechanism for near-saturated flow boiling critical heat flux—I. photographic study and statistical characterization of the near-wall interfacial features*. International Journal of Heat and Mass Transfer, **38**(4), 629-641.

KANDLIKAR, S. G., BALASUBRAMANIAN, P. 2004 *An extension of the flow boiling correlation to transition, laminar, and deep laminar flows in minichannels and microchannels.* Heat Transfer Engineering, **25**(3), 86-93.

PIASECKA, M. 2015 *Correlations for flow boiling heat transfer in minichannels with various orientations.* International Journal of Heat and Mass Transfer, **81**, 114 – 121.

PIKE-WILSON, E. A., KARAYIANNIS, T. G. 2014 *Flow boiling of R245fa in 1.1 mm diameter stainless steel, brass and copper tubes.* Experimental Thermal and Fluid Science, **59**, 166 – 183.

The Effect of Brine Temperature and Salinity on the Rate of Electromagnetic Attenuation within it.

A. Hales[1] *et al.*

[1] *University of Bristol, Queens Building, University Walk, Bristol, BS8 1TR, UK, a.j.hales@bristol.ac.uk*

Extended Abstract

The attenuation rate, through brine, of electromagnetic radiation in the microwave bandwidth is affected by both temperature and salintiy. The rate's dependency on salinity is due to the polarisation of water molecules as NaCl is entirely disassociated in the solution, but it's dependency on temperature is more complex, and also appears to be a function of salinity. Documented literature[1] has concluded that attenuation rate decreases with temperature when the salinity is low (below 5%wt.), but increases with temperature when a larger quantity of salt is present . In this paper, the effect of temperature, close to the freezing points of various brine concentrations is quantified. Additionally, a polynomial function is developed, including a salinity factor, to model the attenuation rate of cooling brine of any salinity. Finally, the results are compared to equivalent data from a sucrose based aqueous solution, which shows attenuation rate is less dependent on the freezing point depressant concentration, and that it decreases with decreasing temperature.

References

[1] Chaplin, M. *Water and Microwaves*. www1.lsbu.ac.uk/water/microwave [Accessed: 16/09/14]

Thermocapillary convection for an evaporating meniscus with changing contact angle

C. Buffone

Microgravity Research Centre, Université libre de Bruxelles, Avenue F. Roosevelt 50, 1050 Bruxelles, Belgium, cbuffone@ulb.ac.be

Extended Abstract

The thermocapillary convection in the liquid phase of an evaporating meniscus interface in open air has been studied for varying imposed contact angles. Ethanol undergoes self-evaporation inside a capillary tube of borosilicate glass with internal diameter of 1mm. The evaporation is not uniform along the meniscus interface pinned at the capillary tube mouth and this generates a gradient of surface tension that is acknowledged to drive the vigorous Marangoni convection. In previous studies of this configuration the meniscus has mainly been concave (Buffone *et al.* 2005), and for this reason other researchers attributed the differential temperature along the meniscus to the fact that the meniscus wedge is closer to the tube mouth (Wang *et al.* 2008) and also further away from the warmer liquid bulk than the meniscus centre (Pan *et al.* 2011). The present study investigates concave, flat and convex menisci. Both flow visualization and infrared temperature measurements have been performed. For concave and convex meniscus the temperature measurements are in line with the predicted trend; the Marangoni vortices for these two menisci shapes spin in the same direction according to the temperature differences along the meniscus (as shown in Fig 1). For a flat meniscus, an intriguing evidence has been found: the temperature difference is inverted with respect to concave and convex menisci but surprisingly the Marangoni vortices spin in the same direction as for concave and convex menisci, as shown in Fig 1.

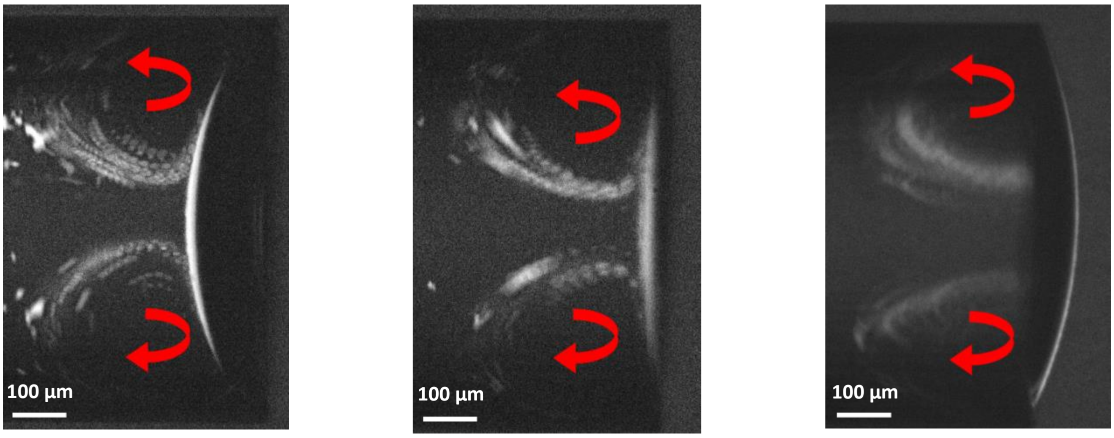


Fig 1: Marangoni flow structure for the three meniscus shapes. Red curved arrows indicate the spinning direction of the vortices.

References

BUFFONE, C., SEFIANE, K., CHRISTY, J.R. 2005 *Experimental investigation of self-induced thermocapillary convection for an evaporating meniscus in capillary tubes using micro–particle image velocimetry*. Phy. Fluids **17**, 052104

WANG, H., MURTHY, J.Y., GARIMELLA, S.V. 2008 *Transport from a volatile meniscus inside an open microtube*. Int. J. Heat Mass Transfer **51**, 3007-3017.

PAN, Z., WANG, F., WANG, H. 2011 *Instability of Marangoni toroidal convection in a microchannel and its relevance with the flowing direction*. Micro. Nano. **11**, 327-338.

Ice Pigging Coolant Jackets: Heat Transfer Into the Ice Pig Body

D.Mcbryde[1,2] et al.

[1,2] *Bristol University, University Walk, Clifton, Bristol, BS8 1TR, daniel.mcbryde@bristol.ac.uk*

Extended Abstract

This paper describes an investigation into the use of ice pigging to clean a coolant jacket from a large storage vessel; sediment deposited on the interior walls of the coolant jacket is limiting the heat transfer, by removing the sediment the heat transfer performance will be restored. The contents of the tank are at a constant 45°C; therefore there will be considerable heat gain into the ice pig body. For the ice pig to remain effective at removing sediment, the ice fraction ϕ_m needs remain higher than 43% at the exit of the cooling jacket.

A large scale model of the coolant jacket/storage tank arrangement was constructed, sharing the same depth and height as the real artefact, the length was reduced by ten times in order to fit in the laboratory. On the back of the coolant jacket, water from a heated storage tank was circulated to replicate the constant internal temperature of the storage tank. The coolant jacket was ice pigged whilst the inlet and outlet temperature of both fluid streams were measured; heat transfer calculations were completed to assess the heat gain to the ice pig and therefore the rate of ice melting.

Ice formation and production in subcooled environments

Xiao Yun et al

University of Bristol, Queen's Building, University Walk, BS8 1TR, Bristol, UK, xiao.yun@bristol.ac.uk

Extended Abstract

Industrial ice manufacture invariably consists of water being exposed to a solid surface which is held at a temperature below the freezing temperature of water. Ice production rate is controlled by the exposed cold surface area, the degree of subcooling, and the thickness of the ice. Increasing surface area and/or the degree of subcooling increases the ice production rate. As the ice thickness on the subcooled surface increases, it acts as an insulator, reducing the rate of formation, necessitating some form of removal of the ice to re-expose the subcooled surface. It is inherently difficult to achieve high rates of ice production in industrial environments with simple, easy to maintain equipment whilst achieving high coefficients of performance, COP. Keeping ice layers thin involves complex mechanical scrapers/ploughs or energy inefficient thermal cycling, using large degrees of subcooling results in poor COPs, and increasing the heat transfer area increases equipment bulk and costs.

An elegant, efficient method of ice production is proposed where ice is generated in a fluid which is below the freezing point of water. This is relatively easy to achieve when the heat sink fluid is immiscible with water, but considerably more difficult if the fluid is water. The paper presents experimental data where pure water is introduced into a chilled brine liquid. Heat transfer rates between the water and the brine are found to be higher than mass transfer rates and so ice is formed in the brine. The solid ice is separated from the liquid bulk fluid by a simple filter sieve arrangement, the active surface area where phase change occurs is very large, and the degree of subcooling can be relatively low. The preliminary and limited experimental work demonstrates that the system works. The data further suggests that the method promises to achieve higher COPs than current ice makers, to be much simpler in that it requires no complex mechanical harvesting equipment, and with the vast liquid-liquid surface areas possible, promises to be able to achieve very high ice production rates per unit volume of equipment.

References

QIN, F. G. F., CHEN, X. D., and RUSSELL, A. B. 2003 *Heat transfer at the subcooled-scraped surface with/without phase change.* AIChE J. vol. 49, no.8, 1947-1955

LAKHDAR, M. B., CERECERO, R., ALVAREZ, G., GUILPART, J., FLICK, D., and LALLEMAND, A. 2005 *Heat transfer with freezing in a scraped surface heat exchanger*. Applied Thermal Engineering, Info. vol. 25, no. 1, 45-60

BEDECARRATS, J.-P., DAVID, T., AND CASTAING-LASVIGNOTTES, J. 2010 *Ice slurry production using supercooling phenomenon.* International Journal of Refrigeration, vol. 33, no. 1, 196-204

Effect of extreme wetting scenarios on pool boiling

T. Valente[1], I. Malavasi[2], E. Teodori[3], A. S. Moita[4], M. Marengo[5] and A. L. N. Moreira[6]

[1]IN+ - Instituto Superior Técnico, Universidade de Lisboa, Av. Rovisco Pais 1049-001 Lisbon, Portugal, tomas.valente@tecnico.ulisboa.pt
[2] Dep. Eng. Appl. Science, University of Bergamo, Viale Marconi 5, 24044 Dalmine, Italy, ileana.malavasi@gmail.com
[3]IN+ - Instituto Superior Técnico, Universidade de Lisboa, Av. Rovisco Pais 1049-001 Lisbon, Portugal, e.teodori@dem.ist.utl.pt
[4]IN+ - Instituto Superior Técnico, Universidade de Lisboa, Av. Rovisco Pais 1049-001 Lisbon, Portugal, anamoita@dem.ist.utl.pt
[5]University of Brighton, School of Computing, Eng.and Mathematics, Lewes Road, BN2 4GJ Brighton, UK m.marengo@brighton.ac.uk
[6]IN+ - Instituto Superior Técnico, Universidade de Lisboa, Av. Rovisco Pais 1049-001 Lisbon, Portugal, moreira@dem.ist.utl.pt

Extended Abstract

Enhancement of pool boiling heat transfer is usually achieved by altering surface properties. Hydrophilic surfaces require larger superheat for the onset of boiling (ONB) but lead to higher Critical Heat Flux (CHF). Hydrophobic surfaces favor the activation of nucleation sites. Given these characteristics, patterned superhydrophilic/superphydrophobic surfaces are shown in recent literature, to offer the best pool boiling performance, but optimum patterns are not defined yet, as they require a better description of the governing processes. This work provides quantitative description of the boiling curves and nucleation processes on superhydrophobic surfaces, made from stainless steel and coated with Glaco. The surfaces are characterized based on the advancing and receding contact angles (ACA and RCA, respectively). Their topography is measured with a profilemeter meter (Veeco) with a vertical resolution of 200A and topographical homogeneity is checked by Laser Scanning Confocal Microscopy. The results show that despite the role of topography is dominant for hydrophilic surfaces, extreme changes in the contact angle lead to the most relevant differences in the boiling processes. Hence, as shown in Fig. 1, for the superhydrophobic surfaces, the ONB occurs almost at saturation temperature (1-2ºC superheat). As soon as the flux increases, bubbles coalesce and a stable vapor film forms over the surface, from which single large bubbles detach. The resulting peculiar curve is consistent with the early “quasi –Leidenfrost” regime, identified by Malavasi et al. (2015). Further increasing the flux while monitoring the surface temperature, one can identify the coating failure, which leads to a sudden variation in the boiling curve, as the surface energy changes and the liquid rewets the surface. Afterwards, the surface becomes biphilic and depicts the highest heat fluxes. Analysis of the surface structure suggests that one may obtain an optimum pattern (dimensions of hydrophilic/hydrophobic regions). The results also stress the importance of the induced fluid motion in the enhanced heat transfer on these surfaces. This study is expected to provide better insight to the description of the governing processes.

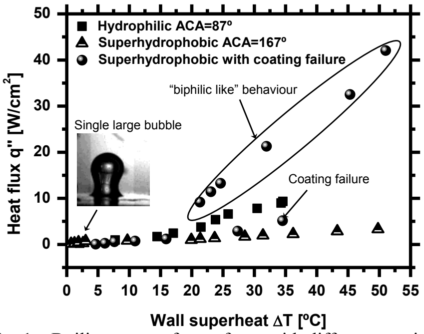


Fig. 1 – Boiling curves for surfaces with different wetting characteristics.

References

MALAVASI, I., BOURDON, B., Di Marco, P., De CONINCK, J., MARENGO, M. *Appearance of a low superheat "quasi-Leidenfrost" regime for pool boiling on superhydrophobic surfaces*. Int. Comm Heat Mass Transf., 63, 1-7.

Experimental study on the thermal performance of two-phase closed-loop thermosyphon with liquid heat transfer agent at high heat flux

Lingjiao Wei[1], Dazhong Yuan[2] and Dawei Tang[3]

1. Institute of Engineering Thermophysics, Chinese Academy of Sciences, Beijing 100190, China, wljustc@126.com
2. Institute of Engineering Thermophysics, Chinese Academy of Sciences, Beijing 100190, China, yuandz@iet.cn
3. Institute of Engineering Thermophysics, Chinese Academy of Sciences, Beijing 100190, China, dwtang@mail.etp.ac.cn

Extended Abstract

In a two-phase closed-loop thermosyphon with liquid heat transfer agent (TPCLTL), the heat transfer mechanism is dominated by the liquid-flow heat transport, and the liquid flow is accelerated by vapor bubbles that are continuously generated at the heated wall. Due to the large difference between sensible heat and latent heat of working fluid, the overall heat transfer capability of a TPCLTL is lower than that of a traditional two-phase closed-loop thermosyphon with vapor heat transfer agent (TPCLTV). However, the heat transfer mechanism between wall and working fluid at the heating section of TPCLTL and TPCLTV are convective boiling and pool boiling, respectively. As the heat transfer coefficient as well as the heat-flux limit of convective boiling is higher than that of pool boiling, TPCLTLs show more promising application at heat exchange conditions with high heat-flux and small heating-area.

In this paper, an experiment has been carried out to investigate the thermal performance of a 6 mm-diameter TPCLTL at different charge ratios (80%, 85%, 90%, 95%) and different heat flux increasing from 20 W/cm^2 to 200 W/cm^2. The heating area of the experimental setup is 1 cm^2. The experimental result shows that the thermal resistance of the TPCLTL is at the magnitude of 10^{-2} K/W, and the best charge ratio which achieves the lowest thermal resistance varies with heating load. In addition, the thermal performance of the TPCLTL is compared with that of the TPCLTV with the charge ratio of 50%, which demonstrates the TPCLTL can operate well when the heating load has exceeded the heat-flux limit of the TPCLTV.

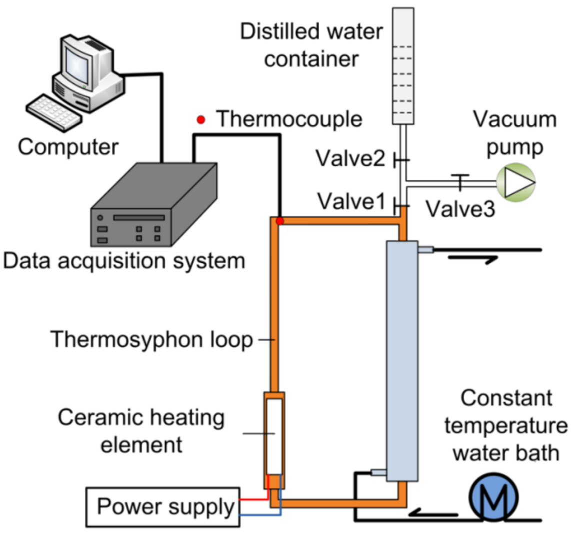


Fig. 1 Schematic diagrams of the rectangular loop and experimental system

Effect of Substrate Temperature on Deposition Pattern from Nanofluid Droplets

X. Zhong[1] and F. Duan[2]

[1] Nanyang Technological University, School of Mechanical and Aerospace Engineering, 639798, Singapore, xzhong002@ntu.edu.sg
[2] Nanyang Technological University, School of Mechanical and Aerospace Engineering, 639798, Singapore, feiduan@ntu.edu.sg

Extended Abstract

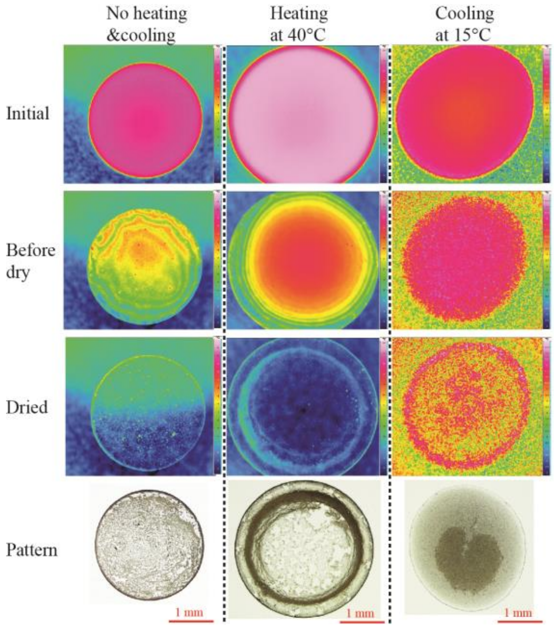


Fig 1: Temperature fields and deposition patterns of nanofluid droplets under no intervention, heating and cooling conditions.

Probing the temperature field along the droplet surface can facilitate our understanding of the internal flow and the corresponding particle motion inside the droplet [1]. It is of great significance as controlling the deposition pattern of particles is the objective of many applications such as micro-coating, inkjet-printing, bio-deposition, etc. We examined the temperature field along the liquid-vapor interface and the final deposition pattern of water-based graphite nanofluid droplets evaporating on a clean silicon wafer substrate under conditions of heating, cooling and no intervention. An infrared technique was employed to visualize the temperature field along the liquid-vapor interface, while the dried pattern was observed and recorded by a bright-field microscopy. Heating and cooling the substrate are shown to vary the temperature and the final deposition pattern of the droplet, as shown in Fig 1. At the self-motivated evaporation condition, the pattern is uniform in the interior area, enclosed by an evident coffee ring. With heating at 40°C, two rings were produced with a thinner one enclosing a thicker one, implying that the heating created a flow regime which brought part of the particles inwardly. The droplet cooled at 15°C exhibits a distinctively different pattern that most particles were gathered in the interior, surrounded by sparsely distributed particles and an almost unnoticeable ring. Notably that for the natural and heating conditions, the contact line receded consecutively at the end of drying; while the one under cooling did not exhibit any receding during the entire lifetime.

References

[1] Parsa, M., Harmand, S., Sefiane, K. Bigerelle, M., & Deltombe, R. 2015 *Effect of substrate temperature on pattern formation of nanoparticles from volatile drops*. Langmuir **31**, 3354−3367.

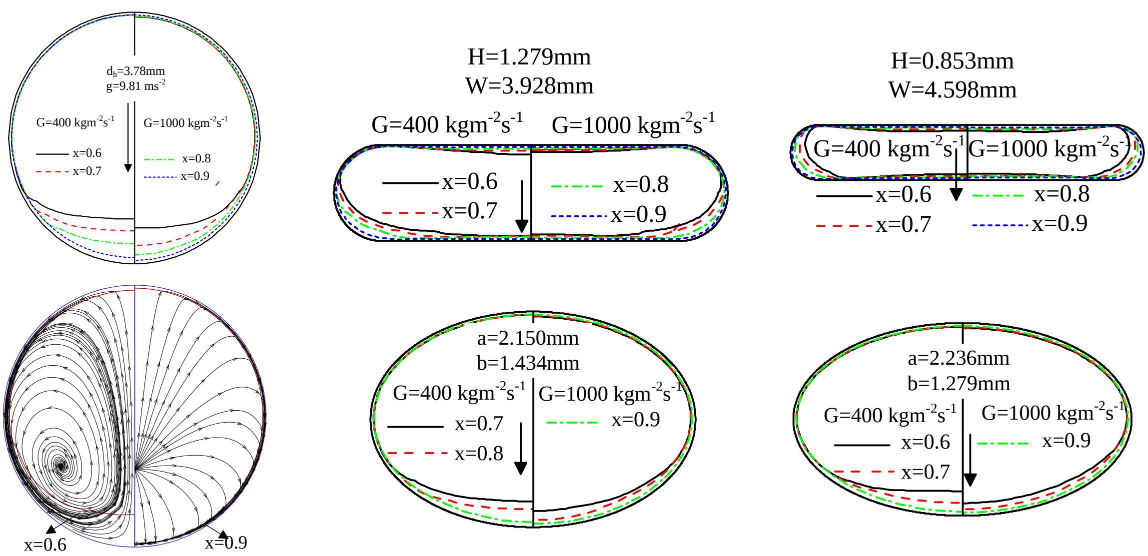
Numerical simulation of condensation in mini horizontal tubes with different cross-section shapes

Jingzhi Zhang[1], Wei Li*[2]

[1]*Department of Energy Engineering, Zhejiang University, Hangzhou, China, zjzsdu@163.com*
2 Department of Energy Engineering, Zhejiang University, Hangzhou, China, weili96@zju.edu.cn

Extended Abstract

Numerical simulations on heat transfer characteristics of condensation in circular tubes (d_h=3.78mm), elliptical tubes and flattened tubes (with the same cross-section perimeter) are conducted using R410a as working fluid at mass fluxes ranging from 300Kgm^{-2}s^{-1} up to 1000Kgm^{-2}s^{-1} and saturation temperature Tsat=320K in a three-dimensional model. The Volume of Fluid (VOF) method is used to acquire liquid-vapor interface, while the lower-Reynolds form of the Shear Stress Transport k~ω(SST k~ω) model is adopted to taking turbulent effect into account. The numerical heat transfer coefficients of circular tube agree well with our previous experimental work. The vapor-liquid interface and stream traces are also presented for better understanding of condensation process inside these tubes. The simulation results indict that heat transfer coefficient increases with increasing mass fluxes and vapor quality for all cases. For circular and elliptical tubes, liquid film thickness at lower mass flux and vapor quality is much thicker in the bottom of the tube due to gravity effect, while that of flattened tubes accumulates at the bottom corner of the tube. Compared with circular and elliptical tubes, the liquid film thickness of flattened tubes at the top part is much higher because of the combinational effect of shear force and gravity. At higher vapor quality, the stream traces start from the core of tubes and point to the vapor-liquid interface, which will enhance heat and mass transfer process; at lower vapor quality, a vortex is observed caused by the gravity effect and the density difference of liquid and vapor. Heat transfer coefficients are much higher at flattened tubes with lower tube height followed by elliptical tubes. Moreover, the cross-section area decreases with decreasing tube height. As a consequence, the heat transfer rate is higher with lower tube height at the same refrigerant charge.



Vapor-liquid interfaces in five tubes at different vapour quality and mass flux

On The Use of Phase Change Materials in Low-Temperature Fischer-Tropsch (LTFT) Reactors

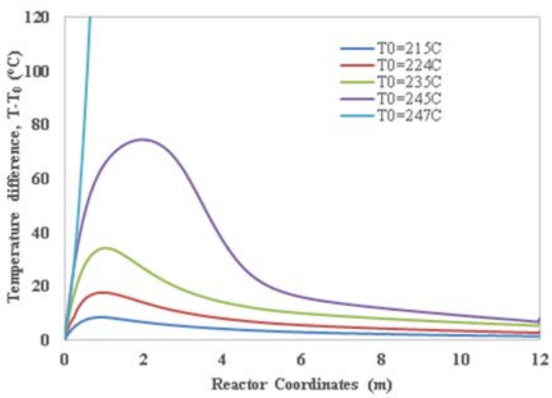
A.O. Odunsi[1], T.S.O'Donovan[2] and D.A.Reay[3]

[1] Heriot-Watt University, Riccarton Currie Edinburgh, UK, aoo39@hw.ac.uk
[2] Heriot-Watt University, Riccarton Currie Edinburgh, UK, T.S.O'Donovan@hw.ac.uk
[3] Heriot-Watt University, Riccarton Currie Edinburgh, UK, dareay@aol.com

Extended Abstract

Exothermic chemical reactions with high activation energy, carried out in wall-cooled, packed bed reactors exhibit parametric sensitivity and multiple stationary states. The effect of this is the formation of hotspots leading to thermal runaway if proper heat rejection is not ensured. One such reaction is the XTL (X to liquid) Low Temperature Fischer Tropsch synthesis (200-250°C). LTFT converts synthesis gas from various precursors such as natural/shale gas, biomass, coal, waste plastics etc. to superior liquid fuels. Temperature control is crucial to the process in order to ensure longevity of the catalyst and optimise the product distribution. Traditional proportional integral and differential (PID) controllers used for temperature control in the FT process are reliant on thermocouple sensors to detect and communicate the development of micro-sized hotspots in the the catalyst pores to the controller. They are however limited by the practicality of their size and location within the reactor.

Our work examines a novel method of using phase change materials (PCM), within a hierarchical supervisory cooling system, to intensify heat transport in such chemical reactors. Chemically encapsulated PCM, with a phase transition temperature lying between a maximum nominal operating temperature and a maximum safe operating temperature can act as a rapid-responding, distributed temperature controller. By packing the encapsulated PCM with the catalyst in a fixed bed, the PCM, an "active-inert", has the two-fold effect of: (i.) providing a temporary, isothermal sink into which the enthalpy of reaction could be dissipated (ii.) and increasing the thermal capacity of the reaction bed. The PCM thus acts as a buffer and thermal flywheel- keeping the propensity for parametric sensitivity and vacillation between multiple steady states in check. Preliminary results from a 2D-axisymmetric pseudo-homogeneous steady state mathematical model (Figure1) show that the PCM has the effect of evening out the temperature profile within the reactor, (at varied inlet temperatures, T_0), thereby reducing temperature spikes in the reactor by a factor of up to 10.



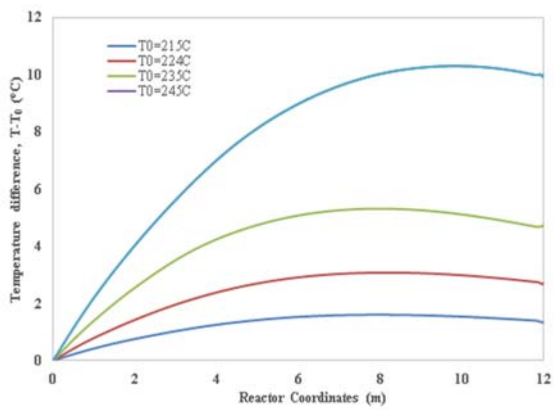


Figure1: Temperature profiles in LTFT reactor without PCM (left) and with PCM (right)

An Experimental Investigation of The Effect of Control Algorithm on The Energy Consumption and Temperature Distribution of a Household Refrigerator

Dr. Tolga N. Aynur[1], G. Peker [2] and Dr. Egemen Tinar[3]

[1] *ARCELIK A.S., E5 Ankara Asfalti Uzeri Tuzla-Istanbul, Turkey, tolga.aynur@arcelik.com*
[2] *ARCELIK A.S., E5 Ankara Asfalti Uzeri Tuzla-Istanbul, Turkey, gokmen.peker@arcelik.com*
[3] *ARCELIK A.S., E5 Ankara Asfalti Uzeri Tuzla-Istanbul, Turkey, egemen.tinar@arcelik.com*

Extended Abstract

In order to determine the energy consumption and cooling characteristics of a domestic refrigerator controlled with various cooling system algorithms, a side by side type (SBS) refrigerator was tested in controlled chamber conditions. Two different control algorithms; so-called drop-in and frequency controlled algorithms for driving the inverter compressor, were tested on the same refrigerator. The most important comparison parameters were taken as; temperature distribution in storage compartments, total energy consumption, evaporation and condensation temperatures, and refrigerator run time ratios (RT). In according to test results, almost the same energy consumption levels have occurred; with a difference of 1.5%. By using two different control algorithms, the power consumption character of the refrigerator was investigated. This paper contains the details of this experimental study conducted with different cooling control algorithms and compares the findings based on the same standard conditions.

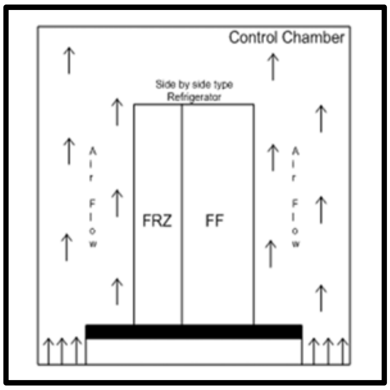


Fig 1: Schematic display of the test

References

Alberto Leva, Luigi Piroddi, Massimiliano Di Felice, Alessandro Boer, Raffaele Paganini, Adaptive relay-based control of household freezers with on–off actuators. Control Engineering Practice 18, pp. 94-102, 2010

M. Alparslan Zehir, Mustafa Bagriyanik Demand Side Management by controlling refrigerators and its effects on consumers. An International Energy Conversion and Management 64, pp. 238-244, 2012..

R. Saidur, H.H. Masjuki, T.M.I. Mahlia Factors Affecting Refrigerator-Freezers Energy Consumption. Journal for Science and Technology Development 19, pp. 57-67, 2002

Zihili Lu, Guoliang Ding, Temperature and time-sharing running combination control strategy of two-circuit cycle refrigerator–freezer with parallel evaporators. Applied Thermal Engineering 26, pp. 1208-1217, 2006.

Miha Mraz, The design of intelligent control of a kitchen refrigerator. Mathematics and Computers in Simulation 56, pp. 259-267, 2001.

ISO 15502:2005 Household refrigerating appliances characteristics and test methods standard.

Experimental Investigation of Bubble Behaviours in a Heat Pump Water Heating System

J.B. Qin[1], Y.T. Ge[2]

[1] *Brunel University London, College of Engineering, Design and Physical Sciences, UK, jianbo.qin@brunel.ac.uk*
[2] *Brunel University London, College of Engineering, Design and Physical Sciences, UK, yunting.ge@brunel.ac.uk*

Extended Abstract

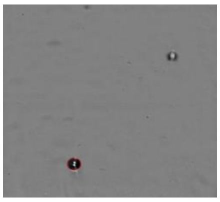


Fig 1: Gas bubbles

Heat pump systems are a widely applied form of green technology to achieve global targets of CO_2 emission reductions. Subsequently, further investigation into potential improvements of the system performance is becoming increasingly significant. One fundamental issue is the examination of bubble behaviours to understand their impact on system performance. In such systems, gas (air or nitrogen) microbubbles of hot water flow, as shown in Fig. 1, nucleate on and detach from the inner surface of the condenser when water is at super-saturated conditions (Fsadni et al. 2012). After exiting the heat exchanger, microbubbles will dissolve gradually if the bulk fluid is at under-saturated conditions (Ge et al. 2013). The broad consensus is that microbubbles reduce the efficiency of a heat pump water heating system and thus require an efficient discharge.

In this paper, experimental investigation of bubble behaviours on a heat pump water heating system will be demonstrated. A horizontal sight-glass was purposely built and installed at the plate condenser outlet to be used for measuring and capturing bubble behaviours at various operating states through a high-speed camera. Bubble sizes are measured at several different depths on a focal plane within the sight-glass under experimental conditions of saturation ratio, fluid velocity, fluid temperature and pressure of the bulk fluid. The experimental results will therefore be analysed to understand bubble behaviours at different conditions; this will help determine which factors strengthen deaeration techniques and thus improve the overall efficiency of the heat pump water heating system.

References

FSADNI , A.M. , GE, Y.T. , LAMERS, A.G. 2012 *Bubble Nucleation on the Surface of the Primary Heat Exchanger in a Domestic Central Heating System*. Applied Thermal Engineering. **45-46** , 24-32.

GE, Y.T. , FSADNI , A.M. , WANG, H.S. 2013 *Bubble dissolution in horizontal turbulent bubbly flow in domestic central heating system*. Applied Energy **108** , 477–485.

Keynote 2

Recent Numerical and Analytical Studies on Effects of Surface Characteristics in Phase-Change Heat Transfer

P. Cheng

School of Mechanical Engineering, Shanghai Jiaotong University

Extended Abstract

Recent numerical and analytical studies on surface characteristics in phase-change heat transfer, carried out at Shanghai Jiaotong University, will be summarized in this lecture. Effects of wettability and roughness in saturated pool boiling heat transfer from a smooth superheated substrate, with a finite thickness at constant wall temperatures at the bottom, are investigated numerically based on a recently developed liquid-vapor phase-change lattice Boltzmann method. Wettability effects on bubble departure modes, spatial/temporal temperature and heat flux distributions on the heating surface as well as the boiling heat transfer mechanisms are discussed. Boiling curves (from onset of nucleate boiling to critical heat flux, to transition boiling to stable film boiling) for smooth hydrophilic/hydrophobic heating surfaces as well as rough surfaces are simulated numerically for the first time. This mesoscale numerical method is also used to simulate dynamic evaporation along the vapor/liquid interface and boiling in the liquid thin film on a microstructured surface consisting of micro pillars, under constant heat fluxes for different wettabilities of solid hydrophilic surfaces. Residual liquid volume, two-phase interface morphology, wall temperature distribution, as well as effective heat transfer coefficient and CHF under different conditions are illustrated.

An improved dropwise condensation heat transfer model modified from previous models, incorporating with a newly developed model for the critical radius is presented. Effects of wettability, degree of subcooling, thickness and thermal conductivity of the coating layer on droplet nucleation radius, nucleation density, condensation heat flux are illustrated. The closed form solution for condensation heat flux without any fitting constants, is shown in good agreement with experimental data. Coalescence induced droplets self-propelled jumping on textured superhydrophobic surfaces (SHS) is simulated using multiple-relaxation-time (MRT), three dimensional (3D) lattice Boltzmann method. For a fixed droplet diameter, the spacing of the microstructure is found to play a key role on jumping velocity of the coalescence droplet, and an optimal spacing of the microstructure exists for a maximum jumping velocity.

14th UK Heat Transfer Conference 2015
Edinburgh (Sept 7 – Sept 8, 2015)

Numerical Simulation of Flow and Heat Transfer around Vertical Cylinder Submerged in Water

Ahmed Ramadan[1], Reaz Hasan[2] and Roger Penlington[3]

[1] *Northumbria University Newcastle, Department of Mechanical and Construction Engineering, UK, ahmed.ramadan@northumbria.ac.uk*
[2] *Northumbria University, Department of Mechanical and Construction Engineering, Newcastle, NE1 8ST, UK, reaz.hasan@northumbria.ac.uk*
[3] *Northumbria University, Department of Mechanical and Construction Engineering, UK, r.penlington@northumbria.ac.uk*

Abstract

Natural convection flows around submerged heated cylinders have gained attention in recent years due to their use in many engineering applications such as flow around tubes and rods (as in nuclear reactors and spent fuel cooling ponds).

The purpose of this paper is to establish the modelling strategy for simulating natural convection heat transfer and flow around vertically positioned cylinders submerged in water tank heated with a constant heat flux. The simulations were conducted for cylinders having different diameters which varied from 10 mm to 165 mm. The problem involved flow transition from laminar to turbulent within the boundary layer as the maximum Rayleigh number based on the cylinder length reached a magnitude of the order of 10^{14}. The outcomes of these calculations were validated against published experimental data[1]. The comparisons were made in terms of heat transfer coefficient and Rayleigh number which showed good agreement between predictions and experimental results.

The predicted results showed sensitivity to various eddy viscosity models. Another important point that emerged from the study was that the specification of a suitable evaporation boundary condition at the top (water) surface was very critical. Our results quantified the sensitivity of the evaporation models. We believe that the results obtained from our study will be useful for modelling the flow in situations where a large number of heated cylinders are involved.

References

1. Kimura, F., Tachibana, T., Kitamura, K. and Hosokawa, T. (2004*). Fluid flow and heat transfer of natural convection around heated cylinders (effect of cylinder diameter)*. JSME International Journal Series B, 47(2), 156-161.

Saturated Boiling of Water on Biphilic Surfaces under Sub-atmospheric pressure

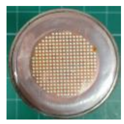
M. Yamada[1], B. Shen[2], H. Hong[1], K. Furusato[1], S. Hidaka[1], M. Kohno[1,2], K. Takahashi[2,3] and Y. Takata[1,2]

[1] *Department of Mechanical Engineering, Motooka 744, Fukuoka 819-0395, Japan*
[2] *International Institute for Carbon-Neutral Energy Research (I²CNER) , Motooka 744, Fukuoka 819-0395, Japan, shen.biao.604@m.kyushu-u.ac.jp*
[3] *Department of Aeronautics and Astronautics, Motooka 744, Fukuoka 819-0395, Japan*

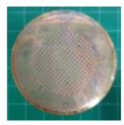
Extended Abstract

The semiconductor and microprocessor industry faces growing demand for cost-effective thermal managerment solutions such as heat pipes and thermosiphones. When water is used as the working fluid, reductions in the saturation temperature (pressure) are often necessary to match the relatively low surface temperature of the electronic component.

Plain surface



Biphilic surface (Φ1, p3)



Biphilic surface (Φ1, p1.5)

Fig.1: Boiling surfaces

In this study, we investigated the effect of wettability on low pressure boiling of water. Specifically, pool boiling characteristics of biphilic surfaces (hydrophilic copper surfaces decorated with an array of hydrophobic spots made by electrolytic plating of fine nickle and TFE particles, see Fig. 1) were measured in open and sealed vessels, which corresponded to atmospheric (P_∞=101.3 kPa) and sub-atmospheric (P_∞=12.35 kPa) conditions, respectively. For comparison, we repeated experiments using a plain copper surface polished to mirror finish. The results showed a general deterioration of heat transfer under low pressure, as shown in Fig. 2, due to much enlarged bubbles, which aggrees with previous experimental observation [Nisiokawa. (1976)]. Nonetheless, biphilic surfaces were found to significantly enhance the boiling performance [Fig. 2(b)]. At low heat flux levels, considerably more nucleation sites were activated on the biphilic surfaces compared with the bare surface, where intermittent boiling (featured with large surface temperature fluctuations) occurred. As the heat flux increased, the surface remained partially wetted despite bubble coalescence.

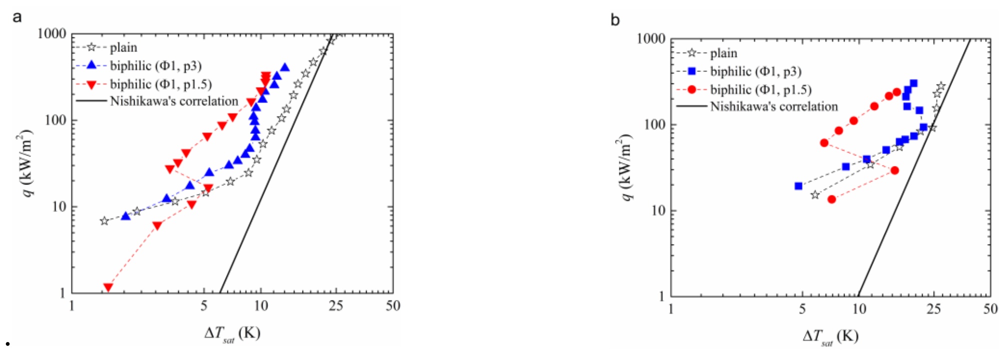


Fig. 2: Boiling curves for (a) atmospheric and (b) sub-atmospheric conditions

References

NISHIKAWA, K., FUJITA, Y. &NAWATA, Y. 1976 *The effect of pressure on heat transfer and bubble behavior at saturated nucleate boiling*. Trans. JSME. **42**, 2879-2891.

14th UK Heat Transfer Conference (UKHTC2015)
Edinburgh (Sept 7 – Sept 8, 2015)

On the evaporation of droplets with related initial and receding contact angles

J. M. Stauber[1], S. K. Wilson[1], B. R. Duffy[1] and K. Sefiane[2]

[1] Department of Mathematics and Statistics, Livingstone Tower, University of Strathclyde Richmond Street, Glasgow G1 1XH, United Kingdom, s.k.wilson@strath.ac.uk

[2] School of Engineering, University of Edinburgh, The King's Buildings, Mayfield Road, Edinburgh, EH9 3JL, United Kingdom
and
International Institute for Carbon-Neutral Energy Research (WPI-I2CNER), Kyushu University, 744 Motooka, Nishi-ku, Fukuoka 819-0395, Japan

Extended Abstract

Droplet evaporation plays a crucial role in many practical applications and, as a consequence, the evaporation of a fluid droplet on a solid substrate has been the subject of extensive theoretical and experimental study in recent years. However, until recently surprising little attention has been paid to the lifetime of a droplet (i.e. the time it takes for a droplet to evaporate entirely) and, in particular, how it depends on the manner in which the droplet evaporates.

In a previous contribution (Stauber et al 2014) (see also our recent related work Stauber et al 2015) we analysed the lifetime of a droplet evaporating in a "stick-slide" (SS) mode of evaporation in which the contact line of the droplet is initially pinned (the "stick" phase) but subsequently de-pins (the "slide" phrase) when the contact angle, θ, reaches its receding value, θ^*. In this work, the initial contact angle, θ_0, and the receding contact angle, θ^*, were assumed to be independent parameters and their values were determined empirically.

In the present work we take an alternative approach and propose a simple relationship between θ_0 and θ^* based on the assumption of a constant maximum pinning force and use it to give a complete description of the lifetime of a droplet evaporating in the SS mode. In particular, it is shown that the dependence of the lifetime on θ_0 is qualitatively different from that described in our earlier work and that it is in unexpectedly good agreement with the results of physical experiments available in the literature.

References

STAUBER, J.M., WILSON, S.K., DUFFY, B.R, & SEFIANE, K. 2014 *On the lifetimes of evaporating droplets.* J. Fluid Mech. **744**, R2.

STAUBER, J.M., WILSON, S.K., DUFFY, B.R. & SEFIANE, K. 2015 *Evaporation of droplets on strongly hydrophobic substrates.* Langmuir **31**, 3653–3660.

Acknowledgements

The financial support of United Kingdom Engineering and Physical Science Research Council (EPSRC), the University of Strathclyde and the University of Edinburgh via a postgraduate research studentship for JMS, and of the Leverhulme Trust via Research Fellowship RF-2013-355 for SKW is gratefully acknowledged.

Generation and Metastability of Interfacial Nanobubbles

K. Takahashi[1], T. Nishiyama[2] and Y. Takata[3]

[1] *Department of Aeronautics and Astronautics, Kyushu University, Fukuoka, Japan,*
International Institute for Carbon-Neutral Energy Research (WPI-I2CNER), Kyushu University, Fukuoka, Japan,
takahashi@aero.kyushu-u.ac.jp
[2] *Department of Aeronautics and Astronautics, Kyushu University, Fukuoka, Japan,*
nishiyama@aero.kyushu-u.ac.jp
[3] *Department of Mechanical Engineering, Kyushu University, Fukuoka, Japan,*
International Institute for Carbon-Neutral Energy Research (WPI-I2CNER), Kyushu University, Fukuoka, Japan,
takata@mech.kyushu-u.ac.jp

Extended Abstract

The superstability of nanobubbles at solid/gas interfaces is one of the most interesting and hotly debated phenomena in the surface science community and also one of the keys to decrease the temperature overshoot of boiling heat transfer. Although nanobubbles have been widely studied in the past few years, the mechanisms by which they form and remain stable have not been clarified. Recently, we conducted atomic force microscopic (AFM) measurement of nanobubbles on both graphite substrate and Teflon amorphous fluoroplastic thin film through the solvent-exchange method. By using focused ion beam technique, very tiny hydrophilic carbon deposits are prepared and nanobubbles in the presence and absence of the hydrophilic domains are observed. On the hydrophobic surface without hydrophilic domains, a small number of nanobubbles are generated and thereafter shrink quickly in size. It is concluded that the presence of the hydrophilic domains has a significant effect on both the generation and stability of nanobubbles, with bubbles remaining on the surface for up to three days. Nanobubbles on graphite substrate are found most stable because the large number of graphite steps work as the effective hydrophillic domains and provide enough number of gas molecules.

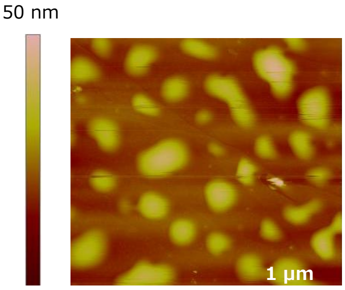


Fig 1: Coalesced nanobubbles

It was also found that the pressing and tapping of the AFM cantilever promotes coalescence of adjacent nanobubbles. Obtained butterfly-shaped nanobubbles (Fig.1) were stable for at least several tens of minutes. To reveal the underlying mechanism of this phenomena, we conducted sub-micron scale observation of advancing and receding contact angles of water-graphite-vapor systems by using environmental SEM and mutiwalled-carbon nanotube. A very large hysterisys was observed, which should attribute the metastability of nanobubbles on graphite surface.

References

NISHIYAMA, T., et al. 2015 *Metastable Nanobubbles at the Solid-Liquid Interface due to Contact Angle Hysteresis*. Langmuir **31(3)**, 982-986.

Optimization of Heating and Condensation System of a Water Condensed Type Washer Dryer Regarding Water Consumption

A.B. Top[1], I. Uslu[2], I. Erdem[3] and S.U. Onbasioglu[4]

[1] Istanbul Technical University, Graduate School of Science - Engineering and Technology, Department of Mechanical Engineering, Maslak Istanbul, TURKEY, burak.top@arcelik.com
[2] Istanbul Technical University, Graduate School of Science - Engineering and Technology, Department of Mechanical Engineering, Maslak Istanbul, TURKEY, isil.uslu@arcelik.com
[3] Istanbul Technical University, Graduate School of Science - Engineering and Technology, Department of Nano-science and Nano - engineering, Maslak Istanbul, TURKEY, ilkan.erdem@arcelik.com
4 Istanbul Technical University, Graduate School of Science - Engineering and Technology, Department of Mechanical Engineering, Maslak Istanbul, TURKEY, onbasiogl1@itu.edu.tr

Extended Abstract

As the technology is developed more in time, home appliances which are manufactured to make our lives easier become more efficient, cheaper and useable. Washing machines and dryers which are two important products among white appliances have been conventionally used by consumers for a long time. Recently, these products are functionally combined in one machine and called as a washer dryer. The key component of washer dryer is a condenser which provides condensation of saturated vapor inside air during the process of removing humidity from clothes. The purpose of this study is optimization of the heating and condensation system in which water is sprayed on drying air directly of water – condensed type washer dryer regarding energy consumption. In order to achieve that purpose, a prototype was set up and related equipment was integrated into the prototype so as to store measurement data. The power of heater device, flow rate of fan and the design of nozzle orifice have been studied and the energy consumption of the washer dryer was minimized while keeping the water consumption of the condensation system and heating durations of the dryer steady. Then, tests results were analysed and discussed.

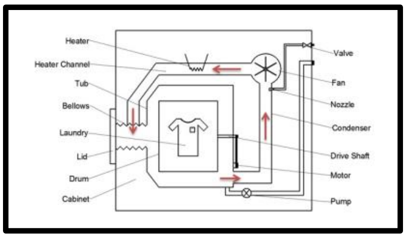


Fig 1: Schematic diagram of the test apparatus.

Thermal Response of a Pulsating Heat Pipe on Board the Rexus 18 Sounding Rocket: PHOS Experiment Chronicles

F. Creatini[1], G. Guidi[1], F. Belfi[1], G. Cicero[1], S. Piacquadio[1], D. Di Prizio[1], D. Fioriti[1], G. Becatti[1], G. Orlandini[1], A. Frigerio[1], S. Fontanesi[1], P. Nannipieri[1], M. Rognini[1], N. Morganti[1], A. Pasqui[1], S. Filippeschi[1], P. Di Marco[1], L. Fanucci[1], F. Baronti[1], M. Manzoni, M. Mameli[2,*], M. Marengo[2,3].

[1] *Università di Pisa, DESTEC, Largo Lazzarino 2, 56122 Pisa,*
[2] *Università di Bergamo, Dept. of Eng. and Applied Science, Viale Marconi 5, 24044 Dalmine (BG), Italy*
[3] *University of Brighton, School of Computing, Eng. and Maths, Lewes Road, BN2 4GJ, Brighton, UK*

**Email: mauro.mameli@unibg.it*

Abstract

This work presents the experimental results of two Pulsating Heat Pipes (PHP) with different internal tube diameter tested on board the REXUS 18 sounding rocket. As shown in Figure 1, both the test cell (experiment box containing the two PHPs) and the electronic hardware (battery pack, power management and data handling) are designed, developed and implemented by the team. The PHPs are both filled with FC-72, thus, being the critical diameter around 1.7mm at ambient temperature and on Earth gravity conditions, one PHP (1.6 mm I.D.) is around the critical diameter while the other one (3 mm I.D.) is larger. The acronym PHOS, Pulsating Heat pipe only for Space, indeed resume the concept of a two phase closed loop that is working as a PHP only in microgravity: surface tension prevails over buoyancy and the flow pattern should switch to the slug and plug PHPs typical operational regime also when the diameter is above the critical. The temperature and pressure trends are expected to reveal such a regime transition and provide further information for future space applications. The tested PHPs consist of a closed end-to-end aluminium tube with fourteen curves arranged on two planes constituting the evaporator or hot section. A heating cable wrapped around the tube in the evaporator section supplies the desired heat flux, while a phase change material allows dissipating the heat in the condenser section. A set of thirty thermocouples for each PHP is located in different tube positions and the local fluid pressure is recorded by means of a mini pressure transducer in the condenser section. Ground and flight results are compared in terms of temperature and pressure temporal trends.

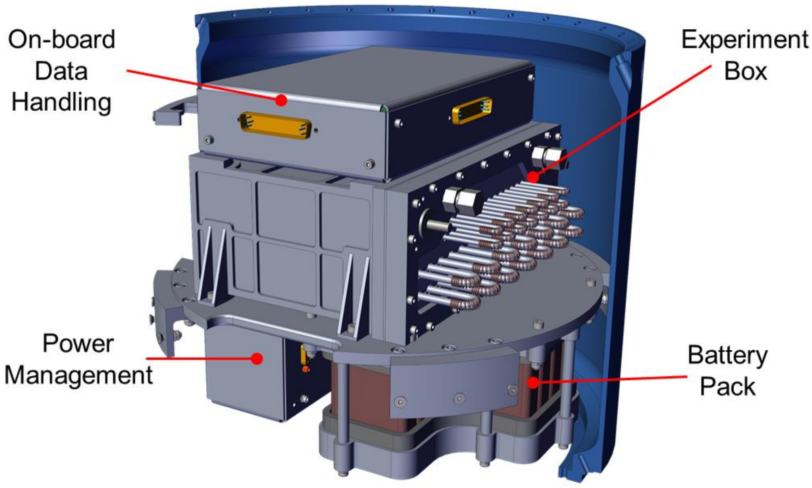


Figure 1: Phos Experiment global layout.

Effect of Dead Volumes on Single Closed Loop Pulsating Heat Pipes

F. Creatini[1], M. Mameli[2,*] S. Filippeschi[1], P. Di Marco[1]

[1] *Università di Pisa, DESTEC, Largo Lazzarino 2, 56122 Pisa, Italy*
[2]*Università di Bergamo, Dept. of Eng. and Applied Science, Viale Marconi 5, 24044 Dalmine (BG), Italy*

**Email: mauro.mameli@unibg.it*

Abstract

Pulsating Heat Pipes (PHPs) are novel promising two-phase passive heat transfer devices that seem to meet all present and possibly future thermal requirements. Although grouped as a sub-class of the heat pipes, the PHPs governing phenomena are quite unique and not fully understood yet. In particular, single closed loop PHPs, that might be considered as the basic constituents of multi turns PHPs, manifest a large number of drawbacks mostly related to the device thermal performance and reliability, i.e. the occurrence of multiple quasi-steady states during device operation. Many attempts have been made in literature in order to face the problem and promote the circulation of the working fluid in a preferential direction. In order to understand the effect of dead volumes on the PHP working regimes and on its stability, two slightly different versions of the single closed loop PHP showed in figure 1 (inner tube diameter of 2.0 mm, filled with ethanol 60 % vol.) are tested.

The device is equipped with nine thermocouples: three in the evaporator zone, one for each copper branch and two for measuring the inlet and outlet temperature of the cooling medium at the condenser. A pressure transducer measures the local fluid pressure fluctuations in one of the four junction between copper and glass tube. The fluid flow regime is visible through the two glass branches composing the adiabatic section.

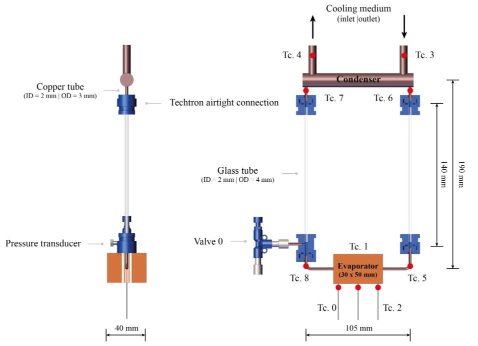


Figure 1: Single Loop Pulsating Heat Pipe layout.

The two circuits basically differ for the design and material of the connections between the copper and glass tubes. On one hand, the connections of the first circuit are designed in order to minimize the overall volume and thermal inertia (Figure 2A). On the other, they are made of brass and introduce a local dead volume that disturbs the working fluid motion (Figure 2B).

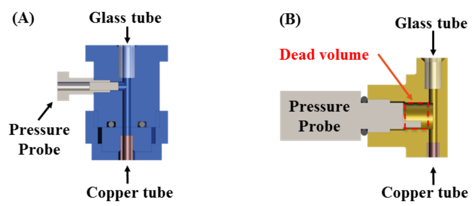


Figure 2: Detail of the connection between copper and glass tubes: A) without dead volume; B) with dead volume.

The equivalent thermal resistances plotted at each heat input level highlight how the operation of the first circuit is possible over a wide heat power range, while the operational range of the second circuit is narrower, thus confirming the detrimental effect of the dead volume.

Additional long-term runs on the first circuit exhibit stable circulation and no multiple steady states are detected.

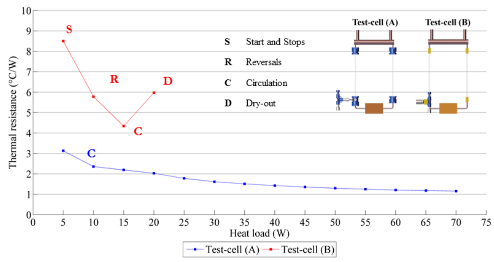


Figure 3: Equivalent Thermal resistances.

Flow Boiling Heat Transfer in a Shallow Metallic Microchannel

M. R. Ozdemir[1,2], M. M. Mahmoud[3,4] and T. G. Karayiannis[5]

[1] *Brunel University London, Uxbridge, UB8 3PH, UK, mehmed.oezdemir@brunel.ac.uk*
[2] *Marmara University, Istanbul, 34722, Turkey, mehmet.ozdemir@marmara.edu.tr*
[3] *Brunel University London, Uxbridge, UB8 3PH, UK, mohamed.mahmoud@brunel.ac.uk*
[4] *Zagazig University, Zagazig, 44519, Egypt, mbasuny@zu.edu.eg*
[5] *Brunel University London, Uxbridge, UB8 3PH, UK, tassos.karayiannis@brunel.ac.uk*

Extended Abstract

High-tech and miniaturized electronic devices brought significant challenges in the thermal management research arena. The heat dissipation rate from these devices has been increasing rapidly with the development in the industry. Mudawar (2001) stated that the heat dissipation rate from supercomputers, electric devices and military avionics can be as high as 1 MW/m^2. Moreover, he reported ultra–high heat flux values of 10^3 MW/m^2 in applications such as cooling fusion reactors, laser and radar systems. Consequently, the traditional cooling methods such as air cooling can not meet these high cooling loads. Zhou et al. (2004) reported that conventional air cooled systems cannot dissipate more than 1.5 MW/m^2. Microchannel heat exchangers offer an effective cooling solution in this area due to their high thermal performance, compactness and small volumes. Also, flow boiling in microchannel heat sinks can achieve high heat transfer rates and better thermal characteristics with less pumping power than single phase flow, see Consolini and Thome (2009). However, several fundamental issues in flow boiling in microchannels are not yet understood.

The present work is a fundamental study on understanding the flow boiling characteristics of de-ionized water in a single rectangular shallow microchannel. This paper presents the experimental results of flow boiling heat transfer and pressure drop. The channel was manifactured by CNC machining with the following dimensions: 1.68 mm wide, 0.34 mm deep and 62 mm long. The experiments were conducted at 115 kPa inlet pressure and 89 ^{0}C water inlet temperature. The mass flux ranged from 200 to 800 kg/m^2s while the heat flux was increased gradually until the occurrence of dryout. The effect of heat and mass flux on the local flow boiling heat transfer coefficient, flow patterns and pressure drop is presented and discussed. Additionally, the heat transfer and pressure drop data were used to assess existing macro and micro scale correlations. The prsent data are also compared with previously obtained results to compare the effect of aspect ratio.

References

Mudawar I. Assessment of high heat flux thermal management schemes. Transactions on Components and Packaging Technologies, 24(2):122-140, 2001.

Zhou P., Hom J., Upadhya G., Goodson K. and Munch M., Electro-kinetic microchannel cooling system for desktop computers. 20th IEEE SEMI-THERM Symposium, 2004.

L. Consolini and J.R. Thome, Microchannel flow boiling heat transfer of R134a, R236fa and R245fa, Microfluid Nanofluid, 6, pp. 731–746, 2009.

Vapour generation of perfectly wetting liquid by cyclone evaporator under variable gravity level

A. Glushchuk[1], V. Grishaev[1] and C. Buffone[1]

[1] *Microgravity Research Centre, Université libre de Bruxelles, Avenue F. Roosevelt 50, 1050 Bruxelles, Belgium*, aglushch@ulb.ac.be

Extended Abstract

The behaviour of a liquid undergoing phase change in microgravity is completely different from that found on Earth. Pool boiling in microgravity suffers from the fact that the bubble growing on top of the heater tends to occupy the whole volume available and it becomes difficult to displace then *(Lee et al. 1996)*. This leads to reduced heat transfer rates and performance. Therefore a novel evaporation system has been investigated. In the presented work a cyclone evaporator of perfectly wetting liquids has been designed and tested for two distinct experiments on condensation during ESA parabolic flights 58th and 61st. Detailed description of condensation experiments can be found in the authors recent paper *(Glushchuk et al. 2014)*. Two liquids (FC-72 and HFE-7100) with surface tension coefficient of 12-15 mN/m were used. The cyclone evaporator is a cylindrical chamber inside which the liquid is heated by an electric heater positioned on the cylindrical bottom face. Inside the cylindrical chamber a magnetic stirrer and a barrier in the top cover are implemented, to make sure that during microgravity for a certain quantity of the working fluid it does not exit the top opening and the vapour bubbles are continuously chopped avoiding that a big bubble occupies the whole chamber and gets stuck with detrimental effects on heat transfer.

Obtained data has proven than the use of only a physical barrier is not enough to keep the liquid inside the evaporator volume during 22 seconds of weightlessness. Vapour pressure is used to analyse the functioning of the evaporator because condensation is sensitive to any pressure fluctuation. An increase of vapour pressure was observed under microgravity condition (Fig. 1). During microgravity the vapour pressure decreases with increase of rotational speed of the magnetic stirrer. According to obtained measurements the pressure fluctuation did not exceed 4 mbar at the rotation of 200 rpm for the cooling power up to 1.8 W.

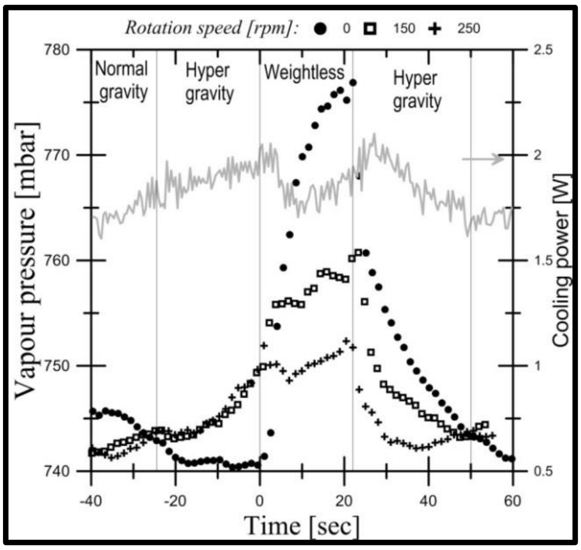


Fig 1: Evolution of vapour pressure and cooling power during one parabola for different cross-shaped magnetic stirrer speed (zero time corresponds to beginning of microgravity).

References

LEE, H.S., MERTE, H. & CHIARAMONTE, F., 1996, The pool boiling curve in microgravity, AIAA Meeting Paper 96-0499.

GLUSHCHUK, A., MINETTI, C. and BUFFONE, C., 2014, Fin condensation in variable gravity environment, Multiphase Science and Technology, Vol. 26 (1), pages 63-81.

14th UK Heat Transfer Conference 2015
Edinburgh (Sept 7 – Sept 8, 2015)

Experimental study on in-tube convective condensation of inclined tubes in a single minichannel with HFE-7100 at low mass fluxes

C. Gallo[1], M. Kwiatkowski[2], A. Glushchuk[1], C. Buffone[1]

[1] *Microgravity Research Centre, Université libre de Bruxelles, Avenue F. Roosevelt 50, 1050 Bruxelles, Belgium, cgallo@ulb.ac.be*
[2] *Mechanical Departament, University of Technology Wrocław, Wybrzeże Stanisława Wyspiańskiego 27, 50-370 Wrocław*

Extended Abstract

From the literature it is clear that relatively little works have been done on the development of two-phase flow regime maps for small diameter tubes and for low mass velocities (*Wang and Rose* 2005) (*Del col et al.* 2014). Moreover, for convective condensation at low mass velocities, flow patterns have been only studied for horizontal flows (*Lips and Meyer* 2011). In this paper, flow patterns and heat transfer were investigated during convective condensation of HFE-7100 in a single microcondenser. The test section consisted of a brass rectangular microchannel, of 3.9 and 7.6 mm cross sections (5.15 mm hydraulic diameter) and of 220 mm length. The transparency of the microchannel top wall allowed the visualization of the different condensation flows using high speed camera. Pressure drop during condensation of HFE-7100 is also measured between the inlet and outlet of the test section. Tests have been performed with HFE-7100 at 60°C saturation temperature, at mass velocity ranging between 10 and 50 kg m^{-2} s^{-1} and varying channel inclination, "α". Annular flow was the dominant flow pattern for vertical downward flow, α = -90°. Annular flow and stratified flow were observed for -45°. Bubbly flow, plug flow, pulsating flow and annular flow were observed for +45° and +90°. Stratified flow, slug flow and wavy-stratified flow occurred for horizontal tube. The results also show that the dependency of the average HTC on inclination has a maximum at -90° (at 10, 20, 40 and 50 kg m^{-2} s^{-1}) and at +90° for mass flux equal to 30 kg m^{-2} s^{-1}.

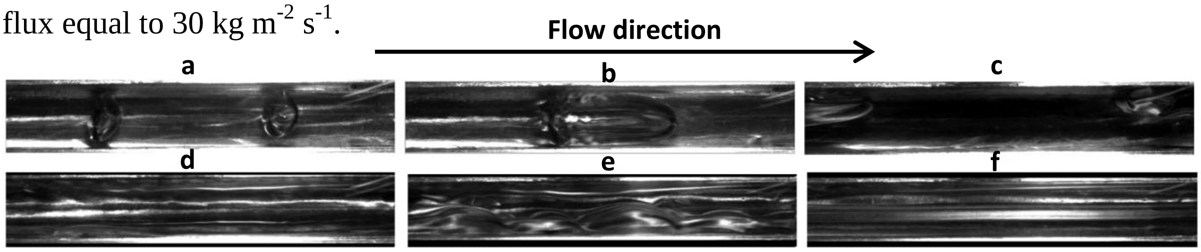


Fig 1 : Different two-phase flow patterns in microchannel of D_h=5.15 mm: (a) bubbly flow in vertical upflow, (b) plug flow in vertical upflow, (c) slug flow in horizontal flow, (d) annular flow vertical downward flow, (e) wavy flow in horizontal flow, (f) stratified flow in horizontal flow.

References

WANG, H.S., ROSE, J. W., 2005 *A theory of film condensation in horizontal noncircular section microchannels*, J. of Heat Transfer **127**, 1096.
DEL COL, D., BORTOLATO, M., AZZOLIN, M., BORTOLIN, S., 2014 *Effect of inclination during condensation inside a square cross section minichannel*, Int. J. of Heat and Mass Transfer **78**, 760-777.
LIPS, S., MEYER, J.P., 2011 *Two-phase flow in inclined tubes with specific reference to condensation: A review,* Int. J. of Multiphase Flow **37**, 845-859.

Flow Boiling Heat Transfer and Pressure Drop of R134a in a Multi Microchannel Metallic Evaporator

A. A. Mohammed[1], E. M. Fayyadh[2], M. M. Mahmoud[3,4],

A. A. Abdulrasool[1] and T. G. Karayiannis[3]

[1]*Al-Mustansiriya University,College of Engineering Baghdad, Iraq, adekef@yahoo.com*
[2]*University of Technology, Department of Mechanical Engineering,10066 Alsina'a Street, Baghdad, Iraq,ikhlase60@yahoo.com*
[3]*Brunel University London, Uxbridge, Middlesex,UB8 3PH, UK, tassos.karayiannis@brunel.ac.uk*
[4]*Zagazig University, Zagazig, Egypt*, mbasuny@zu.edu.eg, mohamed.mahmoud@brunel.ac.uk

Abstract

Flow boiling in multi microchannel evaporators is one of the most promising and efficient methods for cooling electronics and high heat flux devices. However, several fundamental issues such as flow instability and lack of accurate prediction methods are not completely resolved. This study is a part of a long term project for understanding the underlying physical phenomena and designing microchannel metallic evaporators for cooling high heat flux systems. The experimental results of flow boiling pressure drop and heat transfer rate of R134a in a multi microchannel evaporator are presented and discussed in this paper. The evaporator consisted of 25 microchannels with dimensions 297 µm wide, 695 µm deep and 209 µm separating wall thickness. It was made of oxygen free copper by CNC machining and was 20 mm long and 15 mm wide. Experimental operating conditions spanned the following ranges: heat flux 10 –380 kW/m^2, mass flux 50 – 300 kg/m^{2}s and system pressure 8.5 – 12.5 bar. Flow visualization was conducted using a high speed camera at the channel inlet including a small portion of the inlet manifold. The visualization indicated the occurrence of flow reversal at certain experimental conditions. Also, it was found that the heat transfer coefficient increased with heat flux and there was no mass flux and system pressure effect. The experimental data of pressure drop and heat transfer coefficient were used to assess existing prediction models and correlations to help recommend practical design equations.

Thermal Energy Storage Using Composite Phase Change Materials: Linking Materials Properties to Device Performance

Chuan Li[1], Zhiwei Ge[2], Yi Jin[2], Yongliang Li[1], Yulong Ding[1*]

[1] *Birmingham Centre for Thermal Energy Storage, University of Birmingham, Edgbaston, Birmingham B15 2TT, UK*
[2] *Institute of Process Engineering, Chinese Academy of Sciences, Beijing 100190, China*

Extended Abstract

The work reported in this paper aims to establish a relationship between materials properties and device level performance with particular focus on the use of composite phase change materials for medium and high temperature thermal energy storage (TES) (Fig 1). The composite phase change materials consist of a molten salt based phase change material (PCM), a thermal conductivity enhancement material (TCEM) and a ceramic skeleton material (CSM) [1-2]. A mathematical model of transient heat transfer in the composite materials and TES component has been formulated and validated with experimental data obtained by the experimental investigations. The comparison results show that numerical results agree reasonably well with the experimental data, which point out the validity of the numerical model. Extensive modelling was carried out to study the heat transfer performance from TES component to TES device levels. Firstly, the heat transfer performance of TES component was investigated. Influence of materials properties and geometrical design of composite materials module as well as heat transfer fluid (HTF) velocity were examined. The results show that increasing the mass fraction of TCEMs and thickness of composite materials module as well as the HTF velocity can enhance the heat storage and release rate and improve the heat transfer performance of TES component. Then heat transfer performance of TES device made from TES components investigated with three arrangements of TES components-parallel, interlaced and trapezoidal packing using the same amount of composite material modules and components. The results show that trapezoidal packing TES device offers the best performance in terms of heat transfer performance, which gives 43% and 31% reduction in the charging time compared with the parallel and interlaced configurations.

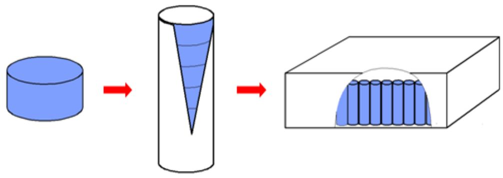


Fig 1: Thermal energy storage using composite materials-linking material to device levels

References

[1] Zhiwei Ge, Feng Ye, and Yulong Ding. 2014. *Composite Materials for Thermal Energy Storage: Enghancing Performance through Microstructures*. ChemSumChem. **7**, 1318-1325.

[2] Zhiwei Ge, Yongliang, Dacheng Li, ZeSun, Yi Jin, Chuanping Liu, Chuan Li, Guanghui Leng, Yulong Ding. 2014. *Thermal energy storage: Challenges and the role of particle technology*. Particuology. **15**, 2-8.

A Novel Type of Multi-Evaporator Closed Loop Two Phase Thermo-syphon: Thermal Performance Analysis and Fluid Flow Visualization

M. Mameli[1], D. Mangini[2], G.F. Vanoli[3], L. Araneo[4], S. Filippeschi[5] and M. Marengo[6]

[1] Università di Bergamo, Viale Marconi 5, Dalmine, Italy, mauro.mameli@unibg.it
[2] Università di Bergamo, Viale Marconi 5, Dalmine, Italy, daniele.mangini@unibg.it
3Politecnico di Milano, Dipartimento di Energia, Via Lambruschini 4A, 20158 Milano, Italy, Giulio.vanoli@mail.polimi.it
4 Politecnico di Milano, Dipartimento di Energia, Via Lambruschini 4A, 20158 Milano, Italy, lucio.araneo@polimi.it
5 Università di Pisa, DESTEC, Largo Lazzarino 2, 56122 Pisa, sauro.filippeschi@den.unipi.it
5 University of Brighton, Brighton BN2 4GJ, Cockcroft Building, C221, Lewes Road, UK, m.marengo@brighton.ac.uk

Abstract

A novel type of multi-evaporator Closed Loop Two Phase Thermo-syphon has been designed and tested at different inclinations and heat input levels. The device consists in an aluminum tube (I.D./O.D. 3/5mm), bended into a planar serpentine with five U-turns in the heated zone, with a 50 mm transparent section for the purpose of visualization. The tube is closed in a loop, firstly evacuated and then partially filled with FC-72, 50% vol. Each turn of the evaporator zone is wrapped with five electrical wiring heaters, giving a symmetrical heat power input at all the branches of the device. The condenser zone is embedded into a heat-sink and cooled by fans blowing air at 20°C. Sixteen T-type thermocouples are located on the external tube wall in the evaporator and condenser zones, while the fluid pressure is measured in the condenser zone adjacent to the transparent tube. The flow pattern is recorded by means of a high-speed camera up to 450 fps, as shown in Fig. 1. The device operational limits (start-up and dry-out heating levels at the different inclinations) are detected and the overall thermal performance is calculated for all the inclination and the heat power input tested. At the vertical orientation, the device shows promising results since it is able to dissipate up to 260 W keeping the evaporator zone below 85 °C.

Fig 1: Two phase flow visualization during tests.

Numerical Simulation of a Sodium Thermosyphon

M. Manzoni[1], T.W. Uhlmann[2], M.B.H. Mantelli[3], P. Eskilon[4] and M. Marengo[5]

[1] *Università degli Studi di Bergamo, Dept. of Eng. and App. Science, Viale Marconi 5, 24044 Dalmine (BG), Italy; University of Brighton, School of Computing, Eng. and Maths, Lewes Road, BN2 4GJ, Brighton, UK, miriam.manzoni@unibg.it*
[2] *Federal University of Santa Catarina, Mech. Eng. Dept., Bairro Trindade 88040-970 Florianópolis (SC), Brazil, tiago@labtucal.ufsc.br*
[3] *Federal University of Santa Catarina, Mech. Eng. Dept., Bairro Trindade 88040-970 Florianópolis (SC), Brazil, marcia.mantelli@ufsc.br*
[4] *Cleanergy, Theres Svenssons gata 15, 417 55 Göteborg, Sweden, per.eskilson@cleanergy.com*
[5] *Università degli Studi di Bergamo, Dept. of Eng. and App. Science, Viale Marconi 5, 24044 Dalmine (BG), Italy; University of Brighton, School of Computing, Eng. and Maths, Lewes Road, BN2 4GJ, Brighton, UK, m.marengo@brighton.ac.uk*

Extended Abstract

Thermosyphons are closed, wickless, two-phase heat transfer devices. They can transport heat at high rates over appreciable distances, virtually isothermally, without any requirement for external pumping devices, by making use of evaporation and condensation.

A gravity assisted conventional thermosyphon consists of an evacuated sealed tube containing a small pool of liquid on the bottom. Heat is applied on the bottom region (hot section), evaporating part of the pool liquid. Local pressure increases, pushing upward the vapor toward the cold section, where condensation occurs and heat is rejected to the external environment. The flow circuit is completed by the condensed liquid, forced back to the pool by gravity in the form of a thin film on the tube wall. High temperature thermosyphons, working above 800K, are already successfully implemented as thermal transport devices, for example, in nuclear or solar plants. Liquid metals with high boiling point, like sodium, lithium or potassium, are usually chosen as working fluids, for these kinds of applications.

However, most of the empirical correlations for the prediction of the thermosyphons internal coefficients of heat transfer, working limits and thermal performances were developed for water and might not be applicable to other fluids, such as alkaline metals. Therefore, a simple mathematical model able to simulate the behaviour of a thermosyphon independently of the employed working fluid is a powerful tool to help engineers to design and optimize thermosyphon assisted equipment. In this paper, a novel one-dimensional lumped parameters numerical code, for the thermo-hydraulic simulation of thermosyphons, is proposed. Solid components and fluidic domains are subdivided into a limited number of control nodes characterized by a particular thermodynamic status and connected by electrically analogical elements, as in Fig 1. The generated set of non-linear ordinary differential equations is then solved numerically. This model has been applied to prototype and optimized an industrial thermal management system for a Stirling Engine Unit powered by a hybrid heating source (i.e. combustion and solar energy). The results of the work will be presented in this paper. The final designed thermosyphons are able to transfer up to 40kW of power with a very high efficiency (the equivalent thermal conductivity computed is over 3000W/mK), at different inclination angles, from quasi horizontal to vertical position.

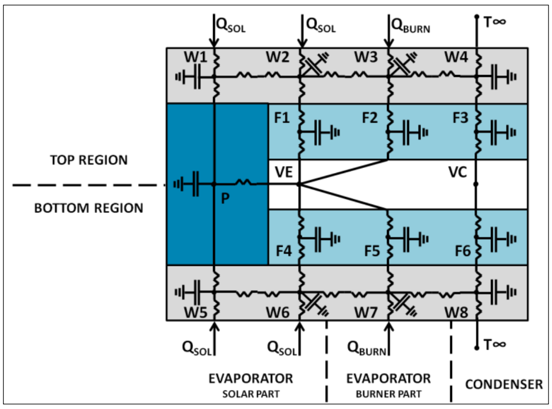


Fig. 1: Network scheme of the thermosyphon Wall, liquid Film and Pool. Vapor nodes are also indicated (VE and VC).

Numerical simulation of water-vapour condensation by means of a flow oriented scheme

D.P. Karadimou[1], N.C. Markatos[2]

[1] National Technical University of Athens, 9 Iroon Polytechniou Street, Zografou Campus, 15780, Greece, dkaradimou@gmail.com

[2] National Technical University of Athens, 9 Iroon Polytechniou Street, Zografou Campus,15780, Greece, N.Markatos@ntua.gr

Extended Abstract

In this study the numerical simulation of the water-vapour condensation by means of the flow oriented discretization scheme SUPER (Karadimou D.P. and Markatos N.C., 2012) is presented. A three-dimensional two-phase flow Eulerian model is developed within the CFD computer program PHOENICS, which considers the phases as interpenetrating continua (Spalding, 1978). The study focused on the heat and mass transfer interaction between humid air and liquid droplets in a real scale indoor space for various humidity conditions (Padfield T.). The model takes into account momentum interaction by means of interphase friction and calculates the humid air properties distribution inside the room. The numerical results are presented in terms of the absolute humidity ratio (kg H2O/kg of dry air) distribution on the cold surface of the walls. It is concluded that there is a qualitative agreement between the numerical solution and the expected humidity formation. Water droplets are forming on the wall surface as soon as the temperature reduces below the dew point of the water vapour, as expected.

The general form of all the governing differential conservation equations is:

$$\frac{\partial}{\partial t}\left(R_i \rho_i \phi_i\right) + div\left(R_i \rho_i \vec{u_i}\, \phi_i - R_i \Gamma_{\phi,i}\, grad\phi_i\right) = \dot{S}_{\phi,i} \qquad (1)$$

The water vapour mass condensed to water droplets is described by equation (2):

$$m_{wv} = 0.062 \cdot 1.0E-05 \cdot p \cdot \rho_{dry \cdot air} \cdot V_{cell} \cdot R_1 \qquad (2)$$

The interphase heat transport is described by equation (3):

$$q_{\text{int}} = C_{q,\text{int}} \cdot \left(T_1 - T_2\right) \qquad (3)$$

where $C_{q,\text{int}}$ the interphase thermal transfer coefficient and T_1, T_2 the bulk temperatures of each phase.

References

KARADIMOU, D.P., MARKATOS, N.C. 2012 *A novel flow oriented discretization scheme for reducing false diffusion in three dimensional (3D) flows: An application in the indoor environment.* Atmospheric Environment**. 61**, 327-339.

SPALDING, D.B. 1978 *Numerical Computation of Multiphase Flow and Heat-transfer. A Contribution to Recent Advances in Numerical Methods in Fluids.* In: Taylor C. & Morgan K, Editors. Pineridge Press, 139-167.

PADFIELD T. *Conservation Physics.* An online textbook in serial form: http://www.conservationphysics.org/atmcalc/atmoclc2.pdf

Surface tension of n-Butanol and steam mixture on metal surface

Saqib Raza Jivani[1], Dr Huasheng Wang[2].

[1] *Queen Mary University of London, Mile End Road E1 4NS, London, United Kingdom, s.r.jivani@qmul.ac.uk*
[2] *Queen Mary University of London, Mile End Road E1 4NS, London, United Kingdom*

Extended Abstract

Many researches have demonstrated that the surfaces with larger contact angle can enhance drop-wise condensation heat transfer due to the rapid removal of droplets and the high surface replenishment frequency (Kim & Kim, 2011). Binary mixture of steam with small concentration of alcohol has shown to promote drop wise condensation with fairly large contact angles. Recently, Fan et al showed the variation of contact angle with the concentration of aqueous solution of ethanol and propanol. The novelty of their investigation was the unstable behaviour of the wetting ability around the azeotropic point of the Propanol aqueous solution (Fan et al., 2011).

In this paper, surface tension of the aqeous solution of n-Butanol and steam were calcualted by measuring the contactangle of the mixture on the copper plate. The variation in the contact angle with different concentration were investigated closely focusing on the point of azeotropic. Contact angle is measured using the Axisymmetric Drop Shape Analysis Profile (ADSA-P).Contactangle hysteresis is also measured by taking the difference of advancing and reeceding cotnact angle. The model to pridict the contact angle of binary mixture on copper surface is derived from youngs equation.

References

Fan, L.-T. et al., 2011. Contact angle of Ethanol and n-Propanol aqueous solution on metal surfaces. *Chemical engineering Technology*, 34(9), pp.1535-42.

Kim, S. & Kim, K., 2011. Dropwise condensation modelling suitable for superhydrophobic surface. *Journal of Heat Transfer*, 133(8).

Ma, X.H., Rose, J.W. & Xu, D.Q., 2000. Advance in dropwise condesnation heat transfer. *Chemical Engineering*, 78, pp.87-93.

Effects of Vapour Velocity and Pressure on Marangoni Condensation of Steam-Butanol Mixtures on a Horizontal Tube

Saqib Raza Jivani[1], Dr Huasheng Wang[2].

[1] *Queen Mary University of London, Mile End Road E1 4NS, London, United Kingdom, s.r.jivani@qmul.ac.uk*
[2] *Queen Mary University of London, Mile End Road E1 4NS, London, United Kingdom*

Extended Abstract

It is well known that drop-wise condensation displays significantly higher heat transfer coefficients than film-wise condensation. This has escalated the interest of many researchers to investigate the surface tension criteria for promoting drop-wise condensation. Ma et al. (2000) proposed surface free energy difference criteria for determining condensation mode of steam on solid surfaces. While other have shown that surfaces with larger contact angle promotes drop-wise condensation (Kim & Kim, 2011). Marangoni condensation of binary mixtures results is pseudo-dropwise condensation when less volatile constituents has the higher surface tension. Surface tension gradient make condensate film unstable resulting in thinner film and lower thermal resistance. Hence, significant increase in heat transfer coefficient.

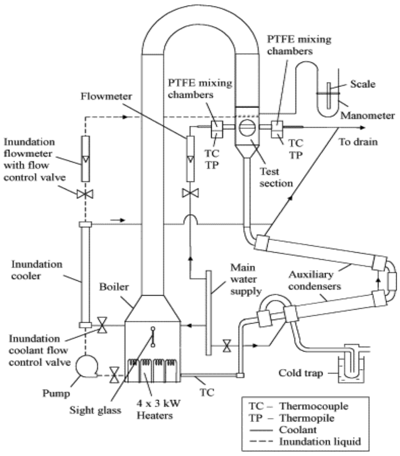


Figure 1: Test Rig for Measuring Condensation Heat transfer on horizontal tubes.

Heat-transfer measurement for condensation of steam-butanol mixture over a horizontal smooth copper tube is conducted. The fluid vapour flows vertically downward over the horizontal tube which is water cooled at atmospheric pressure. The vapour to surface temperature difference was measured by taking the difference of vapour temperature, measured using the K type thermocouple situated at the vapour upstream in the test section, and tube surface temperature, measured using four thermocouples embedded in the tube wall along the circumference of the tube. The maximum vapour velocity is limited due the input electrical power to the boiler. Test was conducted for varying coolant flow rates, varying vapour velocities and different concentrations of the Butanol in the mixture. Special care is taken to make sure there is no air trapped in the vapour. Video and photographs of the Marangoni condensation appearing on the metal surface is also captured.

References

Ali , H., Wang, H., Briggs, A. & Rose, J., 2013. Effects of Vapor Velocity and Pressure on Marangoni Condensation of Steam-Ethanol Mixtures on a Horizontal Tube. *Journal of Heat Transfer*, 108, pp.477-85.

Kim, S. & Kim, K., 2011. Dropwise condensation modelling suitable for superhydrophobic surface. *Journal of Heat Transfer*, 133(8).

Ma, X.H., Rose, J.W. & Xu, D.Q., 2000. Advance in dropwise condesnation heat transfer. *Chemical Engineering*, 78, pp.87-93.

Murase, T., Wang, H. & Rose, J., 2007. Marangoni condensation of steam-ethanol mixtures on a horizontal tube. *International Journal of heat and Mass Transfer*, 50(19-20), pp.3774-79.

Study of a New Wick Material for Capillary-Driven Heat Pipes

S. De Schampheleire[1], K. De Kerpel[2] and M. De Paepe[3]

[1] *Ghent University – Ugent, Sint-Pietersnieuwstraat 41, 9000 Ghent, Belgium, Sven.DeSchampheleire@ugent.be*
[2] *Ghent University – Ugent, Sint-Pietersnieuwstraat 41, 9000 Ghent, Belgium, Kathleen.DeKerpel@ugent.be*
[3] *Ghent University – Ugent, Sint-Pietersnieuwstraat 41, 9000 Ghent, Belgium, Michel.DePaepe@ugent.be*

Extended Abstract

A heat pipe has the advantage over other conventional methods that it can transport heat over a considerable distance with no additional power input to the system. Heat pipes with a wick material consisting of small diameter metal fibres of 12 µm are investigated. The container material is copper and the working fluid is water. The fibre mesh heat pipe is compared with two other wick structures: a screen mesh (145 meshes per inch) and a sintered powder wick. Both these heat pipes are commercially available. All three heat pipes have an outer diameter of 6 mm, a length of 200 mm. The heat pipes are tested in a vertical orientation, both gravity-opposed and gravity-assisted. In the gravity-opposed orientation the heat pipes are tested for a heat input up to 50 W and an operating temperature of 70°C. In the gravity-assisted orientation the heat pipes are tested up to 160 W and 120°C. The thermal resistance is used as performance indicators.

For the gravity-assisted orientation, the screen mesh wick clearly outperforms the fibre and sintered powder wick, due to its higher permeability and better ability to distribute the working fluid over the circumference of the wick. For heat inputs larger than 80 W, the trends for the screen mesh heat pipe and the fibre heat pipe are different. The thermal resistance of the screen mesh remains constant at 0.28 K/W while the fibre heat pipe starts to increase.

For the gravity-opposed orientation, the fibre and screen mesh heat pipe perform equally well. Both have a lower thermal resistance than the sintered powder heat pipe, as the small diameter fibres and fine mesh create more and very small capillary channels in comparison with the sintered powder wick. For high heat transfer rates, there is no significant difference between the screen mesh and the fibre heat pipe. No conclusions can be drawn when the thermal resistance is taken as a performance indicator for low heat transfer rates. Assuming a threshold thermal resistance of 0.55 K/W, the sintered powder heat pipe transfers 20 W of energy, while the other two heat pipes can transfer 50 W.

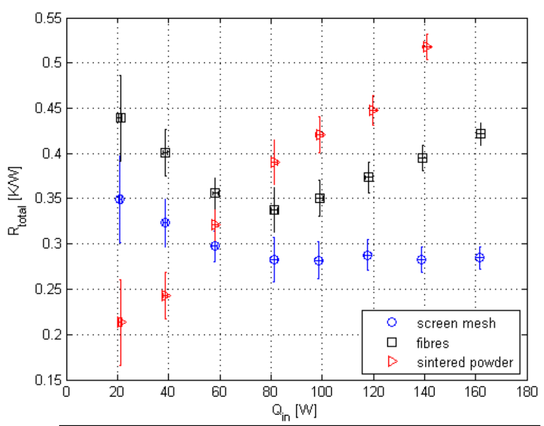


Fig. 1 Total thermal resistance against the input power for the gravity-assisting direction

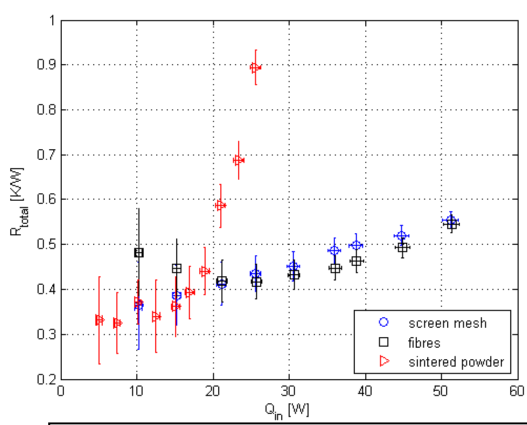


Fig. 2 Total thermal resistance against the input power for the gravity-opposed direction

Numerical Simulation of Flow Boiling in Micro-channels: Bubble Growth, Detachment and Coalescence

A. Georgoulas[1] and M. Marengo[1,2]

[1] *University of Bergamo, Viale Marconi 5, Italy, anastasios.georgoulas@unibg.it*
[2] *University of Brighton, Cockcroft Building,Lewes Road, Brighton, UK, M.Marengo@brighton.ac.uk*

Extended Abstract

Flow Boiling heat transfer within micro-channels has been a subject of extensive investigation during the last decades (see for example the review of Baldassari and Marengo, 2012). Due to the complexity, the development of comprehensive correlations and/or models for flow boiling has not been possible so far. However, more recently, numerical simulations have been proven to be capable of reliably predicting bubble dynamics and heat transfer characteristics. In the present paper, heat transfer and phase-change due to evaporation and/or condensation are coupled with a previously improved and validated, by the authors, Volume Of Fluid (VOF) model for adiabatic bubble dynamics (Georgoulas *et al.*, 2015). The model is initially verified with an existing analytical solution (Scriven, 1959) and with literature available experimental results of pool boiling [Lee *et al.,* 2003]. Then, the model is further applied for the conduction of various numerical simulations of flow boiling in microchannels, with single and multiple nucleation sites identifying some interesting observations regarding the bubble growth and detachment characteristics within the liquid cross-flow as well as regarding the coalescence of bubbles that have detached from different nucleation sites (see Figure 1 for example). The analysis of the numerical results reveals that the proposed numerical model contitutes a quite promising tool for the investigation of the complex sub-processes that occur during flow boiling in microchannels.

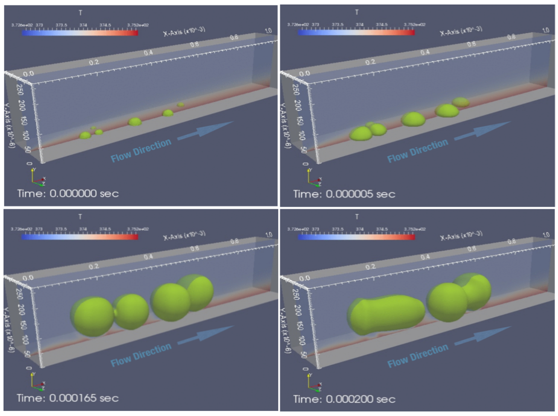


Fig 1: Bubble growth, detachment and coalescence during flow boiling of water in a micro-channel

References

BALDASSARI C., & MARENGO M. 2012 *Flow boiling in microchannels and microgravity.* Prog. Energ. Comb. Sc. **39 (1)**, 1-36.

GEORGOULAS A., KOUKOUVINIS P., GAVAISES M., & MARENGO M. 2015 *Numerical investigation of quasi-static bubble growth and detachment from submerged orifices in isothermal liquid pools: The effect of varying fluid properties and gravity levels.* Intl. J. Mult. Fl. **74**, 59-78.

LEE, H.C., OH, B.D., BAE, S.W., & KIM M.H. 2003 *Single bubble growth in saturated pool boiling on a constant wall temperature surface.* Intl. J. Mult. Fl. **29 (12)**, 1857–1874.

SCRIVEN, L.E. 1959 *On the Dynamics of Phase Growth*. Chem. Eng. Science **10**, 1-3.

Numerical Simulation of Pool Boiling: The Effects of Initial Thermal Boundary Layer, Contact Angle and Wall Superheat

A. Georgoulas[1] and M. Marengo[1,2]

[1] *University of Bergamo, Viale Marconi 5, Italy, anastasios.georgoulas@unibg.it*
[2] *University of Brighton, Cockcroft Building,Lewes Road, Brighton, UK, M.Marengo@brighton.ac.uk*

Extended Abstract

Boiling heat transfer has been a subject of extensive investigation during the last decades. Since the underlying sub-processes in nucleate boiling, involve quite complex physics, the development of comprehensive correlations and/or models has not been possible so far. However, more recently, numerical simulations of the boiling process have proven to be capable of reliably predicting bubble dynamics and heat transfer characteristics. In the present paper, heat transfer and phase-change due to evaporation and/or condensation are coupled with a previously improved and validated, by the authors, Volume Of Fluid (VOF) model for adiabatic bubble dynamics (Georgoulas *et al.*, 2015). The model is initially verified with an existing analytical solution (Scriven, 1959) for cases of evaporating bubble growth in a superheated liquid domain. Apart from this, the predictions of the proposed model regarding the bubble detachment diameter and time are also validated against literature available experimental results of pool boiling [Lee *et al.,* 2003]. Then, the validated and optimised version of the model is further applied for the conduction of wide range of parametric numerical simulations, identifying the effects of (a) the initial thermal boundary layer thickness, (b) the contact angle between the liquid/vapour interface and the heated plate, as well as (c) the plate superheat, on the bubble detachment characteristics. The analysis of the numerical results reveals that the bubble growth and detachment characteristics, are highly sensitive to the thickness of the initially developed thermal boundary layer (see Figure 1 for example) as well as to the wettabilty (imposed contact angle) and superheat of the heated plate.

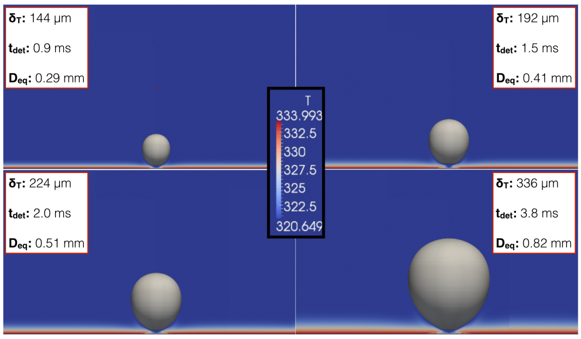


Fig 1: Effect of initial thermal boundary layer thickness on the bubble detachment characteristics

References

GEORGOULAS A., KOUKOUVINIS P., GAVAISES M., & MARENGO M. 2015 *Numerical investigation of quasi-static bubble growth and detachment from submerged orifices in isothermal liquid pools: The effect of varying fluid properties and gravity levels.* Intl. J. Mult. Fl. **74**, 59-78.

LEE, H.C., OH, B.D., BAE, S.W., & KIM M.H. 2003 *Single bubble growth in saturated pool boiling on a constant wall temperature surface.* Intl. J. Mult. Fl. **29 (12)**, 1857–1874.

SCRIVEN, L.E. 1959 *On the Dynamics of Phase Growth*. Chem. Eng. Science **10**, 1-3.

Evaporation/boiling heat transfer characteristics in an artery porous structure

L. Bai[1,2], L. Zhang[1], J. Guo[1],G. Lin[1] and D. Wen[2]

[1] Beihang University, Beijing, China, bailizhan@buaa.edu.cn
[2] University of leeds, Leeds, UK, d.wen@leeds.ac.uk

Extended Abstract

Boiling heat transfer is one of the most efficient heat transfer modes, which has been widely applied in many areas. However, the relatively low critical heat flux (CHF) for boiling heat transfer on a flat surface limits its applications in some special situations, and the enhancement of CHF for boiling heat transfer has become a very important issue to be resolved.

In this paper, based on the concept of "phase separation and modulation", artery porous structure was proposed to enhance the CHF for boiling heat transfer by actively formulating the flow paths for liquid supply and vapor venting. In the experiment, the porous structure was made of sintered copper particles, and pure water was used as the working fluid. Table 1 presents the basic parameters of the artery porous structure, and Fig.1 shows the superheat dependence of heat flux for evaporation/boiling heat tranfer in an artery porous structure. As transparent material (pyrex glass) was used for the boiling pool wall, the liquid/vapor distribution and movement during the evaporation\boiling process at different stages can be easily observed. Experimental results indicate that the CHF for boiling heat transfer can be enhanced considerably through the employment of an artery porous structure, and the maximum heat flux of 380W/cm^2 on a heating area of 0.78 cm^2 was achieved without the occurrence of dryout. Meanwhile, the effects of pore size and artery depth on the evaporation/boiling heat transfer performance were also experimentally investigated.

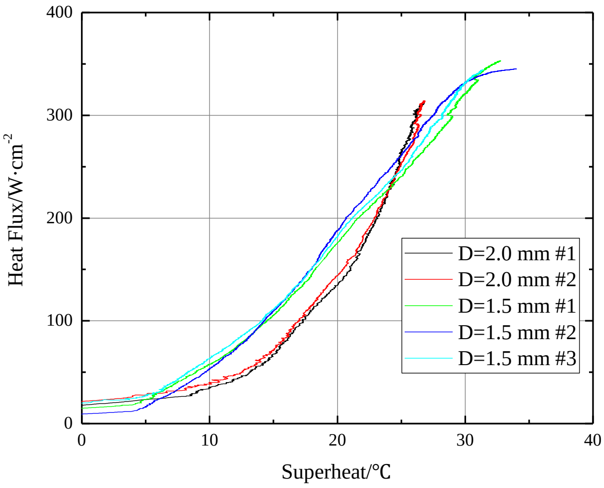


Fig.1 Superheat dependence of heat flux for evaporation/boiling heat transfer in an artery porous

Table 1 Basic parameters of the artery porous structure in the experiment

Item	Parameter
Porous structure thickness(T)/mm	2.0
Artery width(W1)/mm	1.0
Artery depth(D)/mm	1.0-1.5-2.0
Fin width(W2)/mm	1.2

References

Li, C., Peterson, G. P., & Wang, Y. 2006 Evaporation/Boiling in Thin Capillary Wicks (l) - Wick Thickness Effects. Journal of Heat Transfer, **128**, 1312-1319.

Weibel, J. A., Garimella, S. V., & North, M. T. 2010 Characterization of evaporation and boiling from sintered powder wicks fed by capillary action. International Journal of Heat and Mass Transfer, **53**, 4204-4215

Experimental study on direct solar energy absorption of Au-Cu hybrid nanofluids

L. Bai[1,2], A. Zeiny[2], M. Amjad[2], G. Raza[2], G. Lin[1] and D. Wen[2]

[1] *Beihang University, Beijing, China, bailizhan@buaa.edu.cn*
[2] *University of leeds, Leeds, UK, d.wen@leeds.ac.uk*

Extended Abstract

To develop sustainable and renewable energy technologies, especailly solar energy related, becomes important and urgent in securing our energy future. In order to enhance the absorption efficiency of solar thermal collectors, nanoparticle-based direct absorption concept has been proposed and investigated both theoretically and experimentally in recent years.

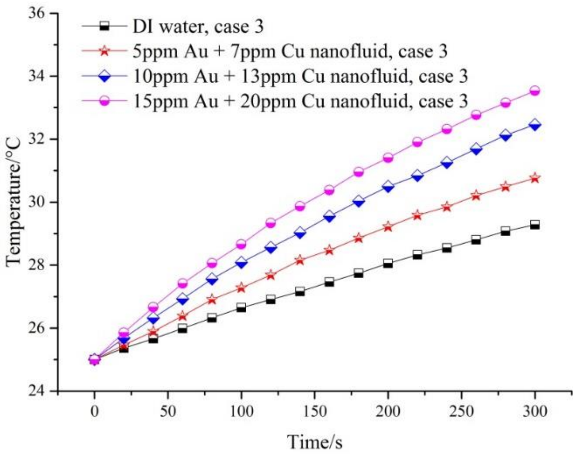


Fig. 1 Temperature variations of Au-Cu hybrid nanofluids with different concentrations

For each kind of nanoparticles, it only has strong absorption capability within a narrow solar spectrum. In order to further enhance the absorption efficiency, it is necessary to improve the solar absorption in the whole solar spectrum, and the application of hybrid nanofluids, i.e. a mixture of different kinds of nanoparticles with the solar absorption peak at different wavelengths dispersed into the base fluid, is a practically feasible method.

In this paper, Au, Cu and Au-Cu hybrid nanofluids with different concentrations were prepared to experimentally investigate their solar absorption performance under a solar simulator. The Au nanofluids were synthesized by the citrate reduction method, and the Cu nanofluids were prepared by dispersing commercial Cu nanoparticles into the base fluid. The Au-Cu hybrid nanofluids was made by mixing the Au and Cu nanofluids with appropriate ratio. The experiment was conducted in 3 cases, and experimental results showed that both Au and Cu nanofluids can enhance the absorption efficiency of solar thermal energy, and the larger the concentration of the nanofluids, the higher the absorption efficiency. Meanwhile, the cover effect and the reflection effect were examined in the experiment. Fig. 1 shows the temperature variations of Au-Cu hybrid nanofluids with different concentrations in case 3. The application of Au-Cu hybrid nanofluids can improve the solar absorption in the whole solar visible spectrum, and for a liquid layer 3mm in thickness, the solar absorption efficiency was enhance from 19.6% of DI water to about 40.0% of hybrid nanofluids with15ppm Au and 20ppm Cu.

References

Zhang, H., Chen, H., Du, X., & Wen, D. 2014 Photothermal conversion characteristics of gold nanoparticle dispersions, Solar Energy **100**, 141-147

Ladjevardi, S.M., Asnaghi, A., & Izadkhast, P.S. 2013 Applicability of graphite nanofluids in direct solar energy absorption, Solar Energy **94**, 327-334

Modelling of a tank containing a paraffin as phase-change material for cold storage applications

Javier Biosca-Taronger, J. Payá and J. M. Corberán

IIE, Universitat Politècnica de València, Camino de Vera s/n, ed. 8E, cubo F 5º, 46022 Valencia, Spain, corberan@iie.upv.es

Extended Abstract

This work aims to develop and validate a numerical model of a storage tank using a paraffin as phase-change material (PCM) for cold storage applications. This model reproduces the performance of a tank with around 235 l PCM immersed around the heat exchanger, which is a set of 8 spiral shaped coils placed parallel wise on horizontal planes. The paraffin used is the RT8 from RUBITHERM, which has a phase-change temperature in the range 3-8ºC. The storage tank has a capacity of around 6 kWh in the temperature range 0-10 ºC. A 2D enthalpy model has been developed, where one dimension corresponds to the axial direction of the heat exchanger tube and the other is the radial direction of the tube along which the PCM solidifies and melts. The volume of PCM is discretized in these two dimensions and the heat transfer between the nodes and their corresponding state and temperatures are calculated. The nodes are characterized by their enthalpy state, and the corresponding temperature is obtained for heat transfer calculations. Therefore a key input to the model is the enthalpy-temperature curve of the tested PCM. The enthalpy-temperature curve has been measured by means of the DSC method with heating and cooling rates of 0.5 K/min at the laboratories of the Centro de Innovación Tecnológica de Acciona Infraestructuras.

The numerical model of the tank has been successfully validated, since the results have shown a good agreement between the predicted and measured thermal power of the tank. The model has been compared with experimentals tests (López-Navarro *et al.* 2014) for different conditions for the heat transfer fluid involving flow rates of 1000 and 2000 kg/h and supply temperatures of -1 and 1 for solidification tests and 9 and 13 for melting tests.

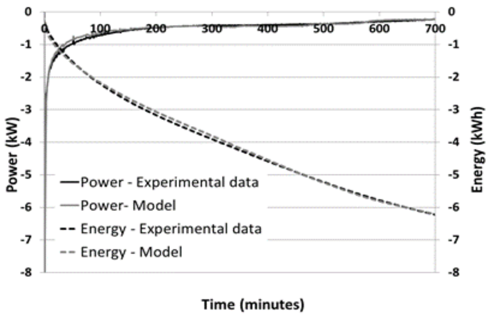


Fig 1: Thermal power and energy in a melting test with 9 ºC and 2000 kg/h.

The model has helped to understand the influence of the phenomena ocurring in the solidifying and melting processes. The melting process is particularly complex since the contact between the solid paraffin and the coils is variable throughout the tests.

References

LÓPEZ-NAVARRO A, BIOSCA-TARONGER J, CORBERÁN JM, PEÑALOSA C, LÁZARO A, DOLADO P & PAYÀ J. 2014 *Performance characterization of a PCM storage tank.* Appl Energy; **119**, 151-162.

Thermal Analysis of a Novel Solar-biogas Hybrid System Integrated with PCM Insulation Closure

Haowei LU[1], Yong LU[2], Ye TIAN[3] Rui XIAO[4]

[1,2,3,4] *Jiangsu Province Key Laboratory of Solar Energy Science and Technology, School of Energy and Environment, Southeast University, Nanjing, 210096, China,*
[1] 491773092@qq.com, [2] Speaker: luyong@seu.edu.cn, [3]1277061434@qq.com, [4]ruixiao@seu.edu.cn

Extended Abstract

It is a key energy policy in China that developing biogas technology is to crack energy short problem in its rural areas. A novel biogas-solar hybrid system intedgred with phase change insulation closure is developed to collect the daily solar energy for satisfying the heat loading required by biogas fermentation tank which should keep about 35℃ optimum efficient fermentation temperature in each winter day. A pilot water tank with 1m^3 volume was used to simulate fermention tank. Its insulation closure inculded an inner 50mm thick polyurethane, a middle 30mm thick paraffin as phase change material (PCM) and an outside 200mm thick polyurethane. Both numerical and experimental results certificate that heat loss disspiated from the clousre in a day can be reduced from 20% to 2% and the system exnergy efficiency can be improved about 27.1% under the condition of using PCM insultion layer. This sugests that the proposed solar-biogas hybrid system could be a promising biogas fermentation devices used in cold rural regions of China.

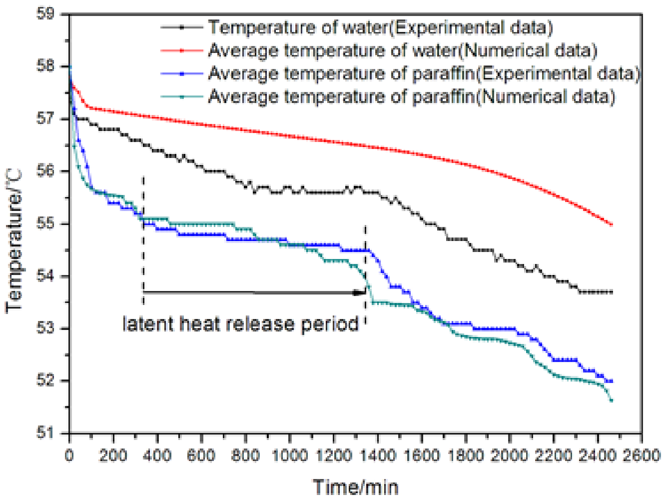


Fig.1. Temperature variation of the solid thermal package in 40 hours

Acknowledgements

Financial support from the National Key Technology R&D Program (No. 2013BAJ10B12) and the National Natural Sciences Foundation of China (No. 51376048) are acknowledged with gratitude.

References

[1] Chen Y, Yang G, Sweeney S, et al. Household biogas use in rural China: a study of opportunities and constraints[J]. Renewable and Sustainable Energy Reviews, 2010, 14(1): 545-549.
[2] Liu Jianmin, Principle of Solar Energy Utilization, Technology and Engineering, Beijing: Electronic industry press, 2010.
[3] Yiannopoulos A C, Manariotis I D, Chrysikopoulos C V. Design and analysis of a solar reactor for anaerobic wastewater treatment[J]. Bioresource technology, 2008, 99(16): 7742-7749.
[4] Kocar G, Eryasar A. An application of solar energy storage in the gas: Solar heated biogas plants[J]. Energy Sources, Part A, 2007, 29(16): 1513-1520.
[5] Abid M, Karimov K S. Biogas Digester with Simple Solar Heater[J]. IIUM Engineering Journal, 2012, 13(2).

14th UK Heat Transfer Conference 2015
Edinburgh (Sept 7 – Sept 8, 2015)

Pool Boling on Superhydrophobic/philic Surfaces

Y. Takata[1,2], M. Yamada[1], B. Shen[1], S. Hidaka[1], M. Khono[1,2] and K. Takanashi[2,3]

[1] *Department of Mechanical Engineering, Kyushu University, 744 Motooka, Nishi-ku, Fukuoka, Japan,* takata@mech.kyushu-u.ac.jp
[2] *International Institute for Carbon-Neutral Energy Research (WPI-I2CNER), Kyushu University, Japan*
[3] *Department of Aeronautics and Astronautics, Kyushu University, Japan*

Extended Abstract

Wettability of the heat transfer surface is one of the key parameters which influences boiling heat transfer. The wettability is categorized as *hydrophilic* and *hydrohobic* depending on the contact angle. Both surfaces have advantage and disadvantage in boling. The hydrophilic surface has high critical heat flux (CHF) and high minimum heat flux (MHF), but has lower heat transfer performance and higher wall superheating at onset of boiling (ONB). On the other hand, the hydrophobic surface has lower wall superheating at ONB and works as excellent bubble nucleation site, but as a disadvantage it has lower CHF. By combining advantages of hydrophilicity and hydrophobicity, we can have an ideal boiling surface that has low ONB wall superheating, high heat transfer performance in nucleate boiling and higher CHF. One such surface is that has hydrophobic spots on hydrophilic surface as shown in Fig.1.

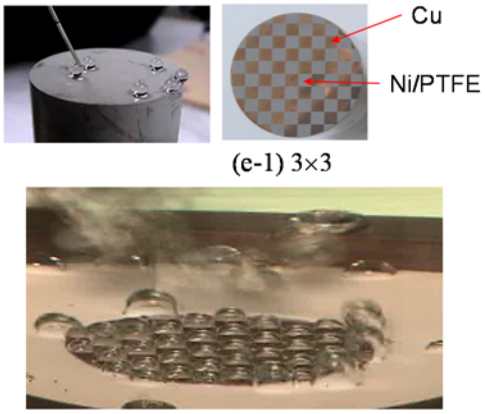


Fig.1 Boiling from hydrophobic spots

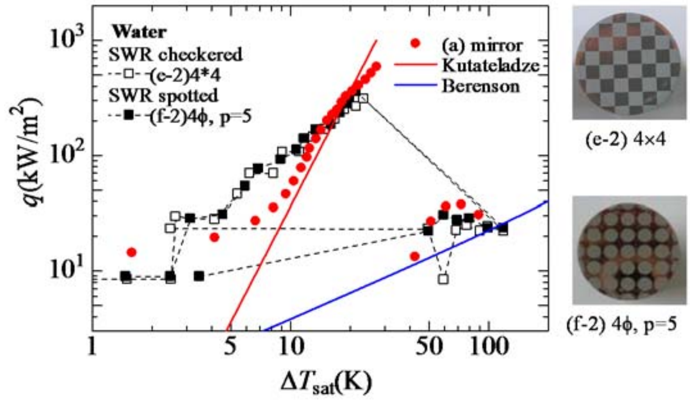


Fig. 2 Boiling performance of superhydrohobic checkered and spotted surfaces at saturated condition

The surface shown in Fig. 1 is a copper with superhydrophobic spots where boiling bubbles are generated. The heat transfer characteristis are shown in Fig.2. The figure compares boiling curves for three surfaces; a mirror finished copper surface, superhydrophobic checkered and spottted surfaces. Both surfaces with checkered and spotted superhydrophobic have by seven times larger heat transfer performance than that of the mirror surface at lower heat flux region. In this boiling mode, the superhydrophobic spots are always covered with vopor domes that are emitted periodically. Therefore, the excellent heat transfer performance in this boiling mode attributes to the bubble agitation effect and not to the heat transport by latent heat of vaporization.

In subcooled condition, we have observed some strange behaviors. (1) the superheating at ONB becomes negative. (2) periodic bubble growth and departure from a hydrohobic spot are observed and its frequency ranges 10^{-3}-10^{-2} s^{-1} depending on surface temperature as shown in Fig. 3. The bubble frequency is by 1000 time smaller than that at saturated boiling.

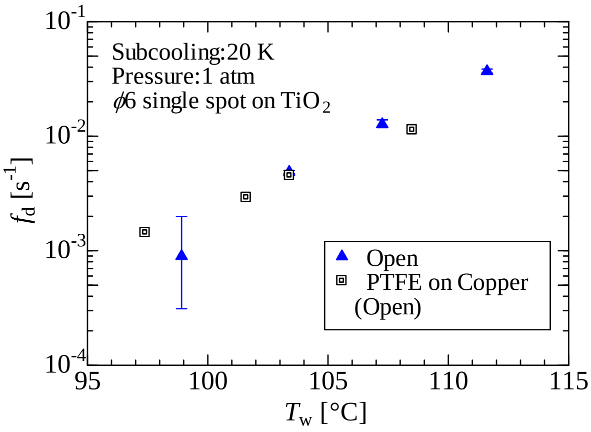


Fig. 3 Frequency of bubble departue

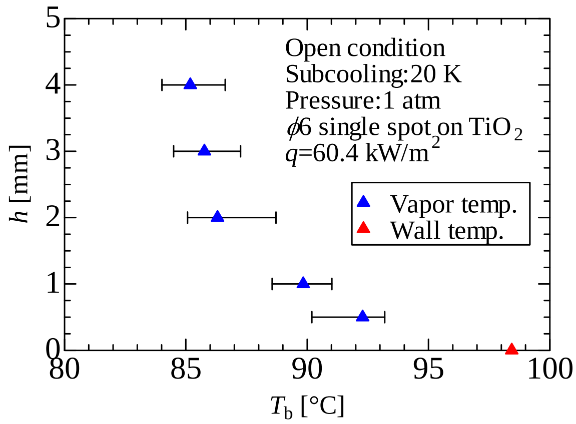


Fig. 4 Temperature profile inside a bubble

The authors think that these strange phenomena are caused by dessolved air. First, we have measured temperature profile inside the bubble by small thermocouple and the result is shown in Fig. 4. Temperature insied the bubble is much lower than saturated temperature and therefore, inside of the bubble consists of a mixture of water vapor and air. To make clear the effect of dissoved air, we have developed a experimental apparatus with vacuum pump and belows. The experiments have been done with and without dessolved air. Fig. 5 shows boiling curves and bubble behaviors both for with and without dissolved air. “Open” and “Closed” are with and without dissolved air, respectively.

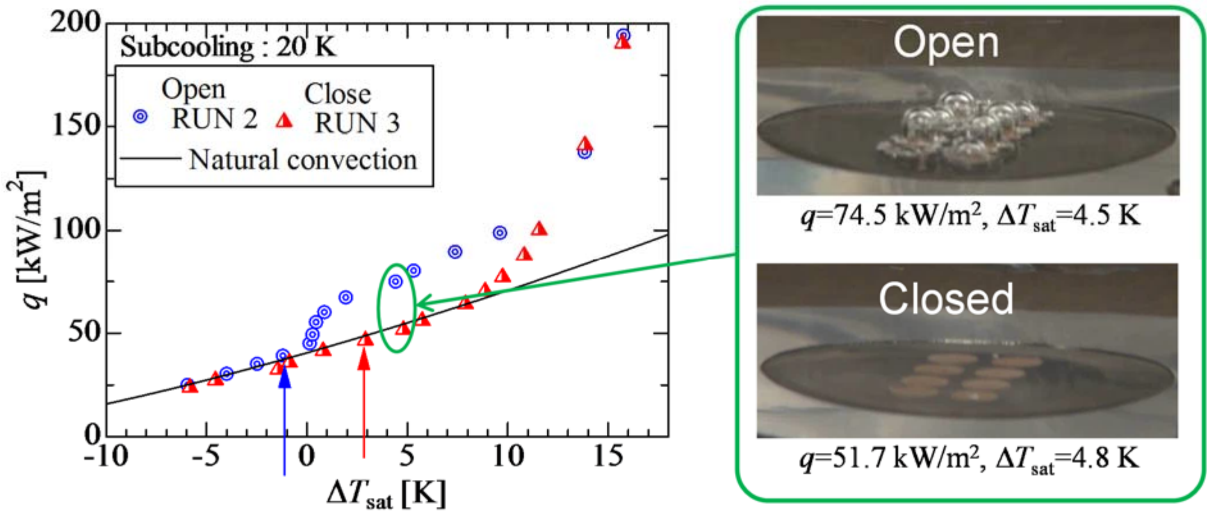


Fig. 5 Boiling curves and bubble behaviors for with and without dissolved air

Some important results were obtained from the experiment. (1) In closed condition, the superheating at ONB is positive, while that in open condition is negative. (2) Heat tranfser performance of open condition is larger than that of closed condition. Both curves meet each other at higher superheating. (3) The size of bubbles for open and closed conditions differ each other. The bubble for open condition is much larger than that of closed condition. In conclusion, the inside of the bubble for open condition is a mixture of water vapor and air that was originally dissolved in the liquid water.

The Effect of Secondary Flow on Developing Flow in the Transitional Flow Regime

M. Everts, J.P. Meyer

Department of Mechanical and Aeronautical Engineering, University of Pretoria, South Africa

Extended Abstract

According to a recent review paper by Meyer (2014), the transitional flow regime has been mainly investigated by Professor Afshin Ghajar from Oklahoma State University and his co-workers, and Professor Josua Meyer from the University of Pretoria and his co-workers. Limited research has been done on tube flow in the transitional flow regime and the previous work that has been done focussed primarily on fully developed flow or the average measurements of developing flow across a tube length (Everts, 2014). Therefore, the heat transfer characteristics of developing flow in the transitional flow regime have not yet received the required attention. The purpose of this study was to experimentally investigate the heat transfer characteristics of developing flow in the transitional flow regime.

Approximately 400 measurements were taken at Reynolds numbers between 500 and 10 000 at three different constant heat fluxes. Water was used as the test fluid and the Prandtl number ranged between 3 and 7. The test section was a smooth circular tube with an inside diameter and length of 11.52 mm and 2.03 m, respectively and a square-edged inlet was used.

Secondary flow was investigated by plotting the ratios of the local heat transfer coefficients at the top and bottom of the test section. It was found that secondary flow effects were supressed near the inlet, but became significant as the thermal boundary layer increased along the tube length. The secondary flow effects increased with increasing heat flux due to the increasing temperature difference between the surface and fluid, but decreased with increasing Reynolds number since the thickness of the thermal boundary layer decreased.

The average heat transfer data included both developing (laminar and transitional flow regimes) as well as fully developed (turbulent flow regime) data. It was concluded that the laminar heat transfer coefficients increased with increasing heat flux since the flow was still developing and due to the buoyancy-induced secondary flow. In the turbulent and low-Reynolds-number-end regimes, the heat transfer coefficients increased with increasing Reynolds number due to the enhanced mixing inside the tube. The turbulent fluid motion also suppressed the secondary flow effects, thus heating had no significant influence on the heat transfer coefficients in these two regimes. Although transition was delayed for increasing heat fluxes, heating did not have a significant influence on the magnitude of the heat transfer coefficients in the transitional flow regime.

References

EVERTS, M. 2014 *Heat transfer and pressure drop of developing flow in smooth tubes in the transitional flow regime*. Univerisity of Pretoria, Pretoria.

MEYER, J.P. 2014 *Heat transfer in tubes in the transitional flow regime.* 15th International Heat Transfer Conference, Kyoto, Japan.

Thermal conductivities of annular packed beds in axial fluid flow

D.H.Glass[1] and G. R. Duursma[2]

[1] *The University of Edinburgh, UK, Don.Glass@ed.ac.uk*
[2] *The University of Edinburgh, UK, Gail.Duursma@ed.ac.uk*

Extended Abstract

The industrial synthesis of organo-metallic substances is typified by the kilomolar scale production of the Grignard reagent magnesium methyl bromide by the reaction of solid magnesium metal with methyl bromide in tetrahydrofuran solution. This is carried out in a batch-wise reactor. The process is hazardous for a variety of reasons, including the very high heat of reaction and the unstable nature of the product. The development of other processes requires the production of a Grignard reagent with the methyl group replaced by an unsaturated radical. Unfortunately, this material can self-react and for this reason the product yield and purity from a batch reactor would be unacceptable. The standard answer to this problem is to carry out the process in a plug flow reactor, a suitable example of which would be a cooling jacketed tube packed with magnesium, subjected to an axial flow of reactant dissolved in tetrahydrofuran. The latter has an atmospheric boiling point of only 65°C and removal of the heat of reaction is a problem. Boiling of the solvent at the tube axis would mix the contents and totally disrupt the plug flow pattern desired.

The literature contains almost nothing on the effective thermal conductivity of packed beds of metal particles perfused by organic liquids at low velocities. For this reason experimental studies were conducted on a small annular packed bed with heat transfer in the radial direction from an axial heating element to a cooling jacket. The bed was perfused with water or kerosene, entering at ambient temperature and leaving at a higher temperature.
The results are presented (e.g. in Figure 1) in the form of effective thermal conductivity of the perfused solid material as a function of the fluid axial velocity. The effective thermal conductivity is seen to vary with axial fluid velocity in a linear way under these conditions. Analysis of the equations of heat transfer in such systems leads to a feasible design for the proposed reactor, avoiding the hot-spot problem that was anticipated.

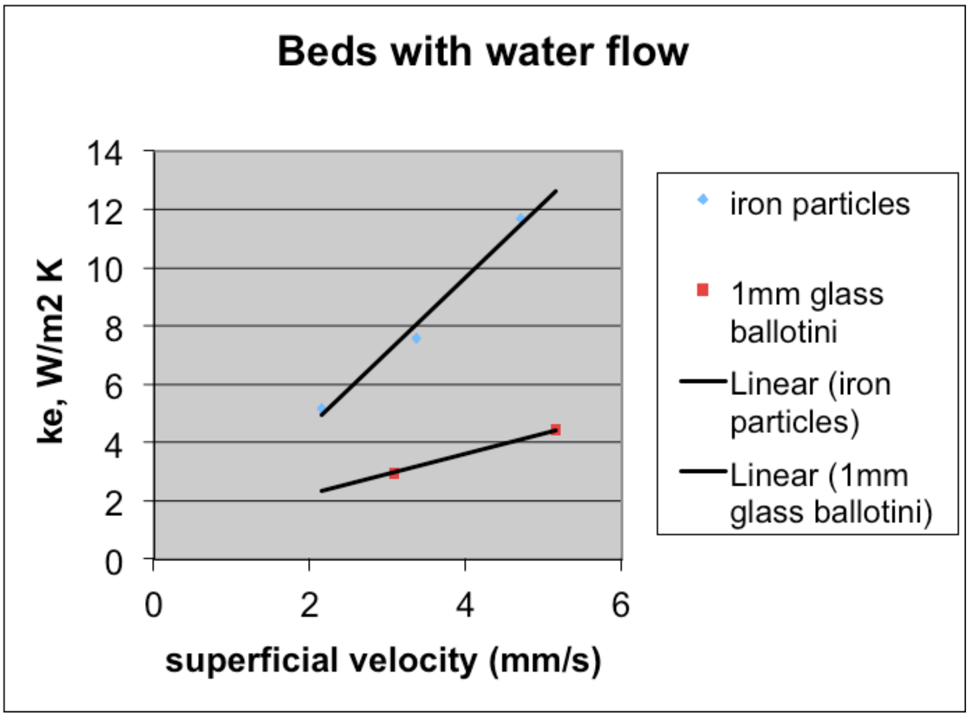


Figure 1 Beds with water flow

Heat Transfer Mechanisms for Single Rising Taylor Bubbles

A. Scammell[1] and J. Kim[1]

[1]University of Maryland,2181 Glenn Martin Hall, College Park, MD, 20740, USA, ascammel@umd.edu

Extended Abstract

As a regime frequently encountered in two-phase thermal management applications, the development of mechanistic models for slug flow heat transfer is important for future heat exchanger designs. This work studied the heat transfer and flow characteristics of individual, rising vapor Taylor bubbles using an infrared thermography technique (Kim et al., 2012) to obtain local heat transfer measurements, high-speed flow visualization, and liquid film thickness measurements. It was found that a slight enhancement in heat transfer over single-phase flow occurs in the liquid film as liquid is accelerated by gravity towards the bubble tail. The largest enhancement, however, was present in the bubble wake, where vortices created mixing and disruption of the wall thermal boundary layer. A time trace of the heat transfer coefficient as a bubble traverses is seen in Fig. 1. The maximum wake heat transfer enhancement was found to decrease with increasing background liquid velocity. Measurements of the fully developed liquid film thickness were shown to agree well with correlations for stabilized, laminar, falling films.

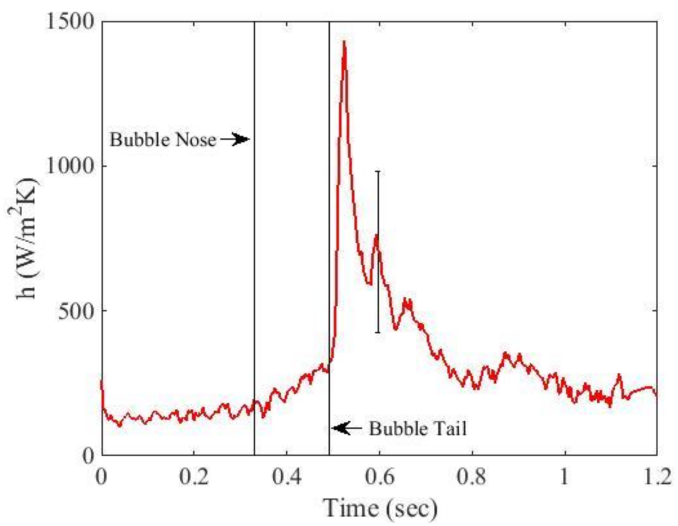


Figure 1: Time trace of heat transfer coefficient at a single location on the tube for a bubble with L=21 mm, rising in co-current flow at U_L=72 mm/s, and imposed heat flux of q"=2.4 kW/m^2.

References

KIM, T.H., KOMMER, E., DESSIATOUN, S., & KIM, J. 2012 *Measurement of two-phase flow and heat transfer parameters using infrared thermometry*. Int. J. Multiphase Flow **40**, 56-67.

Thermal characterisation of compact heat exchangers for automotive air conditioning

B. Torregrosa-Jaime, J. Payá, J. M. Corberán

IIE, Universitat Politècnica de València, Camino de Vera s/n, ed. 8E, cubo F 5º, 46022 Valencia, Spain, corberan@iie.upv.es

Extended Abstract

The use of air-conditioning (AC) in the automotive industry increases the energy consumption by more than 20%. In full-electric vehicles (EVs) the waste heat from the motor and electronics is not enough to warm the cabin, so the AC is also employed in winter as a heat pump. In vehicle applications, simulation tools comprising the whole AC system are essential to predict the performance under the wide range of possible operating conditions. Compact water-to-water reversible heat pumps can achieve high system efficiency while facilitating assembly (Bjurling *et al.* 2014). Key components in these AC systems are the water to air heat exchangers (HEX), which due to space restrictions must be considerably compact and be integrated in a small casing together with the fan, still they must operate efficiently in both heating and cooling modes.

This paper presents the results of a full thermal characterisation of a commercial compact HEX for this novel application in both heating and cooling of a vehicle cabin. In order to do so, a comprehensive experimental campaign has been carried out as a part of the EC funded ICE Project, 2014. A methodology to fit a single correlation of the overall heat transfer coefficient of the HEX for both working modes is explained. The developed correlation has helped to build a HEX model that considers dehumidification and can be integrated into a dynamic model of the whole mobile AC system.

Figure 1 compares the predicted temperature difference with the experimental results. The maximum deviation in the fluid outlet temperature is 0.5 K and in the air outlet temperature, 1.5 K. These results validate the model and confirm the excellent accuracy and robustness of the approach employed.

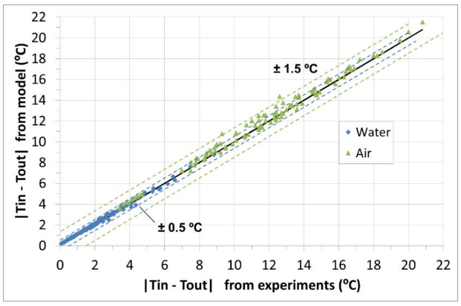


Fig 1: Predicted (dots) and measured (black line) temperature differences across the HEX.

The paper describes the heat exchanger geometry, the experimental campaign, the results and the model employed for the heat transfer characterisation under the wide range of tested operating conditions and the obtained correlations for both heating and cooling modes.

References

BJURLING, F., CORBERÁN, J.M., PAYÁ, J. & TORREGROSA-JAIME, B. 2014 *Control strategies for the air-conditioning in electric vehicles*. Proceedings of the V Iberoamerican and VII Iberian Congress of Cooling Techniques **7**, 769-777. June 18-20 2014, Tarragona, Spain.

ICE Project FP7, 2014, www.ice-mac-ev.eu.

Improving thermocouple measurement accuracy for Solid Oxide Fuel Cell application

F. Barari[1], R. Morgan[2] and P. Barnard[3]

[1] *University of Brighton, , Cockcroft Building, Brighton, BN2 4GJ, UK, farzad.barari@cerespower.com*
[2] *University of Brighton, , Cockcroft Building, Brighton, BN2 4GJ, UK, r.morgan2@brighton.ac.uk*
[3] *Ceres Power Limited, Viking House, Foundry Lane, Horsham RH13 5PX, UK, info@cerespower.com*

Extended Abstract

Ceres Power Limited has developed a novel and highly differentiated Steel Cell SOFC technology which offers low cost, robust and high efficiency systems that converts fuel directly into electrical power. The thermometry system accuracy plays an important role in achieving higher efficiency, better thermal management, longer product life and lower environmental impact. Initial modelling results show that ±10°C temperature measurement inaccuracy in stack inlet and outlet air temperature measurement could reduce the net efficiency and consequently reducing customers' saving by up to £400 over the product lifetime.

In order to manufacture low cost SOFC systems, conventional mineral insulated base metal thermocouples (TC) are the only available cost effective temperature sensor that capable of operating at high temperature. Even these TCs could not withstand corrosion and drift at high temperature without mineral insulated metal sheath. Moreover, the sheath and wire diameter should be large enough to prevent TC drift. However, in non-isothermal condition increasing TC diameter could increases measurement error due to low convection heat transfer between the hot gas and TC tip, larger radiation effect and larger immersion error. Due to thermal interaction between the adjacent components in SOFC systems such as burner, heat exchangers and reformer a large temperature gradient could exist between the gas and the wall which could bias the reading.

Result from the 1D TC model[1] shows that pipe surface temperature, gas velocity, gas measured temperature, TC length, TC diameter and TC emissivity affect temperature measurement accuracy. In order to validate the modelling data and testing the thereof variables a test piece was designed and built. The results show that wall temperature, TC size and flow rate had the most impact on temperature measurement accuracy. The results show that the reading difference between 0.5mm and 1.5mm TC was up to 40°C. A method was developed to estimate the true gas temperature. The results were experimentally validated and good agreement was observed.

References

1- A Design of Experiments (DOE) approach to optimise temperature measurement accuracy in Solid Oxide Fuel Cell (SOFC). F Barari, R Morgan and P Barnard, Journal of Physics: Conference Series, 547 012004 (2014).

Investigation on thermal efficiency and cost-effective mode of a solid thermal package by utilizing off-peak power

Wenjun Duan[1], Yong LU[2], Lei WU[3] Haowei LU[4]

[1,2,3,4] *Jiangsu Province Key Laboratory of Solar Energy Science and Technology, School of Energy and Environment, Southeast University, Nanjing, 210096, China,*
[1] 1277061434@qq.com, [2] luyong@seu.edu.cn, [3] Speaker: leiwu@seu.edu.c , [4] 1277061434@qq.com

Extended Abstract

In China east electric grid, the peak power consumption reached 2.1×10^9 KWH in 2014. But its off-peak load only occupied 1/3 of its maxium value. Thermal storage technique is one of promise ways to shift power grid load from peak to off-peak. In this study, a 216KW solid thermal package system was built for utilizing off-peak power through electric heater. The package was made from 2.5m^3 volumetric magnesia bricks with 300°C storage temperature. Conduction oil was used as thermal medium among the package and thermal terminals. In order to investigate the system thermal performance, parameters of thermal efficiency, heat loss and system exergy were evaluated by experiments under 50%, 80% and 100% electric-heat power and natural cooling processing. Based on the experimental results and exergy efficiency analsysis, a cost-effective mode was proposed for economically running the solid thermal package during charging heat or providing thermal for users.

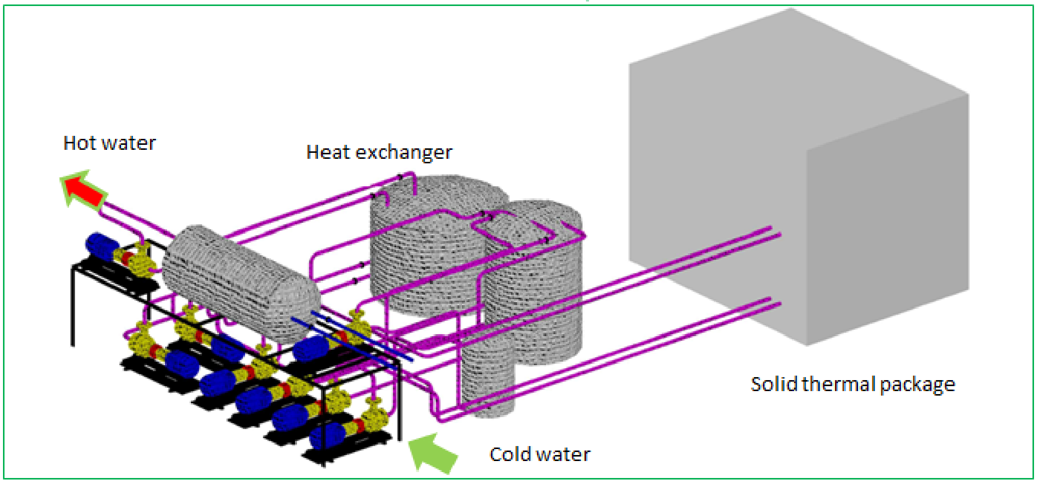


Figure 1 Shematic of a 216kw solid thermal package system

Acknowledgements

Financial assistance from the National Natural Sciences Foundation of China (No. 51376048)

References

[1] Y. X. Yu, W. P. Luan, Basic philosophy of smart grid, Journal of Tianjin University, 44.5 (2011) 377-384
[2] Singh, H., R.P. Saini, and J.S. Saini, A review on packed bed solar energy storage systems. Renewable and Sustainable Energy Reviews, 2010. 14(3): p. 1059 - 1069.
[3] Chai, L., et al. , Performance study of a packed bed in a closed loop thermal energy storage system. Energy, 2014. 77(0): p. 871 - 879
[4] Zanganeh, G., et al. , Design of packed bed thermal energy storage systems for high-temperature industrial process heat. Applied Energy, 2015. 137(0): p. 812 - 822.
[5] Bindra, H., et al. , Thermal analysis and exergy evaluation of packed bed thermal storage systems. Applied Thermal Engineering, 2013. 52(2): p. 255 - 263.

Development of an optical thermal history sensor based on the oxidation of divalent rare earth ion phosphor

A. Yanez-Gonzalez[1], E. Ruiz-Trejo[1], B. van Wachem[1], S. Skinner[1], F. Beyrau[2] and A. Heyes[3]

[1] *Imperial College London, London SW7 2AZ, UK, a.yanez-gonzalez12@imperial.ac.uk*
[2] *Otto-von-Guericke-Universität, Magdeburg 39106, Germany, frank.beyrau@ovgu.de*
[3] *University of Leeds, Leeds LS2 9JT, UK, a.heyes@leeds.ac.uk*

Extended Abstract

The measurement of temperatures in gas turbines is essential to maintain the lifetime of the metallic components as the firing temperatures increase. When on-line measurements, such as those performed with thermocouples and pyrometers, are not possible, the maximum temperatures of operation can be recorded and measured off-line after operation. Although thermal paints have been used for many years for this purpose, a novel technique proposed by Feist et al. (2007), which is based on irreversible changes in the opticlal properties of thermographic phosphors, can overcome some of the disadvantages of thermal paints.

In particular, oxidation of the divalent rare earth ion phosphor $BaMgAl_{10}O_{17}$:Eu (BAM:Eu) has shown great potential for temperature sensing between 700 °C and 1200 °C. The emission spectra of this phosphor changes with temperature, which permits to define an intensity ratio that is temperature dependent as shown in Fig 1.

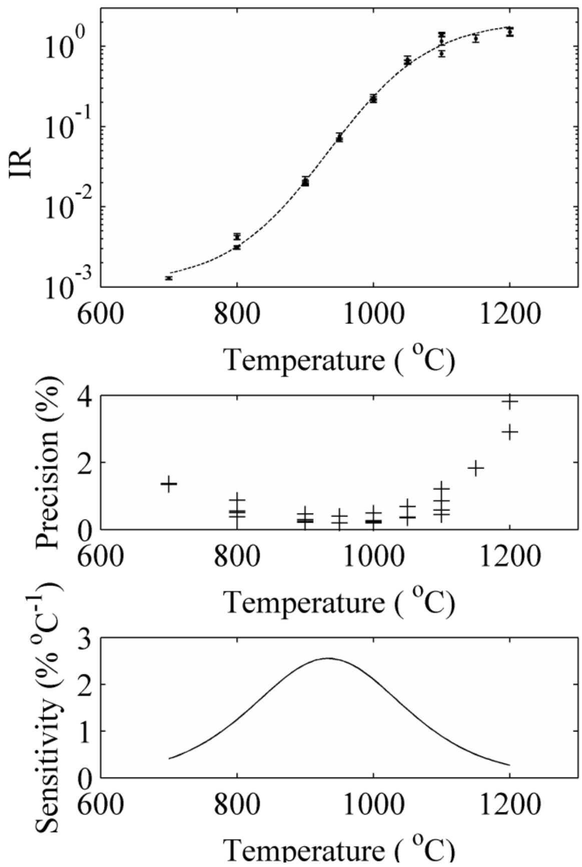


Fig 1: Intensity ratio (IR), precision and sensitivity of the temperature measurements using BAM:Eu powder

The precision of the measurement is typically below 2 % and the sensitivity varies between 0.5 $\%\cdot°C^{-1}$ and 2.5 $\%\cdot°C^{-1}$, which reinforces the suitability of this phosphor to perform measurements in this temperature range. In this paper, the development of a sensor based on BAM:Eu phosphor material is addressed by investigating the influence of several factors on the measurement accuracy and sensitivity.

References

FEIST, J., NICHOLLS, J., & HEYES, A. 2007 *Determining thermal history of components*. Patent WO/2009/083,729.

Measurement and Simulation of Low Temperature Packed Bed Regenerators

E. A. Pike-Wilson[1], T. Gardhouse[2], R. E. Morgan[1] and M. R. Heikal[1]

[1] *School of Computing, Engineering and Mathematics, University of Brighton, Brighton BN2 4GJ, United Kingdom, e.pike-wilson@brighton.ac.uk*

[2] *Highview Power Storage, 1 Northumberland Avenue, London, WC2N 5BW, United Kingdom, tim.gardhouse@highview-power.com*

Extended Abstract

The storage of thermal and electrical energy will become increasingly important with the transition to a low carbon energy distribution network. Packed bed regenerators (PBR) can be used to directly store thermal energy and, when integrated in a cryogenic energy storage system, electricity. PBRs consist of a loosely packed solid material, over which a fluid is passed to receive or transfer thermal energy. The solid and liquid used can vary greatly, depending on the application. PBRs are traditionally operated at elevated temperatures, with inlet temperatures ranging from 350 to 600 K (Cascetta et al. 2014).

The design of PBRs requires accurate material properties which can be used to evaluate the thermodynamic performance of the PBR, usually through simulation. Material properties, including specific heat and thermal conductivity, at ambient and elevated temperatures are well documented. However, there is limited published data for material properties at low (<150 K) temperatures and the performance of a PBR operating at cryogenic temperatures. Future development of PBRs for this application will be limited without an understanding of the material properties at the requisite temperatures (down to -196°C for liquid nitrogen).If the material properties are unknown, assumptions must be made when simulating the thermodynamic performance of the packed bed.

Experiments were conducted using a scaled down PBR, consisting of liquid nitrogen and a gravel packed bed. The experimental temperature profile along the length of the packed bed was compared with the Schumann model (2002) and the White model (2014) from literature. These models are based on a single PBR charge, not a cyclic process. The comparison was conducted at multiple stages during the charging of the PBR, using Matlab. The results showed that the accuracy of the models varied with the charging time. Neither model was able to accurately predict the temperature profile across the whole charging time. At the initial stages of charging, the experimental data closely matched that of the Schumann model (2002). As the charging process continued, the White model (2014) showed a greater accuracy. The accuracy of the models was also a function of the mass flow rate, with the Schumann model (2002) performing better at higher flow rates. We believe that this discrepancy in the agreement is largely due to the lack of accurate material properties at such low temperatures and assumptions made in the simulation. The particle size was assumed to be of uniform shape and diameter but the gravel varies in both size and shape, affecting the void fraction and conduction. The simulations do not include wall effects, which are higher in the lab scale packed bed than those of a commercial packed bed. This is due to a higher particle to packed bed diameter ratio resulting in the path of least resistance for the flow being along the wall rather than through the bed.

References

CASCETTA, M., CAU, G., PUDDU, P., SERRA, F. 2014 *Numerical investigation of a packed bed thermal energy storage system with different heat transfer fluids*, Energy Procedia, **45,** 598 – 607.
WILLMOTT, A. J. 2002 *Dynamics of regenerative heat transfer*, Taylor and Francis.
WHITE, A. J., McTIGUE, J., MARKIDES, C. 2014 *Wave propagation and thermodynamic losses in packed – bed thermal reservoirs for energy storage,* Applied Energy, **130,** 648 – 657.

Variable Conductance Heat Pipes for Managing Thermal Stores

D. Hislop[1], R. Law[2], A. Mustaffar[6], R.J.McGlen[3], D.A.Reay,[7] C. Underwood[4], B. Ng[4], A. Whitaker[5] & X.Yang[3]

[1] Sustainable Engine Systems Ltd., Lewes, East Sussex, UK, hislop@msn.com
[2] Newcastle University, Merz Court, Newcastle upon Tyne NE1 7RU, UK, richard.law2@ncl.ac.uk
[3] Thermcore Ltd, Wansbeck Business Park, Ashington, UK, R.J.McGlen@thermcore.com; X.Yang@thermacore.com
[4] Northumbria University, Newcastle upon Tyne, UK. Chris.underwood@northumbria.ac.uk; bobo.ng@northumbria.ac.uk
[5] TWI Ltd., Cambridge, UK. Andrew.whitaker@twi.co.uk
[6] Newcastle University, Merz Court, Newcastle upon Tyne NE1 7RU, UK. ahmadmustafar@me.com
[7] Newcastle University, Merz Court, Newcastle upon Tyne NE1 7RU, UK. David.reay@ncl.ac.uk

Extended Abstract

This paper reports on the development of a novel thermal management controller (TMC) for micro-Combined Heat and power (mCHP) systems.

Stirling-engine based units for providing heat and electricity for individual houses are increasingly of interest and several units are entering the marketplace. However, their economic operation and their ability to satisfy user heat demands could be much improved by a more sophisticated thermal management system that combines highly effective storage of heat with the ability to release such stored energy in amounts and at times to accurately meet the needs of the consumer.

Fig. 1. Heat pipe and PCM unit.

Using phase change materials with high thermal conductivity (instead of a large water storage tank) and an innovative variable conductance heat pipe (VCHP) for controlling heat release, it is believed that the TMC will accelerate the take-up of domestic micro-CHP, as well as in other applications.

Fig. 1 shows the finned storage tank (upper left) for the phase change material (PCM), in this case being a salt hydrate for low temperature storage or Erythritol for storage around 120°C. The copper heat pipe has a finned evaporator (lower section) that is immersed in the PCM in the tank. The upper condenser section is water-cooled and (not visible) is the inert gas container for activating the variable conductance feature. The units were tested in the Thermacore laboratories before being relocated to Northumbria University, where domestic CHP units were able to provide heat to charge the store.

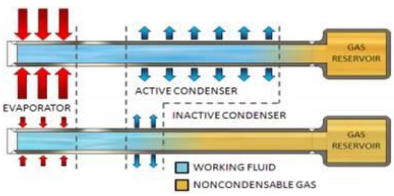


Fig. 2. The VCHP gas front options.

The control using an inert gas is shown in Fig. 2, and in the mCHP system the inert gas position is varied using feedback control to close off the VCHP condenser when the heat demand is low. As demand increases the gas front moves back to the reservoir, releasing condsenser surface area for taking heat from the PCM to the domestic hot water circuit.

The paper will discuss the concept in more detail, give experimental data, and present other applications.

Nanofluids for Heat Transfer Applications

V. Rudyak[1], A. Minakov[1,2], M. Pryagnikov[1,2] and D. Guzey[1,2]

[1] *Novosibirsk State University of Architecture and Civil Engineering, Novosibirsk, Leningradskaya, 113, Russia, valery.rudyak@mail.ru*
[2] *Siberian Federal University, Krasnoyarsk, Svobodny, 79, Russia*

Extended Abstract

Nanofluids are a new class of dispersed fluids consist on the carrier fluid and dispersed nanoparticles. The nanofluids research has both fundamental and application motivation. On the one hand the transport properties of nanofluids are not described as a rule by classical theories (Einstein, Batchelor, Maxwell, etc.). On the other hand the nanofluids may be utilized in several applications, for example, engine cooling, refrigeration, thermal storage, drilling, lubrications, solar water heating, in different biomedical technologies, and so forth. In all cases the thermophysical properties of nanofluids play the key role. The review of the last results of the study of nanofluids transport properties obtained by the authors was presented. The experimental and molecular dynamics simulation data were considered. We analyzed about forty different nanofluids. In particular, it is discussed:

- The results of molecular dynamics simulation of viscosity and thermal conductivity coefficients of nanofluids (Rudyak et al. 2014, 2015).
- The data of the molecular dynamics simulation of the state equation of nanofluid (Rudyak 2015).
- The experimental data of the measurement of the viscosity and thermal conductivity coefficients of the different nanofluids.
- The experimental data of the heat-transfer coefficient of the different nanofluids for laminar (Minakov *et al*. 2015) and turbulent flows.
- The experimental data of measuring of critical density of heat flow during boiling of nanofluids on a cylindrical heater.

This work was supported in part by the Russian Science Foundation (grant No. 14-19-00312).

References

RUDYAK, V.Ya. 2015 *Molecular dynamics simulation of nanofluids equation of state.* Tech. Phys. Lett. (in press)

RUDYAK, V.Ya., & KRASNOLUTSKII, S.L. 2014 *Dependence of the viscosity of nanofluids on nanoparticle size and material.* Phys. Letters A. **378**. 1845-1849.

RUDYAK, V.Ya., & KRASNOLUTSKII, S.L. 2015 *Simulation of the nanofluid viscosity coefficient by the molecular dynamics method*. Tech. Phys. (in press)

MINAKOV A.V., RUDYAK V.YA., GUZEI D.V., LOBASOV A.S. 2015 *Measurement of the heat transfer coefficient of a nanofluid based on water and copper oxide particles in a cylindrical channel.* High Temp. **53**, 246-253.

Flooding behaviour in countercurrent gas-liquid flow in vertical tubes with turbulence promoters

Klaus Spindler

Institute of Thermodynamics and Thermal Engineering, University of Stuttgart
Pfaffenwaldring 6, D-70550 Stuttgart, Germany, Email: spindler@itw.uni-stuttgart.de

Counterflow of gas and liquid in tubes is of great interest in many industrial processes. The liquid flowing under gravity forms a thin film coating the inner tube wall. The gas driven by a pressure difference flows up in the core of the tube. Interfacial shear stress is acting on the film surface. With increasing gas volume flow rate the smooth liquid film becomes progressively more disturbed. The film is getting more rippled. Liquid droplets are breaking off the film. There is a net flow reversal of the liquid. The liquid bridges the flow area. The liquid moves upward along the tube. The phenomenon of flooding is a complex two-phase flow phenomenon. It is a major limiting factor in design and operation of reflux condensors.

Inserts in shape of matrix elements are often used as turbulence promotors for in-tube condensation of mixtures, see Fig.1. These inserts are disturbing and thinning the liquid film and the concentration boundary layer. The heat and mass transfer is enhanced significantly. These inserts also have an impact on the flooding beviour.

A test apparatus was built up to study the flooding behaviour of a countercurrent adiabatic air-water two-phase flow in a vertical glass tube. The tube length is 3000 mm. Two different inner tube diameters of 15 mm and 22 mm have been used. The superficial velocities of air and water have been varied to obtain the flooding curves for the empty tubes with 15 mm and 22 mm inner diameter as well as for the tubes with the low and high density turbulence promoters, see Fig. 2. Counterflow is possible below the flooding curve. The flooding curve of the 15 mm tube is below that of the 22 mm tube. That means flooding occurs at lower superficial gas velocities. The possible region for countercurrent flow is diminished when using turbulence promoters and especially with high density. Detailled experimental results are shown. The increasing water column height in the tube and resulting climbing velocity of the liquid are presented. In addition some experiences with acrylic resin tubes and various liquid distributors are discussed. Correlations from literature are used for comparison.

Fig.1: High and low density turbulence promotors (hiTRAN)

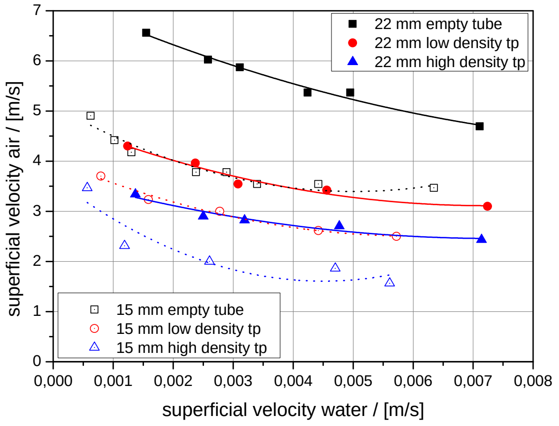


Fig.2: Flooding curves for the 15 mm and 22 mm empty tubes and with low and high density turbulence promoters

Manipulating Phonon Heat Conduction by High Pressure Torsion in Silicon Based Thermoelectrics

Mitsuru Tabara[1], Sivasankaran Harish[1], Yoshifumi Ikoma[2], Zenji Horita[2,3], Yasuyuki Takata[2,3,4], David G. Cahill[5] and Masamichi Kohno[2,3,4]

[1] *Department of Mechanical Engineering, Kyushu University, Fukuoka, Japan*
[2] *Department of Materials Science and Engineering, Kyushu University, Fukuoka, Japan.*
[3] *International Institute of Carbon-Neutral Energy Research (I²CNER), Kyushu University.*
[4] *CREST, Japan Science and Technology Agency*
[5] *Department of Materials Science and Engineering, University of Illinois at Urbana Champaign, Urbana, Illinois, USA.*

Abstract

Materials with length scales on the order of few nanometers exhibit unique abilities to control thermal transport. Tailoring the thermal properties of nanostructured systems have a promising application in the field of micro electronics and thermoelectrics. In this work, we demonstrate a novel technique using high-pressure torsion (HPT) to create a high density of lattice defects on nanometer length-scales in semiconductor materials such as silicon and silicon-germanium alloys (SiGe). We report a dramatic reduction in the thermal conductivity of bulk crystalline silicon and SiGe alloy when subjected to severe plastic strain under a pressure of 24 GPa at room temperature using HPT. Thermal conductivity of the HPT-processed samples were measured using pico-second time domain thermoreflectance. The reduction in thermal conductivity is attributed to the formation of nanograin boundaries and metastable phases which act as phonon scattering sites. Subsequent annealing shows a reverse transformation from metastable phases to cubic diamond phase and a nominal increase in thermal conductivity due to the reduction of the density reduction of secondary phases and nanocrystalline defects.

References

HARISH, S., Tabara, M., Ikoma, Y., Horita, Z., Takata, Y., Cahill, DG., Kohno, M. 2014 *Thermal conductivity reduction of crystalline silicon by high pressure torsion.* Nanoscale Res. Lett. **9**, 1-6.

14th UK Heat Transfer Conference 2015
Edinburgh (Sept 7 – Sept 8, 2015)

Experimental Study of Unsteady and Conjugate Heat Transfer in Wavy Film Flows over an Inclined Heated Foil

A. Charogiannis[1], B. G. Heiles[2] and C. N. Markiedes[3]

[1] Department of Chemical Engineering, Imperial College London, London, SW7 2AZ, United Kingdom, ac1005@ic.ac.uk
[2] Department of Chemical Engineering, Imperial College London, London, SW7 2AZ, United Kingdom, bh1514@ic.ac.uk
[3] Department of Chemical Engineering, Imperial College London, London, SW7 2AZ, United Kingdom, c.markides@imperial.ac.uk

Extended Abstract

Liquid films falling under the action of gravity are encountered in a wide range of industrial applications, such as wetted-wall absorbers, evaporators and heat exchangers, owing to their high surface-to-volume ratios and exemplary heat and mass transfer capabilities. Yet, only a limited number of publications relating to simultaneously acquired and spatiotemporally resolved film thickness and heat transfer data are currently available (see for example, Schagen et al. (2006) and Kabov et al. (2002)), primarily owing to the challenges that arise when performing such measurements (restricted fluid domains under observation and intermittent nature of the moving and wavy interface).

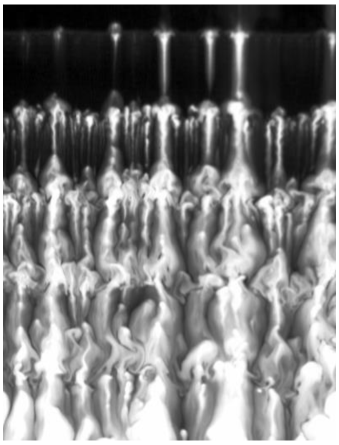


Fig 1: IR image of thermal rivulets forming on the free-surface of a pulsed film-flow over a heated titanium foil.

In order to achieve an improved understanding of these flows, an optical technique based on planar laser-induced fluorescence (PLIF) imaging and infrared (IR) thermography has been developed and applied to the measurement of unsteady and conjugate heat transfer in thin, gravity-driven falling film-flows over a heated metal substrate/foil. Based on the combined technique, spatiotemporally resolved and simultaneously conducted film thickness, free-surface temperature, heated substrate temperature, heat transfer coefficient (HTC) and heat flux measurements will be reported over a range of electrically applied heat input values, flow Reynolds (*Re*) numbers and pulsation frequencies.

In addition, free-surface temperature measurements have been conducted with the IR camera, whereby the formation, interaction and spatiotemporal evolution of thermal “rivulet” structures, associated with significant film thickness variations have been identified. It will be shown that flat thermal rivulets are linked to the development of Marangoni flows on the film free-surface, and that the introduction of flow pulsation has a pronounced effect on these features, instigating a clearly discernible mixing enhancement manifested by the appearance of strongly skewed and intertwined thermal rivulets.

References

KABOV, O.A., SCHEID, B., SHARINA, I.A. & LEGROS, J.-C. 2002 *Heat transfer and rivulet structures formation in a falling thin liquid film locally heated*. Int. J. Therm. Sci. **41**, 664-672.

SCHAGEN, A., MODIGELL, M., DIETZE, G. & KNEER, R. 2006 *Simultaneous measurement of local film thickness and temperature distribution in wavy liquid films using a luminescence technique*. Int. J. Heat Mass Transfer **49**, 5049-5061.

An experimental study of rotational pressure loss in rotor ducts

Y. C. Chong[1], D. Staton[1], M. Mueller[2] and J. Chick[2]

[1] *Motor Design Ltd, 4 Scotland Street, Ellesmere, SY12 0EG, U.K., e-mail: eddie.chong@motor-design.com*
[2] *University of Edinburgh, The Institute for Energy Systems, School of Engineering, Edinburgh, EH9 3JL, U.K.*

Extended Abstract

The power output of an electrical machine is strongly affected by its thermal performance because machine operating temperature limits the electric loading. An electrical machine thermal performance is predominantly affected by two factors. One is amount of heat sources within the machine generated from losses. The other one is how well the generated heat can be transported out from the machine. From the cooling point of view, the most effective cooling method is to duct the coolant to the sources of losses. Therefore, accurate prediction of flow distribution in the cooling paths and their pressure losses are essential for accurate thermal modelling. The coupled thermal-fluid modelling between equivalent thermal network and flow network has been adopted in [1] for dual mechanical port machine and [2] for turbogenerator. This study investigates the pressure loss of flow through straight circular ducts rotating about an axis parallel to the duct axis. The experimental measurements found that the flow passing through the rotating ducts suffered additional pressure loss when compared with stationary condition. For stationary condition, the experimental results agreed with the values of pressure loss estimated using proper theoretical/empirical correlations. The rotational pressure loss is mainly caused by rotor ducts entrance loss and additional friction loss due to the effect of rotation. These losses have been isolated in the present study. The coefficient of entrance loss has been correlated with a dimensionless parameter, rotation ratio, and correlations for entrance loss coefficient of rotor ducts of different proximity to the rotor periphery were proposed. The correlations are useful for predicting flow distribution and thermal-fluid modelling of rotating machines.

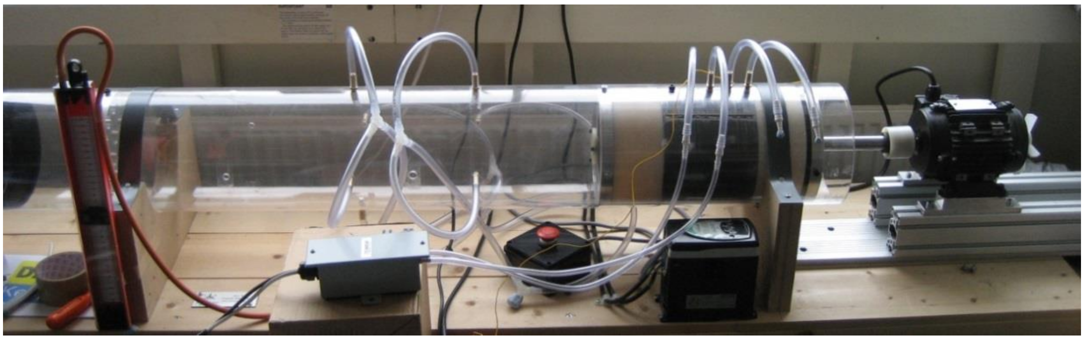


Fig. 1. Air flow test rig

References

[1] X. Sun and M. Cheng, *Thermal analysis and cooling system design of dual mechanical port machine for wind power application*, IEEE Trans. Ind. Electron., vol. 60, no. 5, pp. 1724–1733, 2013.

[2] W. Li, J. Han, X. Zhou and Y. Li, *Calculation of ventilation cooling, three-dimensional electromagnetic fields, and temperature fields of the end region in a large water–hydrogen–hydrogen-cooled turbogenerator*, IEEE Trans. Ind. Electron., vol.60, no.8, pp. 3007–3015, 2013.

The Effect of the Inclination Angle on Heat Transfer Performance in Back-ward Facing Step Utilizing Nanofluid

S. Etaig[1], R. Hasan[2] , N. Perera[3] and R. Mathkor[4]

[1] *Northumbria University, Newcastle Upon Tyne, UK, Etaig.Mahmoud@Northumbria.ac.uk*
[2] *Northumbria University, Newcastle Upon Tyne, UK, Reaz.Hassan@Northumria.ac.uk*
[3] *Northumbria University,Newcastle Upon Tyne, UK, Noel.Pererra@Northumbria.ac.uk*
[4] *Newcastle University, Newcastle Upon Tyne, UK, R.Z.H.Mathkor1@newcastle.ac.uk*

Extended Abstract

This paper investigates the effect of the inclination angle of the vertical face of backward-facing step on the heat transfer performance utilizing Nanofluid, Al2O3 is the nanoparticle used in this investigation. The finite volume technique was used to solve the momentum and energy equation in 2D backward facing step geometry. The effect of Re on Nu is investigated, the volume fraction of the nanoparticle in the Nanofluid is examined, the pressure loss also reported for different Re numbers, the entropy is studied for a range of Re numbers and for different volume fraction of Nanofluid. The effect of the inclination angle on the heat transfer showed that the heat transfer was enhanced with an increase in the inclination angle.

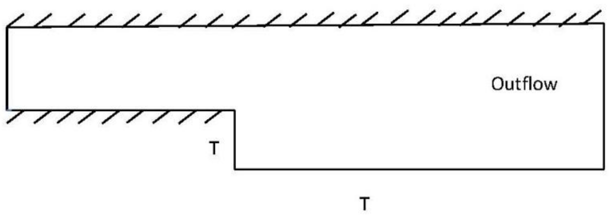


Fig. 1

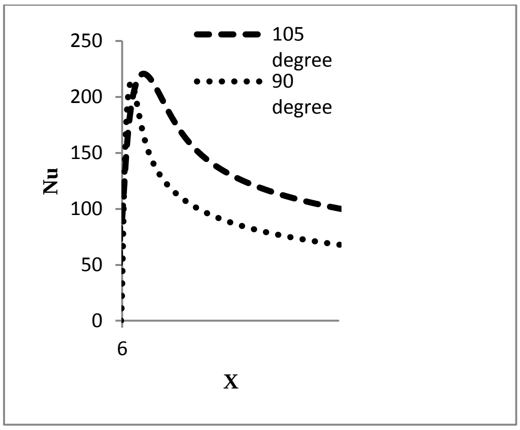


Fig. 2

Fig.1 shows the geometry investigated for 90^o, the dimensions and full boudary conditions will be detailed in the full manuscript. Fig.2 illustrates the variation of Nu along the bottom for 90^o and 105^o the thermophysical properties of Nanofluid were temperature dependent, the thermal conductivity was modelled in Eq. (1) as introduced by (Corcione, 2011).

$$\frac{K_{static}}{K_f} = 1 + 4.4\,\mathrm{Re}^{0.4}\,\mathrm{Pr}^{0.66}\left(\frac{T}{T_{fr}}\right)^{10}\left(\frac{K_s}{K_f}\right)^{0.03}\phi^{0.66} \tag{1}$$

References

CORCIONE, M. 2011. Rayleigh-Bénard convection heat transfer in nanoparticle suspensions. *International Journal of Heat and Fluid Flow,* 32**,** 65-77.

Characterization of the Corona Discharge for Ionic Wind Heat Transfer Enhancement in Internal Flow Channels

N. Gallandat[1] and Prof. J.R. Mayor[1]

[1] *Georgia Institute of Technology, George W. Woodruff School of Mechanical Engineering, 801 Ferst Drive, Atlanta, GA 30322-0405, USA, rhett.mayor@me.gatech.edu*

Extended Abstract

Ionic wind is a generic term used to describe the bulk flow of air induced by the momentum transfer of free ions to neutral air molecules. Several studies have considered ionic wind as a heat transfer enhancement method [1, 2]. However, the work performed thus far focused on the enhancement of external convective heat transfer. The present study considers the configuration of rectangular, internal flow channels as shown in Figure 1. The phenomenon of ionic wind can be subdivided into two steps, as shown in Figure 2: first, positive ions and free electrons are produced through a Corona discharge. Then, positive ions drift towards the grounded electrode and transmit momentum to neutral air molecules through collisions. The first step is a plasma physics phenomenon involving numerous complex reactions that is challenging to model numerically [3]. On the other hand, the physics happening outside the plasma region is well described by a set of partial differential equations. Therefore, the proposed approach combines an experimental study to describe the Corona discharge and a numerical, multiphysics model of the unipolar region. This paper investigates the experimental characterization of the Corona discharge as a function of the spacing between the electrodes, the applied voltage and the ambient humidity and presents a function of the form shown in equation (1), where V_{Corona} is the applied voltage, $\Phi_{Ambient}$ the humidity and d representative dimensions of the ionic wind generator. This function can then be used for the design of new heat sinks utilizing ionic wind to enhance flow in internal channels.

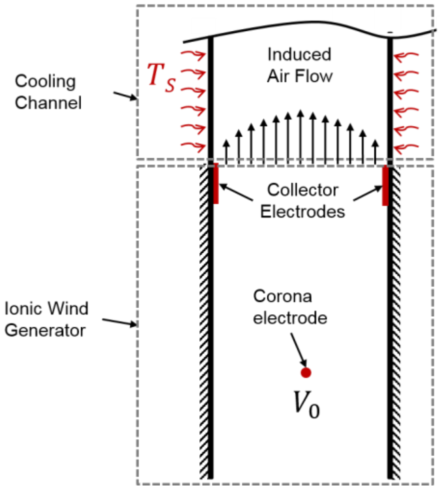


Figure 1: Configuration considered to enhance the air flow in internal cooling channels utilizing ionic wind.

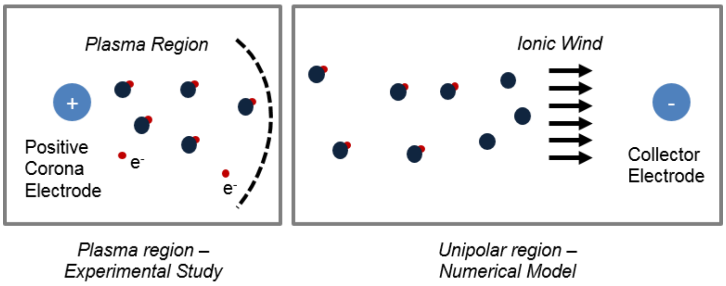


Figure 2: Proposed research approach to describe ionic wind.

$$I_{Corona} = f(V_{Corona}, \Phi_{Ambient}, d) \quad (1)$$

References

[1] D. B. Go, R. A. Maturana, T. S. Fisher, and S. V. Garimella, "Enhancement of external forced convection by ionic wind," *International Journal of Heat and Mass Transfer*, vol. 51, pp. 6047-53, 12/ 2008.

[2] N. E. Jewell-Larsen, C. P. Hsu, I. Krichtafovitch, S. Montgomery, J. Dibene, and A. Mamishev, "CFD analysis of electrostatic fluid accelerators for forced convection cooling," *IEEE Transactions on Dielectrics and Electrical Insulation*, vol. 15, pp. 1745-53, 12/ 2008.

[3] P. Wang, F.-G. Fan, F. Zirilli, and J. Chen, "A hybrid model to predict electron and ion distributions in entire interelectrode space of a negative corona discharge," *IEEE Transactions on Plasma Science*, vol. 40, pp. 421-428, 2012.

Experimental Study of Ionic Wind Heat Transfer Enhancement in Rectangular, Vertical Channels

N. Gallandat[1] and Prof. J.R. Mayor[1]

[1] *Georgia Institute of Technology, George W. Woodruff School of Mechanical Engineering, 801 Ferst Drive, Atlanta, GA 30322-0405, USA, rhett.mayor@me.gatech.edu*

Extended Abstract

Ionic wind is a potential heat transfer enhancement method that has been considered for a variety of applications [1]. The main advantage of ionic wind as compared to standard fans is the absence of any moving part. This is especially important for application fields requiring long lifetime and high reliability, such as the thermal management of grid-scale power routers. In a previous study, the authors presented a numerical model assessing the potential of ionic wind for the thermal management of grid-scale power routers [2]. The present work intends to validate the numerical model by performing an experimental study on the heat transfer in vertical, rectangular cooling channels.

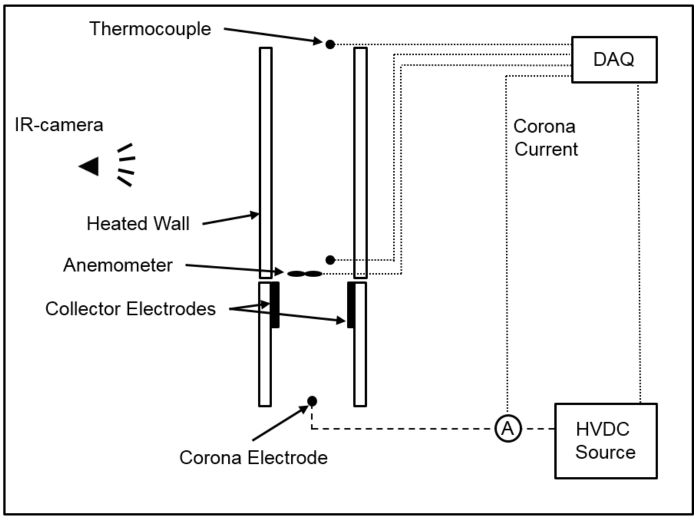


Figure 1: Experimental setup to test the heat transfer performance of rectangular, vertical cooling channel utilizing ionic wind to increase the air flow rate.

The test setup designed to measure the performance of ionic wind heat transfer enhancement is shown in Figure 1. The Corona electrode is subject to a high voltage from a HVDC source. The Corona current is measured using a picoammeter. The data acquisition system collects the measurement data such as the air velocity at the exit of the ionic wind generator and the air temperature and the inlet and outlet of the channel. The temperature of the channel walls is monitored using surface thermocouples and an IR thermal camera. The experiment is run at voltages up to 15kV and different channel width and electrode spacings. An ultra-thin heat sheet is mounted on the walls of the channel to represent an isoflux surface. For each configuration considered, the power dissipated is increased until the wall reaches the critical temperature of 100°C. The maximal allowable heat flux is reported. Finally, the experimental results are compared to the multiphysics numerical model presented in [2].

References

[1] D. B. Go, R. A. Maturana, T. S. Fisher, and S. V. Garimella, "Enhancement of external forced convection by ionic wind," *International Journal of Heat and Mass Transfer,* vol. 51, pp. 6047-53, 12/ 2008.

[2] N. Gallandat and J. Rhett Mayor, "Novel Heat Sink Design Utilizing Ionic Wind for Efficient Passive Thermal Management of Grid-Scale Power Routers," *Journal of Thermal Science and Engineering Applications,* vol. 7, pp. 031004-031004, 2015.

Novel Heat Transfer using Solid Phase Transport Medium

W.D. Alexander[1]

[1] *ACPI Ltd., Stafford, U.K., bill_alexander@btinternet.com*

Extended Abstract

A new heat transfer technology is described which may have significant applications in high heat flux applications, particularly where wide operating temperature ranges are required. Equivalent heat transfer coefficients of greater than 100 000 [Wm-2K-1] are predicted for operation around room temperature, and although this figure is much reduced at cryogenic temperatures, significant improvements are indicated relative to LN2 systems. A simple theoretical analysis is presented to indicate the principles behind the technology, together with some results from prototype testing. The technology is applicable to either heating or cooling situations and over a temperature range from below 100K to above 1000K. An advantage over existing thermal management systems is that the technology does not use toxic or hazardous materials over the temperature range indicated. This may be beneficial in situations where personnel are in close proximity to heat transfer equipment and also where "end of equipment life" disposal issues are important. A further advantage is that a direct interface to ambient air may be included in designs using the technology, which can obviate the need for additional radiators or finned heatsinks. There may be weight and cost benefits as a result of this latter point.

A target application has been the cooling of computer processor chips where the flux levels have led computer manufacturers to move from simple air blast cooling techniques towards the use of heat pipes and liquid cooling schemes to augment the performance of air blast cooling techniques[1]. High power LED lighting chips and RF transmitting transistors are also high thermal flux devices which require tight temperature control and thermal management.

The limited experimental results indicate considerable potential for the innovation, and it is expected that a wide range of applications will be found as the technology is developed. Applications may include : compact heat exchangers for petrochemical and nuclear industries ; computer processor chip cooling ; solar energy harvesting and storage ; air conditioning and heat pump heat exchangers ; and high input flux portable device cooling.

References

1 Bar-Cohen, A., March 14 2012 *Thermal Packaging – The Inward Migration* Technical Symposium on Thermal Management – Maryland, U.S.A.

Dynamic Testing and Modelling of Solar Collectors

I. Guarracino, J. Freeman, and C. N. Markides

Clean Energy Processes (CEP) Laboratory, Department of Chemical Engineering, Imperial College London, South Kensington Campus, SW7 2AZ, UK, ilaria.guarracino12@imperial.ac.uk, j.freeman12@imperial.ac.uk, c.markides@imperial.ac.uk

Extended Abstract

Solar-thermal collectors operating under real conditions rarely reach steady state due to the temporal fluctuation of climate conditions and thermal loads. Consequently, collector models that describe dynamic (together with steady state) behaviour are required for the accurate prediction of the thermal output and optimising the control strategy. Previous studies (Schnieders, 1997), have presented methods for the characterisation of dynamic behaviour that incorporate mainly single-node solar collector models. The results of such models cannot easily be extended to different geometries.

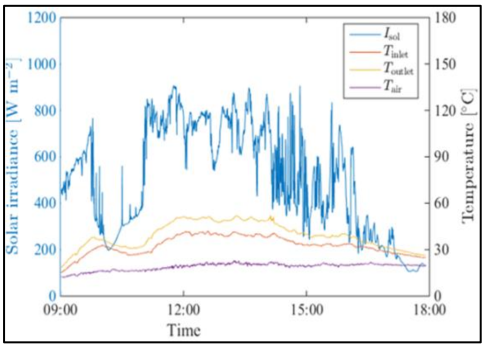


Figure 1: Dynamic performance data for an intermittently cloudy day

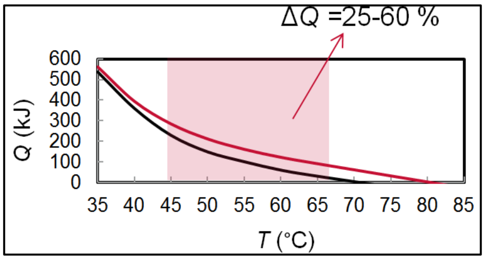


Figure 2: Predicted thermal output of quasi-steady and the dynamic collector

In an earlier paper, Guarracino *et al*. (2015) presented a thermal and eletrical model of a hybrid photovoltaic/ thermal (PV/T) solar collector module. For PV/T modules, knowledge of the temperature distribution on the PV cells is of crucial importance for the estimation of the electrical output (Zondag *et al*., 2002). The aforementioned PV/T model included a detailed 3-D thermal sub-model that can be adapted to various geometries or collector configurations, in which the energy balance equation is solved along the water-flow diection (*y*-plane) and transverse direction (*x*-plane), with each material layer discretized into multiple nodes along each plane.

In this paper, the 3-D numerical model is used to simulate the performance of a solar-thermal evacuated tube collector (ETC) under dynamic conditions. The model is experimentally validated using real performance data for the ETC in operation over three typical weather days: (i) clear sky; (ii) intermittently cloudy (see Fig. 1); and (iii) fully overcast. In the experimental setup, inlet fluid temperature and flow rate are controlled while solar irradiance, wind-speed and temperatures are monitored. The thermal output of the validated dynamic solution is compared with an equivalent quasi-steady solution, showing that the quasi-steady model overestimates the energy generated, particularly during cloudy and overcast days (Fig. 2). The dynamic thermal model is shown to be an invaluable tool for predicting the performance of solar-thermal and PV/T collectors in different climatic conditions and also for evaluating the effectiveness of novel design features for thermal performance enhancement.

References

GUARRACINO, I., MARKIDES, C. N., & EKINS-DAUKES, N. J. 2015 , *Thermal and electrical model of the sheet-and-tube hybrid photovoltaic/thermal (PV/T) collector*. In: Proc. ASME-ATI-UIT 2015 Conf. Therm. Energy Syst: Prod., Storage, Util., Env., Naples, Italy.

SCHNIEDERS, J. 1997 *Comparison of the energy yield predictions of stationary and dynamic solar collector models and the models' accuracy in the description of a vacuum tube collector*. J. Sol. Energy. **61**, 179-190.

ZONDAG, H. A., DE VRIES, D. W., VAN HELDEN, W. G. J., VAN ZOLINGEN, R. J. C., VAN STEENHOVEN, A. A. 2002 *The thermal and electrical yield of a PV-thermal collector*. J. Sol. Energy. **72**, 113-128

A Self-Pumped Heat-Exchanger for Wave-Powered Desalination.

D. Hellenschmidt[1], S.H. Salter[2]

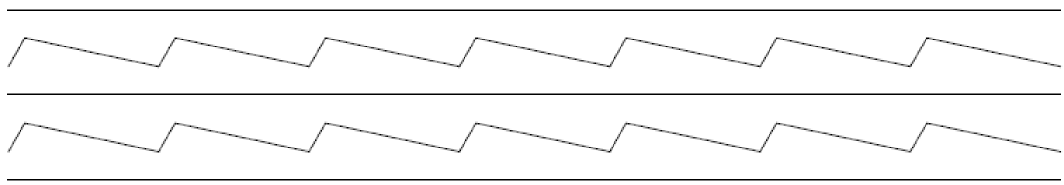
[1] *University of Oldenburg, Ammerländer Heerstraße 114-118, 26129 Oldenburg,Germany, Desiree.Hellenschmidt@uni-oldenburg.de*
[2] *University of Edinburgh, School of Engineering, Mayfield Road, Edinburgh EH9 3HL, Scotland, S.Salter@ed.ac.uk*

Extended Abstract

The alternating rolling motions of the Edinburgh duck can be used to pump the very large volumes of water vapour from the evaporation side to the condensing side of a vapour-compression desalination system without high-speed impellers or machined pistons. All the latent heat is recycled. But the method requires the smaller, liquid-feed volumes to be raised to boiling point. Most of the sensible heat in the outgoing flows of fresh water and enhanced strength brine must then be recovered. The proposed liquid heat-exchanger consists of interleaved scrolls of thin, stainless steel clad with scale-resistant plastic. These are wrapped around an 8 metre diameter boss. One sheet is folded into an asymmetric zig-zag with a steep slope followed by a gentle one. The other sheet is not folded. The gaps between the sheets act like a series of rectangular Venturi tubes with efficient flow in the direction of the gentle expansion but much higher pressure drop in the other direction. Sharp bends help to induce turbulence and flow separation. When the scrolls are given alternating angular movements one side of the zig-zag will pump water in and the other out. The pumping pressure is distributed along the length of both paths and will balance the pressure drop. It is difficult to think of a more satisfactory heat exchanger design.

SolidWorks software was used to select promising dimensions. The continuous changes of gap mean continuous changes of Reynolds, Prandtl and Nusselt numbers. A 1.2 by 0.6 metre model with an airspaced, double-glazed wall was built. Recipocating flows were provided by a wave-maker driven at a 3 second period with horizontal amplitude of about 20 to 70 mm. A heat input of 800 watts was provided by a sheet of Veroboard with opposite a.c. voltages between adjacent copper strips. Theoretical analysis and Computational Fluid Dynamic (CFD) simulations suggest very efficent heat transfer. The laboratory model experiments however were limited due to the inadequate sensitivity of the thermometers available.

The paper will suggest ways in which full size heat exchangers might be built.



References

SALTER, S. 1985. *Wave-powered desalination.* Proceedings of Conference on Energy for Rural and Island Communities, Inverness, September pp 235-241. J. Twidell ed. Pergamon Oxford.

CRUZ, J.M.B.O., SALTER, S. 2006. *Numerical and experimental modelling of a modified version of the Edinburgh Duck wave energy device.* Proc. I.Mech.E. Vol. 220 Part M: J. Engineering for the Maritime Environment. doi: 10.1243/14750902JEME5.

Analysis and Experiment on Forced Convection Heat Transfer Coefficient and Pressure Drop of Diamond-Shaped Fin-Array

S. Hirasawa[1], A. Fujiwara[1], K. Takaoka[1], T. Kawanami[1] and K. Shirai[1]

[1] Dept. of Mechanical Engineering, Kobe University, 1-1 Rokkodai, Nada, Kobe, Hyogo 657-8501, Japan, hirasawa@kobe-u.ac.jp

Extended Abstract

Forced convection cooling of fins on a high-temperature wall has been used to cool high-power electronic devices. We numerically calculated and experimentally measured the forced convection heat transfer coefficient and pressure drop of a diamond-shaped fin-array with water flow. First, we calculated the flow velocity and temperature distribution in a water flow around a diamond-shaped fin-array using a 3-dimensional thermo-fluid computation code called "STAR-CCM+". The leading angle of the diamond-shaped fins was 30° and their length was 7.7 mm in the flow direction. The surface temperature of the fins was 40 °C and the inlet temperature of water was 20 °C. The boundary condition for the outside wall of the flow region was symmetrical except for the inlet and outlet regions. Figure 1 shows the calculated results for the inlet flow velocity of 3 m/s and Reynolds number of *Re* =26900. The separation of fluid flow behind the fins moved periodically and there was a flip-flop flow phenomenon. Then we experimentally examined the water flow velocity and heat transfer coefficient of a diamond-shaped fin-array. The diamond-shaped fin-array was made out of a copper block by cutting crossed grooves of 2 mm in width, 2 mm in depth, and 4 mm in pitch. The flip-flop flow phenomenon was not observed in the experiments. Figure 2 plots the relation between Nusselt number and Reynolds number for our calculation results, experimental results and an equation for turbulent flow in a tube which is known to be Eq. (1).

$$Nu = 0.023 Re^{0.8} Pr^{0.4} \tag{1}$$

We concluded that the heat transfer and pressure drop of the diamond-shaped fin-array could be estimated with equations for turbulent flow in a tube.

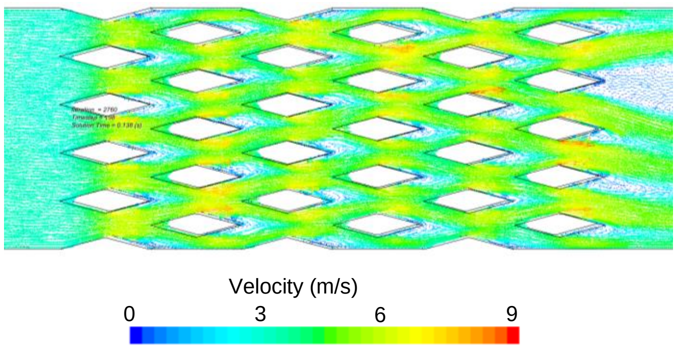


Fig 1: Calculation result of flow velocity distribution

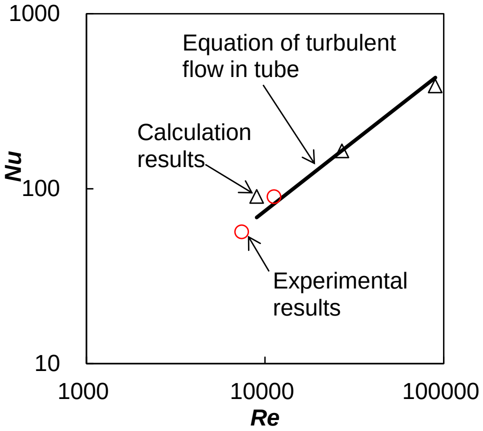


Fig 2: Relation between Nusselt number and Reynolds number

Keynote 4

Ice slurries, the cool, benevolent bringers of sustainable clean living

G L Quarini et al

University of Bristol, Queen's Building, University Walk, Bristol, BS8 1TR, UK, joe.quarini@bristol.ac.uk

Extended Abstract

A semi-analytical model describing the thermalhydraulic behaviour of high ice fraction slurries in ducts is presented. The model is numerically solved and the resultant predictions are compared with experimantal data generated in a laboratory environment for pipes of diameters 12 to 100mm and lengths no greater than 10m. This comparison has lead to the optimisation of a number of components of the model, reulting in an overall model system which is better able to represent the physics associated with phase change in a heat transfer regieme cominated by convection and conduction processes. The 'optimised' model is then used to predict the performance of ice slurries pumped through large diameter pipes (typically 300mm diameter) over large very distances (in some cases greater than 5,000m). The agreement between the measured and the predicted longevity of the ice slurry is surprisingly good, suggesting that the model is probably representing the important/dominant physical mechanisms controlling the heat transfer rates between the cold ice slurry and the initially ambient temperture pipe and its environemtal surroundings.

This work supports the development, application and expansion of the use thick ice slurries, 'ice pigs' as elegant artefacts which can be adopted to clean complex topology ducts where conventional solid pigs cannot be used, and where it is imperative that the 'pig' cannot, under any circumstance, get stuck in the duct.

References

QUARINI, G. L. Ice-pigging to reduce and remove fouling and to achieve clean-in-place. Applied Thermal Engineering vol. 22, 747–753, 2002

CARSLAW, H. S., and JAEGER, J. C., Conduction of heat in solids, 2nd Ed. Oxford University Press, London, 1959.

QUARINI G., AINSLIE E., HERBERT M., DEANS T., ASH D., RHYS D., HASKINS N., NORTON G., ANDREWS S. & SMITH M.,. Investigation and development of an innovative pigging technique for the water-supply industry. Proc. Inst'n Mech. Eng'rs, Part E, Journal of Process Mechanical Engineering, vol. 224, 79-89, 2010

QUARINI, G., AINSLIE, E., ASH, D., LEIPER, A., McBRYDE, D., HERBERT, M. & DEANS, T. Transient thermal performance of ice slurries pumped through pipes, Applied Thermal Engineering Vol 50, 743-748, 2013.

The imitation of the surface temperature variation characteristics of concrete road under periodical ambient conditions

Hong Ye*, Shimin Li

Department of Thermal Science and Energy Engineering, University of Science and Technology of China, Hefei 230027, PR China

*hye@ustc.edu.cn

Abstract

The imitation of the surface temperature variation characteristics of concrete road, a common background, is important in the field of infrared camouflage and has been rarely discussed before. Therefore, one-dimensional heat transfer models for both of the concrete road and the imitative object are proposed, to explore the influence of the latter's thermophysical properties (thermal conductivity and volumetric heat capacity) on the surface temperature difference (STD) of them under the same periodical ambient conditions. It is elucidated that the STD is dominated by the thermal inertial (i.e., the product of the thermal conductivity and the volumetric heat capacity) of the imitative object when the dimensionless thickness, defined as the ratio of the thickness to the heat penetration depth, of the imitative object is not less than 1.0. While the imitative object's dimensionless thickness is less than 1.0, STD is influenced by both the thermal inertial and the dimensionless thickness of the imitative object. If the imitative object can be described with the lumped capacitance method, specially, the STD is determined by the product of the thermal inertial and the dimensionless thickness of the imitative object. Particularly, if the thermal inertial of the imitative object is same as that of the concrete, the surface temperatures of the concrete road and the imitative object are identical at any time, as long as the dimensionless thicknesses of the imitative object and the concrete are both not less than 1.0.

Improving the operation of a geothermal district heating network through the use of a heat storage tank

S. A. KYRIAKIS

University of Glasgow, School of Engineering, James Watt Building, Glasgow, G12 8QQ, United Kingdom, s.kyriakis.1@research.gla.ac.uk

Extended Abstract

Geothermal energy is a proven, cheap and environmentally friendly energy source which provides numerous benefits to local communities (Hepbasli, 2010). In this paper, the application of a hot water storage tank in a geothermal district heating system is studied from a thermo-economic point of view. The storage tank will be used to store excess heat in times of low load which will then be released during peak-load. This contrasts with the traditional approach of covering the peak demands with the use of fossil-fuel boilers as mentioned by Bloomquist (2003). First, an integrated model for the sizing of the installation is built, in which the geothermal data, the heat demand data throughout a whole year and the topology are used as inputs, while the sizing of the installation is the output. Then, the operation of the installation over a randomly-selected day is studied in detail. The outcome of the previous model together with the heat demand of the studied day are used as inputs in this model. The final output is the operational strategy of the installation, which is defined as the complete knowledge of the operation of the installation. For example, the necessary geothermal flow rate, the amount and the time that hot water should be charged or discharged by the storage tank, the amount and the time that the peak-up boiler should be used etc. will all be provided by this algorithm. This model is then used as the basis for the development of an extended model for the study of the operation of the installation throughout a whole year. The main goal is to calculate the basic annual running costs of the installation, comprising the cost of fuel for the peak-up boilers and the cost of electricity for running the pumps, in order to carry out an integrated financial and environmental analysis of the proposed solution. The results show that by using the heat storage all the financial indices of the investment improve compared to the traditional case of not using the heat storage. For example, the levelised cost of heating decreases by around £0.004/KWh which potentially leads to an extra annual income of £200,000 for the studied installation. Furthermore, the annual emissions of CO_2 decrease by 54%, the load factor of the geothermal installation increases by 2.4% and the total heat demand which is covered by geothermal energy increases by almost 3.85%. These indicate that by using the storage tank more geothermal energy is utilised in a more efficient way leading to financial and environmental benefits which are both conducive to sustainable development.

References

BLOOMQUIST, R.G. 2003 *Geothermal space heating*. Geothermics **32**, 513-526.

HEPBASLI, A. 2010 *A review on energetic, exergetic and exergoeconomic aspects of geothermal district heating systems (GDHSs)*. Energy Conversion and Management **51**, 2041-2061.

Sensitivity Analysis of a Capillary Pulsating Heat Pipe: Influence of the Tube Characteristics

M. Manzoni[1], M. Mameli[2], S. Andromidas[3], C. de Falco[4], L. Araneo[5], S. Filippeschi[6], K.-S. Nikas[7] and M. Marengo[8]

[1] *Università di Bergamo, Dept. of Eng. and Applied Science, Viale Marconi 5, 24044 Dalmine (BG), Italy; University of Brighton, School of Computing, Eng. and Maths, Lewes Road, BN2 4GJ, Brighton, UK, miriam.manzoni@unibg.it*
[2] *Università di Bergamo, Dept. of Eng. and Applied Science, Viale Marconi 5, 24044 Dalmine (BG), Italy, mauro.mameli@unibg.it*
[3] *Technological Education Institute of Piraeus, Thivon 250, Egaleo 122 44, Greece, sotiris.andromidas@gmail.com*
[4] *Politecnico di Milano, Maths Dept., Piazza Leonardo da Vinci 32, 20133 Milano, Italy, carlo.defalco@polimi.it*
[5] *Politecnico di Milano, Energy Dept., Via Lambruschini 4A, 20158 Milano, Italy, lucio.araneo@polimi.it*
[6] *Università di Pisa, DESTEC, Largo Lazzarino 2, 56122 Pisa, Italy, sauro.filippeschi@den.unipi.it*
[7] *Technological Education Institute of Piraeus, Thivon 250, Egaleo 122 44, Greece, ksnikas@teipir.gr*
[8] *Università di of Bergamo, Dept. of Eng. and Applied Science, Viale Marconi 5, 24044 Dalmine (BG), Italy; University of Brighton, School of Computing, Eng. and Maths, Lewes Road, BN2 4GJ, Brighton, UK, m.marengo@brighton.ac.uk*

Extended Abstract

The present industry demand of high heat transfer capability, coupled with reasonably cheap and increasingly small components, leads to the evolution of novel two-phase passive devices. As relatively new and promising members of the wickless heat pipes family, pulsating heat pipes (PHPs) represent a promising and a flexible solution for quite high heat flux applications, up to 30W/cm^2.

A PHP consists of a capillary loop with alternated heating and cooling zones, evacuated and partially filled with a working fluid. Despite its relatively simple structure, the PHP thermal-hydraulic behaviour is complex, chaotic and non-linear. Up to now, very few models are able of complete thermal-hydraulic simulations, and even less are partially validated against experimental data.

In this study, a novel 1D lumped parameters numerical tool able to reproduce the performance of PHPs is described. It consists of a two-phase separated flow model which accounts for the phase changes, as well as the thermal and the fluidic phenomena that appear within a PHP when a slug flow can be assumed. This code has already been validated against experimental data both in quasi-steady state and transient conditions under various gravity levels showing good prediction capability (see an example in Fig. 1).

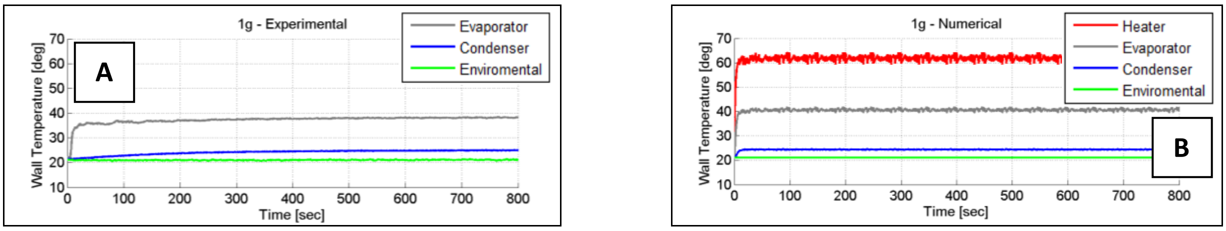


Fig 1: a) Experimental and b) numerical temperature for a ground tested bottom heated mode copper PHP filled with FC-72 (0.5 volumetric filling ratio) when 3.23W/cm^2 are provided in the hot region.

This paper shows the influence of the tube characteristics on the global performance of a planar PHP filled with FC-72 for different values of flux provided in the hot region. The device will be tested in vertical Bottom Heated Mode varying the wall material (e.g. copper, aluminium, PET), the tube internal and external diameter (e.g. ID/OD 1.6/2.5mm, 1.1/2mm, 0.5/1.4mm) and the number of turns in the evaporator sections (e.g. 16, 8, 4). This kind of analyses are very interesting for industries and researches, if the aim is to find ways to improve and optimize PHPs for a large number of applications belonging to different fields, from automotive and aerospace to electronic cooling and house-holding in general.

Numerical Simulation of a Capillary Pulsating Heat Pipe in Various Gravity Conditions

M. Manzoni[1], M. Mameli[2], C. de Falco[3], L. Araneo[4], S. Filippeschi[5] and M. Marengo[6]

[1] *Università degli Studi di Bergamo, Dept. of Eng. and App. Science, Viale Marconi 5, 24044 Dalmine (BG), Italy; University of Brighton, School of Computing, Eng. and Maths, Lewes Road, BN2 4GJ, Brighton, UK, miriam.manzoni@unibg.it*
[2] *Università degli Studi di Bergamo, Dept. of Eng. and App. Science, Viale Marconi 5, 24044 Dalmine (BG), Italy, mauro.mameli@unibg.it*
[3] *Politecnico di Milano, Maths Dept., Piazza Leonardo da Vinci 32, 20133 Milano, Italy, carlo.defalco@polimi.it*
[4] *Politecnico di Milano, Energy Dept., Via Lambruschini 4A, 20158 Milano, Italy, lucio.araneo@polimi.it*
[5] *Università di Pisa, DESTEC, Largo Lazzarino 2, 56122 Pisa, Italy, sauro.filippeschi@den.unipi.it*
[6] *Università degli Studi di Bergamo, Dept. of Eng. and App. Science, Viale Marconi 5, 24044 Dalmine (BG), Italy; University of Brighton, School of Computing, Eng. and Maths, Lewes Road, BN2 4GJ, Brighton, UK, m.marengo@brighton.ac.uk*

Extended Abstract

In the last two decades a new concept of capillary heat pipe without wick structures, commonly known as Pulsating Heat Pipe (PHP), entered the domain of the two-phase passive heat transfer devices. The thermal-hydraulic behavior of this mini-channel with alternate heating and cooling zones, evacuated and partially filled with a working fluid, mainly depends on the interplay between phase change phenomena, capillary and gravity, if present, which may assist or damp the fluid motion.

Numerous are the attempts to simulate PHPs complex behavior, but only a few of them are capable of complete thermal-hydraulic simulations; in addition, none is able to predict the effects of various gravity levels. Nevertheless, validated numerical simulations can constitute useful tools to complete and support experimental studies, and to help the design of new and better performing PHPs.

Thus, a novel lumped parameters numerical code for the transient thermo-hydraulic simulation of PHPs has been developed and validated. It consists of a two-phase separated flow model where capillary slug flow is assumed a priori. A complete set of balance differential equations accounts for homogeneous and heterogeneous phase-changes, as well as thermal and fluid-dynamic phenomena. This novel model shows a very good quantitative and qualitative prediction capability not only when computing the correct measured equivalent thermal resistance, but even when reproducing the experimental trend of temperature when transient conditions are applied (see, for example, Fig 1). This paper presents the comparison between numerical and experimental data, for a copper PHP (I.D./O.D. 1.1mm/2.0mm) filled with FC-72 tested experimentally in micro-gravity (58th Parabolic Flight Campaign), and hyper-gravity conditions (ESA SYT!2013 Programme).

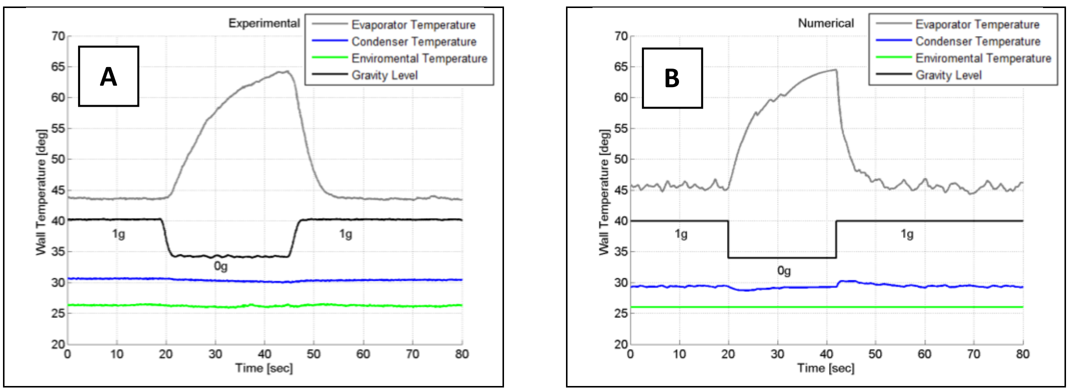


Fig. 1: a) Experimental and b) numerical comparison of the data from the 58th parabolic flight.

Effect of Hydraulic Diameter and Aspect Ratio on Single-Phase Flow and Heat Transfer in a Rectangular Microchannel

A. M. Sahar[1], M. M. Mahmoud[2,3], Jan Wissink[4], T. G. Karayiannis[5]

[1]*Brunel University London, Uxbridge, UB8 3PH, UK, Amirah.MohamadSahar@brunel.ac.uk*
[2]*Brunel University London, Uxbridge, UB8 3PH, UK, mohamed.mahmoud@brunel.ac.uk*
[3]*Zagazig University, Zagazig, 44519, Egypt, mbasuny@zu.edu.eg*
[4]*Brunel University London, Uxbridge, UB8 3PH, UK, jan.wissink@brunel.ac.uk*
[5]*Brunel University London, Uxbridge, UB8 3PH, UK, tassos.karayiannis@brunel.ac.uk*

Extended Abstract

Some researchers such as Dharaiya and Kandlikar (2012), Moharana and Khandekar (2012) and Xing et al. (2013) investigated the effect of aspect ratio on single phase flow and heat transfer in a single microchannel. However, in their study, the hydraulic diameter was also varying with the aspect ratio and hence conclusions are difficult to be assumed as generic, i.e. it is not clear which parameter is the most important one. This work presents a numerical investigation on single-phase flow and heat transfer characteristics in a single rectangular microchannel. Numerical simulations were performed using the CFD software package ANSYS Fluent 14.5. The geometry investigated in this study includes symmetrical cylindrical inlet and outlet plenums and a microchannel. The fluid entered and leaved the channel vertically from the top in a direction normal to the flow direction. The dimensions of the inlet/outlet plenums (diameter and height measured from the channel bottom surface) were kept constant in this study while the width and depth of the channel were varied. The effect of hydraulic diameter was studied by varying the channel width and depth but keeping the aspect ratio constant. The range of hydraulic diameter was 0.1 to 1 mm and the aspect ratio was fixed at 1. In second set of numerical runs, the range of aspect ratio was 0.39 to 10 while the hydraulic diameter was kept constant at 0.56 mm. The simulation was conducted for the range of Reynolds number 100 – 2000 and water was used as the working fluid. A three dimensional thin wall model was used in this study to avoid the effect of conjugate heat. A constant heat flux boundary condition was applied at the bottom wall and both vertical side walls of the channel, while the upper channel wall was considered adiabatic. The simulations results showed that only the hydraulic diameter has a significant effect on the friction factor and heat transfer coefficient.

References

DHARAIYA,V.V & KANDLIKAR,S.G.. 2012. *Numerical investigation of heat transfer in rectangular microchannels under H2 boundary condition during developing and fully developed laminar flow*. Journal of Heat Transfer. **134**, 1-10.

MOHARANA, M.K & KHANDEKAR, S. 2012. *Effect of aspect ratio of rectangular microchannels on the axial back conduction in its solid substrate*. International Journal of Microscale and Nanoscale Thermal and Fluid Transport Phenomena . **4**, 211-229.

XING,D., YAN,C., WANG,C., & SUN,L. 2013. *A theoretical analysis about the effect of aspect ratio on single-phase laminar flow in rectangular ducts*. Progress in Nuclear Energy. **65**, 1-7.

Influence of the microstructure on the transport phenomena on horizontal tubes

C. Tomforde[1], A. Luke[2]

[1] *Kassel University – Department of Technical Thermodynamics, Kurt-Wolters-Str.3, Kassel, Germany, tomforde@uni-kassel.de*
[2] *Kassel University – Department of Technical Thermodynamics, Kurt-Wolters-Str.3, Kassel, Germany, luke@uni-kassel.de*

Extended Abstract

Absorption chillers are used in many industrial applications and offer an opportunity to reduce the emissions of greenhouse gases and to increase the efficiency of technical processes. Although a high efficiency of industrial processes is one of the most important aims in engineering, fundamental mechanisms of the heat and mass transfer have not been completely understood so far. Due to this, the design of all components of an absorption chiller is fraught with a high level of uncertainty. Thereby, the absorber has been identified as the limiting compononent in many investigations (e.g. Killion and Garimella (2001), Beutler (1996)).

It is known that the transport processes are affected by the thermophysical properties of the solution, the thermophysical properties of the material and several geometrical parameters of the absorber heat exchanger. Among the geometrical parameters the surface structure of the tubes seems to have the highest influence on heat and mass transfer. These modifications may be divided into macrostructured and microstructured surface modifications. So far, only very few investigations on the influence of the microstructure have been carried out. Therefore, it must be pointed out that the insight on this kind of modification is very limited.

The influence of microstructured tubes on the heat and mass transfer in an absorber is experimentally investigated in this paper. Aqueous lithium bromide solution is used as working fluid and a polished, a drawn, and a sandblasted surface are compared. The microstructure of the tubes is analyzed with a new three dimensional contactless optical roughness measurement technique based on the focus variation. Several parameters like solution flow rate, solution concentration, or coolant flow rate are varied during the investigations on the transport processes.

The experimental results are discussed and compared with those published in literature in detail. It is shown that an increasing surface roughness leads to higher transport coefficients. One very important reason for this effect is the improved wettability the sandblasted surface compared with polished or drawn tubes. On the basis of the experimental results a modified design method for absorber heat exchangers is suggested.

References

BEUTLER, A., 1997, Wärme- und Stoffübergang bei der Rieselfilmabsorption am horizontalen Rohr. Diss., Technical University Munich

KILLION, J.D., GARIMELLA, S., 2008 A critical review of models of coupled heat and mass transfer in falling-film absorption, Int. J. Refrig. **24** (8), 755-797

14th UK Heat Transfer Conference 2015
Edinburgh (Sept 7 – Sept 8, 2015)

Development of a Solar Cooling System Based on a Fluid Piston Convertor

K. Mahkamov[1] and G. Hashem[2]

[1]*Northumbria University, Newcastle upon Tyne, UK, Email address: khamid.mahkamov@northumbria.ac.uk*
[2]*Northumbria University, Newcastle upon Tyne, UK, Email address: gamal.hashem@northumbria.ac.uk*

Extended Abstract

Solar water pumping and dynamic water desalination has been developed at Northumbria University during the last few years. These systems are built around the fluid piston converter with a simple design and made of low cost materials. In water pump and desalination systems, the fluid piston converter works as an engine driven by solar thermal energy accumulated by flat- plate or evacuated tube collectors. If in the same design the fluid piston is driven using external source of energy without heat input then such a converter works as a cooling device.

In this study, the solar fluid piston converter/engine is coupled with the cooling unit and the fluid piston of the latter is driven by the solar fluid piston converter/engine. The resulting effect is producing cooling effect using solar energy. The operation of such system has been investigated theoretically and experimentally. The thermodynamic model, consisting of a system of ordinary differential equations, was developed in MATLAB/Simulink environment to simulate the operation of this thermal auto-oscillation system. The theoretical results confirm that it is possible to achieve the temperature of the working fluid in the cycle which is below the ambient temperature and that the cooling effect depends on the operational parameters of the engine part of the system.

Performance Evaluation of Room Temperature Magnetic Refrigerator Using Corrugated Plate Regenerator

M. S. Kamran[1,2*], M. Q. Ansari[1], H. Ali[1], N. Hayat[1], Hu Zhang[3], Y. Long[3], H. S. Wang[2]

[1]Department of Mechanical Engineering, University of Engineering and Technology, Lahore, Pakistan
[2]School of Engineering and Materials Science, Queen Mary University of London, Mile End Road, London E1 4NS, UK
[3]School of Materials Science and Engineering, University of Science and Technology Beijing, Beijing 100083, China

*Corresponding author: m.s.kamran@qmul.ac.uk ; +923311492222

Extended Abstract

Magnetic refrigeration at room temperature is a promising new technology with a realistic potential to replace the conventional systems. Active magnetic regenerator (AMR) is the key component determining the cooling performance of a magnetic refrigerator. A numerical simulation of the active magnetic regenerator was performed on two different regenerator geometries (a) corrugated and (b) flat plate for a range of utilization. Gadolinium was taken as the magnetocaloric material and water as the heat transfer fluid. The simulation was performed using ANSYS Fluent with user-defined functions. The peak performance for both the regenerators was found at utilization value of 0.8. It was found that the corrugated plate regenerator exhibited larger temperature spans than the flat plate regenerator in the given range of utilization. The maximum no-load temperature span of corrugated plate regenerator was seen to be around 5% higher than the flat plate.

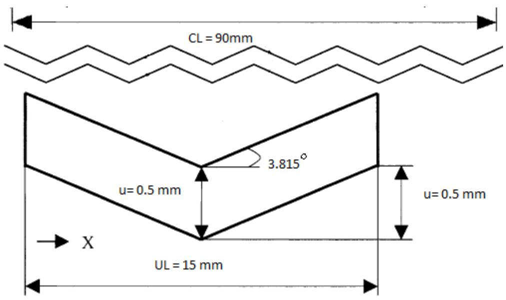


Figure 1 Characteristic dimension of one unit (1/6th) of the complete corrugated channel.

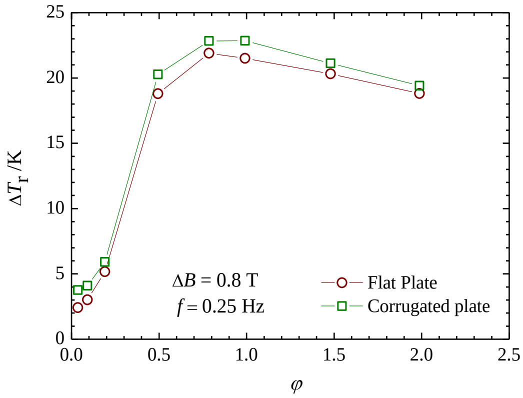


Figure 2 Variation of no-load temperature span with utilization for corrugate and flat plate regenerators

References

Kamran, M. S., J. Sun, J. H. Wu, Y. G. Chen, H. S. Wang, Numerical simulation of room temperature micro-channel regenerator magnetic refrigerator; Fifth IIF-IIR International Conference on Magnetic Refrigeration at Room Temperature, Thermag V, Grenoble, France, 17-20 September 2012.

Nielsen, K. K., Bahl, C. R. H., Smith, A., Pryds, N., Hattel, J., A comprehensive parameter study of an active magnetic regenerator using a 2d numerical model, International Journal of Refrigeration, 33(4) (2010) 753-764.

Heat Exchanger Analysis of Azeotropes in Organic Rankine Cycles

C. J. W. Kirmse[1], O. A. Oyewunmi[2] and C. N. Markides[3]

[1] *CEP Laboratory, Department of Chemical Engineering, Imperial College London, London SW7 2AZ, UK, c.kirmse@imeperial.ac.uk*
[2] *CEP Laboratory, Department of Chemical Engineering, Imperial College London, London SW7 2AZ, UK, oyeniyi.oyewunmi12@imperial.ac.uk*
[3] *CEP Laboratory, Department of Chemical Engineering, Imperial College London, London SW7 2AZ, UK, c.markides@imperial.ac.uk*

Extended Abstract

Organic Rankine cycle (ORC) systems can be used to generate useful work from waste heat and other low- to medium-grade heat sources. Using zeotropic working-fluid mixtures in ORCs reduces the temperature difference between the working fluid and heat source during heat addition. This can lead to an increased power output and thermal efficiency from the cycle (Oyewunmi et al., 2014). However, the average temperature of heat rejection in the condenser is also increased. This might have a negative effect on efficiency and power output. Some mixtures can exhibit azeotropic behavior at certain pressures and compositions. When adding or removing heat at constant pressure, azeotropes evaporate or condense at nearly constant temperature and thus behave like pure fluids. Therefore it could be advantageous to use a mixture which is zeotropic in the evaporator and azeotropic in the condenser. Examples of substances which are azeotropes are R500 (the mixture of R12 and R152a (Jung et al., 1989)) and R504 (mixture of R32 and R115).

In this paper we investigate the effect of using the above mentioned azeotropes in the hot and cold heat exchangers on the performance of a subcritical ORC. The composition of the mixture will be varied, which will affect the pressures at which the mixture is azeotropic. The effects on the following performance indicators of a system based on such a cycle will be examined: the thermal energy added from the heat source; the thermal energy rejected to the heat sink; the power output and the thermal efficiency. From these performance indicators an estimation of important heat exchanger characteristics can be evaluated. Due to the pressure dependence of the azeotrope, it will occur either in the hot heat exchanger, cold heat exchanger or in neither of the two. A significant impact on the performance indicators is expected when an azeotrope exists in either of the two heat exchangers. The compromise between improved thermodynamic performance when small temperature differences in the heat exchangers, and the improved thermal (and cost) performance when large temperature differences in the heat exchangers will be considered.

References

JUNG, D. S., McLINDEN, M., RADERMACHER, R. & DIDION, D. 1989 *A study of flow boiling heat transfer with refrigerant mixtures*. Int. J. Heat Mass Tran. 32 (9), 1751-1764.

OYEWUNMI, O. A., TALEB, A. I., HASLAM, A. J. & MARKIDES, C. N. 2014 *An assessment of working-fluid mixtures using SAFT-VR Mie for use in organic Rankine cycle systems for waste-heat recovery*. Computational Thermal Sciences: An International Journal 6 (4), 301-316.

Parametric Design Study of Vacuum Glazed Windows

H. Ali[1], O. Ali[1], M. Umer[1], M. S. Kamran[1], F. Farukh[2], H U Mughal[1] and H. S. Wang[3]

[1]Faculty of Mechanical Engineering, University of Engineering and Technology Lahore, Pakistan
[2]Wolfson School of Mechanical and Manufacturing Engineering, Loughborough University, Loughborough, UK
[3]School of Engineering and Materials Science, Queen Mary University of London, London E1 4NS, UK
* Corresponding author email: hassan.ali@uet.edu.pk Phone No. +92-333-4436053

Abstract

Windows play an important role in energy flows through buildings. Moreover, provide visibility and natural light while protecting the occupants from dust, noise and infiltration. Windows as compared to walls and doors are considerably less efficient thermally due to limitations of allowing daylight into the building and high visibility for occupants. Vacuum glazed windows have received significant attention due to their low thermal transmittance. Vacuum glazed windows consist of two glass sheets, sealing and support pillars with the separation between the two glass sheets.

The design of vacuum glazing involves consideration of trade-offs regarding decreasing mechanical stresses associated with support pillars (which require more, larger pillars) and reducing heat transfer through pillars (which require fewer, smaller pillars). Moreover, pillars should be small enough to avoid visual obstruction through the vacuum glazed windows. Hence, a novel in-house parametric code has been developed. This paper is focused on the development of method to determine the relationship between the design parameters and corresponding heat transfer in the vacuum glazed windows. The rates of heat transfer per unit length corresponding to certain glass thickness, pillar radius and pillar separation are presented in Table 1.

The parametric feature allows the user to efficiently automate the modeling by changing various design parameters such as size of the glass sheet, width of edge sealing, pillar size and pillar spacing. Moreover, the technique allows the convenient modeling of vacuum glazed windows with non-uniform meshing.

Table 1. Heat flow and thermal coefficient for different glass thickness

Glass thickness (mm)	Pillar radius (mm)	Pillar separation (mm)	Thermal Coefficient (W/m^2K)	Heat Flow (W)
4	1.128	40	1.17	1.4676
6	0.16	40	0.721	0.9016

Key words: Vacuum glazed windows, Energy conservation, Parametric heat transfer study, Thermal performance

Experimental and numerical study on heat transfer and pressure loss in concentric tube heat exchanger with inserts placed on coolant side

S. Abbasi[1], H. Ali*[1], M. Hassan[1], S. A. Nawaz[1], F. Farukh[2] and H.S. Wang[3]

[1]Faculty of Mechanical Engineering, University of Engineering and Technology Lahore, Pakistan

[2]Wolfson School of Mechanical and Manufacturing Engineering, Loughborough University, Loughborough, UK

[3]School of Engineering and Materials Science, Queen Mary University of London, London E1 4NS, UK

* Corresponding author email: hassan.ali@uet.edu.pk Phone No. +92-333-4436053

Abstract

The paper presents experimental and numerical investigation on Thermo-hydraulic characteristics of a concentric tube heat exchanger. The experiments were conducted at a range of Reynolds number and results were in agreement with those obtained from simulation. The results were also compared with those obtained from the analytical method for validation. Further, simulations were conducted by placing non-conventional inserts in the coolant side using Fluent. Various types of inserts were used such as flat perforated plates, curved perforated plates (with different curvatures) and hemispherical shape having different patterns of perforations. Moreover, curved perforated plates were also placed in converging and diverging directions to flow. The effects of Reynolds Number on the Nusselt Number and pressure drop were presented for each case.

It was found that the Nusselt Number and the pressure drop in the tube with the non-conventional inserts are greater than those of tube without inserts. Heat transfer rates and pressure drop for curved perforated plates are found greater than that of flat plates. The optimum angle of curvature with respect to heat transfer enhancement has been reported.

Keywords

Concentric Tube Heat Exchanger, Thermo-hydraulic Characteristics, CFD, Non- Conventional inserts

Turbulent heat transfer between two horizontal planes under inherently stable and unstable conditions

J.D. Jackson[1], S.A. Khan, E.K. Woode and A. McFall

[1]*Emeritus Professor, The University of Manchester, UK,* jdjackson@manchester.ac.uk

Extended Abstract

During the 1960's a strong interest developed in heat transfer to fluids at supercritical pressure which led to considerable experimental work, some of which was driven by practical applications and some was of a fundamental nature aimed at studying the effects of strong non-uniformity of fluid properties on turbulent flow in heated passages. The lead author of the present paper became involved in one such investigation with his close colleague and mentor, the late W.B. Hall, (see Hall, Khan and Jackson, 1966; and Khan, 1966). This very sophisticated and extremely challenging study of fully developed, stably-stratified flow between two horizontal planes, one heated and the other cooled (see Figure 1) produced evidence of strong enhancement of turbulent heat diffusion in the region across the flow where the temperature passed through the pseudo-critical value.

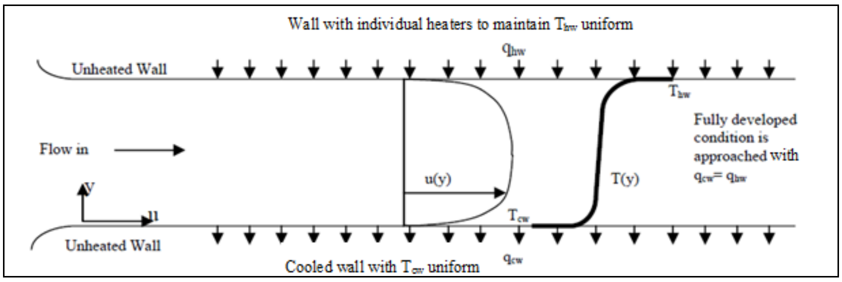


Fig 1: Arrangement for achieving fully developed flow and heat transfer with temperature dependent fluid properties (inherently stable configuration, heated surface uppermost)

This was thought to be due to rapid volume changes of turbulent eddies during mixing. At that time the theoretical treatment of turbulent shear flows was mainly limited to eddy diffusivity and extended mixing length approaches and the topic of computational fluid mechanics was still in its infancy. Further experimental studies using the arrangement referred to above, under conditions where the flow was inherently unstable (cooled surface uppermost), produced results which defied easy explanation and, as a consequence, were written up in Master's degree dissertations (see Woode, 1967 and McFall, 1969) but not published. The present resurgence of interest in heat transfer to fluids at supercritical pressure, at a time when very considerable progress has been achieved on the modelling of turbulent fluid flow and heat transfer, has stimulated the lead author to look again at the early experimental work with a view to it now being simulated using computational modelling techniques. This abstract has been prepared with a view to a paper being offered for presentation at the forthcoming UK National Heat Transfer which describes the early unpublished experimental work. Hopefully, this will lead to attempts being made to reproduce the observed behaviour using advanced turbulence models and also Large Eddy Simulation. The Reynolds numbers involved are much larger than can currently be handled using Direct Numerical Simulation but, eventually as the computing power available for such studies increases this technique could also be used to model the experiments.

References

W.B. Hall, S.A. Khan and, J.D. Jackson 'An investigation of forced convective heat transfer to supercritical pressure carbon dioxide', Paper 25, Vol. 1, pp 257-566, Proceedings of the Third International Heat Transfer Conference, Chicago, USA, 1966

S.A. Khan, Forced convection heat transfer to fluids near the critical point, PhD Thesis, The University of Manchester, UK, 1965

E.K. Woode, Experimental investigation concerning the importance of gravitational body force effects in forced convection heat transfer between horizontal parallel planes with carbon dioxide at slightly supercritical pressure, MSc Thesis, The University of Manchester, UK, 1967

A. McFall, An investigation of combined forced and free convection heat transfer to supercritical pressure CO_2, MSc Thesis, The University of Manchester, UK, 1967

Measuring the Heat Transfer Coefficient in a Direct Oil-Cooled Electrical Machine with Segmented Stator

R. Camilleri[1], P. Beard[2], D. A. Howey[3] and M. D. McCulloch[4]

[1] *Energy and Power Group, Dept. of Eng. Sci., University of Oxford, UK, robert.camilleri@eng.ox.ac.uk*
[2] *Osney Thermo-Fluids Lab., Dept. of Eng. Sci., University of Oxford, UK, paul.beard@eng.ox.ac.uk*
[3] *Energy and Power Group, Dept. of Eng. Sci., University of Oxford, UK, david.howey@eng.ox.ac.uk*
[4] *Energy and Power Group, Dept. of Eng. Sci., University of Oxford, UK, malcolm.mcculloch@eng.ox.ac.uk*

Extended Abstract

This paper presents the measurement of the heat transfer coefficient (HTC) in an oil-cooled electrical machine with a segmented stator using a direct heat flux measurement technique. The quest to improve the torque density in electrical machines requires both the electromagnetic and thermal aspects of the machine to be optimized. As the winding insulation is rated to a maximum operating temperature the hottest spot in the stator windings limits the machine life and torque ratings. Therefore accurate thermal modelling of the machine allows proper choice of material, avoids unnecessary derating and reduces safety factors thus ensuring that high torque densities are achieved. Thermal modelling is often performed using lumped parameter models. These are fast to solve but require parameters such as the HTC. The HTC is difficult to determine and values from empirical correlations and experiments are often quoted. In this paper the double sided direct heat flux gauge shown in Figure 1 is adapted to measure the HTC in an oil-cooled machine stator.

While this type of heat flux gauge has been applied to aerospace components (Jones et al. 1995), to the authors knowledge it has never been applied to electrical machines. The technique was modified by using an internal heater to change the wall temperature, thus adapting it to the boundary conditions presented by oil cooling. The procedure and challenges in measuring the HTC are presented in this paper.

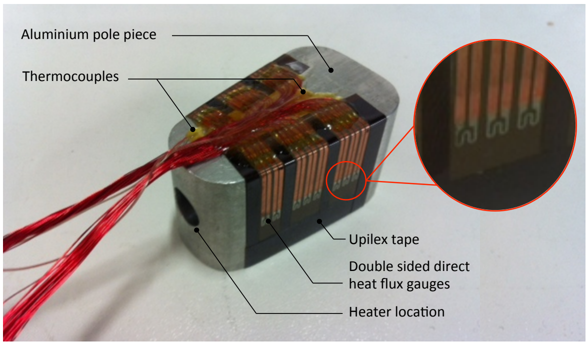


Figure 1: Test piece from a segmented stator with mounted direct heat flux gauges.

Preliminary results suggest that the heat transfer coefficient in the oil cooled stator follows the correleation for a fluid with a Prandtl number > 5 with developing thermal and fluid boundary layers, as presented in Incropera *et al.* 2007, equation (1):

$$\mathrm{Nu} = \mathrm{K}_{\mu,\mathrm{i}} \left[3.66 + \frac{0.0668.\,\mathrm{Pr}.\,\mathrm{Re_i}.\left(\frac{D_\mathrm{h}}{L}\right)_\mathrm{i}}{1 + 0.04.\left(\mathrm{Pr}.\,\mathrm{Re_i}.\left(\frac{D_\mathrm{h}}{L}\right)_\mathrm{i}\right)^{0.667}} \right] \tag{1}$$

References

F. Incropera, D. P. Dewitt, T. L. Bergman and A. S. Lavine, Internal flow: Laminar flow in circular tubes; the entry region, in Introduction to Heat Transfer, 5th ed. New York: Wiley & Sons Inc., 2007, pp. 482-483.

T.V. Jones, The thin film hat transfer gauges – a history and new developments, 4th National UK Heat Transfer Conf., Manchester, UK, 1995

Run-around Coils for Energy Efficiency

E. J. Bentham, P. J. Heggs and T. Mahmud

School of Chemical and Process Engineering, University of Leeds, LS2 9JT UK., p.j.heggs@leeds.ac.uk

Extended Abstract

Run-around coil systems are used to transfer heat between two streams that are spaced apart widely enough to make direct transfer of heat in a single heat exchanger for the two streams a less viable option Reay (1980), Emmerson (1983 &1984) and Wang (1985). In some cases, it is necessary for indirect transfer between the streams, for example air conditioning in hospitals to ensure no cross-contamination of bacteria is possible between the two air streams Eastop and Croft (1990). Also, when the substances in either stream may react with each other, this is a good option to separate them in the interests of safety. Another common example where this type of system is used is the air-cooled radiator system in a car. Although these systems are necessarily less efficient than using a single heat exchanger for normal heat transfer purposes, they are still a useful choice for energy recovery. When placed between two streams that are already set up, for example, it may be easier to install a run-around coil system to connect the two, rather than re-routing the two streams to meet each other Kays and London (1984). Run-around coil systems are used widely in batch processing in the chemical and pharmaceutical industries to heat and cool stirred tank reactors.

A new single equation is developed for the thermal effectiveness of any run-around coil in terms of the thermal effectiveness values of the two individual heat exchangers and the flowing heat capacities of the two process streams and the run-around fluid. A degrees of freedom analysis is developed for the two heat exchangers within the run-around coil systems employing the $E\text{-}N_{TU}$ methodology. This degrees of freedom analysis for design and performance calculations of a run-around coil system reveals that 10 process variables must be specified for a well posed problem. A further 16 variables must be evaluated for the complete system.

Unfortunately in a design problem only 8 thermal variables can be independently specified and two further decisions must be enforced. However for a performance calculation, 6 independent thermal variables can be easily specified along with the two exchanger configurations and flow arrangements. The remaining 4 variables for a well posed problem require the flow and type of pump-around fluid which allow the evaluation of the other two variables: the values of the thermal effectiveness of the two exchangers.

Calculation sequences are proposed for both design and performance calculations in order to obtain the values of the other 16 process variables. The procedures can be repeated in order to maximise energy efficiency.

References

EASTOP T & CROFT D. Energy Efficiency for Engineers and Technologists, Longman Scientific & Technical, 1990.

REAY D. *A review of gas-gas heat recovery systems*, Heat Recovery Systems, 1 (1980), p.18-21.

KAYS W M & LONDON A L. Compact Heat Exchangers – Third Edition, McGraw-Hill Inc. 1984.

EMERSON W. *Designing run-around coils*, Heat Recovery Systems. 3 (1983), p. 305-309.

EMERSON W. *Making the most of run-around coil systems*, Heat Recovery Systems. 4 (1984), p. 265-270.

WANG J C Y. *Practical thermal design of run-around air-to-air heat recovery system*, Heat Recovery Systems. 5, (1985), p. 493-501.

Temperature Dependence of Energy Band Gaps In Triple Junction Solar Cell

Ali M. Maka[1], Tadhg S. O'Donovan[2]

[1] *School of Engineering and Physical Sciences, Heriot-Watt University, United Kingdom, aom2@hw.ac.uk*
[2] *School of Engineering and Physical Sciences, Heriot-Watt University, United Kingdom, T.S.O'Donovan@hw.ac.uk*

Extended Abstract

A multijunction solar cell consisting of three layers of semiconductor materials can be used in concentring solar systems to efficiently and economically convert solar power to electricty and heat. A tandem III-V cell consisting of layers of InGaP/InGaAs/Ge connected in series can have an electrical conversion effieicny in excess of 40% [1], where each layer has a different band gap. A multijunction cell, such as this relies on high optical concentration ratios to achieve high conversion efficiency and economic viability. In this scenario however the heat flux will also be high and can result in high device temperatures or even thermal damage. The current researach concerns the prediction and implication of device temperature of the cell assembly.The Varshni [2] relationship is used to evaluate the semiconductor material band gaps of each layer of the multi-junction solar cell as a function of temperature. Any decrease in band gap leads to a decrease in cell electical conversion efficiency. Cell temperature will reach a steady-state, when power absorbed is equivalent to electrical power and power dissipated in the form of heat to the atmosphere. The heat in solar cell can be either absorbed to increase the cell temperature or released by convection or radiation. An iterative technique to necessary to solve the numerical model to estmiate the cell temperatutre; this is shown to converge after approximately 2000 iterations. Convection heat transfer coefficients were varied from 1.2-1.7 kW/m^2K and the results for an initial cell temperature of 25 ^0C are illustrated in figure 1 below. The band gap has been found to decrease with increasing temperature; the trend is linear, as shows in figure 2. This, in turn leads to reduction in cell performance. Furthermore the decrease in band gap energy of the materials will result in a decrease in open circuit voltage and fill factor. On the other hand, the short current gradually increases and the external quantum efficiency will shift towards the higher wavelengths.

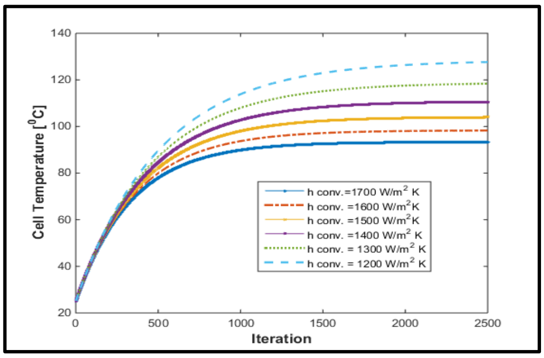


Figure 1: Cell Temperature Convergence

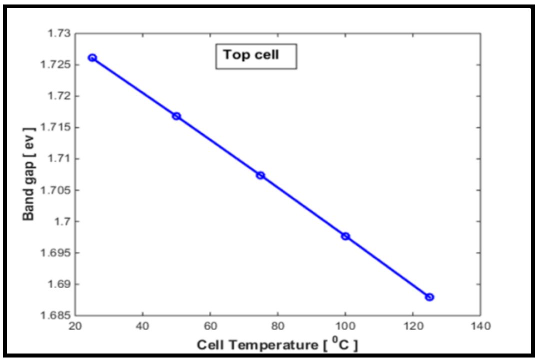


Figure 2: Bandgap vs Cell Temp; InGaP (top layer)

Refrences

[1] R. R. King, D. Bhusari, D, "Solar cell generations over 40% efficiency " Presented at 26TH EU PVSEC, Hamburg, Germany 2011.

[2] Y. Varshni, "Temperature dependence of the energy gap in semiconductors," Physica, vol. 34, pp. 149-154, 1967.

Penetrative convection in a unit aspect ratio enclosure heated by absorption of radiation

I. Amber[1] and T. S. O'Donovan[2]

[1] School of Engineering and Physical Sciences, Heriot-Watt University, Edinburgh, UK, ia196@hw.ac.uk
[2] School of Engineering and Physical Sciences, Heriot-Watt University, Edinburgh, UK, T.S.O'Donovan@hw.ac.uk

Abstract

The problem of penetrative natural convection driven primarily by concentrated solar radiation in a 2D enclosure has been investigated numerically. Penetrative convection is described as the motion of a vertical plume into a fluid layer of stable density and temperature stratification. Understanding the heat transfer and fluid dynamics in penetrative convection is importance to the design of a thermal store for a small scale (circa 5kWe) Concentrated Solar Power (CSP) System.

A unit aspect ratio (H/D) enclosure, filled with molten binary salt KNO_3-$NaNO_3$ (Pr=9), consists of rigid and adiabatic vertical boundaries, a stress free and adiabatic top wall and a rigid lower boundary of finite thickness dx, and known absorption characteristics. The outer surface of the plate is highly adiabatic. The molten salt storage medium is directly illuminated by a concentrated solar flux from the top surface and heats the molten salt by 1) the volumetric absorption of the incident solar radiation; and 2) convection induced from a lower boundary heated by the transmitted radiation, which penetrates the entire depth of the fluid. The aim of this study is to describe the heat transfer and fluid flow within the enclosure and to investigate the effect of natural convection, and aspect ratio on the temperature and flow field. The two-dimensional unsteady state, continuity momentum and energy equations are solved in commercially available Finite Element Analysis (FEA) software (COMSOL).

Results are presented in form of isotherm stream lines and surface plots. A non-linear temperature (fig 1), reveals characteristic zones for penetrative convection: a boundary layer, a mixing layer, the transitional layer and a stable stratified layer which are found to be influenced by the above studied parameters.

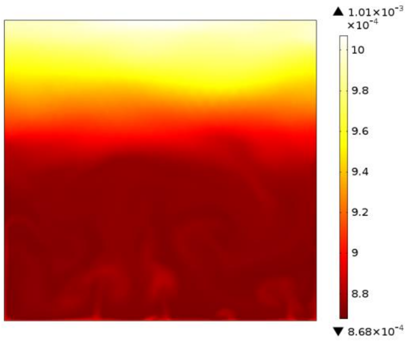


Fig 1: Surface plot for the transient evolution of temperature field for $\tau = 2.5.10^{-4}$ $Ra=8.99\times10^{11}$, Pr=9

References

HATTORI, T., J. PATTERSON, C, AND C. LEI, 2015, *Mixing in internally heated natural convection flow and scaling for a quasi-steady boundary layer.*: J Fluid Mechanics, v. 763, p. 352-368.

MORONI M. AND A. CENEDESE 2006 *Penetrative convection in stratified fluids: velocity and temperature measurements* Nonlin. Processes Geophys., 13, 353–363.

Experimental and Numerical Investigation on Natural Convection in Horizontal Channels Partially Filled with Aluminium Foam and Heated from Below

B. Buonomo[1], L. Cirillo[1], O. Manca[1] and S. Nardini[1]

[1] *Dipartimento di Ingegneria Industriale e dell'Informazione, Seconda Università degli studi di Napoli, via Roma 29, 81031 Aversa (CE), Italy*

Extended Abstract

Natural convection gets a great attention for its importance in practical applications in various modern systems such as electronic cooling, chemical vapor deposition and solar energy systems. Buoyancy force due to the heating of the lower cavity wall induces secondary flows hence the local heat transfer increases. The onset point of the secondary flows is important because it delineates the region after which the two-dimensional laminar flow becomes three-dimensional and a transition motion from laminar to turbulent flow is observed.

In this work natural convection in a horizontal channel partially filled with a porous medium and the lower wall heated at uniform heat flux is studied experimentally.

The experimental test section is made of a horizontal wall and a parallel adiabatic wall. The heated wall consists of a fiberboard plate, 3.2 mm thick, coated with a 17.5 μm thick copper-layer, which is the heater. To reduce heat losses, a 150 mm polystyrene block is affixed to rear face of heated plate. The upper and the side walls of the channel are made of glass rectangular plates in order to take pictures of the flow motion. The distance between the horizontal walls, b, is equal to 40 mm and the distance between the vertical unheated walls, W, is equal to 475 mm. The porous medium is an aluminum foam and it is placed over the heated lower wall. The porous plate has a thickness equal to 20 mm whereas the length and the width are the same of the channel. The aluminum foam has 10, 20 and 30 PPI.

The experiments are performed with working fluid air. Different values of assigned wall heat flux at lower surface are considered in order to obtain some Grashof numbers. The surface temperature is measured by a set of thermocouples embedded in the fiberboard plate in the very proximity of the backside of the copper. Flow visualization is performed to detect the flow patterns in the horizontal channel. Incense smoke is employed for flow visualization. The visualization is made possible by means of a laser sheet, generated by a He-Ne laser source.

A simplified two-dimensional problem is modeled and solved by means of the FLUENT code. The domain is made of the principal channel and two lateral extended domain at the open vertical sections.

The heated wall temperature profiles as a function of Ra values are presented.. Pictures of flow visualization along longitudinal, transversal and horizontal section are given. Average Nusselt numbers are evaluated.

Numerical Investigation on a Latent Thermal Energy Storage with Aluminum Foam

B. Buonomo[1], D. Ercole[1], O. Manca[1] and S. Nardini[1]

[1] *Dipartimento di Ingegneria Industriale e dell'Informazione, Seconda Università degli studi di Napoli, via Roma 29, 81031 Aversa (CE), Italy*

Extended Abstract

Solar energy represents today an important option in energy use and it presents some fundamental peculiarity such as the absence of environmental pollution, a long term availability and a free energy source. However, one of the main drawbacks is the discontinuity of its supply. This problem can be solved by means of thermal energy storage systems (TESS). There are three types of TESS, chemical energy storage system (CESS), sensible heat thermal energy storage system (SHTESS) and latent heat thermal energy storage system (LHTESS).The latest, if possible, is the best choice to increase the energy efficiency because it gives the possibility of storing energy at a quasi-constant temperature with very high energy stored density values.

In this paper, a numerical investigation on Latent Heat Thermal Energy Storage System (LHTESS) based on a phase change material (PCM) is accomplished. The PCM used is a pure paraffin wax having a process melting over a range of temperature and a high latent heat of fusion. However, its thermal conductivity is very low (about 0.2 W/K m) and a method to enhance the heat transfer is putting the PCM into an aluminum metal foam. The geometry of the system under investigation is a vertical shell and tube LHTES made with two concentric aluminum tubes. The internal surface of the hollow cylinder is assumed at a constant temperature above the melting temperature of the PCM to simulate the heat transfer from a hot fluid. The other external surfaces are assumed adiabatic.

A numerical model is employed to simulate the behavior of the PCM embedded with the metal foam. The phase change of the PCM is modelled with the enthalpy porosity theory while the metal foam is considered as a porous media that obeys to the Darcy-Forchheimer law. The momentum equations are modified by adding of suitable source term which it allows to model the solid phase of PCM and natural convection in the liquid phase of PCM. Local thermal non-equilibrium (LTNE) model is assumed to analyze the metal foam and some comparison are accomplished with the local thermal equilibrium model asumption. The governing equations are solved employing the Ansys-Fluent 15 code and verification and validation analysis are accomplished. Numerical simulations for PCM, PCM in the porous medium in LTE and in LTNE assumptions are obtained and their results are compared in terms of melting time and temperature fields. Results as a function of time for the charging phase are carried out for different porosities and assigned pore per inch (PPI).

The results show that at high porosity the LTE and LTNE models have the same melting time while at low porosity the LTNE has a larger melting time. Moreover, the presence of metal foam improves significantly the heat transfer in the LHTES giving a very faster phase change process with respect to pure PCM, reducing the melting time more than one order of magnitude.

14th UK Heat Transfer Conference 2015
Edinburgh (Sept 7 – Sept 8, 2015)

AP40

Comparison of temperature and acoustic monitoring of ice pig passage

E. Lucas[1], et al.

[1] *University of Bristol, University Walk, Clifton, Bristol, BS8 1TR, edward.lucas@bristol.ac.uk*

Extended Abstract

A research aim is to use ultrasound to monitor the passage of an ice pig through metal pipe, with the intention of detecting the onset of ice in order to divert waste from product and also to detect when the ice pig eventually exits the section of tube that is monitored. It is evident that any attendant temperature change as the ice passes the ultrasound detectors could distort the ultrasonic measurements. Temperature can affect the passage of acoustic waves by refracting them where temperature gradients exist. Finally, compensation of temperature effects might correct any tendency for what should be a unit impulse to have an exponential component to the detected signal; Speed of detection might therefore be improved by compensating for temperature. In addition, it was possible that temperature measurement might be a suitable replacement for ultrasound measurement. Invasive temperature measurements with a temperature sensor in the product stream are not precluded but are essentially undesirable in the food processing industry. Instead, external wall temperature may have some utility in detecting any obscuring effect that temperature has on the detection of ice. It was therefore of interest to detect ice flowing within pipes, using ultrasound, in stream and also external wall temperature readings to investigate whether temperature measurements could enhance or even improve on ultrasound measurement. The rate of change of both internal ‘within stream’ and external ‘wall’ temperatures on replacing ice with water was slower than the measured acoustic changes. There is the possibility that while externally sited thermistors are fractionally slower than ultrasound measurements in detecting the ice plug, their use may not materially increase the amount of likely waste.

References

WILSON, W.D. 1959 *Speed of sound in distilled water as a function of temperature and pressure*. J. Acoustic Soc Am. Info. **31**, 1067-1072.

BRUNT, D. 1933 *The adiabatic lapse rate for dry and saturated air*. Quart J. Roy, Meteor, Soc. Info. **59**, 351-360.

AP40

Feasibility Analysis of Molten-Salt Direct Reactor Auxiliary Cooling System

N. Le Brun, C. N. Markides and G. Hewitt

Department of Chemical Engineering, Imperial College London, South Kensington Campus, London, SW7 2AZ, United Kingdom

Extended Abstract

Safety is a primary concern in the next-generation nuclear reactors. One way to assure decay heat removal in the case of LOFC (Loss of Forced Circulation) is to implement passive safety systems. DRACS (Direct Reactor Auxiliary Cooling System) is a passive safety system consisting of a series of heat exchangers capable of removing decay heat by the natural circulation of a working fluid. Molten salts are particularly suitable working fluids for such a system due to their high thermal expansion coefficient and good transport properties. The preliminary design of such a system features a molten salt/air heat exchanger through which the decay heat is ultimately dissipated to the environment (Lv *et al*., 2015). A possible mode of catastrophic failure is the freezing of the salt within the heat exchanger that will interrupt the flow of coolant in the natural circulation loop. The aim of this study is to assess the thermo-hydraulic feasibility of molten salt, natural circulation loops with particular attention to the possible freezing of the salt in the heat exchanger.

A quasi-steady thermo-hydraulic model has been derived and implemented to describe the transient behaviour of DRACS and the transient freezing of the salt. The partial differential equations describing the coupled natural circulation loops of DRACS and the solidification/melting of the salt are solved numerically using a combination of standard explicit and implicit methods. The model has been successfully validated based on previous experimental studies for: 1) transient flow in natural circulation loops (Hallinan and Viskanta, 1986) and 2) transient freezing in pipe flows (McDonald *et al*., 2014). The model was then applied to the preliminary design of a DRACS system. The results of the model show a range of parameters for which the salt is likely to freeze in the heat exchanger and the importance of accurately establishing safe design criteria.

In particular, these results highlight the importance of parameters such as the heat transfer coefficient and the thermal conductivity. In addition, we present a novel technique for the accurate measurements of the thermal conductivity of molten salts up to 700 K, which is necessary to accurately predict the behaviour of DRACS.

References

LV, Q., LIN, H.C., KIM I.H., SUN X., CHRISTENSEN R.N., BLUE T.E., YODER G.L., WILSON D.F. & SABHARWALL P. 2015 *DRACS thermal performance evaluation for FHR*. Ann. Nucl. Engergy. 77, 115-128.

HALLINAN, K. P. & VISKANTA, R. 1986 *Dynamics of a Natural Circulation Loop: Analysis and Experiments*. HEAT TRANSFER ENG. **7**, 43-52.

MCDONALD, A., BSCHADEN, B., SULLIVAN, E., MARSDEN, R., 2014 *Mathematical simulation of the freezing time of water in small diameter pipes.* APPL THERM ENG. **73**, 140-151.

III-V multi-junction cell temperature prediction under concentration and realistic atmospheric conditions

M. Theristis[1], C. Stark[2] and T. S. O'Donovan[3]

[1] Institute of Mechanical, Process and Energy Engineering, Heriot-Watt University, Edinburgh, EH144AS, UK, mt208@hw.ac.uk
[2] Center for Sustainable Energy Systems, Fraunhofer USA, Albuquerque, New Mexico, 87106, USA, cstark@fraunhofer.org
[3] Institute of Mechanical, Process and Energy Engineering, Heriot-Watt University, Edinburgh, EH144AS, UK, T.S.O'Donovan@hw.ac.uk

Extended Abstract

Most installed concentrating photovoltaic (CPV) systems use refractive optics to focus direct sunlight onto a receiver, in order to increase the power output and reduce costs. III-V multi-junction solar cells used in concentrating photovoltaic systems demonstrated electrical efficiencies up to 46% under standard test conditions. However, due to the high heat flux concentration, the temperature rises sharply resulting to suboptimal performance and increases the risk of system failure. Approximately 60% of the incident solar flux is converted to heat on the solar cells. Quantification of the heat power produced on the cell and thus, the temperature is important for the energy prediction of such devices.

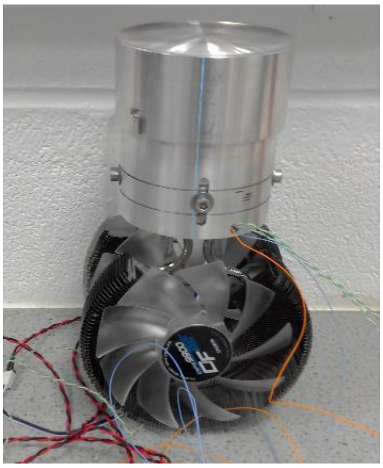


Fig 1: CPV monomodule mounted on a CPU cooling device

In this work, a CPV monomodule (Fig. 1) is exposed on a high accuracy solar tracker in Albuquerque, NM. Current-Voltage (*IV*) characteristics are monitored at maximum power point (i.e. connected to an inverter or a load). Three thermocouples are placed close to the cell. A sunphotometer is used to measure precipitable water (*PW*) and aerosol optical depth (*AOD*); these are then imported into the NREL SMARTS2 model to generate the spectrum. The spectral and weather station data are imported in an electrical numerical model to calculate the performance of the monomodule's triple-junction solar cell as well as the overall electrical characteristics and heat power. Three dimensional finite element analysis is then used to predict the temperature of the CPV monomodule in steady-state. The heat losses are also accounted for through three different mechanisms: conduction, natural convection and surface to ambient radiation. The results are compared with the outdoor measurements.

In order to simulate bulk data for energy prediction, regression analysis is used to predict the heat generated on the cell as a function of irradiance, air mass, *AOD*, *PW*, efficiency. The cell temperature is then calculated as a function of heat power, heat transfer coefficient and ambient temperature.

Identifying Thermal Performance of Two Heat Exchangers for Thermoelectric Generators with CFD

W. Li, M. C. Paul, A. Montecucco, J. Siviter and A. R. Knox

School of Engineering, University of Glasgow, Glasgow, G12 8QQ, UK, Wenguang.Li@Glasgow.ac.uk

Extended Abstract

Thermal performance of heat exchanger of cold side is important for an integrated solar cell and thermoelectric generator (TEG) system. Usually, the thermal performance of a heat exchanger for TEGs is analysed by using a 1D heat conduction theory proposed by Henderson (1979), Yu & Zhao (2007), and Esarte et al (2001). Recently, computational fluid dynamics (CFD) method has been applied by Urbiola and Vorobiev (2013) to characterize the thermal performance of a heat exchanger in the cold side of a TEG. However, thermal performance prediction of heat exchanger with TEG and hot block has not been considered so far. Thus, we have designed two heat exchangers, one is a tube exchanger and another one is a fin exchanger, as shown in Fig. 1. In order to simulate the real working situation of heat exchanger, both heat block with electric heater and TEG are involved. The TEG is simplified by using a 1D heat conduction theory, so its thermal performance is equivalent to the real TEG. The heat transfer in two exchangers is simulated by ANSYS 15.0 CFX by means of steady, 3D turbulent flow ($k-\varepsilon$ model) and heat conduction module under various flow rates. The natural convection effect on the outside surfaces of the computational model is considered.

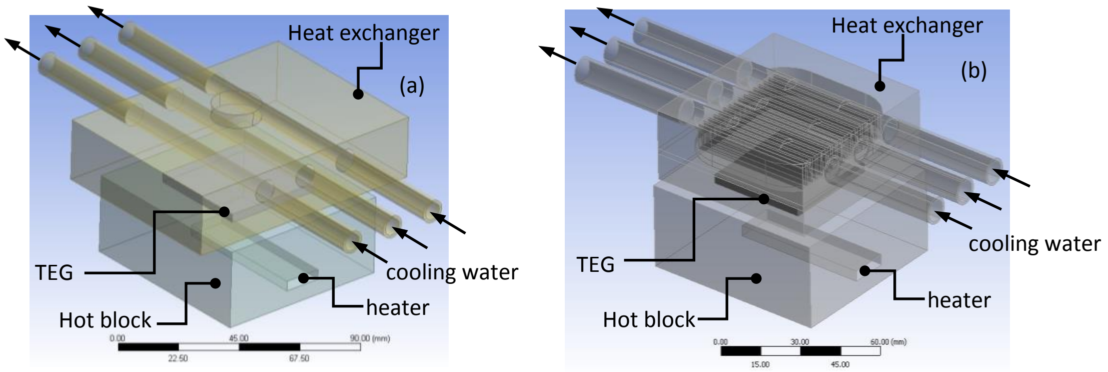


Fig 1. Two heat exchangers for TEG, (a) tube exchanger, and (b) fin exchanger

It is turned out that the two heat exchangers designed have a better thermal performance compared with the existing heat exchanger for TEG. The fin heat exchanger is more compact and has an even better thermal performance than the tube heat exchanger. The CFX heat transfer analysis is an effective way to identify optimal configuration of heat exchanger for TEGs.

References

Henderson J. 1979 *Analysis of a heat exchanger-thermoelectric generator system*, Proceedings of the 14th Intersociety Energy Conversion Engineering Conference, August 5-10, Boston, Massachusetts, USA

Yu J. & Zhao H. 2007 *A numerical model for thermoelectric generator with parallel-plate heat exchanger*, J. of Power Sources **172**, 428-434

Esarte J., Min G., & Rowe D. M. 2001 *Modelling heat exchangers for thermoelectric generators*, J. of Power Sources **93**, 72-76

Urbiola E. A. C. & Vorobiev Y. 2013 *Investigation of sola hybrid electric/thermal system with radiation concentrator and thermoelectric generator*, Int. J. of Photoenergy, http://dx.doi.org/10.1155/2013/704087

Heat Transfer Enhancement Using Partly Porous Channels

R. Nebbali[1] and A. Bousri[2]

[1] *University of science and technology houari boumedien, BP 32 El alia, Algiers, Algeria, rnebbali@usthb.dz*
[2] *University of science and technology houari boumedien, BP 32 El alia, Algiers, Algeria, rnebbali@usthb.dz*

Extended Abstract

The world energy system is almost completely based on the use of fossil fuels. However, these resources are expected to dry up, and have a negative impact on the environment. The use of porous/fluid composite systems is an innovative way that can provide valuable solutions and have a positive impact in domains ranging from preservation of energy resources to limiting global warming. Thus, numerous studies have been dedicated to these issues. Among them, one can quote the earliest work of Hadim (1994) who showed that there is optimal conditions for which heat transfer enhancement could be achieved. Huang et al. (2010) used porous covers on heat sources for electronic ships cooling purpose. Nebbali and Bouhadef (2011) showed that under certain conditions the porous blocks can enhance the heat transfer with the use of less porous matter. This work presents a numerical investigation of fluid flow and forced convection heat transfer in a parallel-plate channel fitted with four porous blocks. Two different arrangements of these blocks have been considered as shown in fig.1. Local heat sources are placed, on the channel upper wall, on the location of the porous blocks. The flow is assumed to be time-independent, bidimensional, and the local thermal equilibrium condition is adopted. The aim of this study is the analysis of the impact of various parameters on the hydrodynamic and thermal caracteristics.

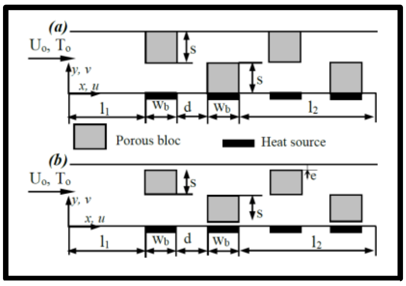


Fig 1: Flow configurations, (a) attached blocks, (b) free blocks.

The results show that the thermal efficiency of the both configurations can be improved in several ways comparatively to the non-porous channel. Thus, the average Nusselt number, Num shows a maximal value of a particular permeability. In addition, compared with the configuration with attached blocks, the narrow gap configuration improves the heat transfer while lowering the pressure drop while using less porous matter. This study confirmed the great potential of the use of partly porous channels to improve heat transfer. However, further studies are needed to optimize the heat transfer gain and keeping the pressure drop at reasonable levels.

References

NEBBALI, R.; BOUHADEF, K. 2011 *Non-Newtonian fluid flow in plane channels: Heat transfer enhancement using porous blocks.* Int. J. Therm. Sci. **10**, 1984-1995.

HUANG Z.F., NAKAYAMA A., YANG K., YANG C., LIU W. 2010. *Enhancing heat transfer in the core flow by using porous medium insert in a tube*. Int. J. heat mass transfer. **53**, 1164-1174.

HADIM A. 1994. *Forced convection in porous channel with localized heat sources*. J. Heat Transfer, **116**, 465-471.

Keynote 5

Ultra-high resolution 3D direct numerical simulations for phase change applications

P. Valluri*, P. Sáenz†, K. Sefiane*, O. Matar‡ and J. Kim‡‡

** School of Engineering, The University of Edinburgh, UK*
† Department of Mathematics, Massachussets Institute of Technology, USA
‡ Department of Chemical Engineering, Imperial College London, UK
‡‡ Department of Mechanical Engineering, University of Maryland, USA

Extended Abstract

This talk will discuss the use of novel 3D, two-phase direct numerical simulation (DNS) methods for phase change problems. The methods concern with solutions of two-phase flow equations (Navier-Stokes) coupled with heat-transport (advection-energy) and mass-transport (advection-diffusion) equations.

The work presented specifically elucidates detailed evolution of thermocapillary instabilities (leading to hydrothermal waves) during evaporation of liquids under three geometrical configurations of pure fluids: i) planar pools ii) hemispherical sessile droplets and iii) non-hemispherical sessile droplets.

We have used our home-grown TPLS2 DNS Solver (http://sourceforge.net/projects/tpls/). We will present our latest findings, which show the break of symmetry and the consequent development of a preferential direction for thermocapillary convection.

As a result, counter-rotating whirling currents emerge in the drop playing a critical role in regulating the interface thermal motion. Our DNS show good agreement with experiments and reveal the intricate drop dynamics due to this geometry-induced phenomenon. The triggering mechanism is analysed along with the resulting bulk flow.

References

1. P. J. Sáenz, K. Sefiane, J. Kim, O. K. Matar and P. Valluri (2015) "*Evaporation of sessile drops: a three-dimensional approach*", Journal of Fluid Mechanics, **772**, 705-739.
2. L. Ó Náraigh, P. Valluri, D. Scott, I. Bethune and P. D. M. Spelt (2014) "*Linear instability, nonlinear instability, and ligament dynamics in three-dimensional laminar two-layer liquid/liquid flows*", Journal of Fluid Mechanics, **750**, 464-506.

Modelling of Polymer Plate Heat Exchangers

Vishwas Wadekar

PS2E Institute, France, Vishwas.Wadekar@Institut-PS2E.com

Extended Abstract

Polymer heat exchangers are emerging as niche market heat exchangers for the duties which cannot be easily performed by conventional metal heat exchangers or the duties that demand expensive alloys or exotic metals. Examples of such duties would include the evaporators used in desalination plants, where titanium would be the standard material of construction and a flue gas heat recovery heat exchanger, where the flue gas containing oxides of nitrogen, sulfur etc, is cooled below the acid dew point temperature. In the latter case because of the extremely corrosive nature of the condensate, a conventional metal heat exchanger is not practical and normally no heat is recovered from these flue gases below the acid dew point temperature.

A polymer heat exchanger is similar in construction to a conventional brazed plate heat exchanger and it offers a number of advantages of plate geometry over the standard shell and tube geometry in terms of better thermal-hydraulic characteristics, compactness and cost effectiveness. A polymer plate heat exchanger, however, is more challenging to model than a conventional metal plate heat exchanger. Commercially available programs, which are used for modelling brazed plate heat exchangers, can be used for modelling a polymer plate heat exchanger to perform design and check rating/simulation calculations. However, when such a program is used for modelling a polymer plate heat exchanger, careful consideration needs to be given to some specific key differences between a polymer plate heat exchanger and a conventional plate and frame exchanger or brazed plate heat exchanger. For example, the conventional brazed plate heat exchanger contains cross-corrugated plate passages, but because of different material properties and the use of different manufacturing technique (i.e. injection moulding) polymer plate heat exchanger passages have a different flow passage structure. The commercial programs contain built-in thermal-hydraulic performance data for cross-corrugated flow passages, which cannot be used for polymer plate heat exchangers. Therefore, specific data on thermal-hydraulic characteristics of plate passage needs to be collected by conducting performance tests with polymer plate heat exchangers to be used with the commercial software for plate exchanger modelling.

The current paper begins by describing the similarities and differences between polymer and metal plate heat exchangers. It then describes an experimental test facility which is currently used to collect the thermal-hydraulic performance data for a polymer plate heat exchanger, along with an innovative arrangement that was used for the flow loop within the experimental facility. Finally, it describes how commercial software for conventional metal plate heat exchanger is used to model a polymer plate heat exchanger.

A reduced numerical model for counter-current two-layer flows

G. Lavalle[1], M. Lucquiaud[2] and P. Valluri[3]

[1] *Institute for Materials and Processes, The University of Edinburgh, EH9 3FB, United Kingdom, g.lavalle@ed.ac.uk*
[2] *Institute for Materials and Processes, The University of Edinburgh, EH9 3FB, United Kingdom, m.lucquiaud@ed.ac.uk*
[3] *Institute for Materials and Processes, The University of Edinburgh, EH9 3FB, United Kingdom, prashant.valluri@ed.ac.uk*

Extended abstract

Many industrial technologies are concerned with liquid-gas channel flows, such as cooling towers, distillation columns and the structured packings for carbon-capture applications.

We present a numerical analysis of a vertical counter-current two-layer flow driven by gravity and pressure in a 2D channel. The gas phase at the top layer is studied with Navier-Stokes equations. We instead model the liquid phase at the bottom with depth-integrated equations, the film thickness being much thinner than the channel length [Ruyer-Quil & Manneville (2000)]. This integral model allows us to reduce the computational cost for the liquid phase, while keeping an accurate wave dynamics. The two-phase coupling is obtained by means of the ALE (Arbitrary Lagrangian-Eulerian) technique, whereat the gas grid follows the interface position [Lavalle (2014)].

We compare the above-described model SWANS (Shallow Water ALE Navier-Stokes) to direct numerical simulations by our in-house solver TPLS [`http://sourceforge.net/projects/tpls/`], and study counter-current, loading and flooding flow regimes. We perturb the interface with a small sinusoidal disturb, and measure the growth of the fundamental wave and the harmonics, showing good agreement with the Orr-Sommerfeld theory and TPLS. We also compare the profile of saturated non-linear waves: Fig. 1 shows convincing agreement between SWANS and TPLS.

Finally, our aim would consist in modeling the mass transfer between the two phases, by accounting for diffusion-convective equations in the liquid and in the gas.

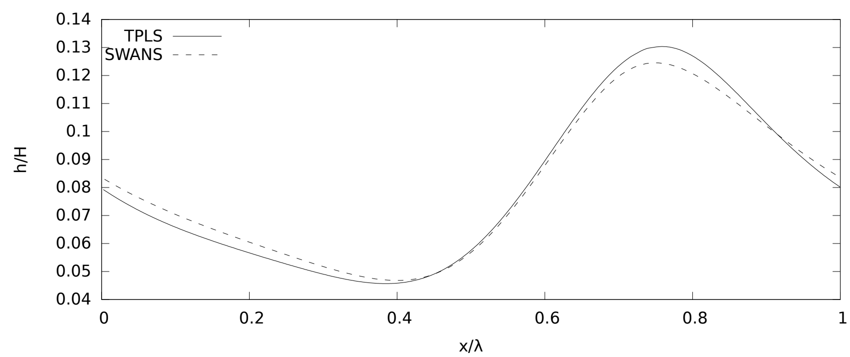


Figure 1: Comparison of the film thickness h between SWANS (dashed line) and TPLS (solid line). Physical parameters: $Re_1 = 0.38$, $H = 10\,mm$ (height channel), $L = 15.7\,mm$ (wavelength), $m = 50$ and $r = 10$ (top/bottom viscosity and density ratios). TPLS data by P.Schmidt.

References

RUYER-QUIL, C. & MANNEVILLE, P. 2000 *Improved modeling of flows down inclined planes*. Eur. Phys. J. B **15**, 357-369.

LAVALLE, G. 2014 *Integral modeling of liquid films sheared by a gas flow*. Ph.D. thesis. Université de Toulouse.

On the generation of nonlinear 3D interfacial waves in gas-liquid flows

L. Ó Náraigh[1,2] and Peter D. M. Spelt[3,4]

[1] School of Mathematical Sciences, University College Dublin, Belfield, Dublin 4, Ireland, onaraigh@maths.ucd.ie
[2] Complex and Adaptive Systems Laboratory, University College Dublin, Belfield, Dublin 4
[3] Université Claude Bernard Lyon 1, Departement of Mechanical Engineering, peter.spelt@univ-lyon1.fr
[4] Laboratoire de Mécanique des Fluides et d'Acoustique (LMFA), CNRS, École Centrale de Lyon

Extended Abstract

We consider the genesis and evolution of unstable interfacial waves in a two-phase stratified laminar gas-liquid flow, from their inception via small-amplitude unstable disturbances right through to ligaments and droplets. For these purposes, we use an in-house parallel two-phase flow solver called TPLS. Interfacial waves are interesting in the context of heat and mass transfer because they promote transport of both heat and mass, not only by advection, but also through enhancement of interfacial surface area. Although the focus of the present work is hydrodynamic, the intention is to extend the work to incorporate heat and mass transfer; for such applications, an understanding of the fundamental hydrodynamics driving the out-of-equilibrium behaviour is essential.

Parallel flows are linearly unstable in the first instance to two-dimensional (streamwise waves), yet three-dimensional waves form in real systems, leading to the formation of extreme nonlinear structures (ligaments, "bags", droplets etc.). A resolution of this paradox has been mooted for liquid-liquid flows [Ó Náraigh *et al.* 2014]. A motivation of the present research is to resolve this paradox for gas-liquid flows as well. Thus, mechanisms for the generation of three-dimensional waves are investigated. First, linear transient growth is examined through Orr-Sommerfeld-Squire analysis and through direct numerical simulation. In this way, transient growth is effectively ruled out as a mechanism for the generation of 3D disturbances. Second, standard linear theory (eigenvalue analysis of the Orr-Sommerfeld-Squire equations) is considered: this is found to play an important role in the generation of the 3D disturbances for suitable parameter values. Finally, the above direct route to 3D disturbances (i.e. standard linear theory) is found to be supplemented in the weakly nonlinear regime by fairly standard weakly nonlinear coupling and Stuart-Landau theory. Beyond weakly nonlinear theory, the generation of extreme non-linear structures is examined in suitable parameter regimes, and novel mechanisms for the breakup of the interface are identified.

References

L. Ó Náraigh, P. Valluri, D. Scott, I. Bethune and P. D. M. Spelt 2014, Linear instability, nonlinear instability, and ligament dynamics in three-dimensional laminar two-layer liquid/liquid flows. *Journal of Fluid Mechanics* 750, 464-506.

Heat transfer in falling liquid films at moderate Reynolds and high Peclet numbers

C. Ruyer-Quil[1], N. Cellier[2] ,M. Chhay[3] and B. Stutz[4]

[1] Université de Savoie Mont-Blanc, Chambéry, France, ruyerquc@univ-savoie.fr
[2] Université de Savoie Mont-Blanc, Chambéry, France, nicolas.cellier@univ-savoie.fr
[3] Université de Savoie Mont-Blanc, Chambéry, France, marx.chhay@univ-savoie.fr
[4] Université de Savoie Mont-Blanc, Chambéry, France, benoit.stutz@univ-savoie.fr

Extended Abstract

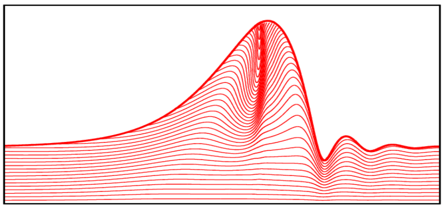


Fig 1: Isotherms under a solitary wave on a vertical wall (Re=2, Pr=7, Bi=0.1, Ka=250)

Falling liquid films offer a promising alternative for the design of efficient heat exchangers as the weakly turbulent wavy dynamics of the film is governed by solitary waves whose interaction greatly enhanced heat transfer. In this work, we consider a liquid film flowing down a vertical plate maintained at a constant temperature. The surrounding gas is assumed to be passive (constant pressure and constant heat transfer coefficient). Marangoni effects are taken into account. We investigate the travelling-wave (TW) solutions of the Navier-Stokes and Fourier equations. TWs are found by continuation through a Hopf bifurcation of the Nusselt uniform-thickness solution using the AUTO07p software [1] after projection on Chebyschev polynomials. An intensification of the waves by the Marangoni effects is observed for roll-waves due to the onset of a thermal layer near the crest of the waves (see figure).

A low-dimensional model, i.e. a set of coupled evolution equations for the film thickness, the local flow rate, the free-surface temperature and the wall temperature gradient has been derived. This set of equations is consistent with the classical long-wave expansion which enables to correctly capture the onset of the hydrodynamic and long-wave thermocapillary instabilities. This new model improves over previous formulations [2] by capturing the main features of the temperature distribution at high Peclet numbers, such as the onset of a thermal boundary layer (see figure).

References

DOEDEL E.J. 2008 *AUTO07p continuation and bifurcation software for ordinary differential equations* Concordia University.

TREVELEYAN P.M.J., SCHEID B., RUYER-QUIl C., KALLIADASIS S. 2007 *Heated falling films*. J. Fluid Mech. **592**, 295-334.

Numercial investigation of a two stage travelling-wave thermoacoustic engine driven heat pump with a hybrid configuration

Ali Al-Kayiem[1]

[1]School of Engineering, University of Glasgow, Glasgow, Scotland, United Kingdom, G12 8QQ, a.al-kayiem.1@research.gla.ac.uk

Extended Abstract

This paper presents the modelling and numerical investigation of a two stage thermally driven travelling wave thermoacoustic heat pump with a hybrid configuration. Hybrid configuration travelling wave thermoacoustic engine essentially employs a near pure travelling wave acoustic resonator to provide the acoustic resonance to the engine unit. Our previous research has demonstrated principle and benefits of such a design.

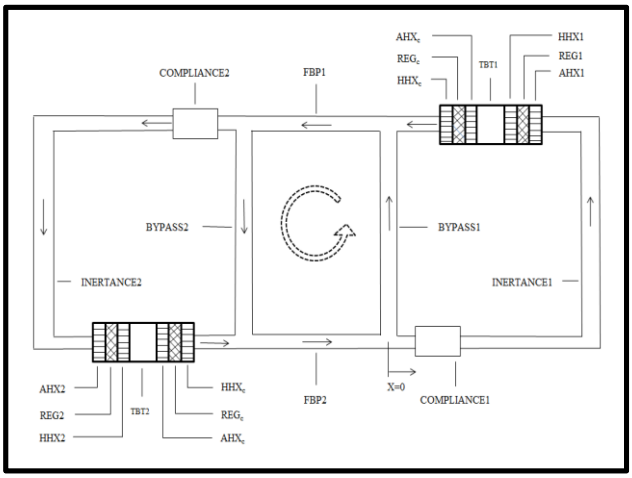


Fig 1: Schematic diagram of the system

The present paper will continue to exploit the benefits of a nearly pure travelling wave acoustic resonator. Two engine units have been installed to share the acoustic resonator to further reduce the acoustic losses, and two thermoacoustic heat pumps have been employed to utilise the acoustic power to upgrade heat. This paper investigates this new concept based on a series of comprehensive numerical simulations using DeltaEC software. The working gas is nitrogen, the mean pressure is 10 bar and the operating frequency is around 75 Hz. The total length of the system is about 5.3m. The integrated system has achieved a 9.2% overall thermal efficiency and a coefficient of performance (COP) of 2.3. This research showed that this new configuration has the potential for developing low cost thermally driven heat pump system.

References

CEPERLEY PH. 1979 *A pistonless Stirling engine-The traveling wave heat engine*. The Journal of the Acoustical Society of America.**66**, 1508.

DE BLOK, K. 2008 *Low operating temperature integral thermo acoustic devices for solar cooling and waste heat recovery*. Journal of the Acoustical Society of America. **123**(5): p. 3541-3541.

AL-KAYIEM A., YU, Z. 2014 *Design of a traveling wave thermoacoustic engine driven cooler with hybrid configuration.* Proceeding of the World congress on engineering Vol **II**, WCE 2014, July 2- 4, 2014, London, U.K.

YU Z., AL-KAYIEM A. 2014 *Numerical Analysis of a thermally driven thermoacoustic Heat Pump for Low Grade Heat Recovery*. Computational Thermal Sciences, **6**(4): 317-327.

Fluid Flow & Heat Transfer Modelling of Adjacent Synthetic Jets

S. Alimohammadi[1], E. Fanning[2], T. Persoons[3] and D. B. Murray[4]

[1,2,3,4] *Trinity College Dublin, Dept. Mechanical and Manufacturing Engineering, Dublin, Ireland*
[1]*alimohas@tcd.ie*, [2]*efanning@tcd.ie*, [3]*tim.persoons@tcd.ie*, [4]*dmurray@tcd.ie*

Extended Abstract

Synthetic jets have become an interesting method of electronic cooling recently, resulting in continued research activities in this field (see References). The formation and interaction of a pair of adjacent synthetic jets is investigated numerically using computational fluid dynamics and experimentally using particle image velocimetry techniques. Both jet actuators are operated at the condition but with an adjustable phase difference $\delta\phi$. The investigation covers the jets issuing into quiescent air (case I) and impinging on a plate (case II).

For realistic computations of flows induced by synthetic jets, a full simulation is performed of the internal flow in the jet cavities. The actuator diaphragm deformation is treated using a moving mesh technique. The results show a reasonable quantitative agreement with our own experiments and PIV measurements done by Smith and Glezer for jets issuing into quiescent air (Fig. 1). These results give confidence that the *CFD* approach can be utilized to predict the intricate flow vectoring phenomenon in adjacent synthetic jets.

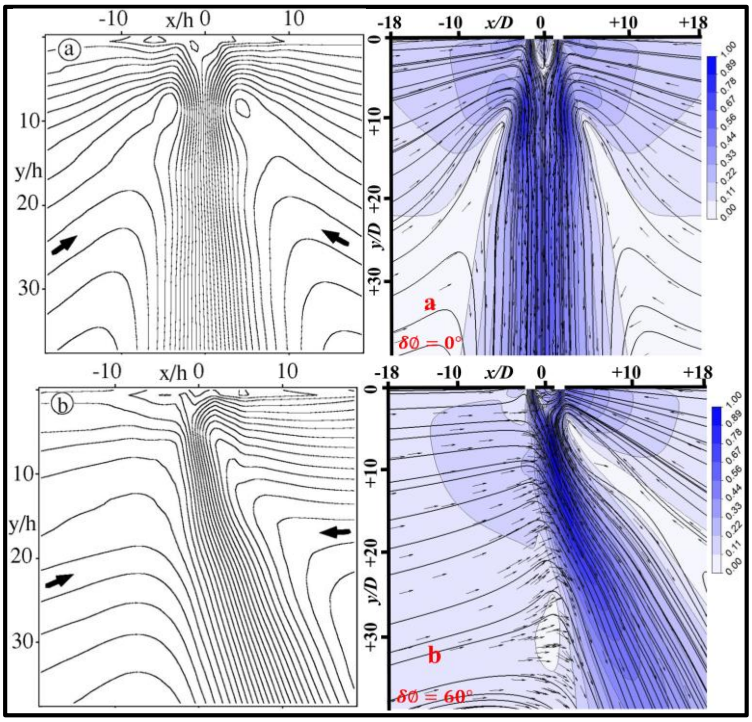


Fig 1. Comparison of PIV (left) versus CFD (right) results

The effect of phase difference between the jets on the vectoring of the merged far field jet is investigated (case I). The merged jet is vectored in the direction of the cavity that is leading in phase, showing a very similar trend between experimental and numerical results on time-averaged streamlines. This leads to a better understanding of the fluid mechanics of adjacent synthetic jets.

Using the validated *CFD* model, impingement heat transfer is investigated (case II). Operating the jets out of phase can lead to enhancement of the local maximum (stagnation) heat transfer coefficient, especially for smaller orifice-to-surface distances (H/D).

References

MCGUINN, A., PERSOONS, T., O'DONOVAN, T. S., & MURRAY, D. 2007 *Surface heat transfer from an impinging synthetic air jet.* 10th UK National Heat Trans. Conf., Edinburgh.
PERSOONS, T., O'DONOVAN, T. S., & MURRAY, D. 2009 *Heat Transfer in Adjacent Interacting Impinging Synthetic Jets*. ASME 2009 Heat Trans. Conference, USA.
SMITH, B. & GLEZER, A. 2005 *Vectoring of Adjacent Synthetic Jets*. AIAA J. 43, 2117–24.

Numerical simulation of Heat Transfer Investigation in New Cooling Schemes of a Stationary Blade Trailing Edge

A. Beniaiche*[1], A. Ghenaiet[2], C. Carcasci[3], M. Pievarolli[3], B. Facchini[3]

[1*] *Laboratory of Thermal Power Systems, Applied mechanics, Ecole Militaire Polytechnique, EMP BP17 Bordj El Bahri, 16046,Algiers, Algeria, Email: beniaiche_ahmed25@yahoo.fr*
[2] *Faculty of Mechanical and Process Engineering University of Sciences and Technology Houari Boumediene USTHB, Al Alalia, BP 32 Bab-Ezouar 16111 , Algiers, Algeria. Email: ag1964@yahoo.com*
[3] *DIEF: Department of Industrial Engineering, University of Florence, Florence, Università di Firenze Via S. Marta, 3. 50139 Firenze – Italia Italy. Emails: carlo.carcasci@unifi.it, marco.pievaroli@htc.de.unifi.it, bruno.facchini@unifi.it*

Extended Abstract

Steady-state RANS computations by means Ansys-Fluent 15.0 using the k-*w* SST turbulence model and the isothermal steady air- flow were conducted to solve the flow field developing inside of both static and rotating passages of 30:1 scaled model (Fig 7.1) reproducing an innovative cooling scheme of a blade trailing edge (TE) of wedge discharging, and a row of enlarged pedestals. The flow results were validated against the PIV measurements for Re = 20000 at stationary and a rotating condition for Ro= 0.23. The aerothermal results are comparable with TLC technique experimental data obtained at Reynolds number = $\{20000, 30000\ and\ 40000\}$ and Rotation number in the range from 0 to 0.15. The results are reported in terms of detailed 2D maps of HTC over the suction side as well as an averaged Nusselt number evaluated inside the inter-pedestals ducts. These results help to better comprehend the complex flow structure and to assess this scheme of cooling system.

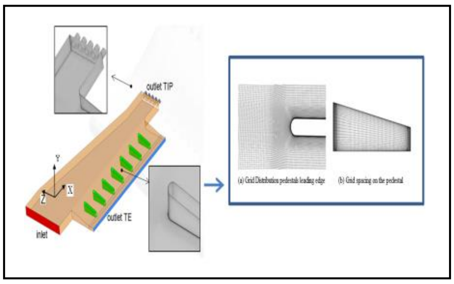


Fig. 1: The studied geometry

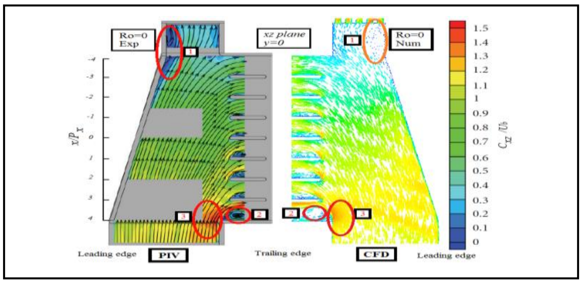


Fig. 2: Cxz/UbVelocity profile distribution, smooth surface, open tip, Re=20000, Ro=0, (left) Experimental PIV data [52] (right) Present numerical prediction.

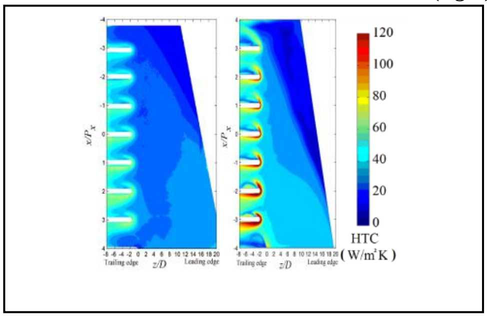


Fig. 3 : 2D HTC [W/m2 K] maps, smooth surface, closed tip, Re=20000, Ro=0.23: (left) TLC Experimentation (right) Numerical predictions.

References

B. Facchini., C. Carcasci, and L. Innocenti., 2003, Heat transfer and pressure drop evaluation in thin wedge shaped trailing edge. ASME GT200338197.

M. Pascotto, A. Armellini, L. Casarsa, C. Mucignat, and P. Giannattasio, 2013, Effects of Rotation at Different Channel Orientations on the Flow Field inside a Trailing Edge Internal Cooling Channel, International Journal of Rotating Machinery Vol 2013, Article ID 765142.

Radiation Effect on Thermal Boundary Layer Flow past a Stretching Plate with Variable Thermal Conductivity

Ravins[1] and Naseem Ahmad[2]

[1]Centre for Interdisciplinary Research in Basic Sciences, Jamia Millia Islamia, New Delhi
ravins@jmi.ac.in
[2] Department of Mathematics, Jamia Millia Islamia, New Delhi-110025, India
nahmad4@jmi.ac.in

Extended Abstract

A closed form solution has been obtained to study the radiation effect on the heat transfer in boundary layer flow past stretching plate with heat transfer with variable thermal conductivity. Applying similarity transformation method, the partial differential equations have been reduced to ordinary differential equations. Due to variable character of thermal conductivity , there is a correction in mean temperature profile. The analysis carried out here shows that the radiated heat influence the mean temperature profile while the correction term remains independent of radiation parameter. To strengthen our claim, we read the radiation effect on mean temperature profile by a graph of mean temperature versus η for randomly chosen values of radiation term N. Also we see the effect of Prandtl on heat transfer for some particular value of Radiation parameter N

SPRS – A Passively Cooled Sellafield Store

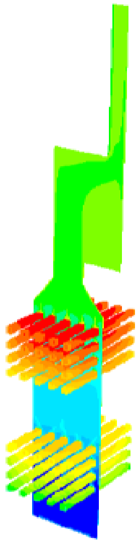
I. Matthew Moorcroft, NNL matthew.moorcroft@nnl.co.uk

II. Paul Cook Sellafield Ltd paul.ma.cook@sellafieldsites.com

Extended Abstract

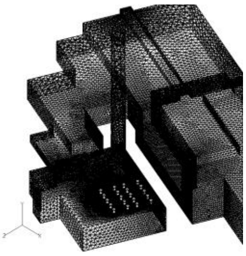
The UK holds a significant stockpile of separated plutonium. This material is mainly stored at Sellafield in purpose built facilities meeting necessary safety and security requirements. Current plans anticipate this material remaining in storage for a number of decades. There are two main streams:- ‘Magnox plutonium’ from the reprocessing of metal ‘Magnox’ fuel; ‘Thorp plutonium’ from the reprocessing of Oxide (AGR, LWR) fuel. Sellafield has existing stores but these are filling and a new store was proposed for a large number of product cans.

The key technical feature of this store is that it was designed and is being operated as one that is passively(naturally) cooled, where the heat of the storage packages drives a buoyancy force within the store. (i.e it has no moving working fluid, no moving mechanical parts, no signal inputs of 'intelligence' and no external power input or forces) This has significant advantages with respect to nuclear safety but also presents challenges in the design and commissioning stages. The design and commissioning also required a detailed understanding of heat transfer withing the store and of fluid flow over a wide range of scales from site modelling of the pressure distributions arising from wind to the flow between and within cells in the store.



A significant program of work was carried out over a number of years and included:-

- Atmospheric modelling– supported by site observations of flows and pressures.



- Building level modelling – supported by measurement.
- Package level modelling – supported by measurement and physical test.

Modelling and calculations were validated by the use of purpose built rigs and by site measurements.

This paper presents an overview of fluid flow and heat transfer aspects of the project concentrating on the contribution of computational fluid dynamics and finite element software to predict behaviour of the store.

Mixed Convection Boundary-Layer Flow near a Stagnation Point in a Nanofluid with Controlled Nanoparticles Volume Fraction

N. A. Yacob[1], N. A. Othman[2] and A. Ishak[3]

[1] *Faculty of Computer and Mathematical Sciences, Universiti Teknologi MARA Pahang, 26400, Bandar Jengka, Pahang, Malaysia, nor_azie@yahoo.com*

[2] *Faculty of Computer and Mathematical Sciences, Universiti Teknologi MARA, 40450, Shah Alam, Selangor, Malaysia, noor.adila.othman@gmail.com*

[3] *School of Mathematical Sciences, Universiti Kebangsaan Malaysia, 43600 UKM Bangi, Selangor, Malaysia, anuar_mi@ukm.edu.my*

Extended Abstract

The mixed convection boundary layer flow of a nanofluid towards over a permeable vertical surface is investigated numerically. We consider the following set of governing equations that represents this particular problem:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$$

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=U\frac{dU}{dx}+\nu\frac{\partial^2 u}{\partial y^2}\pm g\beta\left(T-T_\infty\right),$$

$$u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}=\alpha\left(\frac{\partial^2 T}{\partial y^2}\right)+\tau\left[D_B\frac{\partial C}{\partial y}\frac{\partial T}{\partial y}+\left(\frac{D_T}{T_\infty}\right)\left(\frac{\partial T}{\partial y}\right)^2\right] \tag{1}$$

$$u\frac{\partial C}{\partial x}+v\frac{\partial C}{\partial y}=D_B\frac{\partial^2 C}{\partial y^2}+\left(\frac{D_T}{T_\infty}\right)\frac{\partial^2 T}{\partial y^2},$$

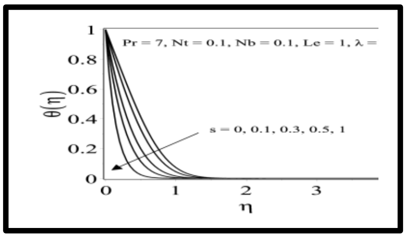


Fig 1: Temperature profile $\theta(\eta)$ for various values of s

subject to

$$u=0,\ v=V_w,\ T=T_w,\ D_B\frac{\partial C}{\partial y}+\frac{D_B}{T_\infty}\frac{\partial T}{\partial y}=0\ \text{ at } y=0,\ u\to U(x),\ T\to T_\infty,\ C\to C_\infty \text{ as } y\to\infty. \tag{2}$$

Eq. (1) subject to Eq. (2) are first transformed into a set of ordinary differential equations (3) using similarity transformation before being solved numerically using a shooting method.

$$f'''+ f\, f''+1-f'^2+\lambda\theta=0,\ \frac{1}{\Pr}\theta''+f\,\theta'+Nb\,\phi'\theta'+Nt\,\theta'^2=0,\ \phi''+Le\,f\,\phi'+\frac{Nt}{Nb}\theta''=0, \text{ subject to boundary conditions } f(0)=s,\ \ f'(0)=0,\ \ \theta(0)=1,\ \ Nb\phi'(0)+Nt\,\theta'(0)=0, \text{ at } y=0;\ f'(\eta)\to 1,\ \theta(\eta)\to 0,\ \phi(\eta)\to 0 \text{ as } \eta\to\infty. \tag{3}$$

Table 1 represents the value of skin friction coefficient for Prandtl number, Pr = 7, mixed convection parameter, $\lambda=0$, suction parameter, $s=0$, nanoparticles volume fraction $\phi=0$ (viscous fluid) which show a favourable agreement with those reported by Rosenhead (1963) and Khan and Pop (2013). Fig 1 shows that the temperature profiles for different values of s. It can be seen that the boundary layer thickness decreases as s increases which in turns increase the temparature gradient and in consequence increase the heat transfer rate at the surface.

Table 1: The values of skin friction coefficient $f''(0)$ for $s=0$ and 0.5

s	Rosenhead (1963)	Khan and Pop (2013)	Present results
0	1.232588	1.2326	1.2326
0.5	-	-	1.5418

References

ROSENHEAD, L. 1963 Laminar Boundary Layers. Oxford University Press, Oxford.

KHAN, W. A. & POP, I. 2013 *Boundary layer flow past a wedge moving in a nanofluid*. Math. Probl. Eng. **2013**, Article ID 637285, doi:10.1155/2013/637285.

Numerical Study on Heat Transfer in Wavy Annular Gas-Liquid Flow

J. Yang[1], F. Sebilleau[2] and G. F. Hewitt[3]

[1] *Imperial College London, London, The Uited Kingdom, junfeng.yang@imperial.ac.uk*
[2] *Imperial College London, London, The Uited Kingdom, frederic.sebilleau11@imperial.ac.uk*
[3] *Imperial College London, London, The Uited Kingdom, g.hewitt@imperial.ac.uk*

Extended Abstract

Early visualisation experiments (Barbosa, et al. 2003 and Hewitt, et al. 1965) shed some light on the interaction between the activity of bubble nucleation sites and disturbance waves in annular flow and stated that as the thickness of the liquid film varied with the presence of the disturbance waves, so did the intensity of the nucleation of the sites. Thereafter, a heat transfer mechanism was postulated that the waves in the film actually trigger bubble nucleation. Several hypotheses, e.g. bubble entrainment and pressure drop, have been proposed to explain the observed nucleation enhancement in the disturbance wave region in the past few decades. The most likely explanation for the effect observed is that (on average) the heat transfer is enhanced under the wave because of turbulence. However, though the average heat transfer coefficient is higher, the fact that the wave is a region of turbulence means that the heat flux on the wall is variable due to wall structures and that there can be small regions of high surface temperature and hence the possibility of nucleate boiling. In the present work, a series of numerical studies have been performed to investigate the heat transfer through a wave/substrate system, see Fig 1a). Modelling results indicated the disturbance waves are packets of turbulence travelling over a laminar film substrate, see Fig 1b). The turbulent fluctuations in the velocity and temperature fields (Figs 1c&d)) might induce local superheated wall zones which further trigger nucleation boiling sites. The current results provide numerical proof for the above hypothesis.

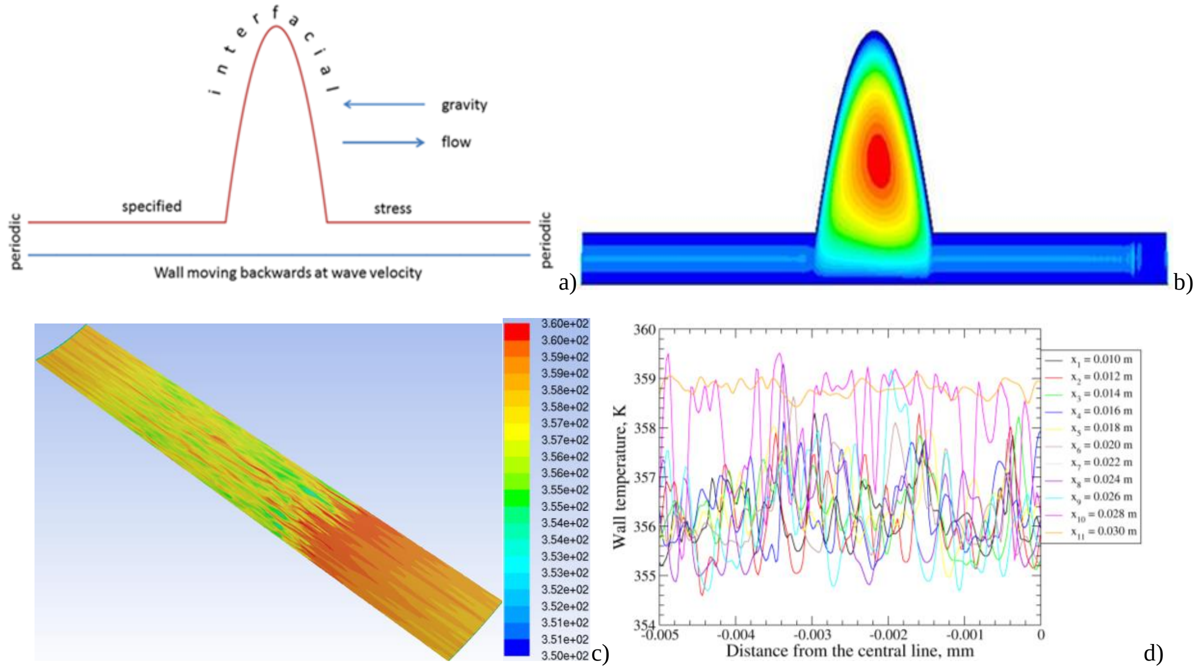


Fig 1: a) the wave/substrate system modelled in the present work, b) predicted turbulent viscosity distribution, c) the instantaneous wall temperature profile underneath the wavy liquid region, d) the spatial temperature distribution.

References

J.R. Barbosa Jr, G.F. Hewitt, S.M. Richardson, High-speed visualisation of nucleate boiling in vertical annular flow, Int. J. Heat and Mass Trans., 46 (2003) 5153-5160.

Developing a scalable and flexible high-resolution DNS code for two-phase flows

I. Bethune[1], T. Collis[1] , M. Jackson[1], L Ó Náraigh[2] and P. Valluri[3]

[1] EPCC, University of Edinburgh, UK, {ibethune, acollis, michaelj}@epcc.ed.ac.uk
[2] School of Mathematical Sciences and CASL, University College Dulbin, Ireland, onaraigh@maths.ucd.ie
[3] Insitute for Materials and Processes, School of Engineering, University of Edinburgh, UK, prashant.valluri@ed.ac.uk

Extended Abstract (Poster Submission)

TPLS (Two-Phase Level Set) is an open-source program for simulation of two- phase flows in 3D channel geometries using high resolution DNS. Due to the high computational cost of these calculations, parallelization is essential, and scaling has now been demonstrated to several thousand CPU cores.

TPLS solves the incompressible Navier-Stokes equations for a two-phase flow. A regular grid finite-volume discretization is employed based on an idealised channel geometry with a range of different inlet conditions that can be prescribed by the user. The interface between phases is tracked with a levelset method. The code evolves the physical variables (pressure, fluid velocities, and interface configuration) through discrete time steps. The pressure is treated using a projection method and the time marching is carried out using a combination of third-order Adams-Bashforth and second-order Crank-Nicholson methodologies. At each time step, the key computational tasks performed amount to the solution of large systems of sparse linear equations with tens of millions of unknowns, for the key physical variables. In addition regular I/O is required to save the system state for later analysis and visualization, or restart in the case of hardware failure.

The code is implemented in Fortran90, initially with MPI parallelization using a 2D domain decomposition and bespoke Jacobi / SOR iterative solvers. Over the last two years, we have improved the TPLS code in several respects to give better performance, scalability and usability, moving from an in-house code specialised for use by the original developers, to a open-source, flexible program which can be easily be used by others, including academic and industrial users.

We present TPLS version 2.0, where we have re-implemented the two most computationally expensive solvers – the pressure and momentum steps – with calls to the PETSc library. Initial tests using the GMRES with a Block Jacobi preconditioner, showed a speedup of 80% in the pressure solve on 2048 cores, along with improved strong scaling behaviour. The original gather-to-master I/O strategy which wrote text files has been replaced with the use of NetCDF. As a result, we have obtained an order-of-magnitude reduction in I/O time, a compression factor of 6.7 and removed the memory bottleneck of requiring rank 0 to gather the entire domain. In addition to the Level Set method, we have added a PETSc implementation of the Diffuse Interface Method (DIM), which is available as an option to users. Finally, with the support of the Software Sustainability Institute, we have added the ability to configure the code through input files or command-line arguments, obviating the need for users to modify and recompile the code for every application. The code has also been refactored to improve extensibility, allowing for currently separate programs that implement counter-current flows and phase change to be included in the main TPLS version in future.

ALE-FEM for two-phase flows with heat and mass transfer

G. Anjos[1], G. Peixoto[2], N. Mangiavacchi[3] and and J. Pontes[4]

[1] GESAR- Department of Mechanical Engineering, State University of Rio de Janeiro, Brazil. gustavo.anjos@uerj.br
[2] gustavo.oliveira@uerj.br, [3] norbert@uerj.br, [4] jose.pontes@uerj.

A numerical method is described to study two-phase flows for single and multiple bubbles with phase change for efficient cooling systems. The fluid flow equations are based on the Arbitrary Lagrangian-Eulerian formulation (ALE) and the Finite Element Method (FEM), creating a new two-phase method with an improved model for the liquid- gas interface in microchannels. A successful adaptive mesh up- date procedure is also described for effective management of the mesh at the two-phase interface to remove, add and repair surface elements, since the computational mesh nodes move according to the flow. The Lagrangian description explicitly defines the two-phase interface position by a set of interconnected nodes, which ensures a sharp representation of the boundary, including the role of the surface tension. The governing equations are shown in dimensionless vector form, where **u**,p and T represent the velocity, pressure and temperature fields respectively; ρ, μ, k and $\mathbf{c_p}$ is stand for density, viscosity, thermal diffusity and the specific heat of the phase Φ, ***Re, Fr*** and ***We*** are dimensionless parameters to characterize the flow regime; t is the time and g is the gravity. On the heat transport equation, **q** stands for the heat flux and H is the Heaviside function.

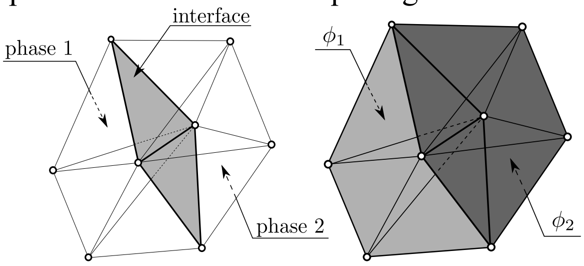


Fig 1: Interface/properties representation in ALE-FEM.

$$\rho \frac{D\mathbf{u}}{Dt} = -\nabla p + \frac{1}{Re}\nabla \cdot [\mu(\nabla \mathbf{u} + \nabla \mathbf{u}^T)] + \frac{1}{Fr^2}\rho \mathbf{g} + \frac{1}{We}\kappa \nabla H$$

$$\rho c_p \frac{DT}{Dt} = \frac{1}{RePr}[\nabla \cdot k\nabla T + \dot{q}|\nabla H|]$$

$$\nabla \cdot \mathbf{u} = \left(1 - \frac{\rho_l}{\rho_v}\right)\dot{q}|\nabla H|$$

The methodology proposed for computing the curvature leads to accurate results with moderate programming effort and computational cost and it can also be applied to different configurations with an explicit description of the interface. The obtained numerical results will be discussed, therefore proving the capability of the proposed new methodology.

References

ANJOS, G. “A 3d ale finite element method for two phase flows with phase change”. PhDthesis, École Polytechnique Fédérale de Lausanne, July - 2012.

ANJOS, G., BORHANI, N., MANGIAVACCHI, N., and THOME, J. “3d moving mesh finite element method for two- phase flows”. *Journal of Computational Physics, 270*, pp. 366–377 - 2014.

JURIC, D. and TRYGGVASON, G. “Computations of boiling flows”. *Journal of Computational Physics, 24*(3), pp. 387–410 – 1998.

Large Eddy & Interface Simulation (LEIS) of Disturbance Waves and Heat Transfer in Annular Flows

J. Yang[1], C. Narayanan[2] and D. Lakehal[3] and G. F. Hewitt[4]

[1] *Imperial College London, London, The Uited Kingdom, junfeng.yang@imperial.ac.uk*
[2] *ASCOMP GmbH, Zurich, Switzerland, chidu@ascomp.ch*
[3] *ASCOMP GmbH, Zurich, Switzerland, lakehal@ascomp.ch*
[4] *Imperial College London, London, The Uited Kingdom, g.hewitt@imperial.ac.uk*

Extended Abstract

A numerical method for forced convective boiling in an annulus needs to be developed in order to elucidate the reason for nucleation enhancement by disturbance waves. The benchmark test case is the experiment of Barbosa et al., in which nucleate boiling in a liquid film, droplet entrainment, disturbance waves of the liquid film, and their interaction were observed. We first develop a numerical strategy to model the development of disturbance waves in annular flows in which the highly turbulent gas core flow drives the laminar liquid flow upwards using advanced CFD tool TransAT. In which, the interface tracking method (e.g. Level-set) combined with a scale-resolving turbulence simulation technique (Large Eddy Simulation) was employed to capture dominant turbulence and interfacial scales using low-to-medium computational costs but reasonable accuracy. The method involves filtering continuity and Navier-Stokes equations *a-priori* defined for the one-fluid formulation. Then, the heat transfer process in the non-boiling annular flow was investigated to provide insight into the temperature gradient underneath the wave region.

Fig 1a) illustrates an instantaneous contour for the liquid-vapour interface and temperature field. The wall temperatures underneath the liquid film are plotted at different time instants, see Fig 1b). The impact of liquid Reynolds number on the film thickness is displayed in Fig 1c). The modelling results are indicative and show that heat transfer is hindered in the wave region. The local overheat zones underneath the disturbance wave could play key roles activating the nucleation boiling sites.

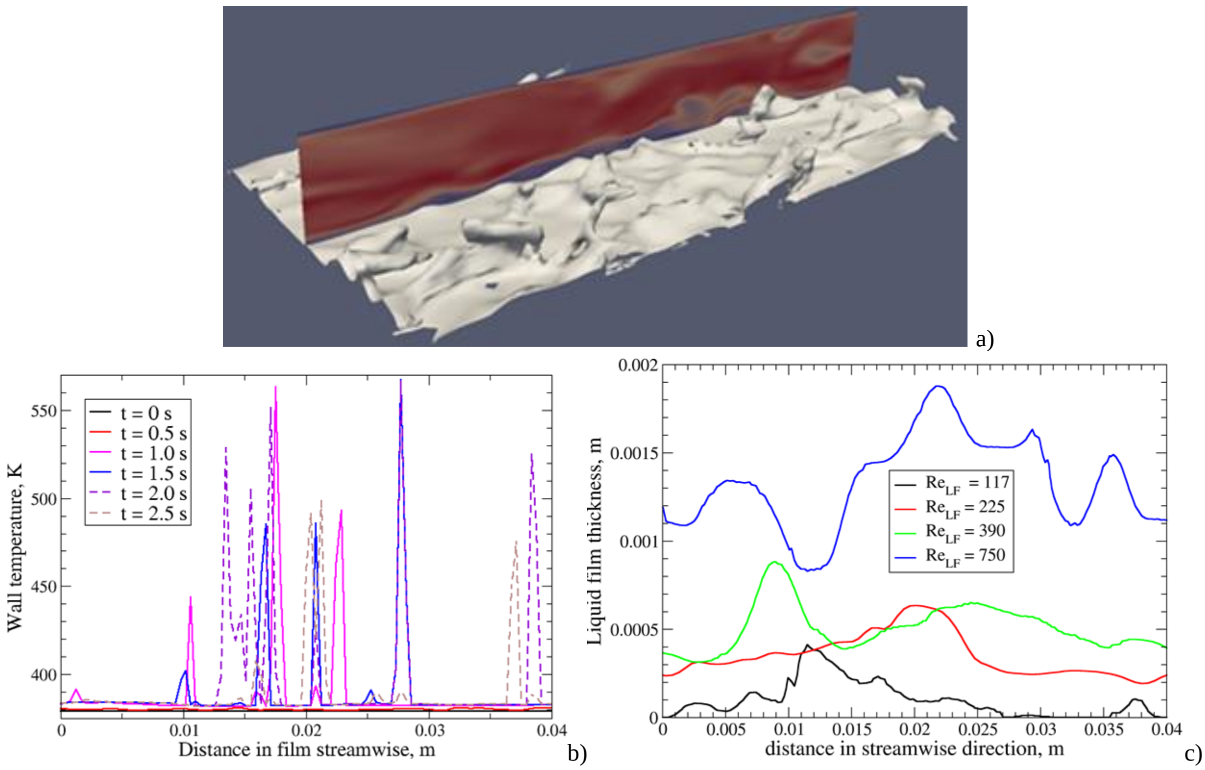


Fig 1: a) the instantaneous contour of liquid-vapour interface and temeprature field, b) the wall temperature underneath the liquid film, c) the impact of liquid Reynolds number on the film thickness.

Impact of Fouling on Thermodynamics Performance of Heat Exchanger: A Computational Fluid Dynamics Study

J. Yang[1] and O.K. Matar[2]

[1] Imperial College London, London, The Uited Kingdom, junfeng.yang@imperial.ac.uk
[2] Imperial College London, London, The Uited Kingdom, o.matar@imperial.ac.uk

Extended Abstract

Crude oil fouling impairs the thermal and hydrodynamic performance of preheat train of crude oil distillation unit. A full-scale simulation approach could improve the understanding of mechanisms responsible for the fouling process and lead to inexpensive and effective mitigation strategies. In the present work, a computationally tractable model consisting of simulations from micro- to meso-scale has been derived for crude oil fouling studies. The micro-scale model based on the SAFT-γ Mie EOS theory has been employed to predict the thermodynamics and phase equilibria of crude oil mixtures. The continuum computational fluid dynamics model has been used to tackle the meso-scale fouling phenomenon, e.g. the spatial and temporal evolution of fouling layer in an industrial heat exchanger, see Figs 1a-c). The heat transfer coefficient (HTC) of fouled heat exchanger has been plotted, see Fig 1d). As can be seen, the wall HCT was hindered at the region adjacent to thick fouling layer. In addition, the 3D CFD modelling also provides a way to understand the relative influence of operating conditions and the interplay between fouling routes on the overall fouling formation rate. This full-scale model was shown to provide an efficient prediction of heat exchanger fouling in crude oil distillation units. Moreover, the modeling results also suggest that a mitigation strategy for fouling can be achieved by controlling the 'interference' between the different fouling routes.

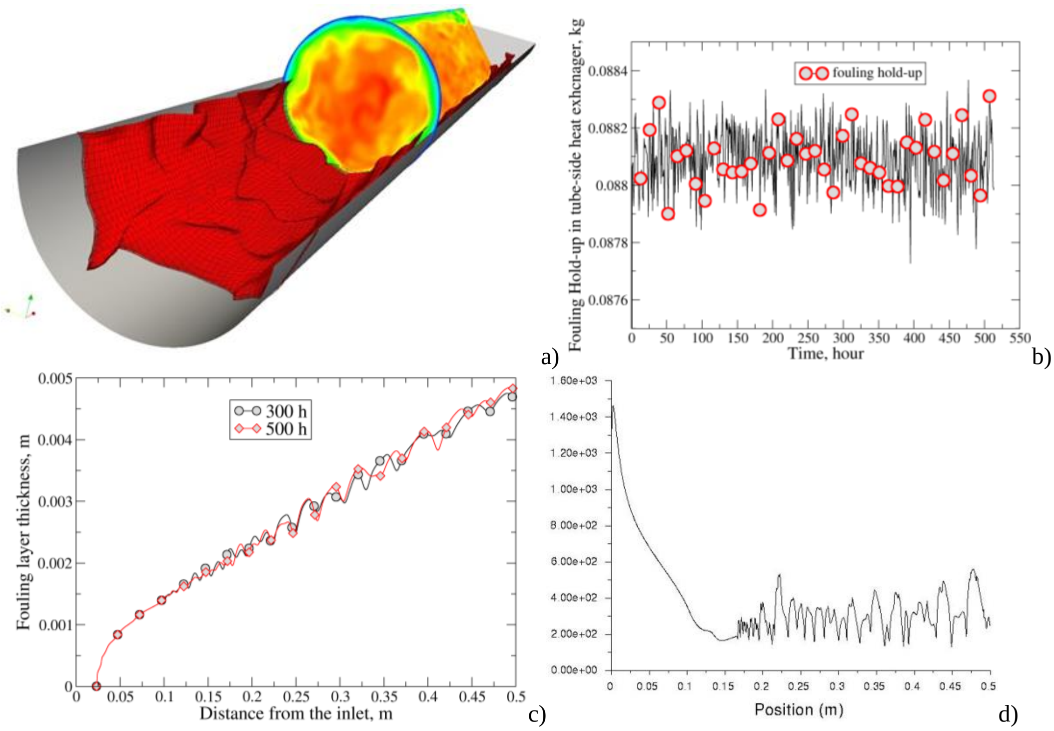


Fig 1: a) Iso-surface of fouling layer and velocity field obtained from LES modeling, b) predicted mass hold-up of fouling in an industrial heat exhcnager, c) predicted spatial and temporal distribution of fouling layer thickness, d) predicted wall heat transfer coeffcient of an fouled ehat exchanger.

Flow analysis in three-dimensional double-diffusive convection in an elongated porous enclosure.

N.Mimouni[1] , O. Rahli[2] , R. Bennacer[3] and S. Chikh[4]

[1] *Laboratoire LTPMP, Fac. GMGP, USTHB, BP 32, Alia, Bab Ezzouar, Algiers, Algeria*
[2] *Laboratoire LTPMP, Fac. GMGP, USTHB, BP 32, Alia, Bab Ezzouar, Algiers, Algeria, rahliomar@yahoo.fr*
[3] *LMT-ENS Cachan, 61 av. du président Wilson F-94235 Cachan Cedex, France*
[4] *Laboratoire LTPMP, Fac. GMGP, USTHB, BP 32, Alia, Bab Ezzouar, Algiers, Algeria*

Extended Abstract

The main purpose of the present study is to analyze the limitation of the parallel flow in 3D double diffusive problem in an elongated enclosure fully filled with a fluid-saturated porous medium subject to constant heat and mass fluxes on the vertical and the horizontal boundaries respectively (Figure1). The Boussinesq approximation is made in the formulation of the problem. The used numerical method is based on the control volume approach with the third order QUICK scheme. Full approximation storage (FAS) with full multigrid (FMG) method is used to solve the problem.

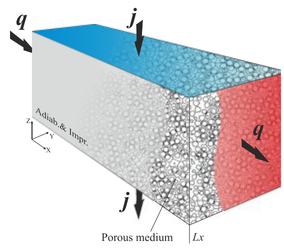


Fig 1: Physical model

For given *Rayleigh number*, the effects of lateral aspect ratio on flow pattern, convective structures, heat and mass transfers are analyzed for a selected range of several controlling parameters; to characterize the bifurcation from 2D to 3D flow.

The steady state equations governing the conservation of mass, momentum, energy and species in non-dimensional form can be written as:

Continuity:
$$\vec{\nabla}.\vec{V} = 0 \tag{1}$$

Momentum:
$$\nabla.(\mathrm{V}\vec{V}) = -\vec{\nabla}\mathrm{P} + \varepsilon^2\Gamma\nabla^2\vec{V} - \frac{\varepsilon^2}{Da}\vec{V} + \frac{\varepsilon^2 Ra_T}{Pr}(\Theta + N\,\Phi) \tag{2}$$

Energy:
$$\vec{V}.\nabla\Theta = \frac{R_k}{Pr}\nabla^2\Theta \tag{3}$$

Species:
$$\vec{V}.\nabla\Phi = \frac{R_D}{Sc}\nabla^2\Phi \tag{4}$$

References

A. Mohamed and R. Bennacer, “Double diffusion, natural convection in an enclosure filled with saturated porous medium subjected to cross gradients; stably stratified fluid,” Int. J. Heat and Mass Transfer **45**, 3725 (2002).

N. Mimouni, R. Bennacer, S. Chikh and O. Rahli, Limitation of parallel flow in double diffusive convection: Two- and three-dimensional transitions in a horizontal porous domain, Phys. Fluids 26 , 074105 (2014).

Identification of a Position and Time Dependent Heat Flux Using the Unscented Kalman Filter in 3D Nonlinear Heat Conduction

C. C. Pacheco[1], H. R. B. Orlande[2], M.J. Colaço[3] and G. S. Dulikravich[4]

[1] *Federal University of Rio de Janeiro, Cid. Universitária, Rio de Janeiro, Brazil, cesar.pacheco@poli.ufrj.br*
[2] *Federal University of Rio de Janeiro, Cid. Universitária, Rio de Janeiro, Brazil, helcio@mecanica.ufrj.br*
[3] *Federal University of Rio de Janeiro, Cid. Universitária, Rio de Janeiro, Brazil, colaco@ufrj.br*
[4] *Florida International University, 10555 West Flagler St., Miami, USA, dulikrav@fiu.edu*

Extended Abstract

This paper addresses the problem of estimating a position and time dependent boundary heat flux with high magnitude in a three dimensional nonlinear heat conduction problem, as shown in Fig. 1, whose dimensions are listed at Table 1. The heat flux is applied on a surface of a flat plate, while transient temperature measurements are taken at the opposite surface. The nonlinear behavior of the problem is due to the material properties, which are functions of local temperature [1]. The inverse problem is solved by using the Unscented Kalman Filter [2], which is a robust technique that provides superior performance compared to the widely-used Extended Kalman Filter, although having the same computational cost.

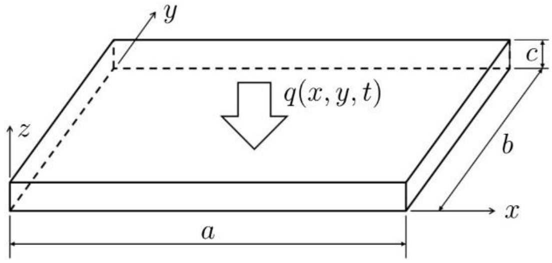


Figure 1. Geometry of the physical problem.

Table 1. Dimensions of the flat plate.

Dimension	Value [mm]
a	120
b	120
c	3

The problem is then approached as a state estimation problem, tracking the temperature values in the three-dimensional domain and the heat flux applied at the surface of the plate. This formulation hinders the use of techniques such as Markov Chain Monte Carlo or Particle Filters, due to the extremely high number of particles needed to obtain good estimates. The UKF, being regarded as an intelligent particle filter [3] allows for solving the inverse problem. Both the temperature and heat flux estimates present an excellent agreement with the reference values, showing the robustness of this approach.

References

ORLANDE H.R.B., DULIKRAVICH G.S., NEUMAYER M., WATZENIG D., COLAÇO M.J. 2014; *Accelerated bayesian inference for the estimation of spatially varying heat flux in a heat conduction problem*. Numers Heat Transf Part A. **65**(1):1–25.

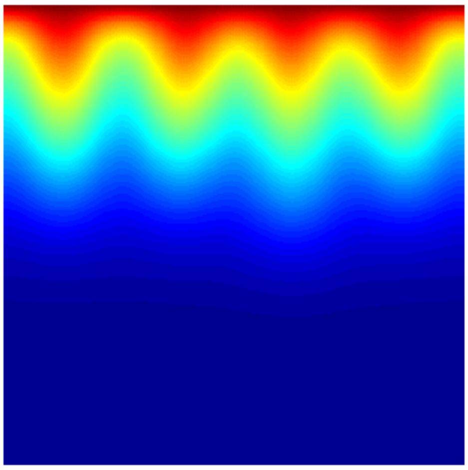
JULIER, S.J.; UHLMANN, J.K., *Unscented filtering and nonlinear estimation*, Proceedings of the IEEE, vol.92, no.3, pp.401-422, Mar 2004

SIMON D. 2006. *Optimal State Estimation: Kalman, H∞, and Nonlinear Approaches*. John Wiley & Sons, Inc.

LSA of fingering in convective dissolution in porous media

R. Lucena[1], N. Mangiavacchi[2], J. Pontes[3], A. De Wit[4]

[1] GESAR- Department of Mechanical Engineering, State University of Rio de Janeiro, Brazil. rachel.lucena@uerj.br
[2] norberto@uerj.br, [3] jose.pontes@uerj.br, [4]Nonlinear Physical Chemistry Unit, Service de Chimie Physique et Biologie Théorique, Faculté des Sciences, Université Libre de Bruxelles (ULB), Belgium. adewit@ulb.ac.be



Fingering refers to hydrodynamic instabilities of deforming interfaces into fingers during the displacement of fuids in porous media. These instabilities are closely linked to changes in viscosity or density between the different layers or within a single phase containing a solute invariable concentration that affects the fuid density or viscosity. In fact, the phenomena are expected to occur in different fields of science and technology, in which flows in porous media are present. We consider the problem of buoyancy-driven fingering generated in porous media by the dissolution of a fluid layer initially placed over a hotter and less dense one in which it is partially miscible. The focus is on the lower layer only where the convective dissolution dynamics takes place. We perform a Linear Stability Analysis (LSA) of the time-dependent temperature profile. The equations describing the dynamics and the linearised evolution equations for the disturbances $\tilde{T}$ and $\tilde{\psi}$ are, respectively:

$$\nabla^2\psi = -\mathrm{R}\frac{\partial T}{\partial x} \qquad \text{and} \qquad \frac{DT}{Dt} = \alpha\nabla^2 T,$$

$$\tilde{\psi}_{yy} - \mathrm{k}^2 = \mathrm{k}^2\tilde{T} \qquad \text{and} \qquad \sigma\tilde{T} + \tilde{\psi}\tilde{T}_y = \tilde{T}_{yy} - \mathrm{k}^2\tilde{T}$$

Finite Element Method is used to solve the equations of the dynamics and we proposed for computing the linear stability analysis the Fourier Transform. The evolution strongly depends on the initial condition. We are currently investigating the nonlinear evolution of instabilities developed with either a flat upper surface or forced by a deformed upper surface.

References

LOODTS, V., RONGY, L., and De WIT, A. *Impact of pressure, salt concentration, and temperature on the convective dissolution of carbon dioxide in aqueous solutions.* Chaos: An Interdisciplinary Journal of Nonlinear Science **24**, 043120, 2014.

BUDRONI, M.A., et al. *Chemical control of hydrodynamics instabilities in partially miscible two-layer systems*. Journal Physical Chemistry Letters **5**, 875-881, 2014.

HOMSY, G.M. *Viscous fingering in porous media*. Annual Review of Fluid Mechanics **19**, 271-311, 1987.

Application of Adomian Decomposition Method for a Stepped Fin Space Radiator with Internal Heat Generation

Rohit K.Singla[1], Ranjan Das[2]

[1]Indian Institute of Technology Ropar, Punjab,India, rohit.singla@iitrpr.ac.in
[2]Indian Institute of Technology Ropar, Punjab,India, ranjandas81@gmail.com

Extended Abstract

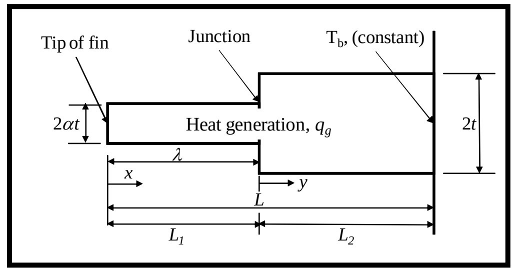


Fig 1: Schematic of stepped in with internal heat generation.

In the present study, a radiative stepped fin with internal heat generation and temperature-dependent thermal parameters (Fig. 1) used in spacecraft or satellite is investigated. Closed form semi-analytical solution is provided using the Adomian decomposition method to evaluate the temperature distributions. The steady state governing equation is formulated by considering conduction and radiation modes of heat transfer. The thermal conductivity and emissivity of the stepped fin material are considered to be linearly temperature-dependent as mentioned below,

$$k(T_1 \,\text{or}\, T_2) = k_b \left\{1+\kappa\left[\left(T_1 \,\text{or}\, T_2\right) - T_c\right]\right\} \quad \text{and} \quad \varepsilon(T_1 \,\text{or}\, T_2) = \varepsilon_b \left\{1+\rho\left[\left(T_1 \,\text{or}\, T_2\right) - T_r\right]\right\} \qquad 1$$

where, T_1 and T_2 are the temperatures at thin and thick sections of the fin, respectively. The governing equations and boundary conditions of the stepped fin for space have been mentioned below,

$$2\alpha t \frac{d}{dx}\left[k(T_1)\frac{dT_1}{dx}\right] - 2\left\{\varepsilon\left(T_1\right)\right\}\sigma\left(T_1^4 - T_r^4\right) + q_g = 0 \qquad 2a$$

$$2t \frac{d}{dy}\left[k(T_2)\frac{dT_2}{dy}\right] - 2\left\{\varepsilon\left(T_2\right)\right\}\sigma\left(T_2^4 - T_r^4\right) + q_g = 0 \qquad 2b$$

$$\frac{dT_1}{dx} = 0; x = 0 \qquad T_2 = T_b; y = L_2 \qquad T_1\big|_{x=L_1} = T_2\big|_{y=0}$$

$$\left[\left\{k(T_2)\right\}(2t)\frac{dT_2}{dy}\right]_{y=0} - \left[\sigma\left\{\varepsilon\left(T_2\right)\right\}\left(2t-2\alpha t\right)\left(T_2^4 - T_r^4\right)\right]_{y=0} = 2\alpha t\left[\left\{k(T_1)\right\}\frac{dT_1}{dx}\right]_{x=L_1} \qquad 3$$

The simplified results from the present case without internal heat generation and with constant conductivity and surface emissivity are validated with those available in the literature (Fig. 2). It is found that the present results are well in agreement with the literature.

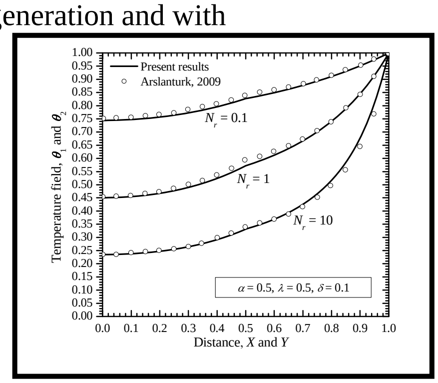


Fig 2: Validation of temperature distribution with Arslanturk, 2010.

References

Arslanturk, C., 2010*Optimization of space radiators with step fins.* P. I. MECH. ENG. G-J. AER. **224**(8), 911-917.

Singla, R. K., & Das, R. 2015*Adomian decomposition method for a stepped fin with all temperature-dependent modes of heat transfer.* Int. J. Heat Mass Transfer **82**, 447-459.

14th UK Heat Transfer Conference 2015
Edinburgh (Sept 7 – Sept 8, 2015)

Heat Transfer Rate from Hydrogen to Tank Wall during Fast Refuelling Process

M. Monde[1], T. Kuroki, N. Sakoda[1] and Y. Takata[1]

[1] Kyushu University, 744 Motooka Nishi-ku, Fukuoka, Japan, 819-0395, monde.masanori. 357@m.kyushu-u.ac.jp

Abstract

For safe and fast refuelling hydrogen within about 3 minutes to fuel cell vehicles (FCVs) at hydrogen refuelling stations (HRSs), a refuelling protocol so called look-up table based on SAE J2601will be introduced as a global standard from 2015. This protocol makes it sure to keep a safety requirement that the hydrogen in the vessel should be less than p = 87.5 MPa and T = 85 °C for any refuelling condition. However, this standard might be inconvenient and over-expected for a hydrogen station due to an existence of some unknown parameters and without any communication between HRSs and FCVs. In order to improve this protocol, one has to understand heat transfer rate during refuelling of hydrogen, since the stored internal energy being function of p and T is mainly governed by the supplied enthalpy and heat transfer between wall and hydrogen.

Keynote 6

Fouling – Is it Still the Major Unresolved Problem in Heat Transfer?

F. Coletti

Chief Technology Officer, Hexxcell Ltd.

Extended Abstract

Over 40 years ago, fouling – the deposition of unwanted material on heat transfer surfaces – has been described as the major unresolved problem in heat transfer. Since then the large economic, environmental, operational and safety impact of fouling on the process industry has been recognised. As a result, significant research efforts have been spent in this area by both academia and industry – but how much progress have we made towards solving the problem and where are we today?

The first part of this lecture will introduce the issues caused by fouling in various process industries, briefly review past and present research programmes dealing with fouling, discuss the current understanding of the underlying phenomena and illustrate the limitations of existing heat exchanger design methodologies.

In the second part of the lecture, the comprehensive framework needed to attack this long standing problem in a systematic way will be presented. It will be shown how deeper understanding of the deposition process can be integrated with software tools to predict fouling as a function of process conditions and time. The potential benefits of this approach will then be demonstrated using crude oil fouling in refinery pre-heat trains as a case study. These include either eliminating fouling by design where possible or appropriately managing its impact by optimising operations.

Final comments will report some thoughts on the future of this exciting and challenging research field. These will address the need to improve our fundamental understanding of the fouling process, the existing heat exchanger design tools and operating practices to provide measurable benefits to industry and, ultimately, the economy and the environment.

14th UK Heat Transfer Conference 2015
Edinburgh (Sept 6– Sept 8, 2015)

Simulation of Droplet Heated by Laser for PCR Application

Zhibin Wang[1,2], Rong Chen[1,2] *,Qiang Liao[1,2], Xun Zhu[1,2], Shuzhe Li [1,2]

[1]Key Laboratory of Low-grade Energy Utilization Technologies and Systems (Chongqing University), Ministry of Education, Chongqing 400030,China
[2]Institute of Engineering Thermophysics, Chongqing University, Chongqing 400030, China

First author and speaker. Tel.: 0086-23-65102474; fax: 0086-23-65102474; e-mail: wangzhibinbill@qq.com
**Corresponding author. Tel.: 0086-23-65102474; fax: 0086-23-65102474; e-mail: rchen@cqu.edu.cn*

Extended Abstract

The droplet based microfluidics for polymerase chain reaction (PCR) has received ever-increasing growth, which provides several advantages over the conventional PCR technologies, such as large specific surface area, small volume, reducing the inhibition of surface, avoiding the contamination and so on. In recent, the incorporation of modern optics into microfluidics, which is so-called optofluidics, has brought new ideas to the microfluidic applications. Among them, the photothermal effect based optofluidics shows the promising potential in the application of the droplet based microfluidics for high speed real time PCR. Using the laser as the heating source can provide quick temperature response and control without significant effect on the carrying fluid and the substrate so that it is attractive in the development of the droplet based microfluidics.

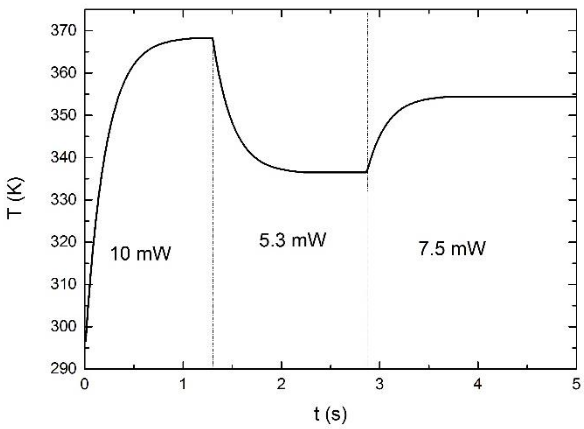


Fig. 1 Temperature response to laser power in one PCR cycle

In this study, we have numerically studied the temperature response of a droplet heated by an infrared laser with the wavelength of 1550 nm in a microchannel. In this simulation, a constant heat transfer coefficient is estimated for the interfacial heat transfer between the droplet and carrying fluid. The Guass heat source is used to model the laser beam and the shear stress on the interface of droplet due to the surface tension temperature coefficient is taken into account. We also study the effect of the laser power, beam size and laser beam location on the temperature response of a droplet. It has been found that the average temperature is linear with the laser power, and the temperature response to the infrared laser is rather quick. At last, we obtain the temperature reponse of the droplet in one PCR cycle, as shown in Fig. 1. Clearly, adjusting the laser power can well meet the conditions for a PCR cycle. The simulated results are helpful for the design of the droplet based microfluidics with the laser as a heating soruce for PCR applications.

Three dimensional simulation of a focused infrared laser heated droplet in microchannels

Rong Chen[1,2], Shuzhe Li[1,2], Qiang Liao[1,2]*, Xun Zhu[1,2], Zhibin Wang[1,2]

[1]Key Laboratory of Low-grade Energy Utilization Technologies and Systems (Chongqing University), Ministry of Education, Chongqing 400030,China
[2]Institute of Engineering Thermophysics, Chongqing University, Chongqing 400030, China
First author. Tel.: 0086-23-65102474; fax: 0086-23-65102474; e-mail: rchen@cqu.edu.cn
Speaker. Tel.: 0086-23-65102474; fax: 0086-23-65102474; e-mail: lishuzhe702@163.com
**Corresponding author. Tel.: 0086-23-65102474; fax: 0086-23-65102474; e-mail: lqzx@cqu.edu.cn*

Extended Abstract

Optofluidics with the photothermal effect has been applied to the droplet based microfluidics and received much attention. In this work, we have numerically investigated the flow and heat transport behavior inside a droplet positioned in a microchannel, which is heated by an infrared laser with the wavelenghth of 1550 nm, as sketched in Fig. 1.

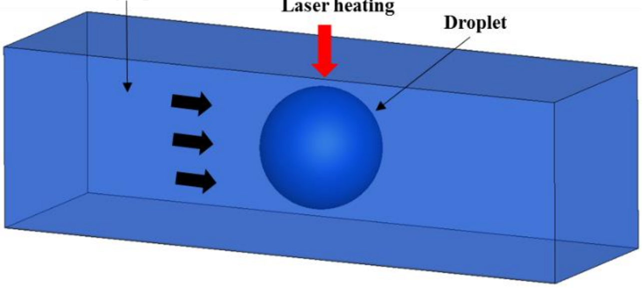


Fig 1: Schematic diagram of the physical model

In this simulation, the carring fluid and droplet simulated are oil and water, respectively. The volumetric Guass heat source is used to model the laser beam and the shear stress on the interface of droplet due to the surface tension temperature coefficient is taken into account. Particular attention is paid to the effect of the laser power and spot location on the Marangoni convection and the temperature response in droplet. For the case of carrying flow under speed of 0.1 *mm/s* in a microchannel with 150 μm in width and 500 μm in length, when the laser spot of 10 μm in diameter located at the center of droplet with the power of 60 *mW* and 100 *mW*, the response of the droplet temperature are shown in Fig. 2. It has been found that the four asymmetry strong vortexs are formed in droplet due to the surface tension gradient. In addition, the different spot locations along and perpendicular to the carring flow direction are simulated to understand the mechanism of the coupled flow and heat transfer. The results show that the droplet temperature is deeply influenced by the spot location as a result of different heat absorption efficiency and the nonuniform heat transfer. Simultaneously, the shape of the rotation flow inside droplet shows a significant transformation depending on the position of the laser spot. We also find that the maximum intensity of vortex exists in the case of the laser spot located between the center and edge of droplet. The simulated results are to benefit of the application of the droplet based microfludics, such as PCR and mixer.

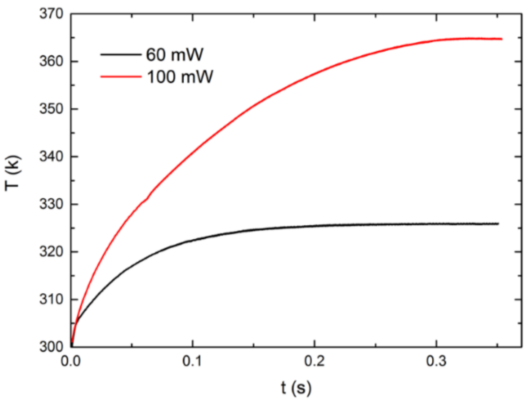


Fig 2: The time evolution of the droplet temperature

A Computational Study of Thermal Losses in a Reciprocating Piston-Cylinder System

A. I. Taleb and C. N. Markides

Clean Energy Processes (CEP) Laboratory, Department of Chemical Engineering, Imperial College London, South Kensington Campus, SW7 2AZ London, UK, a.taleb12@imperial.ac.uk, c.markides@imperial.ac.uk

Extended Abstract

Reciprocating-piston engines offer the potential of high efficiencies for the conversion of low-and medium-grade heat-sources such as solar or waste heat. Once mechanical losses are minimized, unsteady thermal losses become the dominant loss mechanism in piston expanders and compressors and may reach values up to 19% of the total useful work lost (Mathie *et al.* 2014).

Previous work in this area was conducted most notably by Kornhauser & Smith (1993) who performed experiments on a piston-cylinder gas spring to measure losses under different conditions (e.g. of pressure, speed, gas). The losses arise due to the alternating heat transfer across a finite temperature difference between the cylinder wall and the thermal boundary layer. This heat transfer is out of phase with the difference in temperature between the cylinder wall and the bulk fluid. It was found that a non-dimensional loss term was independent of all parameters apart from the non-dimensional Péclet number Pe. The losses were low at both high and low values of Pe with a peak at intermediate values. However, over large variations of the volume ratio, the loss does not correlate well with the Péclet number.

Previous studies, such as Kornhauser & Smith (1994), describe empirical correlations between the Péclet and a complex Nusselt number to account for the phase shift between the temperature difference and heat transfer. Unfortunately, these correlations are not applicable at high volume ratios (above 2) occurring in piston expanders and compressors. Therefore, the objective of this study is to develop a CFD model of a gas spring capable of calculating the thermal loss for different volume ratios and other configuration parameters. The thermal loss is quantified as a non-dimensional loss term defined by Mathie *et al.* (2014) as $\psi = \oint P\mathrm{d}V \,/\, \oint|P\mathrm{d}V|$.

As a first step, the experiments from the literature will be reproduced in the simulations to validate the CFD model. Subsequently, a parametric study will be carried where most importantly the volume ratio is varied. The results will provide information on the work losses and their trends. They can also be used for the further development of analytical models and correlations. Initial results indicate that the losses increase at high volume ratios, especially for high Péclet numbers which has also been observed in the experiments of Kornhauser & Smith (1993).

References

KORNHAUSER, A. A. & SMITH, J. L., Jr. 1993 *The Effects of Heat Transfer on Gas Spring Performance.* J. Energy Resourc. Technology **115**, 70-75.

KORNHAUSER, A. A. & SMITH, J. L., Jr. 1994 *Application of a Complex Nusselt Number to Heat Transfer during Compression and Expansion.* J. Heat Transf. **116**, 536-542.

MATHIE, R., MARKIDES, C. N., & WHITE, A. J. 2014 *A Framework for the Analysis of Thermal Losses in Reciprocating Compressors and Expanders*. Heat Transf. Engineering **35**, 1435-1449.

Numerical and Experimental Investigation of Pulsating Flow for Fabric Drying Application

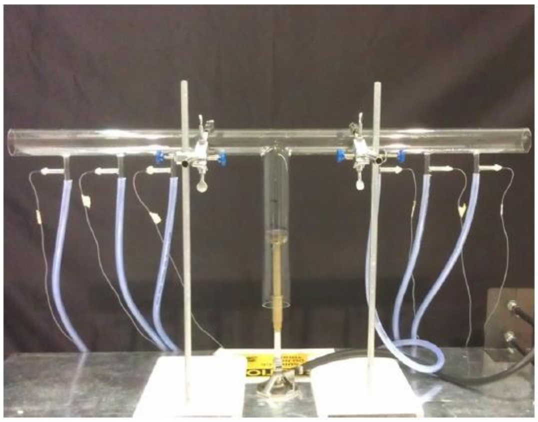
D. Zhao[1*], S. H. Li[1] and S. Chen[1]

[1] *School of Mechanical and Aerospace Engineering, Nanyang Technological University, 50 Nanyang Avenue, Singapore,*
zhaodan@ntu.edu.sg

Extended Abstract

The present work considers a convection-driven T-shaped pulse combustion system. Both experimental and numerical investigations are conducted to study the mechanism of pulsating flow generated by unsteady combustion and its potential application in fabric drying. To gain insight on flame-acoustics dynamic interaction and pulsating flow characteristics, 3D numerical simulation of the pulse combustion process of a premixed turbulent flame in an open-ended T-shaped combustor is performed. Two parameters are examined: (1) fuel-air ratio, (2) inlet mean temperature. Their effects on triggering pulsating flow and Nusselt number are studied. As each of the parameters is varied, Nusselt number characterizing the heat transfer rate and the heat-driven pulsating flow signature is found to change. The main nonlinearity is identified in the heat fluxes. To validate our numerical findings, a cylindrical T-shaped open-ended combustor made of quartz-glass with a Bunsen burner is designed and tested. Two arrays of pressure sensors and an infrared thermal imaging camera are used to monitor the pressure and temperature field along the tube. The experimental measurements show a good agreement with the numerical results in terms of eigenfrequency, mode shape and sound pressure level.

Fig. 1 Experimental setup of the T-shaped combustion system with pulsating flow produced.



References

Dan Zhao, C. Ji, X. Li, 2015 *Mitigation of premixed flame-sustained thermoacoustic oscillations using an electrical heater*. Int. J. Heat Mass Trans. **86**, 309-318.

Dan Zhao, X. Li, 2015 *Minimizng transient energy growth of nonlinear thermoacoustic oscillations*. Int. J. Heat Mass Trans. **81**, 188-197.

Numerical Investigation on an Inclined Ventilated Roof with Different Exit Section

B. Buonomo[1], A. Diana[1], O. Manca[1] and S. Nardini[1]

[1] *Dipartimento di Ingegneria Industriale e dell'Informazione, Seconda Università degli studi di Napoli, via Roma 29, 81031 Aversa (CE), Italy*

Extended Abstract

One of the European Directive priority are represented by the improvement of "building performance requirements" and the development of new strategies for "very low energy buildings". The goals, in particular, is the reduction of the energy consumptions due to the heat flux transmitted through the envelope of residential and commercial buildings. In regions, like Mediterranean region object of this study, with high level of solar radiation ventilation allows to the cooling load during summer period and contributes to the reduction of the energy needs of buildings. The most important advantages is the reduction of the heat fluxes transmitted by the structures exposed to the solar radiation, thanks to the combined effect of the surfaces shading and of the heat removed by the air flow rate within the ventilated air gap.

This paper illustrates a numerical investigation on a prototypal ventilated roof for residential use. Due to geometric and thermal symmetry the system has been studied considering a single side of the pitched roof. The roof is long 6.0 m, inclined from the horizon of 30° and the ventilated channel, under the roof, has a height of 10 cm.

The analysis is carried out on a two-dimensional model in air flow and the governing equations are given in terms of k-ε turbulence model. The investigation is performed in order to evaluate thermofluidodynamic behaviors of the ventilated roof as a function of the following geometric parameters: 1) channel gap or height equal to 5, 10 and 20 cm; 2) Inclination of 15°, 30° and 45°; 3) Hight of the ridge of 5, 10 and 15 cm.

The problem is solved by means of the commercial code Ansys-Fluent and the results are performed for an uniform wall heat flux on the top wall equal to 800 W/m^2. Moreover, some different solutions of the exit section have been investigated. Results are given in terms of wall temperature distributions, air velocity, temperature profiles in different cross section of the roof in order to estimate the differences between the various configurations. Further, the ventilated roof has been compared with other types of roof.

A Comment on Modelling and Analysis of Plasma Gasification as an Emerging Technology for Waste to Energy

M.O. Carpinlioglu[1], A. Sanlisoy[2]

[1] University of Gaziantep , Mechanical Engineering Department Gaziantep 27310 , Gaziantep, Turkey , melda@gantep.edu.tr
[2] University of Gaziantep, Mechanical Engineering Department Gaziantep 27310 , Gaziantep, Turkey , aytacsanlisoy@gantep.edu.tr

Extended Abstract

A critical review of relevant literature on the application of plasma gasification for solid waste treatment is presented in this paper. A criticism on the state of art is provided in terms of the historical background with a brief on the current global solid waste portrait . The particular emphasis is devoted to the thermodynamic process modelling, current terminology, basic equations and the preferred methodology in the analysis . The theoretical background on the manner is also outlined in comparison with the conventional processes of combustion and pyrolysis . A comparative correlation study on the modelling and analysis of plasma gasification is presented in reference to the experimental studies conducted previously as separate and independent research . The proposed methodology on modelling and analysis of plasma gasification is given through a sample test case using available experimental data . An experimental research study concerning the design, construction and the operation of a microwave plasma gasifier " MCwGasifier " which is operated to test the proposed methodology in laboratory scale under controlled conditions is also described .

Table 1: Defined parameters for the performance assessment of the process (a part of Table 1)

Research group	Defined parameter	Equation
[17]	Carbon conversion efficiency	$X_c\% = \frac{\dot{m}_{C\ in\ sysngas}}{\dot{m}_{C\ in\ feedstock}} \times 100$
	Cold gas efficiency	$\eta_{CG}[\%] = \frac{\dot{m}_{syn} \times HHV_{syn}}{\dot{m}_{feed} \times HHV_{feed}} \times 100$
[21]	Conversion for H_2	$H_2\text{Conversion} = \frac{2P_{H_2}}{4F_{PE}}$
	Conversion for C	$\text{C Conversion} = \frac{P_C}{2F_{PE}}$
	Conversion for H_2O based on O element	$\text{Conversion} = \frac{P_{CO} + 2P_{CO_2}}{2F_{H_2O}}$

References

ROTH, J. R.,1994 *Industrial Plasma Engineering*, Institute of Physics
RUJ, B., GHOSH , S., 2014 *Technological aspects for thermal plasma treatment of municipal solid waste-A review,* Fuel Processing Technology, 126, 298-308.

Preliminary CFD Analysis of Natural Convection Fuel Tubes in Molten Salt Nuclear Reactors

A. Cioncolini[1], A.B.M. Pauzi[2], H. Iacovides[3], D. Cooper[4] and I. Scott[5]

[1,3,4] *School of Mechanical, Aerospace and Civil Engineering, University of Manchester, M1 3BB Manchester, UK*
[1] *andrea.cioncolini@manchester.ac.uk*; [3] *h.iacovides@manchester.ac.uk*; [4] *dennis.cooper@manchester.ac.uk*;
[2] *Department of Mechanical Engineering, Universiti Tenaga Nasional, 43000 Kajang, Selangor, Malaysia, anas@uniten.edu.my*
[5] *Moltex Energy LLP, 6th Floor Remo House, 310-312 Regent Street, W1B London, UK, ianscott@moltexenergy.com*

Extended Abstract

Molten salt nuclear reactors have experienced a growing interest in the past decade, boosted by their inclusion as one of the Generation IV nuclear reactor types. Among the several molten salt nuclear reactor designs proposed to date, the *Stable Salt Reactor* proposed by Moltex Energy LLP (www.moltexenergy.com) is particularly attractive. In this design, the reactor core comprises an array of vertical fuel tubes immersed in a pool of molten blanket salt (eutectic mixture, with operating temperature in the range of 700-900 K). The fuel tubes are closed at the bottom, open at the top and are filled with molten fuel salt (sodium cloride plus uranium and/or plutonium salts, with operating temperature in the range of 800-1700 K). The fuel tubes rely entirely on natural convection for internal mixing, and this provides the advantages of a two-fluid molten salt system eliminating the technical hurdle of a pumped design (so called ‘plumbing problem’). In this configuration, the safe power level that can be achieved without overheating the fuel salt is a crucial design parameter.

Preliminary CFD simulations performed on a early reactor design with relatively large fuel tubes (41/45mm inner/outer diameter) show that a natural circulation fuel tube can be safely operated at a power level of 100-150 kW, and with the use of heat transfer enhancement techniques power levels as high as 250-300 kW appear feasible. As such, a cylindrical reactor core (2.5m diameter and 2m active height) comprising 2000 fuel tubes can be safely operated at 200-300 MW_{th}, and power levels as high as 500-600 MW_{th} appear feasible. Moreover, the CFD simulations predict a complex recirculating flow pattern in the molten fuel salt inside the fuel tubes, with three-dimensional and unsteady effects that require careful investigation for a sound design and safe operation of this reactor concept.

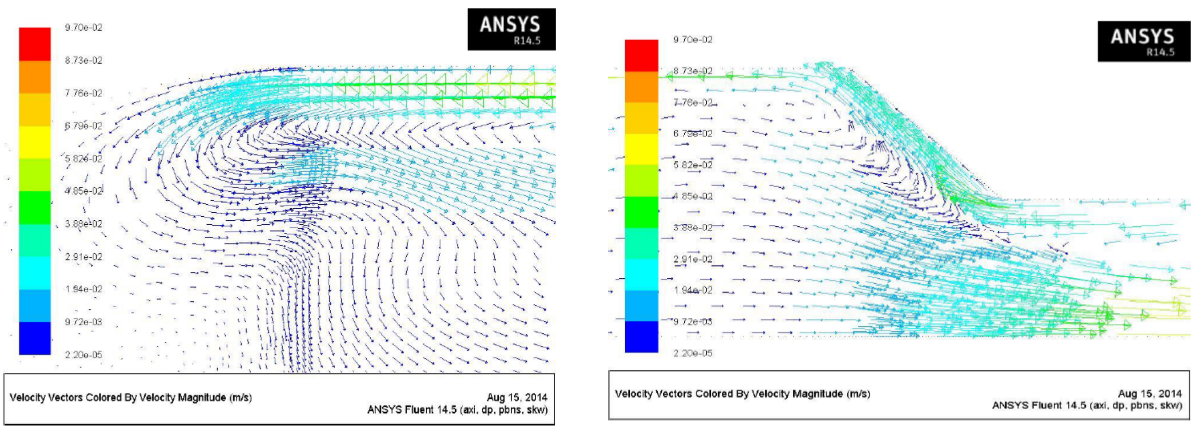


Fig 1: Recirculating flow pattern inside the molten fuel salt at the bottom (left) and top (right) of the fuel tubes.

A three-dimensional numerical model of free convection in a tilted porous cavity

F. J. Guerrero[1,2], P.L. Younger[1,3] and N. Karimi[1,4]

[1] *School of Engineering, University of Glasgow, James Watt Building South, G12 8QQ, Scotland*
[2] *f.guerrero-martinez.1@research.gla.ac.uk*
[3] *Paul.Younger@glasgow.ac.uk* [4] *Nader.Karimi@glasgow.ac.uk*

Extended Abstract

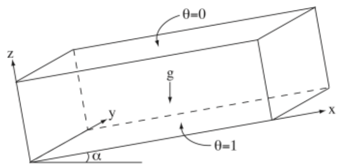


Fig 1: Schematic model

A three-dimensional (3D) numerical model of free convection in a porous medium is presented to study the problem of a rectangular cavity with adiabatic lateral walls, heated from the base (Fig. 1). This is in analogy with the Horton-Rogers-Lapwood problem (Nield and Bejan 2013). Several studies have been carried out in the past to study this problem in 2D (see for example Baytas 2000). The model was formulated assuming local thermal equilibrium; fluid flow is described by Darcy's law and the Boussinesq approximation. Viscous heat generation is considered to be negligible. The dimensionless problem can be described by the following set of equations,

$$\mathbf{u} + \nabla p = Ra_p \theta \mathbf{e} \tag{1}$$

$$\frac{\partial \theta}{\partial t} - \nabla^2 \theta + \mathbf{u} \cdot \nabla \theta = 0 \tag{2}$$

$$\nabla \cdot \mathbf{u} = 0 \tag{3}$$

where Ra_p is the Rayleigh-Darcy number and $\mathbf{e}$=(*sin* α, 0, *cos* α). The differential equations were solved in primitive variables using a Finite Volume numerical method. The steady-state solution of the problem was determined from long simulation times for different Rayleigh numbers up to a maximum of 100, a fixed angle α=10º, and an aspect ratio of 3. Our model was validated against simulation results available in the literature for the 2D version of the problem. Our 3D transient results showed that, regardless of the value of Ra_p, the system initially develops transversal convective rolls in agreement with the two-dimensional version of the problem; furthermore, this 2D distribution of the velocity field prevails for long simulation times when the Rayleigh number is low. For high Rayleigh numbers however, there is a transition from 2D covective rolls to 3D distribution of the velocity field in which a longitudinal coil coexists with two transversal rolls (Fig. 2, Ra_p=80). When this is the case, however, the global Nusselt number in the system does not change significantly during the transition.

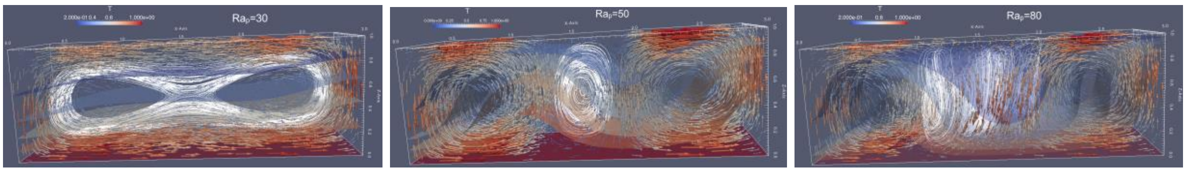


Fig 2: Steady-state solution, isothermal surfaces, velocity field and streak lines for Ra_p=30, 50 and 80.

References

NIELD, D.A., BEJAN, A. 2013 Convection in Porous Media. 4th ed., Springer, New York.

BAYTAS, A.C. 2000 *Entropy generation for natural convection in an inclined porous cavity*. International Journal of Heat and Mass Transfer **43**, 2089– 2099.

Simulating Heat and Mass Transfer in an Aggregate Dryer Using Coupled CFD and DEM

Andrew Hobbs

Edinburgh University, Edinburgh EH9 3JN, United Kingdom, ahobbs@astecinc.com

Extended Abstract

This project describes the use of coupled CFD and DEM methods to simulate drying in an aggregate drum dryer used in the production of hot mix asphalt. To be properly coated by the asphalt binder, aggregate must be completely dried, a process accomplished in a counter flow drum heated by a direct fire burner. Attached flighting is positioned inside the drum to facilitate heat transfer. Because direct observation is impossible simulation provides the best opportunity to optimize flight design for increased drying efficiency. Commercial codes from ANSYS FLUENT and DEM-Solutions were coupled using an open source coupling developed in cooperation by Astec and ANSYS. The coupled CFD DEM simulation used an Eulerian model with heat transfer, mass, and momentum exchanged between the fluid and particle phases. A custom particle property to track the moisture content of the particle was created, and the mass transfer of moisture from the particle to the surrounding fluid was calculated. Results show the coupled model captures the drying process with drying occurring in the region observed in the drum (Figure 1). Further work to include moisture dependent cohesion and particle-particle liquid transfer using a liquid bridge model is on-going.

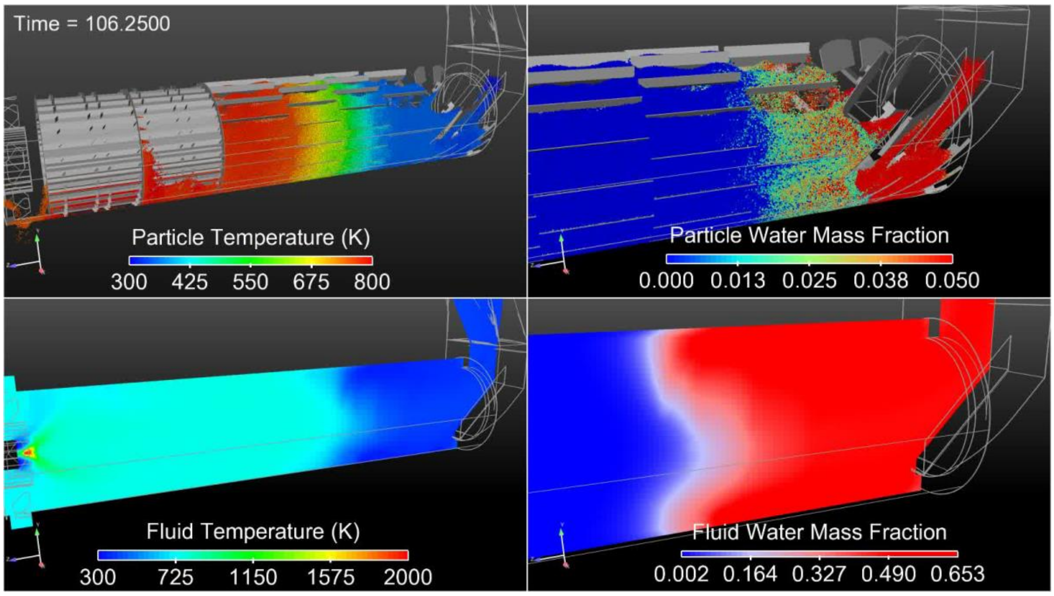


Figure 1. Coupled CFD and DEM results showing particle temperature, particle water mass fraction, fluid temperature, and fluid water vapor mass fraction

References

LI,J. & MASON,D 2002. *Applications of the Discrete Element Modelling in Air Drying of Particulate Solids*. Drying Technology. Vol.20, no. 3, pp 255-282.

SHI,D. 2008 *Advanced Simulation of Particle Processing: The roles of cohesion, mass, and heat, transfer in gas-solid flow*. PhD Dissertation, Universtity of Pittsburgh

CFD Modelling of Thermal Management in Downhole Tools

T. P. Hughes[1], R. Weerasinghe[1]

[1] *Faculty of Environment and Technology, University of the West of England, Coldharbour Lane, Bristol, BS16 1QY*
tom.hughes@uwe.ac.uk

Extended Abstract

Investigation of oil and gas fields using vertical seismic profiling subjects the electronics used to extreme temperatures and pressures. As wells become deeper, the need for effective thermal management in instrumentation and the demands placed on the tools increases. Various techniques including vacuum insulation, heat pipes, phase change media and thermo-electric cooling (TEC) devices are used to maximise the time equipment can be exposed to these hostile conditions. The performance of such devices is temperature dependent and the level of cooling attained is a function of thermal efficiency. This work examines the effect of different TEC devices on the thermal performance of a commercially available Vertical Seismic Profiling tool.

A hot oil bath was used to obtain experimental data for the commercially available tool, supplied by Avalon Sciences Ltd. In the experimental tests, the tool was allowed to reach a steady state after several hours in the oil bath. The temperatures at specific regions and power to the thermal electric cooler were recorded. The vacuum insulation flask and thermo-electric cooler were characterised separately in a laboratory oven.

A computational fluid dynamics model of the system is developed in the commercially available code, Star CCM+. In order to model the heat dissipation from the tool, the oil bath is simulated as a fluid surrounding the tool with fixed temperature boundary conditions and the heat flow at the hot and cold faces of the TEC device modelled according to the Seebeck voltage and thermal conductance at the mean temperature, calculated from linear equations derived from the manufacturer data sheet. The modules used are single stage modules and thus the cold side heat flow, Qc, can be found:

$$Q_c = 2N\left[\alpha I T_c - \left(\frac{I^2}{2G}\right) - \kappa(T_h - T_c)G\right]$$

The hot side heat flux is the sum of the input power (calculated from a specified voltage and the electrical resistance of the unit), and the cold side heat flow. This model is defined as a series of field functions within the software, and the heat flows updated at each iteration.

The results are found to give a temperature field (see figure 1) and heat fluxes comparable with those observed in experimental measurements and provide a platform for developing a tool capable of functioning in 200°C environments. The model is used as a platform to predict the performance of a proposed high temperature TEC device.

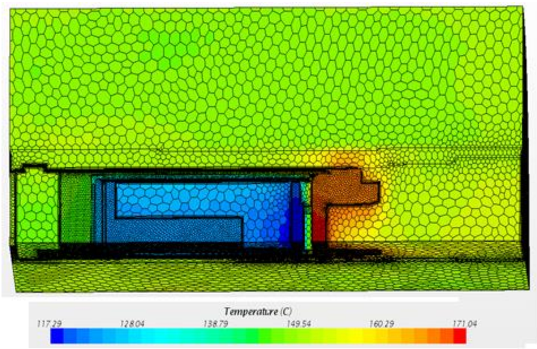


Figure 1: Steady state temperature profile of the system.

Solving Direct and Inverse Nonlinear Heat Conduction Problems by Means of Trefftz Functions and Kirchhoff Transformation

A. Maciag

Kielce University of Technology, Faculty of Management and Computer Modelling,
Department of Applied Computer Science and Applied Mathematics,
al. Tysiaclecia Panstwa Polskiego 7, 25-315 Kielce
maciag@tu.kielce.pl

Extended Abstract

The nolinear problems of heat conduction with teperature dependent thermal conductivity, described by equation (1) with necessary boundary conditions, were considered in the paper.

$$\frac{\partial}{\partial x}\left(\lambda(T)\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(\lambda(T)\frac{\partial T}{\partial y}\right)=0, \qquad (x,y)\in(0,1)\times(0,1) \tag{1}$$

Equation (1) was transformed to the Laplace's equation using Kirchhoff transormation [1]

$$u=\frac{1}{\lambda_0}\int_0^T\lambda(T)dT \tag{2}$$

Laplace equation with transformed boundary conditions was solved by means of Trefftz functions (harmonic polynomials). To get temperature T the inverse transformation for (2) has to be applied. Trefftz functions are suitable for solving boundary inverse problems when one of the boundary conditions (for example for x=1) is not known. Usually, instead of that, the measured temperatures $T_i(1-\varepsilon,y_i)$ in discrete points are known (internal responses). The solution of the boundary inverse problem can be sensitive to errors of measurement. Therefore, to check effectiveness of the method, randomly disturbed internal responses (normal distribution N(0,0.01)) were taken into account. Two cases of the thermal conductivity dependent on temperature were taken into accout

$$a)\ \ \lambda(T)=\lambda_0e^{\lambda_1T}, \qquad b)\ \ (T)=\lambda_0+\lambda_1T\ . \tag{3}$$

For example $\lambda_0=100$ and $\lambda_1=-1/2000$ were taken in the case (3a) with

$$T_{exact}=x+2000\ln(2)-1000\ln\left(\left(2e^{x/1000}-1\right)^2\right)-1000\ln\left(\left(500sin(y/2000)+cos(y/2000)\right)^2\right).$$

The mean relative error of approximation of the boundary condition for $x=1$, defined as $E=100\cdot\int_0^1\frac{\left(T_{exact}(1,y)-T_{approx}(1,y)\right)^2}{\left(T_{exact}(1,y)\right)^2}dy$ [%] is presented in the Table 1.

Table 1: Error E [%]

ε	0	0.1	0.5
Exact data	$5.5\cdot10^{-20}$	$1.5\cdot10^{-19}$	$9.2\cdot10^{-19}$
Noisy data	0.93	1.15	1.97

Obtained results show that the Trefftz method gives accurate solution also for noisy data.

References

CARSLAW, H.S., & JAEGER, J.C. 1959 *Conduction of heat in solids*. Oxford University Press.

Modelling the Free-Surface Turbulent Flow and Heat Transfer in an Unbaffled Vessel Agitated by a Pitched Three-Blade Turbine

T. Mahmud, E. Bentham and P. Heggs

School of Chemical and Process Engineering, University of Leeds, Leeds, England, t.mahmud@leeds.ac.uk

Extended Abstract

This paper investigates a CFD simulation of free-surface flow and heat transfer in a 25 litre unbaffled vessel filled with 20 litres of water at 30°C, agitated by a three-bladed pitched turbine rotating at a typical speed of 180 rpm (Re = 7.67×10^4) and heated through a 6 mm glass wall by an external plain jacket with DW-Therm assumed to be at a constant temperature of 80°C. The simulation was carried out on the software Ansys CFX, using a homogeneous two-phase flow model coupled with the standard k-ε model for turbulence and a volume-of-fluid method for capturing the free surface. The average value of heat transfer coefficient in the simulation is 3199 W m^{-1} K^{-1}, compared to around 2127 W m^{-1} K^{-1} using a correlation by Nagata *et al*. (1972).

The model of a forced vortex in the centre and a free vortex on the outside, joined at a 'critical radius', is widely used in predicting the free-surface profile, for example in Nataga (1975), but the focus has been on Rushton turbines and flat-blade impellers, rather than pitched blade turbines.

Correlations for predicting the vortex depth (h_V) for variuos impeller types have been reported by Rieger et al. (1979). Equation (1) is a correlation derived from an experimental investigation to provide the vortex depth for a pitched three-blade turbine, for a Galileo number (Ga) between 10^8 and 10^{10}, using an empirically derived constant (B_1) of 0.71 ± 0.03 under these conditions.

$$\frac{h_V}{d} = B_1 \mathrm{Ga}^{0.069} \left(\frac{D}{d}\right)^{-0.38} \mathrm{Fr}^{1.14\mathrm{Ga}^{-0.008}(D/d)^{-0.008}} \tag{1}$$

where D and d are the tank and impeller diameter, respectively, and Fr is the Froude number.

Despite the use in of the compressive algorithm for capturing the free-surface in CFX, the definition of the gas-liquid interface is highly dependent on the volume fractions of the two phases in the simulation results obtained for this vessel. Traditionally, liquid volume fractions between 0.5 or 0.9 have been used in previous studies; however, in this case, a somewhat higher value of 0.98 appears to correlate bettter with the analysis in Rieger et al. (1979).

References

NAGATA, S., NISHIKAWA, M., TAKIMOTO, T., KIDA, F., KAYAMA, T., 1972 *Turbulent heat transfer from the wall of a jacketed tank*. Heat Transfer-Jpn Res **1** (1), 66-74

NAGATA, S. 1975 *Mixing: Principles and Applications*. Wiley, New York.

RIEGER, F., DITLE, P., & NOVAK, V. 1979 *Vortex Depth in Mixed Unbaffled Vessels*. Chem. Eng. Sci. **34**, 397-403.

Modeling the effect of thermotherapy on the inner layer of the bladder

Christoph Sadée and Eugene Kashdan,
University College Dublin, Ireland

Abstract

Urothelial carcinoma (UC) is a common disease; it is considered the 4^{th} most common new cancer in men and 8^{th} in women. In the western world around 75,000 new cases of UC are discovered annually, the rate of new cases of UC is constantly increasing with a rate of 0.8% per year. UC is a sporadic disease and rarely inherited genetically, around 70% of the cases can be related to exposure to carcinogens, such as aromatic amines. In the past occupational exposure was common, however today most of the cases are related to smoking.

Non-muscle invasive urothelial cancer (NMIUC) may be considered a chronic disease, due to its natural history and tendency to recur in most of the patients. Some of these patients are also at risk of disease progression to invasive (and potentially life-threatening) disease. Following endoscopic resection of the primary tumor, adjuvant intravesical agents are given to those who are at a higher risk of recurrence. Unfortunately, such treatments are either not effective enough and/or associated with significant side effects, indicating the need for alternative measures.

Radiofrequency induced hyperthermia of the bladder wall in conjunction with intravesical chemotherapeutic agents, so called chemo-hyperthermia. It is a novel treatment used for patients in Europe and Israel with NMIUC showing encouraging results. In 2013, the FDA in the US also approved the treatment.

In this work, we model and analyze one of the novel NMIUC treatments –microwave induced chemo-hyperthermia (Synergo®). With this system, local hyperthermia is administered via a direct irradiation of the bladder by means of a 915 MHz intravesical microwave applicator.

Our model is based on the combination of Maxwell's equation and the Heat equation. It simulates the diffusion of heat on an annulus (representing the bladder wall). In our simulations, we also show the effect of an indent at the inner surface of the bladder (i.e. due to surgery) by placing very high diffusion coefficients in the indentation region. Taking into consideration parameters that correspond to the actual characteristics of bladder we observe the effect of the non-uniform heat distribution within the tissue layers of bladder, which might lead to the "bladder burning" reported in a number of clinical cases.

Decoupling of thermo-physical properties of glycol-water mixtures: insight from nano-scale simulation

J. J. Cannon[1]; T. Kawaguchi[2]; T. Kaneko[3]; T. Fuse[4]; J. Shiomi[5]

[1] *The University of Tokyo, Deptartment of Mechanical Engineering, The University of Tokyo, 7-3-1 Hongo, Bunkyo-ku, Tokyo 113-8656, Japan, cannon@photon.t.u-tokyo.ac.jp*
[2] *DENSO CORPORATION, 500-1, Minamiyama, Komenoki-cho, Nisshin-shi, Aichi, 470-0111, Japan, TOHRU_KAWAGUCHI@denso.co.jp*
[3] *DENSO CORPORATION, 500-1, Minamiyama, Komenoki-cho, Nisshin-shi, Aichi, 470-0111, Japan, TAKASHI_KANEKO@denso.co.jp*
[4] *DENSO CORPORATION, 500-1, Minamiyama, Komenoki-cho, Nisshin-shi, Aichi, 470-0111, Japan, TAKUYA_FUSE@denso.co.jp*
[5] *The University of Tokyo, Deptartment of Mechanical Engineering, The University of Tokyo, 7-3-1 Hongo, Bunkyo-ku, Tokyo 113-8656, Japan, shiomi@photon.t.u-tokyo.ac.jp*

Extended Abstract

Water and the various alcohols each have their own distinctive thermo-physical properties such as thermal conductivity, heat capacity, viscosity, density and so on. Often applications require such thermo-physical properties to be tuned to a value appropriate for the specific application, which is achieved by mixing liquids in various ratios.

It can be noted however that the variation of thermophysical properties is often coupled. For example, addition of a glycol alcohol such as ethylene glycol or propylene glycol to water will result in a decrease in thermal conductivity and a simultaneous increase in viscosity. It would be very useful however to understand the origins of this coupling in order to gain an understanding of how the properties may be varied independently, however understanding of the nature of these fundamental properties is still insufficient.

Although the thermo-physical properties are observed from a macro-scale perspective applicable to industrial processes, this coupling actually originates on the nano-scale, through the structure of the molecules and the interactions between them. Empirical theories typically rely upon experimental parameterisation of variables and do not give firm insight into the molecular origins, while purely experimental observation of the molecular interactions is highly challenging due to the small time and length scales involved. Molecular simulation in contrast offers nano-scale insight into molecular interactions providing information regarding the thermo-physical properties and their coupling. In addition, one is not bound by the "real world", so it is possible to accentuate certain molecular features in order to elucidate the effects of certain characteristics and interactions.

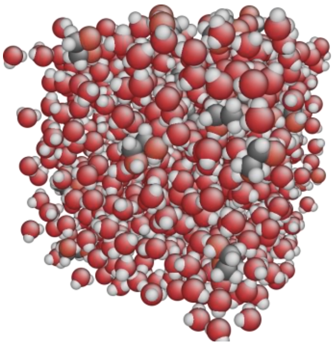


Fig. 1 Snapshot of a simulation containing water and glycol.

It is with these motivations that we have conducted molecular dynamics simulations to elucidate the molecular origins of coupling and how they relate to the macro-scale. In particular, we have considered the relation between thermal conductivity and viscosity, and examined how these two properties are coupled. Our results show that thermal conductivity and viscosity are related via molecular structure and interaction in some ways and not in others, and hence how careful choice of molecule allows decoupling to be induced.

Notes

Notes

Notes

Blank Page

Summary Schedule of the 14th UK Heat Transfer Conference

SUNDAY Evening, 6 September 2015	
18:00	On-site registration and ***Drinks Reception***
MONDAY, 7 September 2015	
7:30	***Poster presenters set up their posters***
8:00	On-site registration
8:30	Welcome
Phase Change Session	
9:00	***Keynote 1: Prof. Tassos Karayiannis (Brunel University London, UK)*** *Chair: Dr. D.A. McNeil, Heriot Watt University*
9:30	Poster presentations PC1-PC22
10:30	Coffee break & poster viewing
11:00	***Keynote 2: Prof. Ping Cheng (Shanghai Jiaotong University, China)*** *Chair: Dr. T. S. Ó Donovan, Heriot Watt University*
11:30	Poster presentations PC23-PC46
12:30	Lunch break & poster viewing (***Poster presenters note switchover at 13:15***)
Applications Session	
13:45	***Keynote 3: Prof. Yasuyuki Takata (Kyushu University, Japan)*** *Chair: Dr. J. R. E. Christy, The University of Edinburgh*
14:16	Poster presentations AP1-AP20
15:15	Coffee break & poster viewing
15:45	***Keynote 4: Prof. Joe Quarini (University of Bristol, UK)*** *Chair: Prof. K. Sefiane, The University of Edinburgh*
16:15	Poster presentations AP21-AP43
17:15	Poster viewing & Day Announcements
18:00	Poster removal
19:30	***Dinner***
TUESDAY, 8 September 2015	
8:00	***Poster presenters set up their posters***
Modelling Session	
9:00	***Keynote 5: Dr. Prashant Valluri (The University of Edinburgh, UK)*** *Chair: Dr. Y. C. Lee, Heriot Watt University*
9:30	Poster presentations M1-M20
10:30	Coffee break & poster viewing
11:00	***Keynote 6: Dr. Francesco Colletti (Hexxcell Ltd., UK)*** *Chair: Dr. P. Valluri, The University of Edinburgh*
11:30	Poster presentations M21-M34
12:12	***Conference Closing Remarks***
12:30	Lunch break (***Poster removal at 13:15***)

Thank You!

$Q = UKA\,\Delta T$

THE UNIVERSITY OF EDINBURGH

HERIOT WATT UNIVERSITY