25th Scottish Fluid Mechanics Meeting

Wednesday 30th May 2012

Book of Abstracts

Hosted by School of Engineering and Physical Sciences
& School of the Built Environment

Sponsored by Dantec Dynamics Ltd.

Programme of Event:

09:00 – 09:30	Coffee and registration*
09:30 – 09:45	Welcome (Wolf-Gerrit Früh)**
09:45 – 11:00	Session 1 (5 papers) (Chair: Wolf-Gerrit Früh)**
11:00 – 11:30	Coffee break & Posters***
11:30 – 12:45	Session 2 (5 papers) (Chair: Stephen Wilson)
12:45 – 13:45	Lunch & Posters
13:45 – 15:00	Session 3 (5 papers) (Chair: Yakun Guo)
15:00 – 15:30	Coffee break & Posters
15:30 – 17:00	Session 4 (6 papers) (Chair: Peter Davies)
17:00 – 17:10	Closing remarks (Wolf-Gerrit Früh)
17:10 – 18:00	Drinks reception

* Coffee and registration will be outside Lecture Theatre 3, Hugh Nisbet Building.

** Welcome and all sessions will be held in Lecture Theatre 3, Hugh Nisbet Building.

*** All other refreshments (morning and afternoon coffee breaks, lunch and the drinks reception) and posters will be held in NS1.37 (the Nasmyth Common room and Crush Area), James Nasmyth Building.

Programme of Oral Presentations

(* denotes presenting author)

Session 1 (09:40 – 11:00):

09:45 – 10:00	***Flow control for VATT by fixed and oscillating flap*** Qing Xiao, Wendi Liu* & Atilla Incecik, University of Strathclyde
10:00 – 10:15	***Shear stress on the surface of a mono-filament woven fabric*** Yuan Li* & Jonathan Kobine, University of Dundee
10:15 – 10:30	***Drops, ridges and rivulets in an external airflow*** Colin Paterson*, Stephen Wilson & Brian Duffy, University of Strathclyde
10:30 – 10:45	***Sediment dynamics in the wake of a tidal current turbine*** Lada Vybulkova*, Marco Vezza, Harshinie Karunarathna & Richard Brown, University of Glasgow
10:45 – 11:00	***Flow analysis for twin-flapping wind energy harvester*** Saishuai Dai* & Qing Xiao, University of Strathclyde

Session 2 (11:30 – 12:45):

11:30 – 11:45	***Dynamic wetting on moving surfaces: A molecular dynamics study*** Konstantinos Ritos*, Nishanth Dongari, Yonghao Zhang & Jason Reese, University of Strathclyde
11:45 – 12:00	***On energetics and inertial range scaling laws of two-dimensional magnetohydrodynamic turbulence*** Luke Blackbourn* & Chuong Tran, University of St Andrews
12:00 – 12:15	***Microdroplet simulations using a multiphase lattice Boltzmann method*** Jonathan Li*, Yonghao Zhang & Jason Reese, University of Strathclyde
12:15 – 12:30	***Revealing small-scale structures in turbulent Rayleigh-Bénard convection*** Yue-Kin Tsang*, Emily Ching, T. N. Fok, XiaoZhou He & Penger Tong, University of St Andrews
12:30 – 12:45	***Numerical investigation of thermal protection system discontinuities*** Rodrigo Palharini*, Thomas Scanlon & Jason Reese University of Strathclyde

Session 3 (13:45 – 15:00):

13:45 – 14:00 ***Extensional flow of viscoelastic fluids in an optimised shape cross-slot extensional rheometer (OSCER)***
Mónica Oliveira*, Simon Haward, Gareth McKinley & Manuel Alves,
University of Strathclyde

14:00 – 14:15 ***Modelling micro-diffusion and autocatalysis in biodegradable polymers***
Yuhang Chen*, Juergen Siepmann, Florence Siepmann & Qing Li,
Heriot-Watt University

14:15 – 14:30 ***The effect of reactive thermosolutal convection on the long-term evolution of a porous medium***
Lindsey Corson* & David Pritchard, University of Strathclyde

14:30 – 14:45 ***Shallow-water vortex equilibria and their stability***
Hanna Płotka* & David Dritschel, University of St Andrews

14:45 – 15:00 ***Squeeze-film flow in the presence of a thin porous bed, with application to the human knee joint***
D. J. Knox, Stephen Wilson*, Brian Duffy & S. McKee,
University of Strathclyde

Session 4 (15:30 – 17:00):

15:30 – 15:45 ***Run-up of long waves on a plane beach***
Yong Sung Park*, University of Dundee

15:45 – 16:00 ***The Stokes boundary layer for a thixotropic or antithixotropic fluid***
Catriona McArdle*, David Pritchard & Stephen Wilson
University of Strathclyde

16:00 – 16:15 ***Coupled modelling of turbidity currents over erodible beds***
Peng Hu*, Zhixian Cao & Gareth Pender, Heriot Watt University

16:15 – 16:30 ***Boundary conditions for molecular dynamics simulations of water transport through carbon nanotubes***
Stephanie Docherty*, William Nicholls, Matthew Borg & Jason Reese
University of Strathclyde

16:30 – 16:45 ***Velocity, turbulence and bed shear stress in oscillatory boundary layer flow over an impermeable and a permeable bed: a comparative study***
Kathryn Sparrow*, Dubravka Pokrajac & Dominic van der A
University of Aberdeen

16:45 – 17:00 ***Ocean dynamics modelling with a hybrid Lagrangian-Eulerian method***
Xavier Perrot & David Dritschel, University of St Andrews

Poster Presentations

Experimental and Numerical Thermal Analysis of a Small Scale Centeo Tg40 Cooling System
Ahmed A. Y. Al-Waaly, Phillip Dobson, Manosh C. Paul & Philipp Steinmann
University of Glasgow

Modelling Lillgrund Wind Farm
Angus C.W. Creech, Eoghan Maguire & Wolf-Gerrit Früh
Heriot-Watt University

The role of Ekman suction in 'rotational augmentation'
Wolf-Gerrit Früh
Heriot-Watt University

Influence of surface roughness and interface wettability on flows through nanochannels
Malgorzata Zimon, David Emerson & Jason Reese
University of Strathclyde

List of Participants

Name	Affiliation	Contact Details
Abdul Ahmad	University of Strathclyde	abdul.o.ahmad@strath.ac.uk
Ahmed Al-Waaly	University of Glasgow	1007238a@research.gla.ac.uk
Luke Blackbourn	University of St Andrews	lb242@st-andrews.ac.uk
Yuhang Chen	Heriot-Watt University	y.chen@hw.ac.uk
Lindsey Corson	University of Strathclyde	lindsey.corson@strath.ac.uk
Alan Cuthbertson	Heriot-Watt University	a.cuthbertson@hw.ac.uk
Saishuai Dai	University of Strathclyde	saishuai.dai@strath.ac.uk
Peter Davies	University of Dundee	p.a.davies@dundee.ac.uk
Stephanie Docherty	University of Strathclyde	stephanie.docherty@strath.ac.uk
Nishanth Dongari	University of Strathclyde	nishanth.dongari@strath.ac.uk
Brian Duffy	University of Strathclyde	b.r.duffy@strath.ac.uk
Gail Duursma	University of Edinburgh	Gail.Duursma@ed.ac.uk
Wolf-Gerrit Früh	Heriot-Watt University	w.g.fruh@hw.ac.uk
Yakun Guo	University of Aberdeen	y.guo@abdn.ac.uk
Peng Hu	Heriot-Watt University	ph66@hw.ac.uk
Jonathan Kobine	University of Dundee	j.kobine@dundee.ac.uk
Yeaw Chu Lee	Heriot-Watt University	y.c.lee@hw.ac.uk
Jonathan Li	University of Strathclyde	jonathan.li@strath.ac.uk
Yuan Li	University of Dundee	y.n.li@dundee.ac.uk
Wendi Liu	University of Strathclyde	wendi.liu@strath.ac.uk
Luke Mason	Heriot-Watt University	lm242@hw.ac.uk
Catriona McArdle	University of Strathclyde	catriona.mcardle@strath.ac.uk
Mónica S. N. Oliveira	University of Strathclyde	monica.oliveira@strath.ac.uk
Rodrigo Palharini	University of Strathclyde	rodrigo.palharini@strath.ac.uk
Yong Sung Park	University of Dundee	y.s.park@dundee.ac.uk

Name	Affiliation	Contact Details
Colin Paterson	University of Strathclyde	colin.paterson@strath.ac.uk
Xavier Perrot	University of St Andrews	xperrot@mcs.st-and.ac.uk
Hanna Plotka	University of St Andrews	hanna@mcs.st-and.ac.uk
David Pritchard	University of Strathclyde	david.pritchard@strath.ac.uk
Konstantinos Ritos	University of Strathclyde	konstantinos.ritos@strath.ac.uk
David Smith	University of Strathclyde/Weir Group	david.a.smith@strath.ac.uk
Jutta Stauber	University of Strathclyde	jutta.stauber@strath.ac.uk
Chuong V. Tran	University of St Andrews	chuong@mcs.st-and.ac.uk
Lada Vybulkova	University of Glasgow	l.vybulkova.1@research.gla.ac.uk
Xue Wang	University of Strathclyde	xue.wang@strath.ac.uk
Craig White	University of Strathclyde	craig.white@strath.ac.uk
Stephen K. Wilson	University of Strathclyde	s.k.wilson@strath.ac.uk
Lei Wu	University of Strathclyde	l.wu@strath.ac.uk
Qing Xiao	University of Strathclyde	qing.xiao@strath.ac.uk
Yonghao Zhang	University of Strathclyde	yonghao.zhang@strath.ac.uk
Malgorzata Zimon	University of Strathclyde	malgorzata.zimon@strath.ac.uk

An updated list of participants will be available on the SFMM2012 website:
http://home.eps.hw.ac.uk/~ycl12/sfmm2012/

SFMM Sponsorship

As always, we are deeply indebted to Dantec Dynamics Ltd., and Graham Hassall in particular, for their continued support of SFMM. Further information on Dantec can be found at their company website: www.dantecdynamics.com.

25 Years of SFMM!

This year, we are celebrating the 25th anniversary of the Scottish Fluid Mechanics Meeting with a drinks reception at the end of today's events. SFMM was the joint idea of Professor Peter Davies (University of Dundee) and Professor Clive Greated (University of Edinburgh) with the aim of bringing together researchers working in all aspects of fluid mechanics in Scotland. The first meeting was held at the University of Dundee in March 1988 (photo 1) and has since visited the Universities of Edinburgh(4), Strathclyde(4), Dundee(3), Aberdeen(3), Heriot-Watt(3), St Andrews(2), Glasgow(2), Napier(1) and Paisley(1), as well as the Scottish Association for Marine Science (SAMS) in Oban in 2009. Last year's event returned to the University of Strathclyde (photo 2) and offers to host SFMM2013 are welcome!

Top: Delegates at the first SFMM Event held in Dundee in 1988. **Bottom:** Last year's event at the University of Strathclyde.

Abstract for 25th Scottish Fluid Mechanics Meeting at Heriot-Watt University 2012

Flow control for VATT by fixed and oscillating flap

Qing Xiao, Wendi Liu* and Atilla Incecik
University of Strathclyde, Glasgow, G4 0LZ, UK

In the past decades, renewable energy industry has been rapidly developed due to the conventional energy resources shortage. Among various ocean renewable energies, tidal energy has its distinct superiorities over other types of ocean energy. It has a much less sensitivity to the climate change and is predictable in both space and time domains. Tidal turbine is becoming one of the most efficient devices which transform tidal energy into mechanical power [1].

Present study is focused on an H-shaped lift force driven Vertical Axis Tidal Turbine (VATT). In comparison with the horizontal axis turbine and drag force driven turbine, H-shaped turbine has many advantages, such as the simple blade design, capable to work in all current flow directions and non-sensitive to water depth. However, the efficiency of existing H-shaped VATT is still relative low as compared to the horizontal axis turbine. It is thus necessary and valuable to carry on a deep research in order to enhance VATT efficiency via various mechanical and control methods [2] [3].

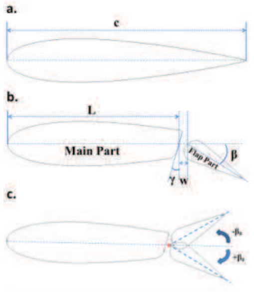


Figure 1 Three different blades in present study (a) Full blade;
(b) Slotted blade with fixed flap; (c) Slotted blade with oscillating flap.

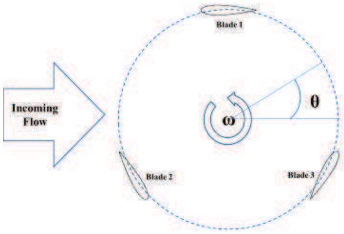


Figure 2 Schematic diagram of flow configuration

The idea of using a fixed flap with slot for aerodynamics flow control was initially developed by Page et al.[4] and Glauert et al.[5] in the 1920s, aiming to achieve a large lift force for a wing at a relatively low flying speed. Weyl et al. [6] also contributed significantly to the development of split flap during 1915 to 1920. In 1927, Fowler flap was invented by Fowler et al. [7] and subsequently tested in NASA wind-tunnel [8]. Experimental measurements proved that the use of Fowler flap, the wing maximum lift coefficient increased to 3.17[9]. Since then, various other types of flaps have been developed such as plain flap, junkers flap, gouge flap and so on [10] [11]. Recently, the slotted aero-wing with flap has been widely adopted in the aircraft design by several famous aircraft companies such as Boeing, Airbus and Douglas to improve the aircraft taking off performance when the high lift is desirable. Apart from the single slot technique, multiple slots method is also used by the modern aircraft designers in order to boost the benefits of slot.

For a VATT, the blade angle of attack usually varies over a wide range when the blade rotates around a rotor, thus large angle of attack and flow separation are inevitable for the traditional turbine constituting of full blades, as shown in Figure 1(a). The concept of slotted blade with flap, which has a success in aerodynamics field, is a good choice to resolve above issues. By adding a slot near the trailing edge of turbine blade, small amount of water stream is forced to pass through the slot from high pressure region to low pressure region. The large momentum associated with the high pressure stream imparts its energy into the boundary layer near the upper surface of blade, resulting in the developed boundary-layer on the base blade breakdown and forming a new boundary layer on the flap part. Such method is believed to reduce the near-boundary turbulence intensity effectively and delay the boundary layer detachment in the vicinity of blade trailing edge. As a consequence, this can reduce blade lift force losses and improve turbine efficiency.

Inspired by the vortex control mechanism, utilized by some aero-/aqua- animals to improve their propulsion performance via their fins or tails flapping motion[12,13], slotted blade with *oscillating* flap is developed and investigated in the present study. The oscillating flap is designed to move periodically around a pitching axis as shown in Figure 1(c). Distinct from a fixed flap, the use of an oscillating flap not only controls the near boundary layer fluid flow, but also regulates the vortex interaction in the wake between each blade. For multiple-blade turbine as studied in this paper, the flow analysis focusing on the blade wake vortex interference is therefore crucial for enhancing the overall turbine performance.

A numerical investigation is carried out with the main objective to explore the potential for enhancing VATT energy extraction efficiency by using fixed and oscillating flap blade. As far as the authors are aware, applying oscillating flap concept on VATT as a flow control tool is quite new, and no research has been conducted on this aspect yet.

---

* Presenting author

References

[1]. Nicholls-Lee. R. F, Turnock. S. R and Boyd. S. W. Simulation based optimization of marine current turbine blades. 7th International conference on computer and IT applications in the maritime industries (COMPIT'08), Liège, Belgium, 21 - 23 Apr 2008.

[2]. Gretton. G. I, Bruce. T and Ingram D. M. Hydrodynamic modeling of a vertical axis tidal current turbine using CFD. 8th European wave and tidal energy conference, Uppsala, Sweden 2009.

[3]. Gretton. G. I. The hydrodynamic analysis of a vertical axis tidal current turbine. PhD thesis, University of Edinburgh, 2009.

[4]. Page. H. The handley page wing. Aeronautical Journal, June 192l, pp. 263-289.

[5]. Glauert. H. The handley page slotted wing. Aeronautical research council, London, R&M No. 834, Mar. 1922

[6]. Weyl. A. R. High-lift devices and tailless aeroplanes. Aircraft Engineering and Aerospace Technology, Vol. 17 Iss: 11, pp.325 – 330, 1945.

[7]. Fowler. H. D. Variable lift. Western flying, November. 1931, pp. 31-33.

[8]. Weick. F. E. and Platt. R. C. Wind-tunnel tests of the fowler variable-area wing. National Advisory Committee for Aeronautics, No. 419, May, 1932.

[9]. Smith. A. High-lift aerodynamics. Vol. 12, No. 6, AIAA 6th aircraft design, flight test and operations meeting, Los Angeles, Calif., August 12-14, 1974.

[10]. Dam. C. V. The aerodynamic design of multi-element high-lift systems for transport airplanes. Progress in Aerospace Sciences, Vol. 38, pp. 101–144, 2002.

[11]. Reckzeh. D. Aerodynamic design of the high-lift-wing for a mega liner aircraft. Aerospace Science and Technology, Vol. 7, pp. 107–119, 2003.

[12]. Xiao. Q. and Liao. W. Numerical investigation of angle of attack profile on propulsion performance of an oscillating foil. Computers & Fluids, Vol. 39, pp.1366-1380, 2010.

[13]. Xiao. Q, Liao. W. and Yang S. C. How motion trajectory affects the energy extraction performance of an oscillating foil. Renewable Energy, Vol.37, pp.61-75, 2012.

The 25th Scottish Fluid Mechanics Meeting

Shear stress on the surface of a mono-filament woven fabric

Y. Li, J. Kobine
Division of Civil Engineering, University of Dundee, Dundee, DD1 4HN

Summary

We present results from a numerical investigation of the shear stress of fluid flow on the surface of a woven fabric. This is part of an EPSRC-funded project on cleaning soiled fabrics at low water temperature. Visser (1976) has shown that the tangential force on the surface contributes significantly to soil particle removal. For an ideal model of a spherical particle resting on a surface, the tangential force F_H can be expressed in terms of the shear stress γ and the radius of particle R, as

$$F_H = 32R^2\gamma.$$

Therefore, the shear stress on the surface of a woven fabric determines in part the cleaning ability for a given fluid flow.

A mono-filament woven fabric with elliptical cross-section yarns (figure 1) is held stationary in an ambient fluid environment so that the fluid-structure interaction is not considered at this stage. The fabric is assumed to be woven tightly so that there is no shifting motion of individual yarns. The geometry of the fabric is constructed using the open source TexGen package. Fluid motion is determined by finite-element calculation using COMSOL Multiphysics. The pressure drop of flow through the fabric, as reported in the literature, is recomputed as validation of our computational model. Then the shear stress on the surface of the fabric is calculated for various configurations of flow fields and structures of woven fabrics. The effect of an oscillatory flow field is also investigated, since the literature suggests that applying an oscillatory flow enhances the cleaning efficiency of fouling on a flat surface or on the inner wall of a cylindrical pipe.

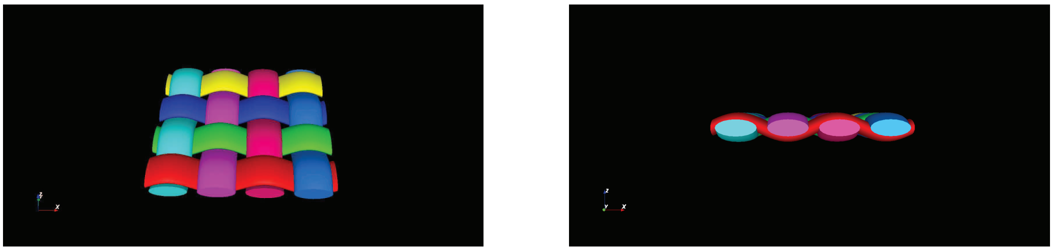


Figure 1: A 4x4 woven fabric and its cross-section.

Drops, Ridges and Rivulets in an External Airflow

Colin Paterson, Stephen K. Wilson and Brian R. Duffy
Department of Mathematics and Statistics, University of Strathclyde, Livingstone Tower,
26 Richmond Street, Glasgow G1 1XH

Abstract

Drops, ridges and rivulets of fluid can occur in a wide variety of practical contexts. In particular, the situations in which they are subject to an external loading caused by a passing airflow are of interest. For example, in the electronics industry, a jet of air is sometimes used to remove drops of water left on the surface of microchips during production. Also of interest is the phenomenon of so-called rain–wind-induced vibrations of cables, in which the formation of water rivulets on the cables of cable-stayed bridges coupled with wind is believed to result in cable vibrations which have a detrimental effect on their durability and can damage the bridge deck.

In this talk I will describe three problems from my PhD. Firstly, a thin ridge of viscous fluid subject to a spatially varying pressure gradient caused by an external airflow will be considered. The external airflow is related to the shape of the free surface of the ridge via thin aerofoil theory, leading to a singular integro-differential equation for the shape. The effect of increasing the strength of the external airflow on a ridge of fixed volume is investigated, and we identify and quantify the conditions for the ridge to de-pin at one contact line.

Secondly, we will consider the gravity-driven draining of a rivulet of fixed width round the outside of a cylinder, and it is found that the rivulet exhibits different behaviour depending on whether it is "narrow" or "wide".

Thirdly, this problem will be extended to include a longitudinal shear stress at the free surface for a rivulet of fixed width or fixed contact angle. Given a strong enough shear, the rivulet will flow round the entire circumference of the cylinder.

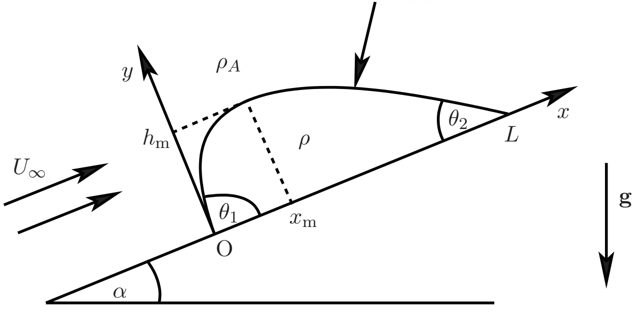


Figure 1: Sketch of a ridge of fluid on an inclined substrate in the presence of an external airflow.

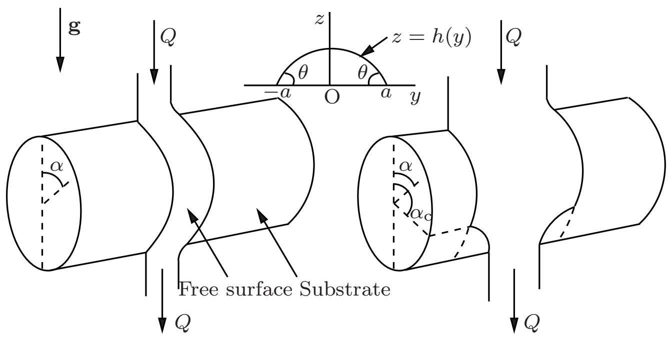


Figure 2: Sketches of a slowly varying rivulet with prescribed flux Q that runs from the top $\alpha = 0$ to the bottom $\alpha = \pi$ of a large horizontal cylinder.

Sediment dynamics in the wake of a tidal current turbine

Lada Vybulkova*[1], Marco Vezza[1], Harshinie Karunarathna[2], and Richard E. Brown[3]

[1]School of Engineering, University of Glasgow, G12 8QQ, Scotland
[2]School of Engineering, University of Swansea, SA2 8PP, UK
[3]Department of Mechanical Engineering, University of Strathclyde, G1 1XJ, Scotland

Abstract

The environmental impact of devices designed to extract power from tidal currents has yet to be thoroughly investigated. The interaction between the wake that is produced downstream of tidal current turbines (TCTs) and the sediment on the seabed is of particular concern given the damage that might be caused to the habitat of marine plants and animals that dwell on the ocean floor. High resolution computational simulations of the hydrodynamics of a TCT and its wake have been conducted using the Vorticity Transport Model (VTM) (Brown and Line, 2005) together with a model for the uplift of sediment from the seabed and its subsequent transport downstream of the turbine. These simulations show the effect of the small-scale, highly intense vortical structures within the wake of the turbine in creating patches of locally-elevated shear stress on the seabed in which sediment uplift is enhanced. The effect of the turbine on the sediment near the seabed is shown to be strongly dependent on the parameters of the device, such as the blade twist distribution, and corresponding power produced by the turbine, but is also influenced by the natural instability within the wake that acts to destroy the coherence of its fine-scale structure some distance downstream of the turbine. These findings suggest that the design of TCTs for deployment in regions with significant sediment mobility needs to be considered with care regarding their impact on the seabed.

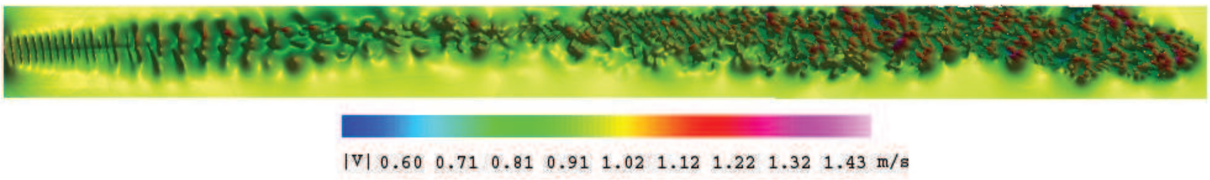


Figure 1: Slice through the wake velocity 10cm above the seabed

References

Brown, R. E., and Line, A. J. (2005). "Efficient High-Resolution Wake Modelling Using the Vortex Transport Equation," *AIAA Journal*, Vol. 43, No. 7, pp. 1434–1443.

---

*presenting author; l.vybulkova.1@research.gla.ac.uk

Flow Analysis for Twin-Flapping Wing Energy Harvester

Saishuai Dai and Qing.Xiao
University of Strathclyde, Glasgow, G4 0LZ, UK

To meet the huge demand of energy, a lot of renewable energy extraction devices have been produced such as turbines and windmills. Recently, a bio-inspired flapping wing based tidal stream energy converter has been developed. Unlike conventional rotational marine turbines, the kinematic energy of tidal stream is converted to mechanical energy by the flapping motion of wing, which imitates the oscillating motion of fish tail or pectoral fins when they swim.

Study on flapping wing energy harvester has attracted more and more attentions since the pioneer work done by McKinney and Delaurier[1]. For a single two-dimensional foil with sinusoidal pitching and plunging motion, it is observed by Kinsey and Dumas[2] that, the total efficiency of energy extraction could be as high as 35% with an optimized combination of plunging amplitude and frequency. Jones and Platzer, Jones et al. and Simpson et al. have investigate the performance potential for single and tandem arranged two oscillating foil over a large range of different parameters by means of water tunnel experiments. They concluded that for a single foil, maximum power was reached when the motion phase between heaving and pitching is around 90 degree and the angle of attack has important impact on power extraction efficiency. For twin foils using panel method, they found that the tandem arranged foils had a larger output than single foil.

Most numerical studies mentioned above are either focused on single wing or ignore the viscous effect. In this study, NACA 0015 foils with tandem configuration are examined. Simulations are carried out using commercial CFD software package FLUENT coupled with secondary developed User Defined Function (UDF). Investigation is concentrated on how gap ratio between two foils affects the device power output and the difference between motion 1 and motion 2 which are illustrated in Fg.1 and Fig. 2, respectively. Our computed results (plotted in Fig. 3) showed that an optimal gap ratio exists at which the twin wing device generates more power than single wing. This gap is also influenced by the flapping frequency and phase difference between up-stream and down-stream wings.

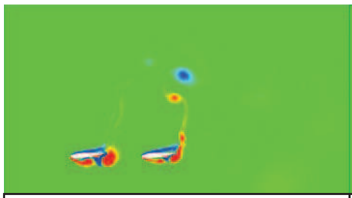


Fig..1 Vorticity contour (phase lag between two wing is zero)

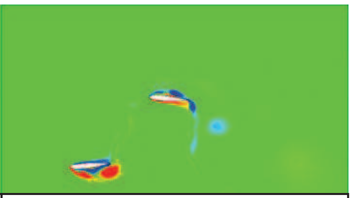


Fig..1 Vorticity contour (phase lag between two wing is 180 degree)

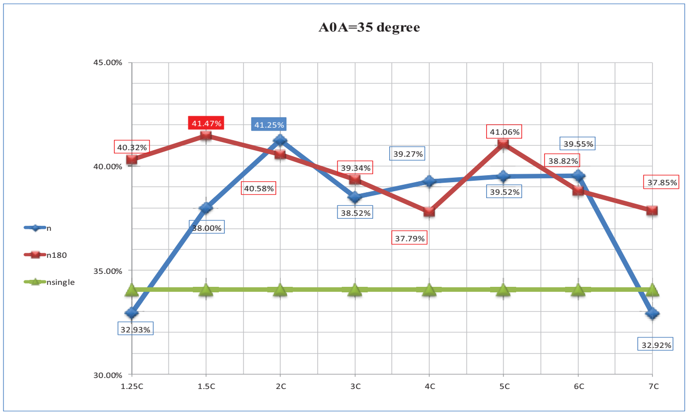


Fig.3 Power efficiency versus gap ratio (effective AOA=35 degree)

References

1. McKinney,W.& DeLaurier, J., "The Wingmill: An Oscillating-Wing Windmill," *Journal of Energy*, **5**, No. 2, pp. 109-115, 1981
2. T.Kinsey &G.Dumas, "Parametric Study of an Oscillating Airfoil in a Power-Extraction Regime," *Journal of AIAA*, Vol.46, No. 6, June 2008

Dynamic Wetting on Moving Surfaces:
A Molecular Dynamics Study

Konstantinos Ritos*, **Nishanth Dongari, Yonghao Zhang and Jason M. Reese**

University of Strathclyde
Mechanical & Aerospace Engineering Department
E-mail: konstantinos.ritos@strath.ac.uk
* Presenting author

We present molecular dynamics (MD) simulations of the dynamic wetting of nanoscale droplets on moving surfaces. The dynamic water contact angle and contact angle hysteresis are measured as a function of capillary number on smooth silicon and graphite surfaces. The hydrogen bonding and density profile variations are also reported, and the width of the water depletion layer is evaluated for droplets on three different static surfaces: silicon, graphite and a fictitious super-hydrophobic surface. Our results show that molecular displacements at the contact line are mostly influenced by interactions with the solid surface, while the viscous dissipation effects induced through the movement of surfaces are found to be negligible, especially for hydrophobic surfaces. This finding is in contrast with the wetting dynamics of macroscale droplets, which show significant dependence on the capillary number. This study may yield new insight into surface-wettability characteristics of nano droplets, in particular, developing new boundary conditions for continuum solvers for liquid flows in micro- and nanoscale devices.

On energetics and inertial range scaling laws of two-dimensional magnetohydrodynamic turbulence

Luke A. K. Blackbourn & Chuong V. Tran
School of Mathematics and Statistics, University of St Andrews

Abstract

The magnetohydrodynamic equations, which govern the motion of an electrically conducting fluid and the evolution of the internally generated magnetic field, have been actively studied for decades. In three dimensions, the usual vortex-stretching mechanism provides a route to allow the downscale transfer of kinetic energy, while the advection of the magnetic field twists the field lines, creating sharp gradients and producing a downscale magnetic energy flux. This leads to the expected direct cascade of the inviscidly-conserved total energy, $\langle |\boldsymbol{u}|^2 \rangle/2 + \langle |\boldsymbol{b}|^2 \rangle/2$.

In two dimension, the situation is different in that, in the absence of the Lorentz force, kinetic energy is preferentially transferred to large scales. Since the total energy is still transferred downscale (see e.g. Fyfe & Montgomery (1986)), the lack of a vortex-stretching term necessitates another route for the direct transfer of kinetic energy. It is only through the Lorentz force, via the conversion of kinetic to magnetic energy and back again, that a direct transfer of kinetic energy can be established.

By examining the equations governing the individual Fourier-mode interactions produced by the nonlinear terms, the mechanisms that result in this forward energy transfer can be deduced. It is seen that dynamo action (defined here as the conversion of kinetic into magnetic energy), and anti-dynamo action (the conversion of magnetic to kinetic energy) are intrinsically linked with the forward energy transfer. This examination naturally leads to the concept of *dynamo saturation*, where dynamo action stalls due to the depletion of the kinetic energy reservoir. This could be expected to lead to a depletion of nonlinearity in the system, in agreement with the Iroshnikov-Kraichnan (IK) picture of energy transfer reduction due to Alfvén wave effects (see Kraichnan (1965)).

These ideas were tested by carrying out a series of pseudo-spectral simulations, up to 4096^2 resolution, at unity magnetic Prandtl number, for different ratios of initial magnetic to kinetic energy. It is seen that certain aspects of IK theory, such as energy equipartition, do not hold up to scrutiny, and the total energy spectrum seems to be shallower than the predicted $k^{-3/2}$ spectrum. However the IK nonlinearity depletion can be seen under the guise of dynamo saturation, which is achieved through weakening cycles of dynamo and anti-dynamo action. One result of this depletion of nonlinearity is that as the viscosity is decreased, the maximum energy dissipation rate $\epsilon_\nu(T)$ decays, and the time of maximum dissipation T diverges. This suggests that in the inviscid limit, the maximum energy dissipation rate slowly, probably logarithmically, converges, although whether it converges to a non-zero constant, as sugested in Biskamp & Welter (1993), or to zero, is unclear.

REFERENCES

Biskamp, D. & Welter, H. 1989 Dynamics of decaying two-dimensional magnetohydrodynamic turbulence. *Phys. Fluids* B **1**, 1964–1979.

Fyfe, D. & Montgomery, D. 1976 High-beta turbulence in two-dimensional magnetohydrodynamics. *J. Plasmas Phys.* **16**, 181–191.

Kraichnan, R. H. 1965 Inertial-range spectrum of hydromagnetic turbulence. *Phys. Fluids* **8**, 1385–1387.

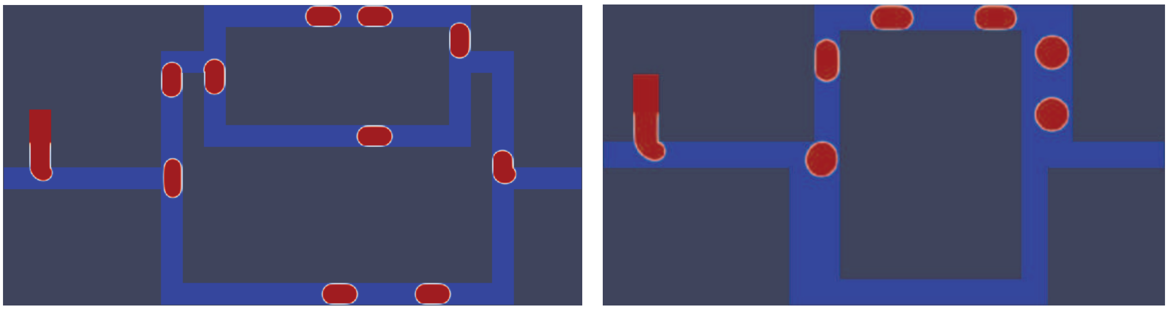
Microdroplet simulations using a multiphase lattice Boltzmann method

Jonathan Li, Yonghao Zhang, Jason M Reese

Department of Mechanical & Aerospace Engineering, University of Strathclyde, Glasgow

In recent years, droplet-based microfluidic devices have attracted significant interest within biology and chemistry communities due to their huge potential for cost and time savings, enhanced analysis sensitivities, and accuracy. However, much of this potential remains untapped due to insufficient understanding of droplet behaviour, which means that microdroplet devices are designed predominantly through trial and error. Although many experimental studies have been carried out to characterise individual droplet behaviour, computational studies at the device level are important to provide more detailed information that is difficult to measure experimentally.

We have recently proposed a lattice Boltzmann (LB) multiphase model using a colour-fluid approach [1, 2]. Here, we demonstrate its capabilities for simulating two-phase microfluidic flows in microchannel networks. We show droplet simulation results for a number of geometries and compare them to experimental data. As Figure 1 indicates, droplets at branch intersections travel along the channel with the highest local flow velocity, which is similar to the observations of Choi et al. [3] for bubble flow in microchannel networks.



(a) (b)

Figure 1: Simulation of droplet traffic in microchannel networks. Carrier (blue) fluid inlet capillary number $Ca_c=0.0125$, Reynolds number Re=0.05. Droplet (red) fluid inlet capillary number $Ca_d=2.5\text{x}10^{-4}$, Reynolds number $Re_d=0.1$. Flow rate ratio $Q=Q_c/Q_d=1/5$. Viscosity ratio $\lambda=\lambda_c/\lambda_d=1/10$. The smaller channels are of width 100μm, while the larger channel in (b) is 200 μm wide.

References

1. HH Liu, YH Zhang (2011) Droplet formation in microfluidic cross-junctions. *Physics of Fluids* 23: 082101.
2. HH Liu, YH Zhang, AJ Valocchi (2012) Modeling and simulation of thermocapillary flows using lattice Boltzmann method. *Journal of Computational Physics* 231, 4433-4453.
3. W Choi, M Hashimoto, AK Ellerbee, X Chen, KJM Bishop, P Garstecki, HA Stone, GM Whitesides (2011) Bubbles navigating through network of microchannels. *Lab on a Chip* 11, 3970-3978.

Revealing small-scale structures in turbulent Rayleigh-Bénard convection

Yue-Kin Tsang,[1,*] Emily S. C. Ching,[2] T. N. Fok,[2] XiaoZhou He,[3] and Penger Tong[3]

[1]*School of Mathematics and Statistics, University of St Andrews, UK*

[2]*Department of Physics, The Chinese University of Hong Kong, Hong Kong*

[3]*Department of Physics, Hong Kong University of Science and Technology, Hong Kong*

Abstract

Turbulent Rayleigh-Bénard convection has been a paradigm of thermally driven turbulence and its small-scale properties has been the focus of numerous research since the original works of Bolgiano and Obukhov in 1959. Small-scale flow properties can be characterized by the temperature structure functions $S_T^{(p)}(\tau)$, defined as the moments of temperature difference $\delta_\tau T(t)$ between time t and $t+\tau$. Based on the assumption that the flow is thermally driven, i.e. temperature is an active scalar, the Bolgiano-Obukhov phenomenology (with intermittency corrections) implies the following scaling in some range of τ,

$$S_T^{(p)}(\tau) \equiv \langle |\delta_\tau T(t)|^p \rangle \sim \langle \chi_\tau^{2p/5} \rangle \, \tau^{p/5} \, .$$

Above, $\langle \cdot \rangle$ denotes time average. $\chi_\tau(t)$ is the average of the (intermittent) thermal dissipation rate over a period of τ about time t. In a Rayleigh-Bénard convection cell, temperature is expected to be active in the thermal boundary layers near the top and bottom plates of the cell. However, the Bolgiano-Obukhov scaling has never been observed in laboratory experiments or numerical simulations. One of the difficulties in identifying the Bolgiano-Obukhov scaling is the lack of a scaling range in the data. Using new experimental data where simultaneous temperature signals from four nearby probes are available, hence making it possible to obtain χ_τ, we introduce the *conditional* temperature structure function,

$$\hat{S}_T^{(p)}(X,\tau) \equiv \langle |\delta_\tau T(t)|^p \, \big| \, \chi_\tau = X \rangle \, ,$$

i.e., a certain $\delta_\tau T(t)$ is included in the average only if the corresponding $\chi_\tau(t)$ equals X. For measurements made at the bottom plate, our analysis shows that $\hat{S}_T^{(p)}(X,\tau)$ has a clear power-law dependence on X and the scaling exponents are consistent with the Bolgiano-Obukhov theory. We also perform analysis on data obtained at the cell center and provide support for the idea that temperature in the bulk region is being passively advected (which results in a different scaling for the structure functions).

Acknowledgement: This work was supported by the Hong Kong Research Grants Council under Grant No. CUHK-400708 (ESCC) and HKUST-604310 (PT).

---

[*] Presenting author

Numerical investigation of thermal protection system discontinuities

Rodrigo C. Palharini, Thomas J. Scanlon and Jason M. Reese

Department of Mechanical and Aerospace Engineering, University of Strathclyde, Glasgow G1 1XJ, UK

During re-entry, aerospace vehicles are exposed to extreme mechanical and thermodynamic loads requiring accurate design of a heat shield. Generally, designers try to keep the surface smooth, but discontinuities between thermal protection plates have to be expected due to sensor installation, fabrication tolerances and differential expansion, ablation rates of non-similar materials or even micrometeorite impacts. Such surface discontinuities, as shown in Figure 1, may constitute a source of heat flux to the surface or even in a premature transition from laminar to turbulent flow.

Hypersonic flow over cavities [1, 2] may cause local thermal and aerodynamic loads dramatically exceeding the ones of a smooth surface. In order to operate safely, these loads have to be predicted correctly. This can be done either by experiments which are often very expensive for real flight conditions, or by numerical simulation, which are now of continually increasing importance.

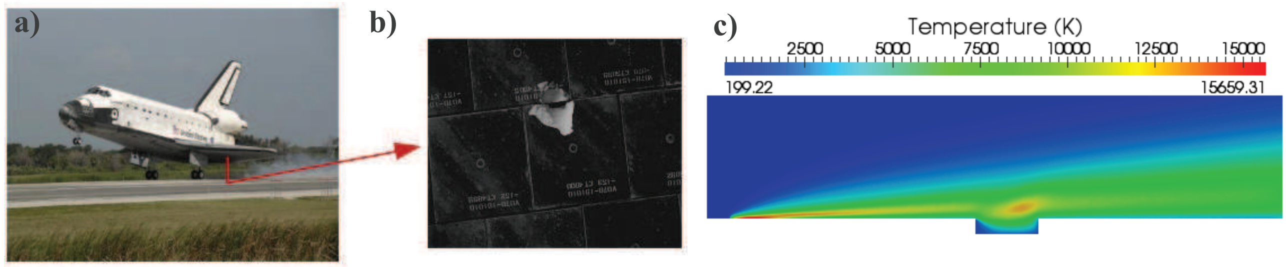


Figure 1: a) Space shuttle; b) Thermal protection system damage; c) 2D simulation of rarefied hypersonic flow over a cavity.

In order to understand the physical aspects of hypersonic flow over cavities, the majority of the available studies have considered only laminar or turbulent flow in the continuum flow regime. However, there is little understanding of the physical aspects of rarefied hypersonic flow past cavities associated with a re-entry environment. Computations of a rarefied hypersonic flow over 2D/3D cavities have been performed using the direct simulation Monte Carlo (DSMC) method [3] aiming to assess the sensitivity of heat transfer, pressure and skin friction coefficients for a family of cavities. In the present work, simulations are performed using a rarefied hypersonic non-reacting gas, consisting of 76.3% N2 and 23.7% O2 that represents the freestream conditions at 80 km of altitude. The cavities considered in this study are modelled using a flat plate of infinite length and cavity length-to-depth ratios (L/D) ranging from 1 to 4.

The analysis shows that the aerodynamic quantities acting on the vehicle surface on the cavity surface depend heavily on the three dimensional flow characteristics. It is shown that a two-dimensional cavity study will over-predict heat and pressure loads, with these being several times larger than those observed for a three-dimensional cavity.

[1] Everhart, J. L., Alter, S. J., Merski, N. R., and Wood, W. A., **Pressure Gradient Effects on Hypersonic Cavity Flow Heating,** 44th AIAA Aerospace Sciences Meeting and Exhibit, AIAA Paper 2006–0185, 2006

[2] Jackson, A. P., Hillier, R., and Soltani, S., **Experimental and Computational Study of Laminar Cavity Flows at Hypersonic Speeds,** Journal of Fluid Mechanics, vol. 427, pp. 329–358, 2001

[3] Bird, G. A., **Molecular Gas Dynamics and the Direct Simulation of Gas Flows**, Oxford University Press., 1994

Extensional flow of viscoelastic fluids in an optimised shape cross-slot extensional rheometer (OSCER)

Mónica S.N. Oliveira[1*], **Simon J. Haward**[2], **Gareth H. McKinley**[2], **and Manuel A. Alves**[3]

[1]Departament of Mechanical and Aerospace Engineering, University of Strathclyde,
Glasgow, G1 1XJ, UK
monica.oliveira@strath.ac.uk

[2]Hatsopoulos Microfluids Laboratory, Department of Mechanical Engineering, Massachusetts
Institute of Technology, Cambridge, MA 02139, USA
shaward@mit-edu; gareth@mit.edu

[3]Departamento de Engenharia Química, CEFT, Faculdade de Engenharia da Universidade do Porto,
Rua Dr. Roberto Frias s/n, 4200-465, Porto, Portugal
mmalves@fe.up.pt

Key words: optimal shape design, cross-slot, extensional flow, complex fluids

ABSTRACT

Shape optimisation techniques together with computational fluid dynamics (CFD) simulations were used to optimise the kinematics of a cross-slot microdevice. The goal is to generate a strong extensional flow comprising a large region of constant strain-rate along the centreline of the device; a fundamental, but often challenging, requirement for meaningful extensional rheometry measurements. The optimised shape cross-slot extensional rheometer (OSCER) has an unexpected shape (based on the canonical cross-slot shape) with a central stagnation point, where the fluid velocity is zero but the local extension rate is finite. The OSCER is able to generate a homogeneous elongational flow with large regions of nearly uniform extensional strain-rate, which is a significant advantage over classical cross-slot devices, in which the imposed strain rate is well-defined only in the region very close to the stagnation point and decays rapidly along the exit channels. Numerical calculations with different viscoelastic models revealed that the optimised shape was very insensitive to the imposed flow rate (and the corresponding dimensionless Weissenberg number), potentially making this a universal design capable of generating an ideal planar extensional flow at low Reynolds number for a range of fluids.

A prototype of the OSCER was constructed and tested using a number of different fluids, including Newtonian fluids, synthetic polymer solutions, wormlike micellar solutions and biopolymeric fluids. The device consists of a stainless steel plate in which the optimised shape was precision-machined by wire-electro-discharge machining (EDM). The OSCER geometry is assembled between two glass discs to confine the flow and allow for optical access. Micro-particle image velocimetry (μ-PIV) is used to quantify the flow kinematics, showing that indeed a large region of homogeneous kinematics is obtained, in which the outflow velocity increases linearly with distance from the stagnation point. For the micellar and macromolecular solutions, full-field birefringence microscopy is used to evaluate the local state of stress in the fluid. Above a critical Weissenberg number, a narrow and uniform birefringent strand is formed along the exit channel centreline, highlighting the extent of localised microstructural orientation along the streamlines passing close to the stagnation point. Furthermore, birefringence and bulk pressure measurements are shown to provide, for the first time, self-consistent estimates of the true steady planar extensional viscosity of the fluids over a range of deformation rates.

Modelling Micro-Diffusion and Autocatalysis in Biodegradable Polymers

Yuhang Chen[1], Juergen Siepmann[2], Florence Siepmann[2], Qing Li[3]

[1] Institute of Mechanical, Process and Energy Engineering, Heriot-watt University, Edinburgh EH14 4AS, UK
[2] College of Pharmacy, Univ. Lille Nord de France, Lille 59006, France
[3] School of Aerospace, Mechanical and Mechatronic Engineering, University of Sydney, NSW 2006, Australia
y.chen@hw.ac.uk

Biodegradable polymers have attracted significant attention recently for their broad use as scaffolding materials and drug carriers. In both scenarios, such biomaterials require not only proper biocompatibility, but controllable degradation rate and pattern that are suitable for material functionality **[1]**. Although extensive experimental studies have been conducted to investigate the mechanisms of polymeric degradation, how to regulate and control the degradation process of biopolymer devices remains rather challenging **[2]**.

The aim of this study is to model the micro/nano-diffusion and autocatalysis effect within biodegradable polymers, which enables the design and optimisation of polymer micro/nanostructures towards controllable degradation and drug release. The hydrolysis reaction is modelled in a discrete fashion by a fundamental stochastic process and an additional autocatalytic effect induced by the local carboxylic acid concentration in terms of the continuous diffusion equation. Illustrative examples of microparticles and tissue scaffolds demonstrate the applicability of the model. It is found that diffusive transport plays a critical role in determining the degradation pathway, whilst autocatalysis makes the degradation size dependent. The modelling results show good agreement with experimental data, in which the hydrolysis rate, polymer architecture and matrix size actually work together to determine the characteristics of the degradation and erosion processes of bulk-erosive polymer devices **[3]**.

After the establishment of the abovementioned mathematical models for characterising the degradation and autocatalysis of biodegradable polymers, a series of tests have been conducted to seek the critical size of degradation, at which the autocatalysis effect can be ignored and the μpH environment approaches neutral. Such critical sizes of different polymers signify an important criterion when designing and fabricating biomaterials for tissue engineering since the degradation rate and μpH environment would make strong impact on neo-tissue mechanobiology and growth. Our results showed that different biodegradable devices have slightly different degradation rates and μpH environment, with the critical sizes ranging from 8μm to 12μm depending on their distinct microstructures and micro-diffusion, in both numerical and experimental tests.

The future work will be devoted to the study on controlled degradation for both tissue engineering and drug delivery systems by approaching optimal micro/nanostructures for those bio-devices utilising numerical design optimisation and 3D biofabrication technique

References:

[1] Chen YH, Zhou SW and Li Q (2011). Microstructure design of tissue scaffold and its effect on tissue regeneration. ***Biomaterials***, 32:5003-5014.
[2] Siepmann J and Goepferich A (2001). Mathematical modeling of bioerodible, polymeric drug delivery systems. ***Advanced Drug Delivery Reviews***, 48:229-247.
[3] Chen YH, Zhou SW and Li Q (2011). Mathematical modeling of degradation for bulk-erosive polymers: applications in tissue engineering scaffold and drug delivery system. ***Acta Biomaterialia***, 7:1140-1149.

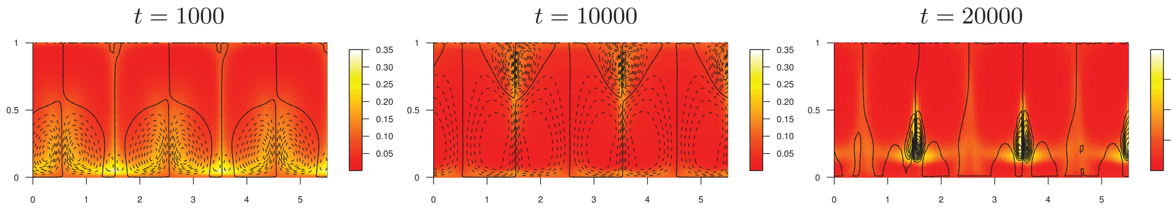
The effect of reactive thermosolutal convection on the long-term evolution of a porous medium

L. T. Corson & D. Pritchard
Department of Mathematics & Statistics, University of Strathclyde,
26 Richmond Street, Glasgow G1 1XH

The circulation of a fluid carrying dissolved minerals through porous rock, under the influence of buoyancy forces associated with thermal or solutal gradients, is believed to occur in many geological settings. The amount of solute in the fluid is unlikely to be conserved, however, as it may be dissolved from or precipitated onto the rock matrix as its solubility varies with temperature, pressure and the rock chemistry.

This study considers thermosolutal (double-diffusive) convection of a reactive fluid in a saturated porous medium, subject to fixed temperatures and chemical equilibrium on the bounding surfaces, where the solubility of the dissolved component depends on the temperature, and the porosity and permeability of the medium can evolve through precipitation and dissolution.

Using numerical methods we investigate the long-term evolution of the porous layer. We find there are two characteristic evolutionary behaviours which result from different dominances of the underlying thermal and solutal gradients. When the solutal gradient dominates, the reaction-permeability feedback triggers a secondary instability, resulting in the lateral migration of the concentration and temperature fields and rapid reversals in the direction of circulation. However, when the thermal gradient dominates, the reaction-permeability feedback tends to suppress the circulation, although it does re-emerge after a quiescent period.



convection suppressed   slow evolution   convection re-emerged

Figure 1: Porosity evolution with dominant thermal gradient. Shading indicates porosity and solid and dashed lines are streamlines.

Scottish Fluids Meeting 2012

Shallow-water vortex equilibria and their stability

Hanna Płotka and David G. Dritschel
School of Mathematics and Statistics, University of St Andrews, St Andrews

Abstract

Vortices are self-organized fluid dynamical structures which spontaneously arise in turbulent flows (see for example [McWilliams, 1984]). They are of great importance in geophysical fluid dynamics, and are an omnipresent feature of the Earth's oceans and atmosphere, as well as the atmospheres of the giant gas planets.

Vortices may be regarded as 'balanced' flow features, whose velocity and thermo-dynamical fields are largely self-induced, essentially by their distribution of 'potential vorticity'. Geophysical flows may meaningfully be split into two components: (1) a dominant, long-lived (compared to a characteristic time such as a day) balanced component consisting of vortices, fronts and jets, and (2) a weaker, higher-frequency, relatively disorganised unbalanced component consisting of inertia-gravity waves.

Turbulent geophysical flows are hugely complicated. As a first step towards understanding them, therefore, it may be useful to examine them through this concept of balance. A tool to do this is the quasi-geostrophic (QG) approximation, which filters out the unbalanced, gravity-wave component, and retains only the balanced, vortical motions.

In this study, we use the QG model as a starting point to investigate the forms of vortex equilibria and their stability, which to date have not been fully documented even for the simplest case of a single vortex patch consisting of uniform potential vorticity (PV). In 1893, Love examined the 'barotropic' (or 2D Euler) case of an elliptical vortex patch, and showed analytically that all vortices having an aspect ratio $\lambda \geq 1/3$ are linearly stable, while those having $\lambda < 1/3$ are unstable [Love, 1893]. Nearly a century later, this was confirmed numerically, for finite-amplitude disturbances [Dritschel, 1986]. Nearly all studies to date have considered only the barotropic model, applicable only when the Rossby deformation radius $L_D = c/f$ (where $c = \sqrt{gH}$ is a characteristic gravity-wave speed, g is gravity, H is the mean active layer depth, and f is the Coriolis frequency) is infinite. The few exceptional studies include Polvani et al. [1989] and Flierl [1988]. Here, we fully document the steadily-rotating equilibria together with their linear and nonlinear stability for a single vortex patch. In the QG model, such equilibria are spanned by two parameters, L_D and λ.

This study goes further to examine vortex 'quasi-equilibria' in the parent shallow-water equations, a model which does permit gravity waves. Using a novel ramping procedure within the full equations of motion, we are able to generate approximate equilibria (unsteady only due to extremely weak gravity wave radiation), and at the same time investigate their (nonlinear) stability. That is, at some critical amplitude of the PV inside the vortex (related to the Rossby number), the vortex exhibits a rapid change in shape, often splitting into two or more parts.

References

D. G. Dritschel. The nonlinear evolution of rotating configurations of uniform vorticity. *J. Fluid Mech.*, 172:157–182, 1986.

G. R. Flierl. On the instability of geostrophic vortices. *J. Fluid Mech.*, 197:349–388, 1988.

A. E. H. Love. On the stability of certain vortex motions. *Proc. London Math. Soc.*, 25:18–42, 1893.

J. C. McWilliams. The emergence of isolated coherent vortices in turbulent flow. *J. Fluid Mech.*, 146: 21–43, 1984.

L., M. Polvani, N. J. Zabusky, and G. R. Flierl. Two-layer geostrophic vortex dynamics. part 1. upper-layer v-states and merger. *J. Fluid Mech.*, 205:215–242, 1989.

Squeeze-Film Flow in the Presence of a Thin Porous Bed, with Application to the Human Knee Joint

D. J. Knox[1], **S. K. Wilson**[2], **B. R. Duffy**[3] **and S. McKee**[4]
Department of Mathematics and Statistics, University of Strathclyde,
26 Richmond Street, Glasgow G1 1XH, Scotland, United Kingdom

Motivated by the desire for a better understanding of the lubrication of the human knee joint, the squeeze-film flow of a thin layer of Newtonian fluid (representing the synovial fluid) filling the gap between a flat impermeable surface (representing the femoral condyles) and a flat thin porous bed (representing the articular cartilage) coating a stationary flat impermeable surface (representing the tibial plateau) is considered. Figure 1 shows sketches of the main anatomical features of the human knee joint.

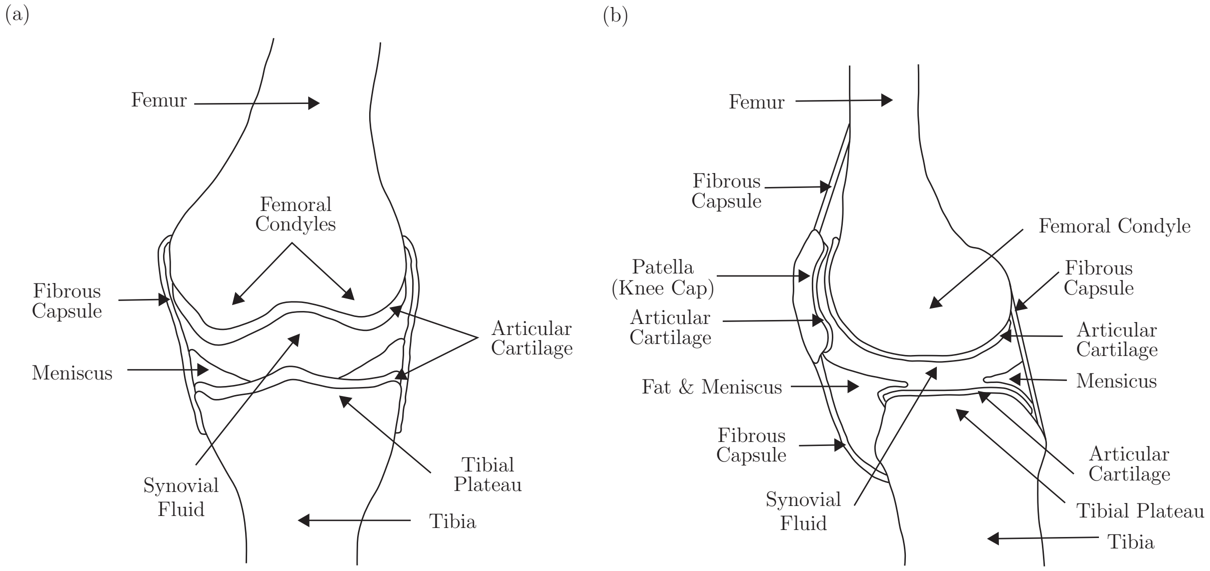


Figure 1: Sketches of the main anatomical features of the human knee joint. A slice through (a) the front of the knee (behind the patella) and (b) the side of the knee (through the patella)

As the impermeable surface approaches the porous bed under a prescribed constant load all of the fluid is squeezed out of the gap in a finite contact time, denoted by $t = t_\mathrm{c}$ (where t denotes time). Figure 2(a) shows plots of the fluid layer thickness, h, as a function of $\ln t$ for different values of the permeability k, and Figure 2(b) shows a plot of the logarithm of the finite contact time, $\ln t_\mathrm{c}$, as a function of $\ln k$. In the context of the knee, the size of the contact time suggests that when a person stands still for a short period of time their knees may be fluid lubricated, but that when they stand still for a longer period of time contact between the cartilage-coated surfaces may occur.

---

[1] Email: `darren.knox@strath.ac.uk`

[2] Presenting author. Presently also a Visiting Fellow in the Oxford Centre for Collaborative Applied Mathematics (OCCAM), University of Oxford, Mathematical Institute, 24–29 St. Giles', Oxford OX1 3LB. Email: `s.k.wilson@strath.ac.uk`

[3] Email: `b.r.duffy@strath.ac.uk`

[4] Email: `s.mckee@strath.ac.uk`

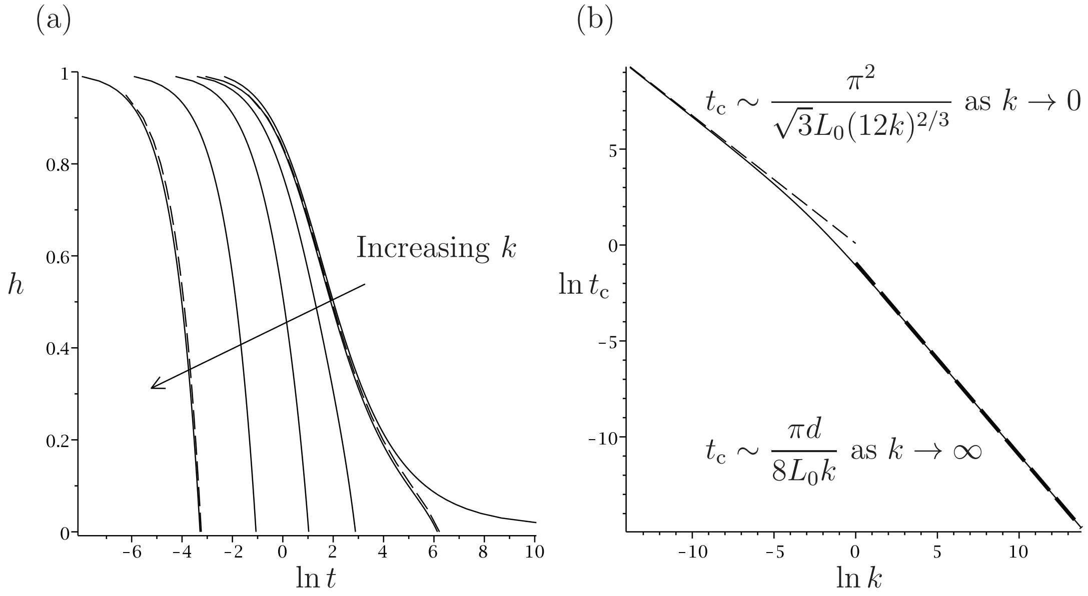


Figure 2: (a) Fluid layer thickness, h, as a function of $\ln t$ for $k = 0$, 10^{-4}, 10^{-2}, 10^{-1}, 1 and 10 (solid lines), the uniformly valid leading order small-k asymptotic solution for $k = 10^{-4}$ (dashed line), and the leading order large-k asymptotic solution for $k = 10$ (dashed line). (b) Logarithm of the finite contact time, $\ln t_\mathrm{c}$, as a function of $\ln k$ (solid line), the leading order small-k asymptotic solution (dashed line), and the leading order large-k asymptotic solution (bold dashed line).

The fluid particle paths are calculated, and the penetration depths of fluid particles into the porous bed are determined. In the context of the knee, these penetration depths provide a measure of how far into the cartilage nutrients are carried by the synovial fluid, and suggest that when a person stands still nutrients initially in the fluid layer penetrate only a relatively small distance into the cartilage. However, the model also suggests that the cumulative effect of repeated loading and unloading of the knees during physical activity such as walking or running may be sufficient to carry nutrients deep into the cartilage.

The first author (DJK) gratefully acknowledges the financial support of the University of Strathclyde via a University Postgraduate Research Scholarship. Part of this work as undertaken while the presenting author (SKW) was a Visiting Fellow in the Department of Mechanical and Aerospace Engineering, School of Engineering and Applied Science, Princeton University, USA, and part of it was undertaken while he was a Visiting Fellow in the Oxford Centre for Collaborative Applied Mathematics (OCCAM), Mathematical Institute, University of Oxford, United Kingdom. This publication was based on work supported in part by Award No KUK-C1-013-04, made by King Abdullah University of Science and Technology (KAUST). All four authors gratefully acknowledge valuable discussions with Dr Richard Black, Dr Philip Riches and Mr Anthony Herbert from the Bioengineering Unit at the University of Strathclyde on the biomechanics of the knee.

D. J. Knox, S. K. Wilson, B. R. Duffy and S. McKee, “Squeeze-Film Flow in the Presence of a Thin Porous Bed, with Application to the Human Knee Joint,” manuscript submitted for publication (2012).

Run-up of long waves on a plane beach

YONG SUNG PARK

Department of Civil Engineering, University of Dundee, Dundee DD1 4HN, United Kingdom
Email: y.s.park@dundee.ac.uk

The surf similarity parameter, or Iribarren Number, is known to characterize breaking of periodic waves including maximum run-up on a plane beach, which is measured vertically from the initial still water level. Mei (1989) pointed out that the surf similarity parameter for the periodic waves appear in the breaking criteria using the nonlinear shallow water equation, and in this talk it will be shown that a similar parameter can also be found for nonlinear long waves such as solitary waves and cnoidal waves. In particular, the cnoidal wave is expressed as the exact sum of solitary waves, which makes analysis much simpler. Experimental data of long wave run-up are well characterized by the new parameter.

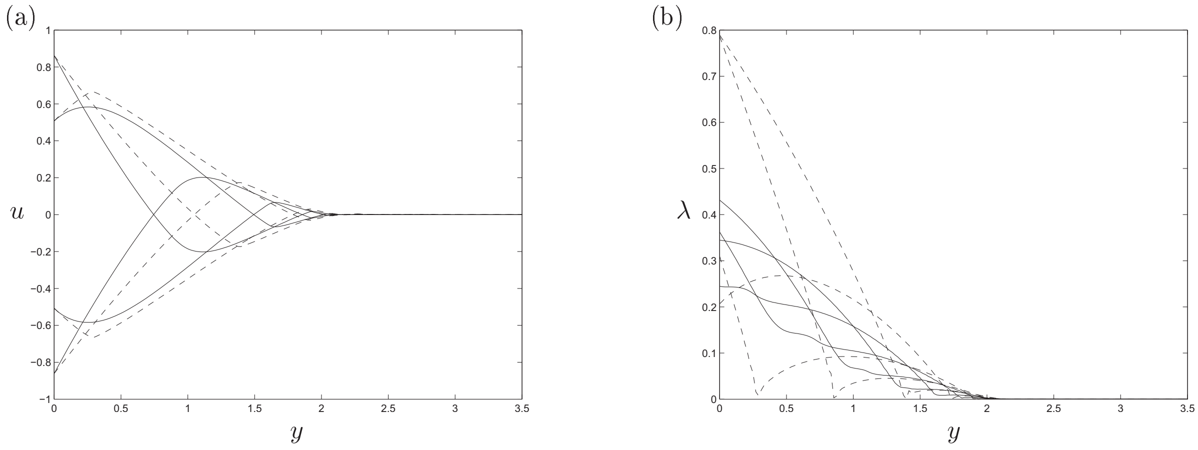
The Stokes boundary layer for a thixotropic or antithixotropic fluid

Catriona McArdle, David Pritchard and Stephen K. Wilson
Department of Mathematics and Statistics, University of Strathclyde, Glasgow

We present a mathematical investigation of the behaviour of the oscillatory boundary layer ("Stokes layer") in a semi-infinite fluid bounded by an oscillating wall (the so-called Stokes problem), when the fluid has a thixotropic or antithixotropic rheology described by the structure parameter model of Mewis and Wagner [1]. Numerical solutions are compared successfully with asymptotic solutions obtained in the limit of small oscillations.

These solutions differ significantly from the classical solution for a Newtonian fluid in the behaviour both of the velocity and of the structure parameter. Three regimes of behaviour for small amplitude are identified, in which the structure parameter takes values defined instantaneously by the shear rate, or by a long-term average, or behaves hysteretically. The regime boundaries depend subtly on the exponents in the structure evolution equation. Additionally, as for a power-law fluid [2], for antithixotropic fluids the velocity reaches zero at a finite distance from the wall, in contrast to the exponential decay for a thixotropic or Newtonian fluid.

These results illustrate the subtle behaviour of complex fluid models in non-rheometric settings.



"Snapshots" of the (a) velocity u and (b) structure parameter λ for a fast-adjusting antithixotropic fluid.

[1] J. Mewis and N. J. Wagner. Thixotropy. *Advances in Colloid and Interface Science,* 147-148:214-227, 2009.

[2] D. Pritchard, C. R. McArdle, and S. K. Wilson. The Stokes boundary layer for a power-law fluid. *Journal of Non-Newtonian Fluid Mechanics,* 166:745-753, 2011.

Coupled Modelling of Turbidity Currents over Erodible Beds

By Peng Hu[1], Zhixian Cao[2], and Gareth Pender[3]

Abstract

Turbidity current, which is subaqueous sediment-laden flow, have been recognized to be critical for effective sediment management in lakes and reservoris and known to be the principle mechanism shaping submarine morphological features and turbidites. Yet previous understanding is mostly based on flume experiments, indirect back-estimation from the resultant morphological features, and decoupled numerical modelling ignoring the feedback impacts of bed deformation partly or completely. Here a layer-averaged coupled numerical model is developed of turbidity currents over erodible beds, which is applicable to both regular and irregular topographies and explicitly accounts for the interactions between turbidity current, sediment transport and bed topography. The governing equations of the model are built upon the complete mass and momentum conservation laws, and are numerically solved by a well-balanced version of the Slope Limiter Centred (SLIC) scheme in the framework of finite volume method. The significance of the coupled modelling approach is demonstrated for turbidity currents featuring active sediment transport and rapid bed deformation, as compared to the previous decoupled modelling. Two applications of the coupled model are outlined on turbidity currents in the Xiaolangdi reservoir, Yellow river, China; and those responsible for the formation of submarine channel-levee morphology. The usefulness of the model to faciliating effective sediment management in reservoirs and examining the formation process of submarine morphological features are highlighted.

---

[1] Institute for Infrastructure and Environment, Heriot-Watt University, UK. E-mail: ph66@hw.ac.uk

[2] Professor, Institute for Infrastructure and Environment, Heriot-Watt University, UK; and State Key Laboratory of Water Resources and Hydropower Engineering Science, Wuhan University, China. E-mail: z.cao@hw.ac.uk

[3] Professor, Institute for Infrastructure and Environment, Heriot-Watt University, UK, and Visiting Professor, State Key Laboratory of Water Resources and Hydropower Engineering Science, Wuhan University, China.

SFMM 2012
25th Scottish Fluid Mechanics Meeting
30 May 2012
Heriot-Watt University

Boundary Conditions for Molecular Dynamics Simulation of Water Transport Through Carbon Nanotubes

Stephanie Y. Docherty*, William D. Nicholls, Matthew K. Borg, and Jason M. Reese
*Presenting Author
Department of Mechanical and Aerospace Engineering,
University of Strathclyde, Glasgow G1 1XJ, UK
E-mail: stephanie.docherty@strath.ac.uk

ABSTRACT

Efficient desalination of salt water is an increasingly important issue, as the World Health Organization estimates that four billion people in 48 countries will not have access to sufficient fresh water by the year 2050. Aligned carbon nanotubes (CNTs) as part of a membrane have been found to possess properties that are potentially of use in filtration and desalination applications. Key characteristics observed are very fast mass flow rates (much faster than is predicted from the Hagen-Poiseuille equation), along with excellent salt rejection capabilities [1]. Although advances are being made in nano-fluid experimental work, there is still significant difficulty in experimental measurements for devices at this scale [2].

Although computationally intensive, non-equilibrium molecular dynamics (MD) simulation has recently been adopted as the numerical procedure of choice for nanoscale fluid dynamics due to its high level of detail and accuracy. Simulation of fluid transport through a CNT in MD requires generation of quasi-steady pressure-driven flow, which can be achieved by application of boundary conditions to the flow domain. Various boundary condition configurations exist, each with advantages and disadvantages, and selection often depends on the desired balance between computational efficiency and accurate representation of physical experiment.

In this work, we compare both new and commonly-used boundary conditions for generating pressure-driven flows through CNTs in MD simulation. Three systems are considered: a finite CNT membrane with streamwise periodicity and gravity forcing; a non-periodic finite CNT membrane with reservoir pressure control; and an infinite CNT with periodicity and gravity forcing. The first system is simple to implement in common MD codes, while the second system is more complex to implement, and the selection of control parameters is less straightforward. The required level of user-input for such a system was found to be largely dependent on selection of state controllers used in the reservoirs. A large pressure difference is required across the realistic membrane system reservoirs to compensate for large pressure losses at the entrance and exit of the nanotube. Despite a dramatic increase in computational efficiency, an infinite length CNT does not account for these significant inlet and outlet effects, suggesting that a much lower pressure gradient is required to achieve a specified mass flow rate. Observation of radial density profiles suggest that explicit control of the fluid inside the infinite nanotube may over-constrain the radial movement of the water molecules. Use of a finite membrane system, however, allows for control to be performed in the reservoirs only, resulting in natural flow development throughout the CNT.

[1] Corry, B. (2008) Designing carbon nanotube membranes for efficient water desalination. *Journal of Physical Chemistry B*, **112**, 1427–1434.

[2] Mattia, D. and Gogotsi, Y. (2008) Review: Static and dynamic behavior of liquids inside carbon nanotubes. *Microfluidics and Nanofluidics*, **5**, 289–305.

Velocity, Turbulence and Bed Shear Stress in Oscillatory Boundary Layer Flow over an Impermeable and a Permeable Bed: a comparative study

Kathryn Sparrow[1], Dubravka Pokrajac and Dominic A. van der A

1. Introduction

Oscillatory boundary layer flow has previously been investigated through detailed laboratory studies such as those by Sleath (1987) and Jensen et al. (1989). These experiments were conducted over a single layer of material which was glued to an otherwise impermeable bed. Natural beds, such as gravel and to some extent sand beds, are permeable and so these experiments are not entirely representative of the conditions to which they are often applied. This study intends to further the work conducted by Sleath and Jensen et al. by looking at the effect of bed permeability on some of the key flow parameters, such as velocity, turbulence and shear stress.

2. Experimental Setup

The study consists of a series of experiments over an impermeable bed and a permeable bed which have been undertaken in the Aberdeen Oscillatory Flow Tunnel (AOFT). The impermeable test bed consisted of a single layer of 9mm gravel which was fixed to the marine plywood base whilst the permeable test bed consisted of a 250mm deep bed of 9mm gravel which was also fixed. A total of 6 sinusoidal flow conditions were generated, these were split into two groups of three flows, whose Reynolds numbers were in the range of 7-10 x10^5 with a relative roughness of 35, and the other whose Reynolds numbers were in the range of 9-12x10^5 with a relative roughness of 40. The resulting horizontal and vertical flow velocities were recorded using Particle Image Velocimetry (PIV) and Laser Doppler Anemometry (LDA) techniques.

3. Results

The data has been phase and spatially-averaged over the measurement area to obtain the average flow velocities in both the horizontal direction (*u*) and the vertical direction (*w*). Comparisons between the permeable and impermeable bed will be performed to note any variations in the velocity, turbulence and shear stress. Figure 1.1 demonstrates the difference in velocity profiles over an impermeable and permeable bed.

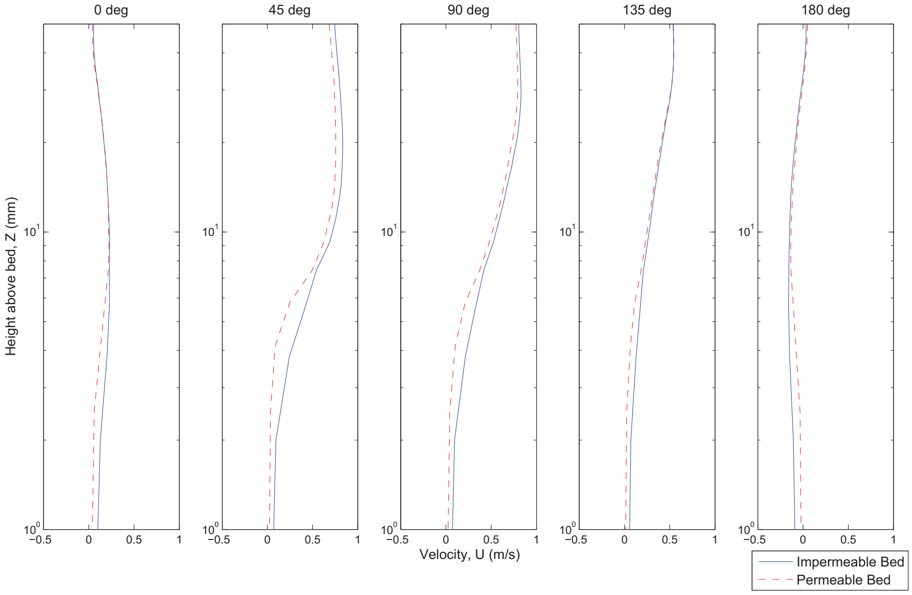


Figure 1.1. Comparison of the near bed velocity profile from the PIV data for half flow cycle over both an impermeable bed and a permeable bed when U_∞=0.9m/s.

References

Jensen, B. L., Sumer, B. M., & Fredsoe, J. (1989). Turbulent oscillatory boundary-layers at high reynolds-numbers. *Journal of Fluid Mechanics, 206*, 265-297.

Sleath, J. F. A. (1987). Turbulent oscillatory flow over rough beds. *Journal of Fluid Mechanics, 182*, 369-409.

[1] ksparrow@abdn.ac.uk, University of Aberdeen, School of Engineering, Aberdeen, AB24 3UE, Scotland.

Ocean dynamics modelling with a hybrid Lagrangian-Eulerian method

X. Perrot, D.G. Dritschel

University of St Andrews, Mathematical Institute North Haugh
St Andrews, KY16 9SS, Scotland

Because of the non-linearity in geophysical fluids equations we have to use numerical scheme to integrate them, even in idealized context. Here we proposed to explore more specifically the ocean dynamics with a continuously stratified QG model with buoyancy boundary condition at the surface. This model was preciously used in the oceanic context recently (*Perrot et al.*, 2010; *Roullet et al.*, 2011). We implement this model with the Combined Lagrangian Advection Method (CLAM) which is new a hybrid Lagrangian-Eulerian method using material contours together with an underlying grid (*Fontane and Dritschel*, 2009). After presenting the key point of this new modelling we will present diagnosis and comparison of our results with previous study to validate the model. Thus we will show different configuration giving an overview of the different possibility of forcing we could implement.

References

Fontane, J., and D. G. Dritschel (2009), The hypercasl algorithm: a new approach to the numerical simulation of geophysical flows., *J. Comput. Phys.*, **228**, 6411–6425.

Perrot, X., J. N. Reinaud, D. G. Dritschel, and X. Carton (2010), Homostrophic vortex interaction under external strain, in a coupled qg-sqg model., *Reg. Chaotic Dyn.*, **15**, 66–83.

Roullet, G., J. C. McWilliams, X. Capet, and M. J. Molemaker (2011), Properties of steady geostrophic turbulence with isopycnal outcropping., *J. Phys. Oceanogr.*, **42**, 18–38.

EXPERIMENTAL AND NUMERICAL THERMAL ANALYSIS OF A SMALL SCALE CENTEO TG40 COOLING SYSTEM

Ahmed A. Y. Al-Waaly[12], Phillip Dobson[2], Manosh C. Paul[1*]& Philipp Steinmann[3]

[12]System, Power & Energy, [2]Electronics & Nanoscale Engineering, School of Engineering, University of Glasgow, G12 8QQ

[3]Centeo Biosciences Ltd, Polaroid Building, Vale of Leven Industrial Estate, Dumbarton, G82 3PW

**Email: Manosh.Paul@glasgow.ac.uk*

ABSTRACT thermal analysis of a single chamber of *CENTEO's* TG40 cooling system is presented. The TG40 cooling system has five rows, each formed by a rectangular piece of copper, with eight chambers per row. A single chamber consists of two wells containing aqueous liquid. The concentration of NaCl is 0.5M in the small well and 1M in the big one. The range of working temperatures of the copper block is between 4-60°C. For this work, the first row is adjusted to 4°C and the last one is set to 20°C with 4°C temperature difference between two successive rows. This system is designed to keep a protein solution, which is placed inside the small well at a uniform temperature. A free convection is considered for thermal analysis inside the chamber, i.e. the driving force is the density variation due to a temperature difference. Therefore, a body force term is added to the Navier-Stokes equation to include this effect. A finite-element method is used as a numerical method for modelling which is included in COMSOL MULTYPHYSICS software. Different sizes of thermocouple are used (300 µm, and 80 µm) to measure the temperature inside the large well. Experimental results show there is a heating effect of using the thermocouple becuase the environment temperature is higher than the measured temperature of the well.

Modelling Lillgrund Wind Farm

Angus C.W. Creech[1,2], *Eoghan Maguire*[3], *Wolf-Gerrit Früh*[3]
[1] School of Engineering and Physical Sciences, Heriot-Watt University
[2] Institute for Energy Systems, School of Engineering, University of Edinburgh
[2] Eoghan Maguire, Vattenfall Research & Development, Edinburgh Office

Computational Fluid Dynamics, using the adaptive Finite-Element solver *Fluidity* in combination with a wind turbine model is used to model the flow through the Lillgrund Wind Farm, an offshore wind farm consisting of forty-eight 2.3MW turbine under a set of typical wind conditions.

The main aim was demonstrate the ability of the model to simulate the operation of the turbines and the development of the wakes behind the turbines, and compare the performance results to measurements from the wind farm.

One key issue in the design of large wind farms, especially those currently developed offshore is the reduced output from the wind farm due to the front row of turbines shading the down stream turbines. Actual observations of the relative performance of down stream turbines against the wind direction is shown in Figure 1. A typical CFD simulation is shown in Figure 2.

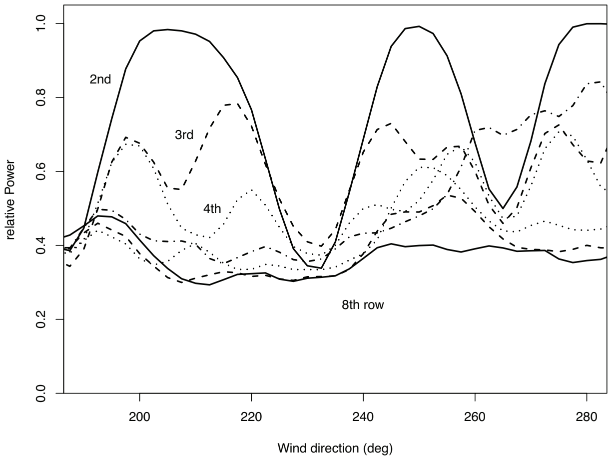


Figure 1: Relative power output of turbines downstream of first row; full alignment of turbines with wind at around 230°

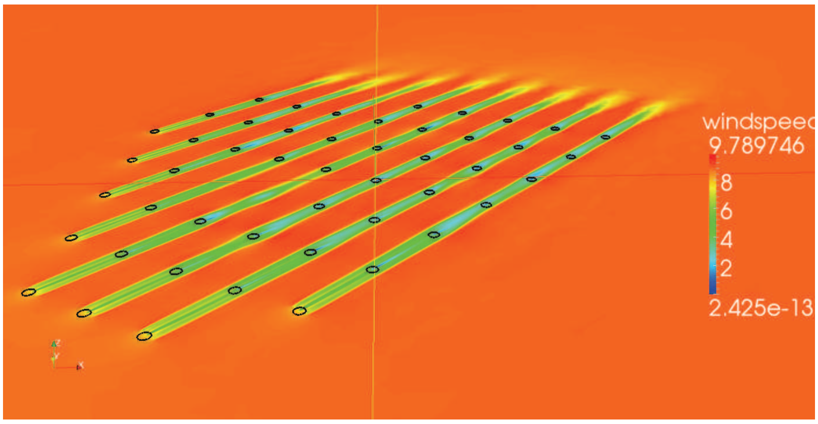


Figure 2: Velocity field in Wind Farm

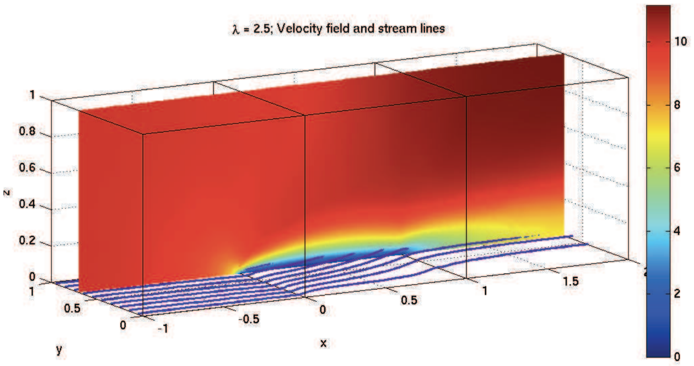
The role of Ekman suction in ‘rotational augmentation’

Wolf-Gerrit Früh
School of Engineering and Physical Sciences, Heriot-Watt University
w.g.fruh@hw.ac.uk

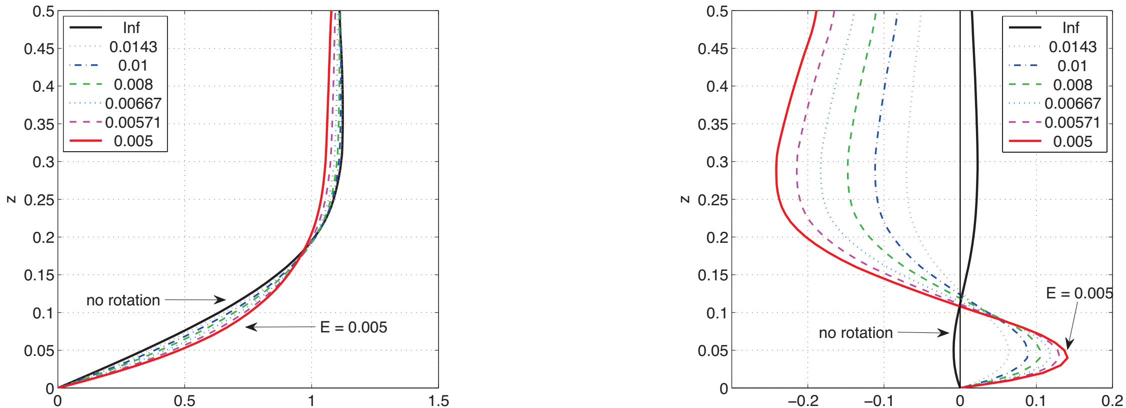
Rotational augmentation is a well-known phenomenon of rotating aerofoils on wind turbines and helicopters where the lift or torque is improved compared to that of stationary aerofoils.

This paper presents an analysis which is based on the concept of the combination of a developing boundary layer with an Ekman boundary layer using a 3D Finite-Element Analysis of the developing boundary layer over a flat or curved plate in a rotating frame of reference. A parametric analysis addresses the Reynolds number, the Ekman number and the speed ratio in turn.

The results demonstrate that the surface's rotation limits the boundary layer to the smaller of the two boundary layers, and that it induces a span-wise, or radial, which in turn leads to Ekman suction. This Ekman suction provides a mechanism to counteract an adverse pressure gradient and thus prevent stall or even enables the separated boundary layer to re-attach.



Typical velocity field in vertical section and streamlines starting in horizontal line.



a)

b)

Streamwise (a) and spanwise (b) velocities for various values of Ekman number

Topic: Influence of surface roughness and interface wettability on flows through nanochannels

Authors: Malgorzata Zimon, David Emerson, Jason Reese
STFC Daresbury Laboratory
University of Strathclyde

A very active field of research in fluid mechanics and material science is predicting the behaviour of Newtonian fluids flowing over porous media and surfaces with different wettabilities. In this work, molecular dynamics (MD) simulations were carried out to investigate the behaviour of water drops deposited on individual nanopillars with circular and square cross sections. Different wettabilities were applied by optimising the Lennard-Jones parameters to observe the ability of the surface to sustain the drop.

A further investigation examined the influence of solid surface roughness on fluid flow. The rough wall was formed by periodically spaced rectangular protrusions. The MD simulations were conducted to study the flow of water in a nanochannel with hydrophobic carbon walls, as well as the liquid argon flowing over a hydrophilic kryptonian corrugated surface.

The results of surface-roughness experiments are in agreement with previous studies, showing a decrease in the slip length near the rough surface. Reduction of the velocity values inside the cavities compared to the flow over smooth boundaries has been confirmed.

In the future work, further study on fluid-solid interaction and the influence of irregular topographies in nanochanels will be conducted in order to determine the best surface treatments for maximising slip. One of the potential applications involves smart coatings with controlled wettability.