Inaugural UK Fluids Conference

7-9 September 2016
Imperial College London

EPSRC
Engineering and Physical Sciences Research Council

Imperial College London

UNIVERSITY OF LEEDS

Inaugural UK Fluids Conference

Contents

Page 4-7: Conference schedule

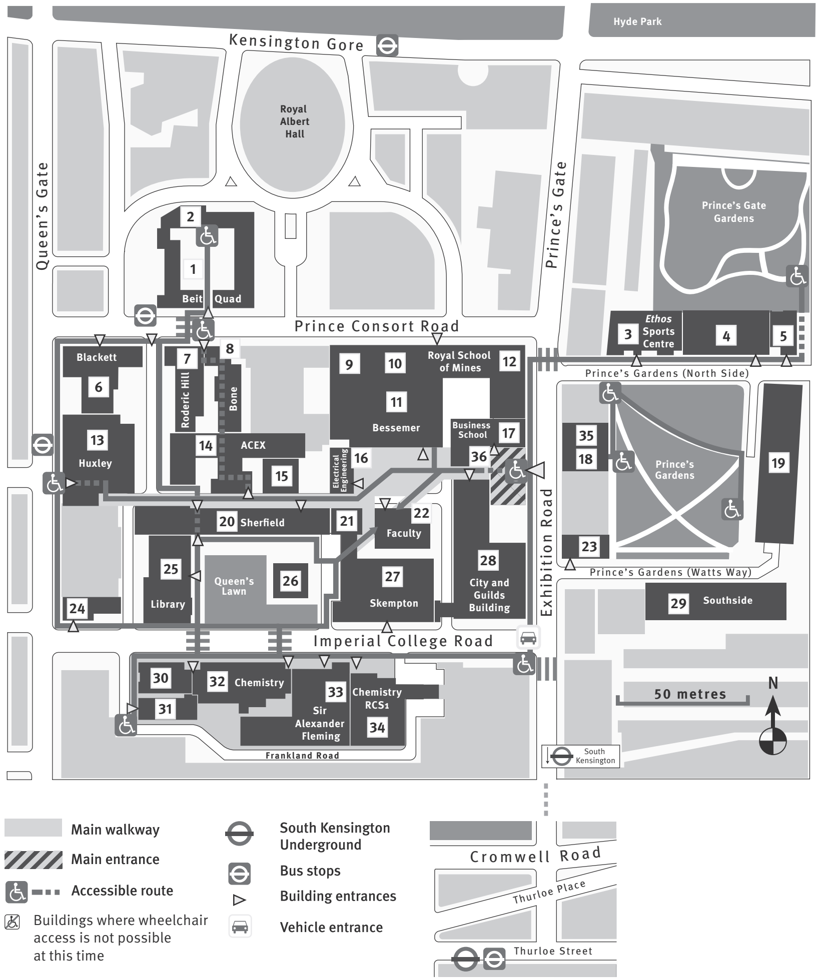
Page 8-11: Plenary speakers

Page 12-60: Sessions

Page 61-74: Posters

Imperial College London

South Kensington Campus



1 Beit Quadrangle
2 Imperial College Union
3 *Ethos* Sports Centre
4 Prince's Gdns, North Side
5 Weeks Hall
6 Blackett Laboratory
7 Roderic Hill Building
8 Bone Building
9 Royal School of Mines
10 Aston Webb
11 Bessemer Building
12 Goldsmiths Building
13 Huxley Building
14 ACE Extension
15 William Penney Laboratory
16 Electrical Engineering
17 Business School
18 53 Prince's Gate
19 Eastside
20 Sherfield Building Student Hub Conference Office
21 Grantham Institute – Climate Change and the Environment
22 Faculty Building
23 58 Prince's Gate
24 170 Queen's Gate
25 Central Library
26 Queen's Tower
27 Skempton Building
28 City and Guilds Building
29 Southside
30 Sir Ernst Chain Building – Wolfson Laboratories
31 Flowers Building
32 Chemistry Building
33 Sir Alexander Fleming Building
34 Chemistry RCS1
35 52 Prince's Gate
36 Alumni Visitor Centre

Wednesday 7 September

Venue: LT G16		Time	Venue: Seminar rooms 119 & 120	
Welcome	Welcome and introduction: Christos Vassilicos	1330-1345		
PLENARY TALK 1	Chair: Christos Vassilicos			
Andy Woods (Cambridge)	Ash flows, turbidites and blowouts; multiphase buoyancy-driven flows	1345-1430		
SESSION S1	Chair: Christos Vassilicos		SESSION S2	Chair: Chris Jones
Burridge (Cambridge)	The significance of engulfment in the process of turbulent entrainment by plumes	1440-1453	Shin (Edinburgh)	Self-similar states in accelerating and decelerating turbulent jets
Van Reeuwijk (ICL)	On a slippery slope	1453-1506	Jameel (Cardiff)	A DNS study of the liquid jet burst phenomena
Hughes (ICL)	Mixing in a lock-exchange gravity current	1506-1519	Witzke (City)	Evolution of forced shear flows in polytropic atmospheres
Horsley (Cambridge)	On two layer channel flow	1519-1532	Paul (ICL)	Genesis and spatial evolution of velocity gradients from production to decay regions of grid turbulence
Sher (Cambridge)	Gravity currents: entrainment, stratification and self-similarity	1532-1545	Stagg (Newcastle)	Classical-like wakes in superfluid Bose-Einstein condensates
Kelly (Leeds)	Turbidity current dynamics and their control on submarine channel development	1545-1558	Hammer (Southampton)	The influence of unsteady wakes generated by rotating bars on a low pressure turbine cascade
Jacobs (Southampton)	An improved quantitative measure of the tendency for volcanic ash plumes to form in water: implications for the deposition of marine ash beds	1558-1611	Jasak (Zagreb)	Harmonic balance method for incompressible turbomachinery applications
Break		1611-1640	Break	
SESSION S3	Chair: Christos Vassilicos		SESSION S4	Chair: Peter Jimack
Esteban (Southampton)	Large irregular particles falling in quiescent liquids	1640-1653	Muscutt (Southampton)	The mechanism of the thrust augmentation of in-line tandem flapping foils
Zainal Abidin (UCL)	Liquid-liquid flows past a bluff body in pipes	1653-1706	Baddoo (Cambridge)	Sound generation in aircraft engines
Bartholomew (ICL)	Modelling of turbulent gas-solid flows in the Eulerian framework	1706-1719	Baker (Cambridge)	Mechanisms of trailing-edge noise reduction
Morgan (ICL)	Coupled Chemistry-CFD simulations of nuclear meltdowns	1719-1732	Ioannou (ICL)	High-Fidelity Simulation of Turbulent Jets and Noise Predictions using Lighthill's Analogy
Xie (ICL)	The next-generation predictive tools for multiphase flows	1732-1745	Jamieson (Cambridge)	Experimental sensitivity analysis and control of thermoacoustic systems via a secondary heat source
Lee (ICL)	Incompressible active fluids: Emergence and universality	1745-1758	Xia (ICL)	Combining low order network modelling with incompressible flame LES for thermoacoustic instability in an industrial gas turbine combustor

Venue: Seminar rooms 121 & 122

POSTER SESSION and drinks reception	1805-1915

Thursday 8 September

Venue: LT G16		Time	Venue: Seminar rooms 119 & 120	
PLENARY TALK 2	Chair: Cath Noakes			
Janet Barlow (Reading)	The fluid dynamics of urban meteorology	0930-1015		
SESSION S5	Chair: Cath Noakes		SESSION S6	Chair: Chris Jones
King (Leeds)	The Silsoe Cubes : CFD modelling single sided and cross-ventilation in naturally ventilated buildings in isolation and in an array configuration	1025-1038	Mesgarnezhad (Newcastle)	Helicity of a small patch of quantum vorticity
Fuka (Southampton)	LES of scalar dispersion from localized sources in a regular array of buildings	1038-1051	Galantucci (Newcastle)	Quantum vortex dynamics and reconnections in trapped Bose-Einstein condensates
Boghi (Cranfield)	CFD simulations of methane diffusion from landfill	1051-1104	Jougla (St Andrews)	On the Energy transfer during jets and vortices formation and evolution in turbulent planetary atmospheres
Pennells (Leeds)	Model comparison of velocity perturbations through wind farms	1104-1117	Arredondo-Galeana (Edinburgh)	Vortex flow of yacht sails
Paul Lynch (Arup)	Designing cool stadiums for hot climates with CFD	1117-1130	Perez Torro (Southampton)	Flow characteristics of deep stalled aerofoils with a wavy leading edge
	Break	1130-1155		Break
SESSION S7	Chair: Andrew Ross		SESSION S8	Chair: Omar Matar
Williams S (Leeds)	Numerical and experimental simulation of atmospheric downbursts	1155-1208	Servini (UCL)	Roughing up Wings—A Promising Technique in Laminar Flow Control
Rodriguez Lopez (ICL)	Influence of the near field on the far- field development of single and multiscale wall-mounted porous fences	1208-1221	Sanchez (ICL)	Computational fluid-structure interaction analysis of membrane wings for Micro-Air-Vehicles
Wise (Inst. of High Performance Computing, Singapore)	Experimental and numerical assessment of turbulent flow through arrays of multi-scale, surface-mounted cubes	1221-1234	Evstafyeva (ICL)	Feedback control of an Ahmed Body flow exhibiting symmetry-breaking regimes
Higham (Sheffield)	Turbulent characteristics produced by a blockage of multi-scale emergent obstacles	1234-1247	Berk (Southampton)	Vectoring of parallel synthetic jets
Steiros (ICL)	Flow field characteristics and power draw of regular and fractal turbines	1247-1300	Brauner (ICL)	High-Fidelity simulations of dielectric barrier discharge plasma actuators in a turbulent channel flow
	Lunch	1300-1400		Lunch

Continued...

Thursday 8 September continued

Venue: LT G16		Time	Venue: Seminar rooms 119 & 120	
PLENARY TALK 3	Chair: Peter Schmid			
Neil Sandham (Southampton)	Compressibility and pressure fluctuations in turbulent flows	1400-1445		
SESSION S9	Chair: Peter Schmid		SESSION S10	Chair: Berend van Wachem
Ferreira (Southampton)	Turbulent flow past three-dimensional patches of roughness	1455-1508	Satya (Southampton)	Automated finite difference modelling on structured grids, and a variety of compute architectures
Van der Kindere (Southampton)	Surface-mounted ribs of varying length in turbulent flows	1508-1521	Evrard (ICL)	A novel curvature evaluation method for the Volume-of-Fluid method based on interface reconstruction
De Angelis (Cardiff)	Energy paths in high Reynolds number wall-turbulence	1521-1534	Abdol Azis (ICL)	Immersed boundary method with exact forcing for unstructured mesh and coupled solver
Bhagat (Cambridge)	Flow field created by a coherent turbulent water jet impinging on a vertical wall	1534-1547	Curran (ICL)	Large Eddy Simulation model for particle tracking in turbulent flows
Laskari (Southampton)	The time signature of the turbulent/non-turbulent interface over a turbulent boundary layer	1547-1600	Xiao CN (ICL)	A fully-coupled Navier-Stokes Solver framework applicable for flows of all speeds
Dogan (Southampton)	Interaction of large-scale free-stream turbulence with turbulent boundary layers	1600-1613	Yao (Sydney)	Validation of a correlation-based transition model for the Spalart-Allmaras turbulence model
de Giovanetti (ICL)	Scale-wise skin friction generation in turbulent channel flow	1613-1626	Li (Cardiff)	An efficient implementation of the CIP-CSL3 method
Break		1626-1650	Break	
SESSION S11	Chair: Omar Matar		SESSION S12	Chair: Andrew Ross
Hwang (ICL)	Traveling wave solutions of minimal large-scale structures in turbulent channel flow	1650-1703	Wang (Leeds)	The accurate and efficient numerical simulation of general fluid-structure interaction
Williams A (Manchester)	Three-dimensional boundary layer states forced by transpiration over short spanwise scales	1703-1716	Sewerin (ICL)	An LES-PBE-PDF approach with PBE-grid adaptivity for modelling particle formation in turbulent reacting flows
Zauner (Southampton)	Linear stability analysis of the boundary-layer over a high pressure turbine blade	1716-1729	Mackenzie (Manchester Met.)	A cut-cell based overset meshing approach for incompressible viscous flow
Robins (Leeds)	Stability Limits and Viscous Behaviour of the Strato-Rotational Instability	1729-1742	Al-Johani (Leeds)	Multilevel Solution Algorithms for a Numerical Model of Thin Film Flow
Atthanayake (Warwick)	On instability of vortices generated by a free-jet flow, in the presence of background	1742-1755	Alrehaili (Leeds)	Efficient iterative solution algorithms for numerical models of multiphase flow
Brewster (Cambridge)	Shape optimisation for hydrodynamic stability using adjoint based methods	1755-1808	Agnese (ICL)	Fitted ALE scheme for Two-Phase Navier-Stokes Flow
Xiao D (ICL)	Nonlinear optimal control of bypass transition in a boundary layer flow	1808-1821	Perrier (ICL)	Modelling of Corium Spreading in a Core Catcher

Venue: Queen's Tower Rooms, Sherfield Building	
Dinner	1830 - 2030

Friday 9 September

Venue: LT G16		Time	Venue: Seminar rooms 119 & 120	
SESSION S13	Chair: Berend van Wachem		SESSION S14	Chair: Peter Schmid
Uppal (ICL)	Structure build-up and evolution in the drying of sessile blood droplets	0930-0943	Lisicki (Cambridge)	Viscoelastic synchronisation
Krams (ICL)	Mechanosensitive signalling pathways detection during TCFA formation	0943-0956	Beeson-Jones (Cambridge)	Optimising the deployment of a time dependent polymer to minimise growth of viscous fingers
De Canio (Cambridge)	Swirling transition in *Drosophila* oocytes	0956-1009	Voulgaropoulos (UCL)	Shear-thinning effects on a coalescing drop
Reigh (Cambridge)	Squirming in a droplet	1009-1022	Roumpea (UCL)	Plug formation of non-Newtonian liquids in microchannels
Dauparas (Cambridge)	Flows Around Bacterial Swarms	1022-1035	Gorbatenko (Leeds)	Turbulent combustion and auto-ignition of alternative engine fuels
	Break	11035-1100		Break
SESSION S15	Chair: Peter Jimack		SESSION S16	Chair: Cath Noakes
Spelman (Cambridge)	An artificial cilium	1100-1113	Hendry (Newcastle)	Computational Fluid Dynamic Modelling of Benzene Abatement using Cryogenic Condensation
Cummins (Edinburgh)	The Stokes-flow parachute of the dandelion fruit	1113-1126	Petrie (Cambridge)	Nonlinear acoustics in a viscothermal boundary over an acoustic lining
Nesbitt (ICL)	Stability of active fluids with application to wound healing	1126-1139	Yang (ICL)	An analytical model for the acoustics of short circular holes
Kerr (City)	The onset of double-diffusive convection with evolving background gradients	1139-1152	Font Garcia (Southampton)	A two-dimensional model for three-dimensional symmetric flows
Goodfellow (Leeds)	Low Order Models of Layer Formation in Oscillatory Double-Diffusive Convection	1152-1205	Booker (Leeds)	Modelling internal wave focusing
McKenna (Belfast)	A reduced order modelling methodology for natural convection in a differentially heated enclosure with heated cylinder using the proper orthogonal decomposition while bypassing a Galerkin projection	1205-1218	Moulopoulou (Leeds)	Modelling the motion of waves in a "slice" of beach
PLENARY TALK 4	Chair: Peter Jimack			
Oliver Jensen (Manchester)	Grow with the flow: applications of viscoplasticity	1230-1315		
CLOSE	Closing remarks: Peter Jimack	1315-1330		
UK FLUIDS NETWORK EVENT	UK Fluids Network launch announcement and lunch reception	1330-1430		

Ash flows, turbidites and blowouts : multiphase buoyancy-driven flows

Andy Woods

Abstract

In this presentation, I will describe a number of multiphase geological and geophysical processes in which the effects of buoyancy, as well as the motion of particles or bubbles suspended in the flow, are central to the evolution of the flow. Important examples include turbidity currents, ash flows, volcanic plumes and oil blowouts.

I will explore how the buoyancy influences the motion and the mixing in these flows, and will present a series of experimental and mathematical models of the evolution of the flows, as the different phases separate, thereby changing the buoyancy and hence the evolution of the flow.

Biography

Prof Andy Woods has been the BP Professor at the BP Institute for Multiphase Flow at the University of Cambridge since 2000, where he has been engaged in research into (a) single and multi-phase turbulent plumes and gravity currents, with applications to volcanic and environmental processes, and (b) problems of flow in porous media including buoyancy driven flows, dispersion, and the effects of rock heterogeneity, with applications for CO_2 sequestration, geothermal power and enhanced oil recovery. Prior to the BP Institute, we was Professor of Applied Mathematics at University of Bristol for 3 years and, before this he was a lecturer in the Institute for Theoretical Geophysics in DAMTP, Cambridge. His PhD, at DAMTP in University of Cambridge, focussed on Geological Fluid Dynamics.

The Fluid Dynamics of Urban Meteorology

Janet Barlow

Abstract

Whilst the weather might seem to consist of large scale vortex and jet dynamics swirling somewhere above our heads, the local flows created by buildings in urban areas have immediate impact on us (!) and a range of applications, such as sustainable building design, air quality modelling and wind engineering. The interaction between bluff buildings and flow is complex, as wakes from individual elements overlap or flow is channelled along streets. The roughness sub-layer flows that dominate within 2 to 3 building heights of the ground have similarities and differences to the better-known turbulent flows encountered in vegetation canopies. At the scale of the city, the heterogeneity of the surface causes overlapping internal boundary layers, which challenges modelling and measurement efforts. This talk will give an overview of current understanding of one of the most complex boundary layers.

Biography

Professor Janet Barlow has research interests in urban meteorology and renewable energy. She graduated with a degree in Physics with German from UMIST in 1994 followed by an MSc in Meteorology in 1995 and PhD in 2000 from Reading, where she has remained ever since. She has conducted well over a decade of observations of London's atmosphere, as well as Birmingham and Manchester, and has completed idealised wind tunnel and field experiments to examine turbulent processes around buildings. She has been a member of the Board for the International Association of Urban Climate, the American Meteorological Society Board of the Urban Environment, and is currently a member of the UK Wind Engineering Society Steering Committee.

Compressibility and pressure fluctuations in turbulent flows

Neil Sandham

Abstract

Compressibility effects are present in many practical turbulent flows, ranging from shock-wave/boundary-layer interactions on the wings of aircraft operating in the transonic flight regime to supersonic and hypersonic engine intake flows. Besides shock wave interactions, compressible flows have additional dilatational effects and, due to the finite sound speed, pressure fluctuations are localized and modified relative to incompressible turbulent flows. Such changes can be highly significant, for example the growth rates of mixing layers and turbulent spots are reduced by factors of more than three at high Mach number. In this talk we will review some of the basic effects of compressibility on canonical turbulent flows and rationalise the different effects of Mach number in different flows using flow instability concepts. We will then turn our attention to a fully three-dimensional problem of shock-wave/boundary-layer interaction in a closed duct, considering direct effects of shock waves, due to their penetration into the outer part of the boundary layer, as well as indirect effects due to the high convective Mach number and shock-induced shear-layer curvature.

Biography

Neil Sandham has been Professor of Aerospace Engineering at the University of Southampton since January 1999. Previously he was at Stanford University (PhD 1989), DLR Göttingen and Queen Mary and Westfield College. His research uses direct numerical simulations of the governing equations of fluid motion to study transitional and turbulent fluid flows, with current projects on transitional separation bubbles, rough surface flows and shock-wave/turbulence interactions. He currently leads the UK Turbulence Consortium.

Grow with the flow: applications of viscoplasticity

Oliver Jensen

Abstract

Viscoplastic materials sit at the interface of solid and fluid mechanics, undergoing irreversible deformations characteristic of a viscous fluid only when subject to a sufficiently large imposed stress or strain. I will explain how plant growth can be interpreted and modelled as a viscoplastic process, involving highly regulated expansion and reshaping of multicellular tissues in response to environmental conditions. I will also present an everyday example of how viscoplastic deformations are exploited to engineer decorative shape changes in a man-made material.

Biography

Oliver Jensen is the Sir Horace Lamb Professor in the School of Mathematics at the University of Manchester. He has held previous positions at the Universities of Cambridge, Newcastle and Nottingham. His research interests span continuum mechanics, transport processes, multiscale methods and mathematical modelling in medicine and biology.

The significance of engulfment in the process of turbulent entrainment by plumes

H.C Burridge[1, 2], D.A. Parker[2], E.S. Kruger[2], J.L. Partridge[2] and P.F. Linden[2]

[1]Dept. of Civil and Environmental Engineering, Imperial College London, [2]Dept. of Applied Mathematics and Theoretical Physics, University of Cambridge

Turbulent plumes are a canonical flow of significance to the environment and industry. By making particle image velocimetry measurements whilst simultaneously detecting the scalar edge of high Peclet number saline plumes we investigate the significance of engulfment within the process of turbulent entrainment. We show that a significant proportion of the vertical mass transport occurs within the irrotational ambient environment, i.e. outside the plume. This vertical mass transport within the ambient typically occurs in the vertical space between turbulent coherent structures (eddies) at the plume edge. The vertical momentum imparted on these pockets of ambient fluid enable it to be easily engulfed into the plume. We argue that turbulent entrainment is initiated by the engulfment of this (relatively high momentum) ambient fluid which occurs at the largest scales within the plume. This engulfed fluid is then stretched, and vorticity imparted, at the plume edge; after which, at far smaller scales, irreversible molecular mixing takes place. Hence, unlike a number of studies which conclude `nibbling' at the interface dominates entrainment, we conclude that engulfment should be viewed as the initial, and indeed a vital, stage of the process of turbulent entrainment by plumes.

On a slippery slope

M. van Reeuwijk[1], M. Holzner[2], C. Caulfield[3] and H. Jonker[4]

[1]Dept. of Civil and Environmental Engineering, Imperial College London, [2]Institute of Environmental Engineering, ETH Zurich, [3]BP Institute and DAMPT, University of Cambridge, [4]Faculty of Geosciences, Delft University of Technology

We study the mixing and entrainment in inclined temporal gravity currents using Direct Numerical Simulation. The Richardson number is varied by changing the slope. We find that the entrainment is dominated by processes occurring in the outer layer. We decompose the entrainment into various physical processes using a recently developed method (Van Reeuwijk and Craske, JFM 2015) and find that 1) entrainment is dominated by turbulence production; and 2) the direct influence of buoyancy on entrainment is small. We find that the entrainment law differs substantially from the model proposed by Ellison and Turner (1959) correlation and offer suggestions into what may cause the differences.
By parameterising the turbulence in the flow, we develop an entrainment law which is in good agreement with the simulations.

Mixing in a lock-exchange gravity current

G. O. Hughes[1] and P. F. Linden[2]

[1]Dept. of Civil and Environmental Engineering, Imperial College London, [2]Dept. of Applied Mathematics and Theoretical Physics, University of Cambridge

Gravity currents are a common buoyancy-driven flow, arising when fluid of one density propagates along a boundary into an environment of different density. Turbulence and mixing is an obvious feature of these currents in environmental, geophysical and industrial settings, but is typically overlooked in developing theoretical models and predictions of the flow.

We present here measurements of mixing in a gravity current created in a long channel by suddenly removing a lock that separates two miscible fluids of different density. Experiments were designed to reach particularly high Reynolds number (of order 10^5, based on the current depth), at which mixing is no longer affected significantly by viscosity. Under these conditions, the subsequent density profiles in the channel become fully self-similar and the proportion of initial (available) potential energy supplied to the flow that results in mixing is found to asymptote to 8%. We develop a model of the mixing in the exchange flow that is based on idealised mean profiles of velocity and density that become stable to stratified shear instability. The associated energy budget yields predictions that are consistent with the measured amount of mixing.

On two layer channel flow

M. Horsley, C. Caulfield and A.W. Woods

University of Cambridge

We have developed a model to describe the evolution of a two layer density interface connecting two reservoirs of different stratification through a turbulent channel flow. A series of steady state solutions for the exchange flow are developed subject to different boundary forcing, and the stability of these solutions are then discussed. This leads to prediction of wave steepening and the formation of travelling shocks which migrate along the channel and may be responsible for mixing. We consider the relevance of the model for mixing in hydraulic exchange flows in long channels.

Gravity currents: entrainment, stratification and self-similarity

D. Sher and A.W. Woods

BP Institute, University of Cambridge

We present new experiments of the motion of a turbulent gravity current produced by the rapid release of a finite volume of dense aqueous solution from a lock of length L into a channel x > 0 filled with a finite depth, H, of fresh water. Using light attenuation we measure the mixing and evolving density of the flow, and, using dye studies, we follow the motion of the current and the ambient fluid. After the fluid has slumped to the base of the tank, there are two phases of the flow. In the first regime, the main part of head of the current retains its original density and the flow travels with a constant speed. In a second regime the position of the head increases with time as $x_n \approx 1.7B^{1/3}t^{2/3}$ while the depth-averaged reduced gravity in the head decreases through mixing with the ambient fluid according to the relation $g'_n \approx 4.6H^{-1}B^{2/3}t^{-2/3}$. Dye studies show that fluid with the original density continues to reach the front of the current, at a speed which we estimate to be approximately 1.35 times that of the front. We derive a new class of self-similar solution which models the lateral structure of the flow.

Turbidity current dynamics and their control on submarine channel development

R. Kelly[1], R. Dorrell[2], A. Burns[3] and W. McCaffrey[2]

[1]EPSRC CDT in Fluid Dynamics, University of Leeds, [2]School of Earth and Environment, University of Leeds, [3]School of Chemical and Process Engineering, University of Leeds

Turbidity currents transport vast amounts (10s to 100s of cubic km) of sediment to the deep ocean. Their main conduit are seafloor channels. However, the hydrodynamic-morphodynamic relationship between turbidity currents and their containing channels, involving several different feedback systems, is poorly understood. Processes dictating the evolution of the channel (deposition and erosion occurring over long timescales) and the dynamics of individual currents (forcing from the channel over relatively short timescales) are interlinked. However the scales of co-dependency of flow and topography are unknown. It is therefore unapparent to what extent flow dynamics and the development of a channel are pre-determined. To answer this, we must consider several factors, perhaps foremost being the relative spatial and temporal dominance of both the current and the channel. Both laboratory and numerical results, focussing on basal processes, will be presented. By analysing the changing interaction between a variety of channel and flow types, the relative dominance of each can be assessed.

An improved quantitative measure of the tendency for volcanic ash plumes to form in water: implications for the deposition of marine ash beds

C.T. Jacobs[1], T.J. Goldin[2], G.S. Collins[1], M.D.Piggott[1], S.C. Kramer[1], H.J. Melosh[3], C.R.G. Wilson[4] and P.A. Allison[1]

[1]Dept. of Civil and Environmental Engineering, Imperial College London, [2]Nature Geoscience, Nature Publishing Group, [3]Dept. of Earth, Atmospheric and Planetary Sciences, Purdue University, [4]Lamont-Doherty Earth Observatory, Columbia University

Explosive volcanic eruptions produce vast quantities of searing hot gas and ash, which poses extreme hazards; recent eruptions have resulted in the closure of airports, damage to infrastructure and natural habitats, and loss of life. Predicting and mitigating such events requires knowledge of their frequency, intensity and ash distribution. The layers of ash deposited by past eruptions can uncover a wealth of information pertaining to this. Ash particles either settle slowly and individually, or rapidly and collectively as a plume/density current. Since the settling rate has implications on the accuracy of information inferred from the layer, one needs to determine whether plumes were likely to have formed. Existing theoretical formulae for doing so assume that plumes obey Stokes' law, which is not the case for subaqueous ash plumes. This work presents a new formula for measuring the tendency for subaqueous ash plumes to form. A parameter sweep is undertaken, and a suite of high-resolution, computationally-intensive simulations is produced to evaluate the effectiveness of this new formula. By taking the inertia-dominated nature of plumes into account, the new formula allows settling times to be more accurately and practically determined, thereby permitting improved interpretation of volcanic ash layers.

Self-similar states in accelerating and decelerating turbulent jets

D. Shin[1] and E.S. Richardson[2]

[1]School of Engineering, University of Edinburgh, [2]Faculty of Engineering and the Environment, University of Southampton

The effects of unsteady injection on the mixing in turbulent gaseous jets are investigated using Direct Numerical Simulation (DNS) and a theoretical analysis. The simulated turbulent jets are subjected to a starting transient, statistically stationary injection, and a stopping transient with Reynolds number of 7,200. This study is relevant to the split-injection of compression-ignition engines where mixing rates and fuel distribution control the rate of heat release and pollutant formation. The theoretical analysis identifies various properties of self-similar unsteady jet. The DNS of the decelerating jet showed that some of velocity profiles reach new self-similar states after a transient time. Especially, the radial velocity profile is significantly different from that of the steady jet, and the difference explains the large increase of the entrainment rate observed in diesel engines at the end of fuel injection. Furthermore, the spatial/temporal evolution of centreline velocity agrees well with the theoretical prediction. This agreement implies that the flow field resulting from general unsteady fuel injection profiles might be characterized by a simple one-dimensional model to provides insight into the mixing of fuel and air in compression ignition (e.g. diesel) engines where the fuel can be introduced by multiple pulses.

A DNS study of the liquid jet burst phenomena

A. Jameel, P. Bowen and K. Yokoi

Cardiff School of Mechanical Engineering, Cardiff University

Generally, liquid jet experiences four distinguished breakup regimes: the Rayleigh breakup regime, the first wind-induced breakup, the second wind-induced breakup, and the atomization. However, there can be found different laminar breakup phenomena to which those regimes do not apply successfully. Rupe [1] experimentally and Pan [2] numerically found that a laminar jet with a relatively high injection speed at Re=2200 could be more unstable and break up in an extremely violent fashion (liquid jet burst) much sooner than a fully developed turbulent jet.

The DNS direct numerical simulation [3] of multiphase flows is investigated to characterize the specific case of a high velocity of a free laminar liquid jet. The models and numerical methods rely on the incompressible Navier-Stokes equations, a Finite Volume approach and Volume of fluid methods coupled with level set method [4] for interface tracking by using the CFD framework OpenFOAM [5]. A fully developed parabolic velocity profile is implemented at the inlet boundary. Focusing mainly on the physical mechanisms liquid jet bursting, the following conclusions are noted:

- The inlet boundary profile has a significant effect on the burst phenomena.
- Significant liquid vortex structures ensue, enhancing the dynamics and instability of the liquid jet leading up to the burst phenomena.
- Strong transverse and axial gas vortices generated upstream produce eddies in the gas-liquid interfacial region which are considered influential in enhancing downstream instabilities and an important influence giving rise to the burst phenomena.

References

[1] Jack H. Rupe, Jet Propulsion Laboratory –California Institute Of Technology, Jan 15, 1962.
[2] Pan, Yu. and Kazuhiko, Suga., *Physics of fluids* 18:052101-1 (2006).
[3] F. Dos Santos and L. Le Moyne, *Oil & Gas Science and Technology – Rev. IFP Energies nouvelles*, Vol. 66 (2011), No. 5, pp. 801-822.
[4] Albadawi A, Donoghue DB, Robinson AJ, Murray DB, Delauré YMC. Influence of surface tension implementation in volume of fluid and coupled volume of fluid with level set methods for bubble growth and detachment. Int J Multiphase Flow 2013; 53:11–8.
[5] *OpenFOAM Foundation and OpenCFD Ltd* [online]. [Copyright © 2011-2014]. http://www.openfoam.org

Evolution of forced shear flows in polytropic atmospheres

V. Witzke[1], L.J. Silvers[1] and B.Favier[1, 2]

[1]Dept. of Mathematics, City, University of London,
[2]CNRS, Aix-Marseille University

Shear flows are ubiquitous in astrophysical objects including planetary and stellar interiors, where their dynamics can have significant impact on thermal transport and chemical processes. Here three-dimensional direct numerical calculations of unstable shear flows are performed to study the long-time evolution of the resulting turbulent motions in a compressible polytropic fluid. At present, there exist several different forcing methods to sustain large-scale shear flows in local models.

Here we examine and compare various methods used in the literature. These techniques are compared during the exponential growth of a shear flow instability, such as the Kelvin-Helmholtz (KH) instability, and the subsequent non-linear evolution. For this we used the concept of available potential energy to analyse the energy budget of the system. The forced shear flows that we consider here can be used to model flows in stellar interiors. Hence, we examine the effect of varying the Prandtl number and thermal diffusivity on the non-linear evolution of an unstable shear flow. Finally, preliminary calculations of a low Péclet number instability with large Richardson number are briefly discussed.

Genesis and spatial evolution of velocity gradients from production to decay regions of grid turbulence

I.Paul, G. Papadakis and J.C. Vassilicos

Dept. of Aeronautics, Imperial College London

The dynamics of velocity gradients for a spatially developing turbulence generated by a single square grid at low inlet *Reynolds* number (*Re*) is explored using direct numerical simulation. We study the generation and spatial evolution of symmetric and anti-symmetric parts of the velocity gradient tensor using the transport equations of mean strain-product and mean enstrophy respectively. Detailed analysis of small scale statistics in the turbulent production and decay regions demonstrates that some universal turbulent flow features can be observed even at low *Re* values. In the lee of the grid, strain is the first velocity gradient generated by the action of pressure Hessian, is transported by turbulent fluctuations, and strain self-amplication is activated a little later. Further downstream, vorticity from the bars is brought towards the center line, and through the interaction with strain leads to production of enstrophy. Vortex stretching dominates vortex compression from the onset point of -5/3 slope in the energy spectrum. Yet, Q-R diagram remains undeveloped in the production region, and both the intermediate and extensive strain-rate eigenvectors align with the vorticity vector there. The usual signatures of velocity gradients are detected only in the decay region.

Classical-like wakes in superfluid Bose-Einstein condensates

G.W. Stagg, N.G. Parker and C.F. Barenghi

Joint Quantum Centre Durham-Newcastle, Newcastle University

We show that an elliptical obstacle moving through a superfluid (in the form of a weakly-interacting Bose-Einstein condensate) generates wakes of quantum vortices which resemble those of classical viscous flow past a cylinder or sphere. Initial steady symmetric wakes, similar to those observed in classical flow at low Reynolds number, lose their symmetry and form clusters of like-signed vortices, in analogy to the classical Benard-von Karman vortex street.

The key ingredient to produce classical-like wakes is that vortices are generated at a sufficiently high rate that they undergo strong interactions with their neighbours (rather than being swept away). The role of ellipticity is to facilitate the interaction of the vortices and to reduce the critical velocity for vortex nucleation. Our findings, demonstrated numerically in both two and three dimensions, confirm the intuition that a sufficiently large number of quanta of circulation reproduce classical physics.

The effects which we describe (dependence of the critical velocity and cluster size on the obstacle’s size, velocity and ellipticity) are also relevant to the motion of objects (such as vibrating wires, grids and forks) in superfluid helium, as the obstacle’s ellipticity plays a role which is analogous to rough boundaries.

The Influence of unsteady wakes generated by rotating bars on a low pressure turbine cascade

F. Hammer[1], R.D.Sandberg[2] and N.D. Sandham[1]

[1]Faculty of Engineering and the Environment, University of Southampton, [2]Dept. of Mechanical Engineering, University of Melbourne

Large Eddy Simulations were carried out in order to investigate the influence of unsteady wakes on the loss mechanisms of the T106 linear LPT cascade. The unsteady wakes entering the cascade, usually created by preceding rotor and stator rows within a turbomachine, are generated by vertically sliding bars upstream of the blades. However, unlike actual rotor and stator blades, these bars do not generate circulation due to their symmetrical shape. In order to be able to investigate the effects of more realistic incoming wakes, and additionally, without increasing the numerical costs by simulating a full turbine stage, the Magnus effect is exploited. This means that the upstream bars are set into rotation around their axis, and thus generating circulation. Four different rotation rates were simulated yielding different drag and lift coefficients. The isentropic Reynolds number, based on the blade chord length C, and the reduced frequency were chosen to be $Re_{is} = 100,000$ and $F_{red} = 0.61$. Increasing the rotation rates of the bars results in an increase of the inlet flow angle by up to 6 degrees. For the two highest rotation rates a separation bubble occurs on the suction surface at the leading edge, which further increases the losses of the LPT.

Harmonic balance method for incompressible turbomachinery applications

G. Cvijeti´c and H. Jasak

Faculty of Mechanical Engineering and Naval Architecture, Zagreb

Harmonic Balance method for non–linear, temporally–periodic turbulent incompressible flows is presented in this work. Assumption of time periodic flow allows us to formulate $2n + 1$ coupled steady state problems using Fourier series expansion. Solving $2n + 1$ equations yields quasi–steady state flow fields with transient features which depend on the number of harmonics, n. The main advantage of the Harmonic Balance is CPU time decrease compared to transient simulation, which shall be presented in detail. Higher number of harmonics used will increase accuracy, but also prolong the simulation time. The implementation is carried out in a second–order accurate, polyhedral Finite Volume framework developed within foam–extend, a community driven fork of the OpenFOAM toolkit. The Harmonic Balance method is validated on canonical test cases and will be applied on turbomachinery test cases. 2D and 3D pumps and turbines will be simulated and compared with available experimental data. The comparison will also include CFD results obtained with Multiple Reference Frame (MRF) steady state approach, conventional transient simulation and Harmonic Balance. To investigate the potential of the Harmonic Balance method, global pump parameters shall be compared. Furthermore, local flow features such as pressure distribution on blades, flow instability and detailed flow features will also be examined.

Large irregular particles falling in quiescent liquids

L.B. Esteban, J.S. Shrimpton and B. Ganapathisubramanii

Faculty of Engineering and the Environment, University of Southampton

The physics of dispersed multiphase flow are complex due to the range of scales present, ranging from the thickness of the boundary layer around a single particle to the domain length scale. For non-spherical particles the shape of the boundary is a function of the particle orientation which is often chaotic even in a quiescent fluid. In the present paper the free-fall behaviour of quasi-planar particles in quiescent liquid are observed with respect to particle dispersion, fall pattern and fall velocity.

Individual particles are dropped in a vertical Plexiglas tank and their trajectories recorded. The purpose of this experiment is to gain understanding of the physics present in a waste separation device that relies on gravitational settling in the turbulent flow.

The variation in the particle trajectory is generally accepted to be dependent upon the stability of the pressure forces in the wake of the particle, which are in turn dependent on the dimensionless particle inertia and the Reynolds number. Thus, different sets of particles are manufacture to identify the effect of particle inertia on their trajectory. Results are presented and compared with analogous experiments found in the literature.

Liquid-liquid flows past a bluff body in pipes

Mohd.I.I.Z. Abidin, K.H. Park, M. Chinaud and P. Angeli

Dept. of Chemical Engineering, University College London

Flow past a bluff body above a critical Reynolds number gives rise to a time-periodic regime characterized by an alternate shedding of vortices behind the bluff body and a von Kármán vortex street. In this work, the generation of instabilities behind a bluff body bounded by a pipe wall and its effects on flow pattern transitions in two-phase oil-water flows from separated to dispersed flow are studied experimentally and numerically. The work is conducted in a horizontal 37 mm ID acrylic pipe at various flow conditions. The bluff body is a cylinder with a diameter of 5 mm and is located in the water phase, transverse to the flow direction. A high-speed camera is used for flow visualization while the velocity profiles in the water phase are determined by high-speed Particle Image Velocimetry (PIV). Numerical studies were conducted with a commercial code FLUENT to study the effects of pipe curvature and bluff body position. Results are compared with experimental data.

Modelling of turbulent gas-solid flows in the eulerian framework

P. Bartholomew, F. Denner, A. Marquis and B.G.M. van Wachem

Department of Mechanical Engineering, Imperial College London

Turbulent gas-solid flows can be found in a range of engineering, scientific and environmental applications; the ability to accurately predict their behaviours is therefore a vital tool in design and analysis. The large number of particles in a system of practical interest leads to the use of continuum models, derived by analogy with kinetic theory of gases, being frequently used in the context of Reynolds Averaged Navier Stokes. To capture more detailed turbulent dynamics, it is desirable to use Large Eddy Simulation (LES) as the modelling framework for turbulence. The use of sub-grid scale closures developed for single-phase flows is questionable and a novel LES model for turbulent gas-solid flows is proposed. This is motivated by the observation that most turbulence models are based on the assumption that at small scales, fluctuations are dissipated as heat. A turbulence model for dispersed gas-solid flow should capture the exchange of fluctuating energy between the bulk solid and the fluid, and the bulk solid and individual particles as '*granular heat*'. The model effectiveness is evaluated by application to a range of gas-solid flows of practical interest including a circulating fluidised bed riser.

* Funding provided by the EPSRC is gratefully acknowledged.

Coupled chemistry-CFD simulations of nuclear meltdowns

G. Morgan[1], V. Badalassi[1,2], M. Eaton[1] and B.G.M. van Wachem[1]

[1]Department of Mechanical Engineering, Imperial College London, [2]Rolls-Royce plc, Derbyshire

Nuclear reactor designs, safety systems and procedures have come under renewed scrutiny in the wake of the 2011 disaster at the Fukushima Daiichi Nuclear Power Plant in Japan. Complex multiphase reactive flows govern the progression of the accident from beginning to end.

This work aims to couple chemical models to computational fluid dynamics to allow three-dimensional modelling of the molten mixture of nuclear fuel, cladding and structural materials known as "corium". Such models will enhance the testing of safety systems using realistic physics and chemistry, leading to safer reactor designs and emergency procedures in future.

We present a reactive flow model of the oxidation of zircaloy cladding by steam, with surface reactions and coupled heat-and-mass transfer based on relations developed by Spalding. This system is relevant to the early stages of accident progression yet encapsulates the majority of the thermo-chemistry required to tackle the more complex late-stage accident scenarios.

* Funding provided by EPSRC and Rolls-Royce is gratefully acknowledged.

The next-generation predictive tools for multiphase flows

Z. Xie[1,2], D. Pavlidis[2], C. C. Pain[2] and O. K. Matar[1]

[1]Dept. of Chemical Engineering, Imperial College London, [2]Dept. of Earth Science and Engineering, Imperial College London

The ability to predict the behaviour of multiphase flows accurately, reliably, and efficiently addresses a major challenge of global economic, scientific, and societal importance. These flows are central to micro-fluidics, virtually every processing and manufacturing technology, oil-and-gas, nuclear, and biomedical applications. Significant advances have been made in the numerical procedures to simulate these flows; examples of these include the use of Large Eddy Simulations to simulate turbulence, and interface-capturing or tracking techniques to deal with the free surface. These codes have made progress in simulating the interaction of a turbulent flow field with an interface, however, there remains a large gap between what is achievable computationally and 'real-life' systems; the latter are beyond what can be addressed with current methods. As a result, the use of empirical correlations to bridge this gap remains the norm. We will present the latest on the modelling framework that we are currently developing as part of the *Multi-scale Examination of MultiPHase physIcs in flowS* (MEMPHIS) programme in order to minimise the use of correlations and shift towards the use of numerical simulations as a truly predictive tool that can be used as a sound basis for design. The framework features model-driven experimentation, massively-parallelisable interface-capturing methods, 3D, adaptive, unstructured meshes, and sophisticated multi-scale, multi-physics models.

*Support from the Engineering & Physical Sciences Research Council, UK (grant no. EP/K003976/1) is gratefully acknowledged.

Incompressible active fluids: emergence and universality

L. Chen[1], J. Toner[2] and C.F. Lee[3]

[1]College of Science, China University of Mining and Technology, [2]Dept. of Physics and Institute of Theoretical Science, University of Oregon, [3]Dept. of Bioengineering, Imperial College London

The incompressible Navier-Stokes equation describes the motion of a collection of interacting particles in the incompressible limit. What if each particle can now move on its own, or in other words every fluid element can generate its own forces? The situation is akin to a flock of birds or a collection of motile cells constituting a tissue. Indeed, in the hydrodynamic limit, the motion of these, and many other biological systems are described by the 'active' version of the Navier-Stokes equation. Because of the internally generated stresses, active fluids can move on their own without an externally imposed pressure gradient, as for instance manifested by the flocking behaviour in birds. Incompressible active fluids (**IAF**) are rich in physics and in this presentation, I will show that in the flocking phase, IAF in 2D can be mapped on the classic Kardar-Parisi-Zhang model that describes many surface growth models [1], and at the onset of flocking, i.e., the critical transition point, the behaviour of the system constitutes a new universality class in non-equilibrium physics [2].

References:

[1] L. Chen, C. F. Lee, and J. Toner, "Birds, magnets, soap, and sandblasting: surprising connections to incompressible polar active fluids in 2D," arXiv:1601.01924, 2016.
[2] L. Chen, J. Toner, and C. F. Lee, "Critical phenomenon of the order-disorder transition in incompressible active fluids," New Journal of Phyics, 17, 042002, 2015.

The mechanism of the thrust augmentation of in-line tandem flapping foils

L. E. Muscutt, G.D. Weymouth and B. Ganapathisubramani

University of Southampton

The propulsive performance of a pair of tandem flapping foils is highly dependent on the spacing and phasing between them. Large increases in thrust and efficiency of the hind foil are possible but only under certain combinations of parameters. Two-dimensional computer simulations of tandem and single foils oscillating in heave and pitch at a Reynolds number of 10,000 are performed over a broad and dense parameter space, allowing the effects of inter-foil spacing and phasing to be investigated over a range of Strouhal numbers. Results indicate that the hind foil can produce up to twice the thrust of a single foil depending on its spacing and phasing with respect to the fore foil. Examination of instantaneous and cycle-averaged flowfields indicate that high thrust occurs when the hind foil weaves in between the vortices that have been shed by the fore foil, and low thrust occurs when the hind foil intercepts these vortices. A simple quasi-steady model elucidates the mechanism behind the observed thrust augmentation of the hind foil, and shows that thrust augmentation is primarily determined through modification of the instantaneous angle of attack of the hind foil due to the vortex street established by the fore foil.

Sound generation in aircraft engines

P. Baddoo

University of Cambridge

The problem of sound pollution near airports is of great industrial and social significance. Aircraft noise represents a costly and inefficient use of energy for airlines and has a negative impact on local communities. One of the key sources of sound produced by the modern turbofan engines used by aircraft is a result of the interaction between wakes produced by the rotors and the stators downstream of the flow. The engine stators are typically modelled as a cascade of staggered aerofoils with finite chord length. In this presentation previous analysis is extended to include a small amount of thickness and camber in the aerofoil geometry. The response of this arrangement to an incident three-dimensional gust is analysed. In this realistic geometry, the height is much smaller than the chord length of the aerofoil and the ratio is O(t). A perturbation approximation in t for the flow field is made and the Wiener-Hopf method is used to give an analytical solution to the resulting differential equations. Some preliminary results demonstrating the effect of camber and thickness are presented.

Mechanisms of trailing-edge noise reduction

D.I. Baker and N. Peake

Dept. of Applied Mathematics and Theoretical Physics, University of Cambridge

The trailing-edge of an aeroplane wing or turbine blade is an unavoidable source of noise, scattering hydrodynamic turbulence into audible noise. With inspiration from the wing of silent owls, demonstrably sound-reducing modifications to the trailing-edge have been developed, with their success (or not) apparently depending sensitively on their construction. Mathematical models are developed to explain the mechanisms behind the success of these devices and to optimise their properties, potentially paving the way to quieter aeroplanes and wind turbines.

High-fidelity simulation of turbulent jets and noise predictions using Lighthill's analogy

V. Ioannou[1], F.Margnat[2] and S.Laizet[1]

[1]Dept. of Aeronautics, Imperial College London, [2]P'Insittute, University of Poitiers

In order to help in reducing the environmental impact of aerospace travel, as set up by Flightpath 2050, Europe's Vision for Aviation, it is important to achieve accurate simulations of turbulent jets. Such simulations will aid in better understanding the underlying mechanisms of acoustic sound generation for a turbulent jet. For this study, the massively parallel high-order flow solver Incompact3d is used. The innovative methods used in the code to perform high-fidelity simulations of turbulent jets such as a customized IBM (Immersive Boundary Method) to model the nozzle inside the computational domain, the specific lateral boundary conditions for a more realistic representation of entrainment and the highly scalable parallel strategy will be detailed in this talk. A high-fidelity simulation of a turbulent jet at a Reynolds number of 460,000 will be presented and compared with experimental data. The state of the nozzle-exit boundary layer will be investigated in detail as it is known to be a key parameter for the flow development. Finally, using Lighthill's analogy, preliminary acoustic predictions of the sound generated by the jet will be presented.

Experimental sensitivity analysis and control of thermoacoustic systems via a secondary heat source

N. Jamieson and M. Juniper

Dept. of Engineering, University of Cambridge

We present the results of an experimental sensitivity analysis on a vertical electrically heated Rijke tube. We examine the shift in linear growth rate, linear decay rate, and the shift in frequency during periods of linear growth and decay of the thermoacoustic oscillations due to the introduction of a secondary heater. The rate of growth is slow, so the growth rate and frequency can be measured very accurately over many hundreds of cycles in the linear regime with and without control. The measurements agree qualitatively well with the theoretical predictions from adjoint-based methods of Magri & Juniper (*J. Fluid Mech*., vol. 719, 2013, pp. 183-202). This agreement supports the use of adjoint methods for the development and implementation of control strategies for more complex thermoacoustic systems.

Combining low order network modelling with incompressible flame LES for thermoacoustic instability in an industrial gas turbine combustor

Y. Xia[1], A.S. Morgans[1], W.P. Jones[1] and G. Bulat[2]

[1]Dept. of Mechanical Engineering, Imperial College London,
[2]Siemens Industrial Turbomachinery Ltd, Lincoln

Low NOx emissions from gas turbine combustors can be achieved using lean premixed combustion, which makes the combustor highly susceptible to the damaging thermoacoustic instability, arising from a two-way coupling between acoustic waves and unsteady flame heat release. Predicting thermoacoustic instability is difficult due to the range of length scales involved, from tiny turbulence eddies to long acoustic wavelengths. A low-order network model is therefore developed, modelling the acoustic waves using linear wave-based theory. The unsteady flame heat release rate is assumed to respond weakly nonlinearly to upstream acoustic forcing. The resulting frequency response is known as a Flame Describing Function (FDF), and obtained by an incompressible LES code BOFFIN. By combining the FDF with the low-order acoustic modes into a thermoacoustic solver OSCILOS, the frequencies and growth rates of a combustor's thermoacoustic modes are predicted. The accuracy of above combined approach has been verified by a lab scale combustor, and in this work this approach is extended to a full-scale industrial Siemens combustor. The frequencies of all the thermoacoustic modes are accurately predicted under 3 bar and 6 bar pressure compared to experimental data, giving us confidence in this coupled approach for real industrial combustors.

The Silsoe Cubes : CFD modelling single sided and cross-ventilation in naturally ventilated buildings in isolation and in an array configuration

M.F. King[1], H.L. Gough[2], C.J. Noakes[1] and J.F. Barlow[2]

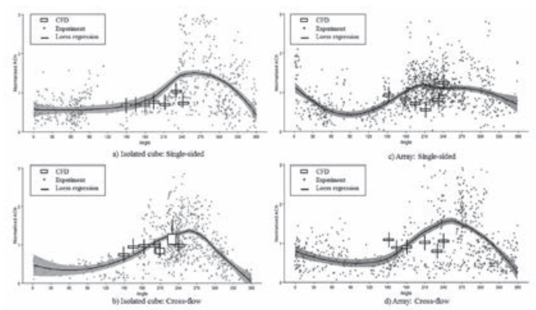
[1]Institute for Public Health and Environmental Engineering, University of Leeds, [2]Dept. of Meteorology, University of Reading

Introduction
The surface-mounted "Silsoe cube" (6mx6mx6m) is an idealised experimental hollow metal cube with windows and has been used for wind engineering research since the early 1990's. But simulating natural ventilated buildings though computational fluid dynamics (CFD) simulations is difficult. This research compares openFoam simulations of experimental isolated and novel array cube configurations under two ventilation types and seven wind angles.

Methodology
OpenFoam 2.3.1 compared the K-ω SST scale adaptive simulation (hybrid RANS-LES) turbulence model to other RANS and LES models for façade pressures and air change rates under for single and cross-flow ventilation.

Results and Discussion
The K-ω SST SAS model was found to perform best overall. Good comparison is found for isolated cases with a clear peak in experimental air change rate values (Figure 1) appearing where windows are facing the wind (240°). Similar trends are not quite so obvious in an array configuration meaning that upwind cubes may complicate ventilation predictions. A relationship between wind angle and ventilation rate can be seen for the isolated case for both window configurations but is less clear in the array. A possible explanation may be due to the vortex shedding of downwind cubes causing airflow to be more of a pulsatile nature, whereby averaged ventilation rates may not reflect this. As a result future simulations must take this into considerations and a different ventilation metric may need to be developed.

Figure 1 shows normalised (by median) air change rates compared against CFD predictions



LES of scalar dispersion from localized sources in a regular array of buildings

V. Fuka[1], Z.T. Xie[1], I.P. Castro[1], P. Hayden[2], M. Carpentieri[2] and A.G. Robins[2]

[1]Aerodynamics and Flight Mechanics Group, University of Southampton, [2]EnFlo, University of Surrey

We report some results from the EPSRC project DIPLOS (www.diplos.org) which aims to increase understanding of dispersion processes in street networks and improve street network models. The street network studied consists of an array of identical buildings of dimensions 1 h (length) × 2 h (width) × 1 h (height) separated by streets 1 h wide. The LES is validated against measurements from the EnFlo wind tunnel. A domain of 12 h × 12 h, height H = 12 h, and resolution h/16 was used in the large eddy simulation.

The relative contribution of the advective and turbulent vertical scalar fluxes varies strongly depending on whether the scalar source is placed in a recirculation zone or in a channel parallel to the flow. Dispersion results show an almost symmetric plume for a wind direction normal to the long streets. A sensitivity study shows how a small difference in the wind direction yields a plume similar to the measured one.

LES on a smaller domain (6 h × 6 h × 6 h) shows significant spanwise flow within the canopy but small asymmetry remains even with the larger domain (e.g. in the vertical velocity along a spanwise line at z=h).

In addition to the regular array, a similar case but with one tall building (height 3h) was computed.

Acknowledgment:

The research is supported by the EPSRC DIPLOS project (Grant EP/K04060X/1). The authors are grateful to colleagues Drs Omduth Coceal, Glyn Thomas and Denise Hertwig for their various contributions and useful comments. The authors also acknowledge the use of the IRIDIS High Performance Computing Facility and associated support services at the University of Southampton and the UK National Supercomputing Service ARCHER (http://www.archer.ac.uk).

CFD simulations of methane diffusion from landfill

A. Boghi and N. Harris

Centre for Atmospheric Informatics and Emissions Technology, Cranfield University

Ozone-depleting gases, such as methane, will be generated from the landfilling of municipal waste. The emissions of noxious gas from landfills and other waste disposal areas can present a significant hazard to the environment and to the health of the population if not properly controlled. In order to have the harmful gas controlled and mitigate the environmental pollution, we must estimate the extent to which the gas will be transported into the air at some time in the future. The emission estimates (inventories) are combined with atmospheric observations and modelling techniques. In this work we perform a series of Computational Fluid Dynamics (CFD) Simulations to determine the dispersion of methane in the atmosphere. The methane is modeled as a passive scalar which diffuses from the landfill with a given mass-flux. The Boussinesq approximation has been used to embed the effect of the buoyancy in the momentum equation. A logarithmic velocity profile has been used to model the wind velocity. The results, expressed in terms of the instantaneous and mean concentration field of methane, can be used to provide a better understanding of the phenomenon and suggest a better position of the atmospheric receptors.

Model comparison of velocity perturbations through wind farms

J. Pennells[1], A. Ross[1], I. Brooks[1] and S. Vosper[2]

[1]University of Leeds, [2]Met Office

Numerical models parametrise a wind turbine either as a sink of kinetic energy and source of turbulent kinetic energy, or by using an actuator disk method. A third linear option is described in Smith (2009). The linear model represents the boundary layer as a uniform flow, with the drag from the wind turbine being vertically distributed. This work aims to compare the results from the linear model to the Met Office boundary layer model BLASIUS. BLASIUS has been used extensively for examining orographic flow and flow through forest canopies and has been extended to include the wind farm parametrisation of Fitch et.al (2012). Simulations are carried out to examine how Froude number, $Fr = u/\sqrt{(g'z_i)}$, and vertically propagating gravity waves, defined by $Z = Nz_i/u$, alter the agreement between the linear model and BLASIUS. Analysis will be presented to show how the flow statistics are altered by varying velocity, inversion height and inversion strength.

The boundary layer in the linear model is represented by a 1D flow. This assumption has limitations, not only from the wind turbine drag being distributed uniformly within the boundary layer but also there is an implication to the total drag from the wind farm.

Designing cool stadiums for hot climates with CFD

P. Lynch

Arup Ltd.

For open-roofed stadiums in hot climates, maintaining acceptable thermal conditions within the stadium bowl is a significant challenge. In a typical stadium design, any cool air supplied to the bowl is subject to regular scouring from the wind, resulting in hot, uncomfortable conditions for players and spectators.

This presentation demonstrates how CFD modelling of the interaction of the turbulent wind with the stadium bowl was used to demonstrate the performance of Arup's design solution to this emerging problem, as a core part of the stadium design process. The results of large-eddy simulations were combined with dynamic thermal modelling and solar analysis to predict player safety and spectator comfort conditions within the stadium bowl.

We will also show how our approach was validated through comparison with physical testing in a wind tunnel, with detailed quantitative data from Irwin probe and PIV measurements, to provide confidence that our innovative design would result in a stadium that meets the client's ambitions.

Helicity of a small patch of quantum vorticity

M. Mesgarnezhad, C.F. Barenghi and A.W. Baggaley

Newcastle University

Tangled filamentary structures occur frequently in turbulent flows. In the absence of viscous or ohmic dissipation, the governing equations of motion (the Euler equation and the magnetic induction equation) conserve helicity, hence preserve the initial topology. Quantum fluids, such as superfluid helium and atomic Bose Einstein condensates, are convenient contexts to study the topology of turbulent flows, as superfluid vorticity is not a continuous field but is constrained to individual vortex lines of fixed circulation.

Recent work suggests that, in the case of small dissipation, helicity is partially preserved when vortex knots untie (Klechner et al, Nature Physics 2016). We present numerical simulations in which we show that, in the presence of small dissipation, a small patch of superfluid vorticity away from boundaries (thus consisting of closed vortex loops) can be driven to a statistical steady state. We find that length, energy and helicity of this turbulent system fluctuate around well-defined average values which depend on the intensity of the drive. We also show results for the "knot spectrum", the distribution of vortex loops of given complexity.

Quantum vortex dynamics and reconnections in trapped Bose-Einstein Condensates

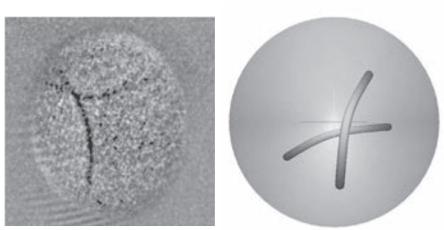
L. Galantucci[1], S. Sarafini[2], F. Dalfovo[2], G. Lamporesi[2], G. Ferrari[2] and C.F. Barenghi[1]

[1]Joint Quantum Centre Durham-Newcastle, Newcastle University, [2]INO-CNR BEC Center, Università di Trento

Vortex dynamics and reconnections play a fundamental role in quantum turbulent flows [1]. In particular, quantum vortex reconnections are responsible for the redistribution of energy and helicity amongst scales [2, 3] and represent the ultimate process for incompressible kinetic energy dissipation into acoustic modes. Experimentally, direct visualization of vortex–line dynamics in elongated Bose-Einstein Condensates (BECs) has been recently achieved [4] (see Fig. 1 (left)), suggesting the possibility of *double reconnection events*.

In this work we numerically solve the non-homogeneous Gross–Pitaevskii Equation for the condensate wavefunction inside an anisotropic harmonic trapping potential, employing an algorithm based on the pseudo vorticity field [5] to trail the core of the vortices (Fig. 1 (right) shows an instantaneous snapshot of the condensate with two reconstructed vortices). We observe different two–vortex interaction regimes depending on the orientation of the vortices, their orbits and their initial distance: unperturbed precession around the centre of the condensate; *bounce dynamics*; double and single reconnection events. The key ingredients driving the dynamics are the anti–parallel preferred alignment of the two vortices and the impact of density gradients and vortex images. The last two factors are peculiar to non–homogeneous trapped BECs, determining hence different reconnection dynamics with respect to homogeneous BECs [6].

Fig. 1: (left): Experimental observation in a trapped BEC of two vortices produced via the Kibble–Zurek mechanism [4]; (right): snapshot of the results achieved via the numerical simulations performed in the present study.



References

[1] C. F. Barenghi, L. Skrbek, and K. P. Sreenivasan. Introduction to quantum turbulence. *Proc. Natl. Acad. Sci.* USA, 111(1):4647–4652, 201
[2] C. F. Barenghi, R. J. Donnelly, and Vinen W. F. *Quantized vortex dynamics and superfluid turbulence*. Springer Science & Business Media, 2001.
[3] M. W. Scheeler, D. Kleckner, D. Proment, G. L. Kindlmann, and W. T. M. Irvine. Helicity conservation by flow across scales in reconnecting vortex links and knots. *Proceedings of the National Academy of Sciences*, 111:15350, 2014.
[4] S. Serafini, M. Barbiero, M. Debortoli, S. Donadello, F. Larcher, F. Dalfovo, G. Lamporesi, and G. Ferrari. Dynamics and interaction of vortex lines in an elongated bose-einstein condensate. *Physical Review letters*, 115(17):170402, 2015.
[5] C. Rorai, J. Skipper, R. M. Kerr, and K. R. Sreenivasan. Approach and separation of quantum vortices with balanced cores. *arXiv preprint arXiv:1410.1259*, 2014.
[6] S. Zuccher, M. Caliari, A. W. Baggaley, and C. F. Barenghi. Quantum vortex reconnections. *Phys. Fluids*, 24:125108, 2012.

On the Energy transfer during jets and vortices formation and evolution in turbulent planetary atmospheres.

T. Jougla and D.G. Dritschel

University of St. Andrews

The formation, evolution, and co-existence of jets and vortices in turbulent planetary atmospheres is examined using a two-layer quasi-geostrophic β-channel shallow water model. The study in particular focuses on the vertical structure of jets. Following Panetta 1988, a vertical shear representing heating variations on a planet is imposed on the flow and maintained by thermal damping. Idealised convection between the bottom and top layers is implemented by adding cyclonic/anti-cyclonic pairs, called hetons, to the flow. Following the energy diagram from Salmon 1980, we analyse the energy correlation and transfer between the different energy components. With this analysis we are able to characterise the main energy components, the main energy transfers and draw a new energy diagram, more precise and detailed than the Salmon diagram.

Vortex flow of yacht sails

A. Arredondo-Galeana and I. Maria Viola

Institute for Energy Systems, University of Edinburgh

The flow around a 1:115th model scale spinnaker of an America's Cup yacht was studied with particle image velocimetry (PIV). Four cross sections across the span were recorded at a Reynolds number of 1.7×10^4. We found that the flow separates at the leading edge followed by turbulent reattachment, forming a leading edge vortex (LEV). The LEV is stably attached to the leading edge and its diameter grows from the foot to the tip of the sail. On the lower half of the sail, the LEV has a negligible diameter while large trailing edge separation occurs from a quarter of the chord. At ¾ of the span, the diameter of the LEV is a quarter of the chord and trailing edge separation does not occur. The industrial exploitation of these results is enabled by the development of a low-order model (based on potential flow) which will underpin future design tools. This work is the first experimental evidence of the existence of a stable LEV on yacht sails, while it was previously predicted numerically at Reynolds number 6×10^5 using Detached Eddy Simulation (Viola et al., 2014, OE, 90:93-103).

Flow characteristics of deep stalled aerofoils with a wavy leading edge

R. Perez Torro

Dept. of Aerodynamics and Flight Mechanics, University of Southampton

This study investigates stalled flow characteristics of a NACA0021 aerofoi with a sinusoidal wavy leading edge (WLE) at $Re_\infty=1.2\times10^5$ and $\alpha=20°$. It is observed that laminar separation bubbles form in parts of the trough sections. However, for a proper aerodynamic simulation of an aerofoil with a WLE, it is found that the spanwise domain size should contain at least 4 leading edge undulation wavelengths since the distribution of laminar separation bubbles is strongly dependent on this variable. The existence of pairs of counter rotating vortices is also manifested in this study and it is concluded that this leads to turbulent transition and flow reattachment at the rear of the laminar separation bubbles. At the simulated angle of attack the straight leading edge (SLE) case undergoes a deep stall and produces strong wake vortex shedding that results in large fluctuations of aerodynamic force. On the other hand the WLE cases show a clear attenuation of the vortex shedding mechanism which is found to be related with the loss of flow homogeneity/coherence in the spanwise direction in comparison with the SLE case.

Numerical and experimental simulation of atmospheric downbursts

S. Williams

University of Leeds

The atmospheric downburst is born from a complex interaction of microphysics in convective clouds, evaporation and condensation induce regions of negatively buoyant air, which in combination with precipitation loading from hydrometeors (rain, hail, ice etc.), force air downwards. Through baroclinic instabilities, the leading edge of the downdraught forms a vortex ring which then impacts the ground and diverges radially. These radially propagating winds are known as the "gust front", and are often responsible for the peak winds in convective storms.

Motivation:
The interaction of the downburst with the surrounding environment is important from:

i) an engineering perspective – downbursts are responsible for unconventional wind loading and hazardous windshear that affect infrastructure (particularly off-shore) and aircraft.

ii) an atmospheric perspective – the interaction with the gust front has important implications for the generation of subsequent convection and the uptake of atmospheric dust, which in turn affects cloud formation and radiative forcing.

Despite the importance of downburst in both engineering and the atmospheric cycle, there are still some fundamental features of the downburst that remain poorly understood.

Impact:
Recent work by Rooney (2015) combined theory of buoyant thermals with axisymmetric gravity currents to propose a similarity solution that could be applied to downbursts. A novel and unique experimental study conducted at the University of Leeds tested this theoretical framework. Results show that the major features of Rooney (2015) are reproduced in the laboratory but with several important inconsistencies: the importance of flow symmetry of descending downburst and the influence of the vortex ring in the impact zone.

Influence of the near field on the far-field development of single and multiscale wall-mounted porous fences.

E. Rodriguez-Lopez[1], L.E. Brizzi[2], P.J.K. Bruce[1]

[1]Dept. of Aeronautics, Imperial College London, [2]P'Insittute, University of Poitiers

Wall-mounted porous fences are employed to passively control wall-bounded flows. Particularly, they can be designed to modify the incipient turbulent boundary layer (TBL) in wind tunnel experiments in order to generate a bespoke TBL far downstream. Previous studies [*Rodrguez-Lopez et al*. DOI 10.1007/s10546-016-0139-8] have shown that the far field development of an articially generated TBL is strongly dependent on the interaction between the detached wake of obstacles and the near-wall region. Accordingly, the present study considers four single- and multi-scale grids and explores how their far-field behaviour is linked to the near-field flow structure. Both hot-wire anemometry and acoustic measurements show that, despite having the same blockage, the various grids present significant dierences from both dynamic and acoustical points of view. Particle Image Velocimetry and wall-pressure measurements in the close vicinity of the grids are employed to assess the formation mechanisms of the TBL and how the relative position between various-sized bars and the wall aects the formation and far-eld development. Two main results are highlighted: (i) The acoustic field is modulated by the spanwise variation of the grid's blockage. (ii) The near-wall region is more strongly influenced by near-wall originated wakes rather than by further (although stronger) ones.

Experimental and numerical assessment of turbulent flow through arrays of multi-scale, surface-mounted cubes

D. Wise[1] and W. Brevis[2]

[1]Dept. of Fluid Dynamics, A*Star Institute of High Performance Computing, Singapore, [2]Dept. of Civil and Structural Engineering, University of Sheffield

Turbulent shear-flow through and over periodic arrays of multi-scale cubes is examined via tomographic particle image velocimetry (PIV), and large-eddy simulation (LES). Four cube arrangements are investigated: the Sierpinski carpet fractal at generation levels one to three, and a randomized Sierpinski carpet at the third generation. It is shown that the transition from the first to the third generation of the Sierpinski carpet causes a transition from "wake interference" to "skimming" flows, an expected feature due to the increase in plan area density. Interestingly however, regions of the modified Sierpinski carpet also show a return to "wake interference" flow, despite having the same frontal and plan-area density as the original pattern. Analysis of point-wise time-series from the PIV measurements reveal that within and immediately above the obstacle arrays the cubes have an unexpected effect on the energy spectra. This is corroborated by acoustic doppler velocimetry. The same behaviour is not observed in either the outer flow above the cubes or in reference smooth wall measurements.

Turbulent characteristics produced by a blockage of multi-scale emergent obstacles

J.E. Higham, W. Brevis and C.J. Keylock

University of Sheffield

Stereo Particle Image Velocimetry (SPIV) measurements were undertaken to quantify the turbulent characteristics created by the blockage of a turbulent open channel flow by four dierent sets of emergent obstacles, all with the same porosity. Four sets of obstacles were tested including a set of in-line regular sized obstacles, and three arrangements of multi-scale pre-fractal obstacles; one arranged as a Sierpinski carpet, and the other two ordered by obstacle size (i.e. large to small and visa versa). It is found that the pre-fractal Sierpinski carpet creates a protracted peak of turbulent kinetic energy, when compared with a regular obstacle. Likewise, by rearranging the Sierpinski carpet it is possible to tweak the position of the peak turbulent kinetic energy. Using modal analysis further investigations are undertaken to determine the mechanisms underpinning the physics of these phenomena.

Flow field characteristics and power draw of regular and fractal turbines

K. Steiros, P.J.K. Bruce, O.R.H. Buxton and J.C. Vassilicos

Dept. of Aeronautics, Imperial College London

Experiments have been performed in an octagonal un-baffled water tank, stirred by three radial turbines with different geometry impellers: (1) regular rectangular blades; (2) single-iteration fractal blades; (3) two-iteration fractal blades. Shaft torque was monitored and the power number was calculated for each case. Both impellers with fractal geometry blades exhibited a decrease of turbine power number compared to the regular one (15% decrease for single-iteration and 19% for two iterations).

Pressure measurements on the blade surfaces showed that the form drag coefficient is larger, while the centre of pressure radius is smaller for the regular blades case compared to the fractal case.

Phase locked PIV in the discharge region of the blades revealed that the vortices emanating from the regular blades are more coherent, have higher kinetic energy, and advect faster towards the tank's walls where they are dissipated, compared to their fractal counterparts. This suggests a strong link between vortex production and behaviour and the energy input for the different impellers. Planar PIV measurements in the bulk of the tank showed an increase of turbulence intensity of over 20% for the fractal geometry blades, suggesting higher mixing efficiency.

Roughing up wings—a promising technique in laminar flow control

P. Servini and F.T. Smith

University College London

Laminar flow control—the process of influencing the structure of a laminar boundary layer as it moves over a surface—is important in the field of aerodynamics for the purposes of drag reduction and safety: new and more effective techniques are constantly being sought. Thus the work of Huebsch, Rothmayer and others on dynamic roughnesses is exciting. These small, oscillating bumps seem to have the potential for delaying or suppressing the separation of a boundary layer from a surface.

In this talk, I'll present their numerical and experimental results; as well as our work on dynamic roughnesses in the context of marginal separation.

Computational fluid-structure interaction analysis of membrane wings for micro-air-vehicles

R. Sanchez and R. Palacios

Dept. of Aeronautics, Imperial College London

Membrane wings have shown superior aerodynamic performance in the Low-Reynolds number regime at which MAVs generally operate. They are able to adapt their shape to the surrounding flow, which allows them to delay stall. At the same time, control mechanisms may be improved by using smart materials such as Dielectric Elastomers (DEs), which offer a noticeable increase in performance with respect to mechanical devices [1, 2]. However, the flexibility of these wings induces large deformations in the structural domain, which significantly complicates the computational analysis of the FSI problem. To investigate the dynamics of the coupled system, but also to define a framework that would allow future studies in coupled optimisation, a non-linear structural solver able to deal with large deformations has been natively developed within the opensource code SU2 [3], and coupled with the original fluid solver for FSI applications [4]. We will present our efforts in the development and validation of this tool, which shows a good performance in the simulation of two relevant test cases (Figs. 1, 2), and its applicability in the simulation of the behaviour of membrane wings.

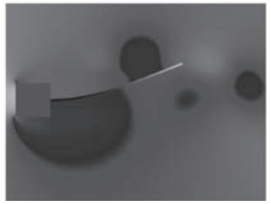


Figure 1: Pressure contours in benchmark [5].

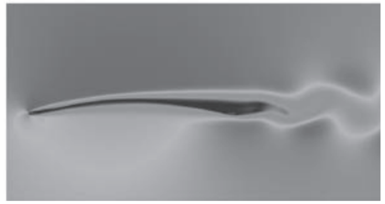


Figure 2: Flow velocity contours in a membrane wing, $\alpha = 4°$.

References

[1] Buoso, S. and Palacios, R., Electro-aeromechanical modelling of actuated membrane wings, *Journal of Fluids and Structures*, Vol. 58, pp. 188-202, 2015.
[2] Buoso, S. and Palacios, R., Viscoelastic effects in the aeromechanics of actuated elastomeric membrane wings, *Journal of Fluids and Structures*, Vol. 63, pp. 40-56, 2016.
[3] Palacios, F., et. al. Stanford University Unstructured (SU2): An open-source integrated computational environment for multi-physics simulation and design, AIAA *51st Aerospace Sciences Meeting*, 2013.
[4] Sanchez, R., et. al. Towards a Fluid-Structure Interaction solver for problems with large deformations within the open-source SU2 suite, *57th AIAA/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference,* 2016
[5] Wall W.A. and Ramm E., Fluid-Structure interaction based upon a stabilized (ALE) finite element method., *Computational Mechanics. New Trends and Applications*, 1998.

Feedback control of an ahmed body flow exhibiting symmetry-breaking regimes

O. Evstafyeva and A.S. Morgans

Dept. of Mechanical Engineering, Imperial College London

At highway speeds most of the usable engine energy in ground vehicles is consumed overcoming aerodynamic drag. Specically for square-back vehicles majority of the aerodynamic drag comes from form-drag, which arises due to the pressure dierence between the vehicle front and the vehicle rear, the latter subjected to the low pressure of the wake flow. Therefore drag-reduction control strategies, targeting pressure recovery on the rear base of the vehicle, oer great potential for significant CO_2 emissions savings. The present work investigates numerically the flow past a simplied square-back vehicle geometry - the Ahmed body. Special attention is paid to understanding the reflectional symmetry breaking behaviour in the wake, with reduced order modelling tools used to track the onset of asymmetry as the Reynolds number increases. A feedback control strategy for reducing drag is developed with actuation in the form of unsteady synthetic jets around the base of the body. To ensure the practical relevance of the approach, body mounted sensors recording base pressure force fluctuations are used. Open-loop forcing is used to characterise the response of the sensor signal to actuation, based on which a feedback controller is designed to suppress base pressure force fluctuations, yielding a concomitant drag reduction.

Vectoring of parallel synthetic jets

T. Berk, G. Gomit and B. Ganapathisubramani

Aerodynamics and Flight Mechanics Group, University of Southampton

A pair of parallel synthetic jets can be vectored by applying a phase difference between the two driving signals. The resulting jet can be merged or bifurcated and either vectored towards the actuator leading in phase or the actuator lagging in phase. In the present study, the influence of Strouhal number on this vectoring behaviour is examined experimentally.

Phase-locked vorticity fields, measured using Particle Image Velocimetry (PIV), are used to track vortex pairs. The physical mechanisms that explain the diversity in vectoring behaviour are observed based on the vortex trajectories. For a fixed phase difference, the vectoring behaviour is shown to be primarily influenced by pinch-off time of vortex rings generated by the synthetic jets. For a given actuator, the pinch-off time is shown to be a function of Strouhal number only. Beyond a certain formation number, the pinch-off timescale becomes invariant. In this region, the vectoring behaviour is determined by the distance between subsequent vortex rings, which is also a function of the Strouhal number.

We acknowledge the financial support from the European Research Council (ERC grant agreement no. 277472).

High-Fidelity simulations of dielectric barrier discharge plasma actuators in a turbulent channel flow

T. Brauner and S. Laizet

Dept. of Aeronautics, Imperial College London

In recent years, the development of devices known as plasma actuators has advanced the promise of controlling flows in new ways that increase lift, reduce drag and improve aerodynamic performance. Dielectric barrier discharge (DBD) actuators are electrohydrodynamic devices which can be used to create a strong electric field and to give momentum to the fluid around them. In this talk we will present high-fidelity simulations of turbulent channel flows at moderate Reynolds numbers with several arrangements of DBD plasma actuators located at the walls of the channel. The aim is to manipulate the turbulent structures near the wall in order to achieve a reduction in skinfriction drag. The simulations are performed with the high-order flow solver Incompact3d and an extra forcing term, based on a phenomenological model, is added to the incompressible Navier-Stokes equations to reproduce the effect of the DBD plasma actuators.

Turbulent flow past three-dimensional patches ofroughness

M.A. Ferreira and B. Ganapathisubramani

Faculty of Engineering and the Environment, University of Southampton

Understanding the impact of surface roughness in frictional drag has long been a lively topic of research that directly benefits a wide range of engineering applications. Most previous investigations mainly focus on homogeneous type of roughnesses, which do not often mimic the surface conditions found in nature. In this work, the aim is to study the effects of a finite patch of roughness on the turbulent flow. Wind tunnel experiments were conducted of flows over surfaces with 3D flow perturbations. Randomly generated rough patches with large relative hight ($h/\delta \approx 0.1$) are placed within a turbulent boundary layer. The characteristics of the finite patch of roughness are systematically varied by altering the frontal and the plan solidities (λ_F and λ_P) of the patch ranging $\lambda_F = 0.05$-0.25 and $\lambda_P = 0.10$-0.38. This covers the entire range for both solidities from sparse to dense. Measurements were made using a floating-element force balance and high-resolution Particle Image Velocimetry (PIV). The experimental data is then compared against Townsend's theory of self-preserving flow inside a boundary layer. The talk will present further detailed analysis of the PIV data.

Surface-mounted ribs of varying length in turbulent flows

J.W. Van der Kindere and B. Ganapathisubramani

Experimental Fluid Mechanics Research Lab, University of Southampton

We study two-dimensional rectangular obstacles (ribs) in a turbulent flow. It is believed that turbulence promotes the breakdown of the shear layer formed at separation on sharp corners. This in turn affects the separation properties and the overall effect of the rib on a boundary layer. An experiment was carried out with three ribs (aspect ratio: 1, 4, and 8) placed in a boundary layer downstream of an active grid which produces three levels of freestream turbulence (u'/U = 3.7%, 7%, 9%). The Reynolds number based on rib height is 20000, and the boundary layer thickness varied between 1:3H and 3:4H, where H is the rib height. The results of this study discuss differences in recirculation region dimensions and the fluctuation of these dimensions, a measure of the perturbation caused by the ribs. Secondly results discuss surface pressure distribution and fluctuations as rib length and freestream turbulence vary. The aim is to confirm whether the behaviour of ribs is influenced by freestream turbulence in comparison with the laminar freestream conditions observed in most comparable publications.

Energy paths in high Reynolds number wall turbulence

E. De Angelis[1], A. Cimarelli[2], J. Jimenez[3] and C.M. Casciola[4]

[1]School of Engineering, Cardiff University, [2]DISMI, University of Modena, [3]School of Aeronautics, Universidad Politecnica de Madrid, [4]DIMA, University of Rome

The current understanding of the wall-turbulence is based on two classical theories, one concerning the dynamics of the small scales and the other concerning the description of the near-wall layer. The first one is based on the phenomenology of the energy cascade from large to small fluctuations. The second one follows from the division of wall flows in a near-wall, viscous region, and an outer inertial region. The dual nature of these theories arises from the fact that one description is given in the space of scales alone while the other is given in physical space. The present work describes the multidimensional behaviour of scale-energy production, transfer and dissipation in wall-bounded turbulent flows with the aim of understanding the cascade mechanisms by which scale energy is transmitted scale-by-scale among different regions of the flow.

We will address the inhomogeneous Kolmogorov equation evaluated on a numerical dataset of a turbulent channel flow at Re_τ = 2000: Two driving mechanisms are identified: a strong scale-energy source in the buer layer related to the near-wall cycle and an outer scale-energy source associated with an outer turbulent cycle in the overlap layer. The topology of these two sources leads to a complex redistribution of scale energy.

References

[1] Cimarelli A., De Angelis E., Jimenez J. and Casciola C.M., Analysis of the Kolmogorov equation for filtered wall-turbulent flows, *J. Fluid Mech.* in press.
[2] Cimarelli A., De Angelis E. and Casciola C.M., Paths of energy in turbulent channel flows, *J. Fluid Mech.* 715, 436-451, (2013).
[3] Marati N., Casciola C.M. and Piva R., Energy cascade and spatial fluxes in wall turbulence, *J. Fluid Mech.*, 521, 191-215, (2004).

Flow field created by a coherent turbulent water jet impinging on a vertical wall

R.K. Bhagat and D.I. Wilson

Dept. of Chemical Engineering and Biotechnology, University of Cambridge

When a coherent liquid jet impinges on a vertical wall, it forms a thin film, spreading radially away from the point of impingement, until a point where the outward momentum is balanced by surface tension. At this point, the liquid film changes its thickness abruptly giving a jump. A model for the jump location, based on Nusselt's film theory, was presented by Wilson *et al.* (Chem. Eng. Sci, 2012, 68, 449-460). In this study, the flow field created by a turbulent liquid jet and the location of the jump are studied and a new model is presented.
The liquid film passes through three zones, namely the laminar boundary layer, the laminar film and the turbulent zone. The location of the laminar to turbulent transition is described theoretically.
In addition, the analysis explains why the location of film jump is observed to be insensitive of the substrate type at high flow rate.
The model is compared with existing as well as previously reported data for the film jump and film thickness. The average velocity of the liquid was estimated from the initial growth of the film. Good agreement is obtained between the measurements and the model, which has no adjustable parameters.

The time signature of the turbulent/non-turbulent interface over a turbulent boundary layer

A. Laskari, R.J. Hearst, R. de Kat and B. Ganapathisubramani

Aerodynamics and Flight Mechanics Group, University of Southampton

The turbulent/non-turbulent interface (TNTI) between a turbulent boundary layer and an approximately laminar free-stream is investigated with time-resolved planar particle image velocimetry (PIV). The turbulent boundary layer ($Re_\tau \approx 5000$), was formed on the floor of a water channel and was captured by a PIV system composed of a Phantom v641 4 mega-pixel camera and a Litron LDY 304 Laser. Images were acquired at 800 Hz, which was sucient to resolve the motions of the TNTI in time. The instantaneous TNTI can be located by using a turbulent kinetic energy decit threshold. The time-resolved data set allows for instantaneous tracking of the TNTI topology and detection of its convection velocity, and thus provides novel insight into the TNTI. Preliminary findings suggest the interface is convected at a velocity $U_C \approx U_\infty$ in the streamwise direction and $Vc \approx 0{:}15U_\tau$ in the wall-normal. The nal study will include analysis of the full data set of over 400,000 time-resolved images and a more accurate estimate of the convection velocity of the interface.

Interactions of large-scale free-stream turbulence with turbulent boundary layers

E. Dogan, R.J. Hearst and B. Ganapathisubramani

Aerodynamics and Flight Mechanics Group, University of Southampton

The scale interactions in a turbulent boundary have been investigated in the presence of free-stream turbulence (FST). An active grid was used to generate the turbulence in the free-stream. Single-component hot-wire anemometry measurements were performed simultaneously in the boundary layer and in the free-stream. It has been shown that the free-stream large scales penetrate into the boundary layer and increase the streamwise velocity fluctuations throughout the boundary layer. The extent of the penetration has been observed to depend on the level of fluctuations in the free-stream. In addition, the large-scales dominating the log-region have demonstrated a modulating effect on the small-scales of the near-wall region and this effect has been observed to increase with turbulence intensity level of the free-stream. This modulating interaction of the scales has noted similarities between a turbulent boundary layer under the effect of FST and a canonical turbulent boundary layer at high Reynolds numbers. For further investigation, multi-channel single-component hot-wire measurements in the boundary layer will also be performed using a four-probe array. The scale interactions in the boundary layer will be presented in more detail from these hot-wire measurements.

Scale-wise skin friction generation in turbulent channel flow

M. de Giovanetti[1], Y.Y. Hwang[1] and H. Choi[2]

[1]Dept. of Aeronautics, Imperial College London, [2]Dept. of Mechanical and Aerospace Engineering, Seoul National University

Recently, a growing body of evidence has emerged for the existence of self-similar energy-containing motions in wall-bounded turbulent flows, as predicted by Townsend (i.e attached eddy hypothesis). However, the role of these energy-containing motions in drag generation is still vastly unknown. We investigated the contribution to skin friction generation of each of these motions, up to $Re_\tau \approx 4000$. Three dierent methods have been employed, namely: 1) FIK identity with an existing spanwise spectra of Reynolds stresses; 2) connement of the spanwise computational domain; 3) articial damping of the motions to be examined. The energy-containing motions in the logarithmic region are found to carry a very significant percentage of the total skin friction, whereas the near-wall structures are observed to decrease their contribution as the Reynolds number is increased, consistent with previous results at lower values of *Re*. Comparatively, the outer structures account for less skin friction than the logarithmic region, and their contribution is almost unchanged for the range of *Re* investigated here. The outcome from dierent methods shows that non-trivial scale interactions play an important role in the drag production.

Automated finite difference modelling on structured grids, and a variety of compute architectures

S.P. Jammy, C.T. Jacobs and N.D. Sandham

Aerodynamics and Flight Mechanics Group, University of Southampton

The path to exascale computational fluid dynamics requires novel and disruptive hardware architectures that are more powerful than ever. Unfortunately, most numerical modelling frameworks are not in a position to readily exploit such architectures to their full potential. The 'static' nature of the hand-coded discretisation schemes in languages such as C or Fortran means that entire codebases often have to undergo non-trivial modifications in order to run efficiently on more exotic compute platforms such as GPUs and the recently-introduced Intel Xeon Phi cards. This places a huge unsustainable burden on computational scientists to not only be domain specialists, but also experts in numerical methods, parallel computing paradigms, and their efficient implementation. This work introduces a new framework, OpenSBLI, which unlike the majority of existing finite difference models, allows users to specify the equations they want to solve in Einstein notation, and details of the numerical methods, at an abstract level. From this specification, the C code that performs the discretisation is automatically generated. By coupling with the OPS execution framework, the generated code is then tailored towards a desired backend to enable the efficient execution of the model on a wide variety of compute hardware.

A novel curvature evaluation method for the Volume-of-Fluid method based on interface reconstruction

F. Evrard[1], F. Denner[1], F. Serfaty[2] and B.G.M. van Wachem[1]

[1]Dept. of Mechanical Engineering, Imperial College London, [2]PetroBras, CENPES, Cidade Universitaria

The Volume-of-Fluid method (VOF) is widely adopted for simulating interfacial flows. A critical step in VOF modelling is the evaluation of the mean curvature of the fluid-fluid interface for the computation of surface tension. Most of the existing curvature evaluation techniques exhibit a bias due to the type of discretisation employed or to the smoothing applied to the volume fraction field. This leads to the production of inaccurate or unphysical results. We present a novel curvature evaluation method which aims at greatly reducing this bias. The interface is reconstructed from the discrete volume fraction field using the Marching-Cubes algorithm, and the mean curvature is directly evaluated on the resulting triangulation. The mean curvature is then integrated back on the Eulerian mesh in order to compute the surface tension force acting on the fluid. The method does not require smoothing of the discontinuous volume fraction field and can be relatively easily extended to unstructured meshes. The local nature of the surface reconstruction makes the parallelisation of our method eortless, and its computational demand is of the order of existing methods.

* Support from Petrobras is gratefully acknowledged.

Immersed boundary method with exact forcing for unstructured mesh and coupled solver

Mohd H.A. Azis and B.G.M. van Wachem

Dept. of Mechanical Engineering, Imperial College London

Immersed boundary method (IBM) is an efficient method to approximate moving solid boundaries in flows, and are extensively employed in fully-resolved particulate flow simulations. To enforce the no-slip condition at the solid-fluid boundary, various strategies can be followed, where using forcing based on the discrete delta functions is the most applied. Discrete delta functions have historically been derived for Cartesian meshes and for specific applications.

In this work, an IBM for unstructured meshes using Reproducing Kernel Particle Method (RKPM) that is adapted to a fully coupled solver will be presented. The implemented adaptation leads to a correct velocity-pressure coupling in a solver with collocated variable arrangement. In addition, the presented method directly uses the terms in the discretised Navier-Stokes equations in order to improve the convergence rate of the no-slip forcing in a coupled solver.

Validation and performance of the formulation will be demonstrated for typical cases, such as flow through settling spheres. This method presents an accurate alternative that is applicable for generic flows and meshes.

Large Eddy Simulation model for particle tracking in turbulent flows

T. Curran and B.G.M. van Wachem

Dept. of Mechanical Engineering, Imperial College London

The interaction of solid particles with turbulent flows has for long been a topic of interest for predicting the behaviour of industrially relevant flows. For the turbulent fluid phase, Large Eddy Simulation (LES) methods are widely used for their low computational cost, leaving only the sub- grid scales of turbulence to be modelled. Concerning the particulate phase, Lagrangian methods are appreciated for their accuracy in tracking particle trajectories and in taking into account particle- particle collisions. However, understanding the eects of the unresolved scales on the discrete phase, especially when regarding particle pair dispersion is still subject to ongoing work. In parallel, Direct Numerical Simulations (DNS) are undertaken as a mean of comparison with the LES results. Statistics of both phases are retrieved in order to validate the current model, with focus brought on particle-particle dispersion, velocity correlations and kinetic energy. We are also interested on the influence of particle size (i.e. Stokes' number) on the interaction with turbulent structures. This model is validated by comparing the results with DNS data. We highlight the importance of interaction between the particles and subgrid turbulence, depending on the Stokes number.

* Support from Petrobras is gratefully acknowledged.

A fully-coupled Navier-Stokes solver framework applicable for flows of all speeds

C.N. Xiao, F. Denner and B.G.M. van Wachem

Dept. of Mechanical Engineering, Imperial College London

In this contribution, a new pressure-based nite volume framework with collocated variable arrangement is introduced. The main feature of the framework is its ability to handle single-phase flows of all speeds, ranging from the incompressible limit to the supersonic regime. This objective is achieved by modifying the transformed continuity equation as well as the coupling of flow and thermodynamic variables to an energy equation and equation of state. The solver operates on collocated variable arrangement and applies a momentum-weighed interpolation method adapted for compressible flows to prevent pressure-velocity decoupling. This approach can easily deal with structured as well as unstructured meshes. Another distinctive feature of the framework lies in its fully-coupled approach which, by solving all equations of the linearized algebraic system simultaneously, guarantees a strong coupling between pressure and velocity at each iteration and ensures additional robustness. Shock-capturing is accomplished via nonlinear spatial discretization schemes or an appropriate blending of rst-order upwind and second-order centered-dierencing schemes. A selection of standard validation cases will be presented to demonstrate the framework's capability of simulating flows of all speeds, its accuracy in predicting transient flows as well as its robustness in handling strong supersonic shock waves.

Validation of a correlation-based transition model for the Spalart-Allmaras turbulence model

H. Yao and B. Thornber

School of AMME, The University of Sydney

A local correlation-based $\gamma - \overline{Re_{\theta t}}$ transition model is coupled with Spalart-Allmaras turbulence model in a structured parallelised Navier-Stokes solver. The correlations of Critical Reynolds number,$Re_{\theta C}$, and the transition length, F_{length}, are as derived in previous models designed for the $\kappa - \omega$ Reynolds-Averaged Navier-Stokes method. However, the proposed model includes significant changes to the $\gamma - \overline{Re_{\theta t}}$ approach to permit coupling with the Spalart-Allmaras turbulence model. This paper provides a detailed and complete summary of the modified governing equations, and a suite of validation and verification tests including the ERCOFTAC T3 at plate series and several two dimensional airfoil cases.

An efficient implementation of the CIP-CSL3 method

Q. Li and K. Yokoi

School of Engineering, Cardiff University

We present a fully conservative upwind multi-moment method for the conservative equation. The proposed method is a variant of the CIP-CSL (Constrained interpolation profile conservative semi-Lagragian) scheme which is based on a third-order polynomial interpolation function and semi-Lagrangian formulation. The proposed method can robustly simulate linear and nonlinear scalar transport problems, and shock tube problem. We also proposed a simple and efficient slope limiter in the presented method which an take advantages of the CIP-CSL2 scheme and CIP-CSL3 scheme.

Number of benchmark tests are presented. The numerical results show that the presented method is accurate and can eliminate numerical oscillation efficiently.

Traveling wave solutions of minimal large-scale structures in turbulent channel flow

Y.Y. Hwang[1], A.P. Willis[2] and C.Cossu[3]

[1]Dept. of Aeronautics, Imperial College London, [2]School of Mathematics and Statistics, University of Sheffield, [3]IMFT Toulouse

Discovery of very-large-scale motions (large-scale streaky structures) in the outer region of wall-bounded turbulent flow has attracted significant interest of many researchers over the last decade. It was initially proposed that this energetic outer motion is formed by concatenation of large-scale motions (or bulges) given in the form of hairpin vortex packet originating from merger and/or growth of the near-wall hairpin vortices [1]. There is growing evidence, however, that an alternative scenario explains the origin of the long streaky structures: we have recently shown that, by artificially removing active motions in the near-wall and the logarithmic regions using an over-damped large-eddy simulation [2, 3], large-scale structures in the outer region are able to sustain themselves in the absence of any 'bottom-up' energy transfer. Self-sustaining large-scale structures consist of very-large-scale motions (long streaky motion) and several large-scale motions (quasi-streamwise vortical structures) aligned to it, and their self-sustaining process is remarkably similar to that of the near-wall motions. Here, we show that the self-sustaining large-scale structures are organized around a set of invariant solutions in the form of traveling wave by extending the recent approach where a set of stationary invariant solutions were computed in plane Couette flow with over-damped large-eddy simulations [3].

References

[1] R. J. Adrian, Hairpin vortex organization in wall turbulence, 2007, *Phys. Fluids.*, 19, 041301.
[2] Y. Hwang and C. Cossu, 2010, Self-sustained process at large scales in turbulent channel flow, *Phys. Rev. Lett.*, 105, 044505.
[3] S. Rawat, C. Cossu, Y. Hwang & F. Rincoln, 2015, On the self-sustained nature of large-scale motions in turbulent Couette flow, *J. Fluid Mech.*, 782, p515.

Three-dimensional boundary layer states forced by transpiration over short spanwise scales

A. Williams

University of Manchester

We consider the three dimensional laminar boundary layer on a semi-infinite plate in a uniform stream. The three dimensionality is induced by a wall transpiration that exists over a spanwise scale that is comparable to the boundary layer thickness. This short-scale transpiration requires the inclusion of spanwise and transverse diffusion in the boundary layer, therefore building upon previous analyses of the corner boundary layer equations. We examine the case in which the spanwise extent of the transpiration region maintains a constant ratio with the boundary layer thickness, allowing similarity solutions to be developed in any cross sectional plane a fixed distance from the leading edge of the plate. Numerical computation is shown to require careful consideration of the far field flow to ensure a global mass balance consistent with the inviscid outer flow. For a blowing transpiration, numerical solutions highlight three regimes of attached flow depending on the magnitude of the mass flux through the boundary. These regimes are differentiated by the (transverse and spanwise) spatial extent of a low-speed streamwise streak. Asymptotic descriptions are presented in the limits of large and small mass flux through the plate, which are seen to be in agreement with the full numerical solutions.

Linear stability analysis of the boundary-layer over a high pressure turbine blade

M. Zauner and N.D. Sandham

Aerodynamics and Flight Mechanics Group, University of Southampton

A better understanding of laminar/turbulent boundary-layer transition in high pressure turbines (HPT) is crucial for increasing aerodynamic efficiencies. Recent direct numerical simulations (DNS) have shown evidence of unstable boundary-layer modes, despite the high disturbance environment. The current contribution considers two approaches for identifying such unstable modes in averaged velocity proles originating from a three-dimensional fully-resolved DNS of a generic turbine blade at representative flow conditions. Results are obtained using an in-house local linear stability and parabolised stability solver, the NoSTRANA code. Firstly, a temporal as well as spatial local linear stability analysis (LST) of the flow is performed, indicating that the boundary-layer on the suction side first becomes unstable at about 82% of the axial cord-length, where the primary instability is an oblique mode with a wave-angle of 31.6°. The phase-speed of the most unstable mode is about 34% of the local free-stream velocity and the group-velocity is of the order of the streamwise velocity at the inflection-point of the corresponding velocity proles. Secondly, the parabolised stability equations (PSE) are solved. Both methods show good agreement as the deviation of the spatial growth-rate close to the trailing edge is about 10%.

Stability limits and viscous behaviour of the strato-rotational instability

L. Robins

University of Leeds

Recently it was shown that the presence of a stable stratification can destabilise an otherwise stable flow. The instability is non-axisymmetric, and is now known as the Strato-Rotational Instability (SRI). The SRI can bypass an axisymmetric stability limit known as the Rayleigh line, which makes it of particular interest for astrophysical Keplerian flows such as accretion discs.

The SRI's stability limit has not yet been analytically derived, but here we show a numerical approach to finding it within the context of Taylor-Couette flow. We also show that, with the SRI, the presence of viscosity can destabilise stratified flows that would be stable in an inviscid context.

On instability of vortices generated by a free-jet flow, in the presence of background rotation

I.U. Atthanayake, J.H.A. Vlaskamp, P. Denissenko, Y.M. Chung and P.J. Thomas

School of Engineering, University of Warwick

We present results of PIV measurements conducted on freely developing jets which are subject to Coriolis forces induced by background system rotation. The jets are generated inside a large water-filled rotating tank (height, 2m; diameter, 1m) by injecting water from a nozzle in the direction of the vertical axis of rotation. In this study we investigate the flow characteristics of the cross sectional flow field at dierent heights above the nozzle from where the jet emerges and for dierent injection Reynolds numbers. It is found that these cross-sectional flow-eld structures are vortical structures and they undergo transient, regularly recurring instability between two dierent flow modes. It is moreover found that further away from the source, these modes are characterised by an initial mono-polar vortex structure developing into co-rotating bipolar vortices. One of the main goals being to establish the relationship between the transition frequency of the observed two alternating flow modes and the Rossby number.

Shape optimisation for hydrodynamic stability using adjoint based methods

J. Brewster and M.P. Juniper

University of Cambridge

Linear stability analysis of flows results in an eigenvalue problem where the real and imaginary parts of the eigenvalue characterise the growth rate and frequency of a given mode shape.

We examine the flow around an object at a Reynolds number of 90. We calculate the gradient of the eigenvalue with respect to the shape of the object. This calculation uses adjoint methods taken from shape optimisation and linear stability analysis. We construct a Hadamard form for the shape gradient by introducing an adjoint global mode and an adjoint base flow. The sensitivity of the eigenvalue with respect to each shape parameter is then found without needing to perform finite differencing, which is expensive. The cost of our approach is equivalent to solving another eigenvalue problem together with one flow calculation. Shape optimisation for eigenvalues using gradient-based methods becomes more feasible for shapes defined by many parameters.

In this study we demonstrate the above, applying it to the flow initially over a cylinder using the finite element package FEniCS. We optimise for stability from an initial condition of a laminar flow just after the onset of vortex shedding. Additionally, we demonstrate how the above approach can be used in conjunction with a variety of other objective functions whilst constraining the growth rate of the most unstable mode.

Nonlinear optimal control of bypass transition in a boundary layer flow

D. Xiao and G. Papadakis

Dept. of Aeronautics, Imperial College London

Bypass transition is observed in a flat-plate boundary-layer flow when high levels of free-stream turbulence are present. This scenario is characterized by the formation of stream-wise elongated streaks, their break down into turbulent spots and eventually fully turbulent flow. In the present work, DNS is employed to simulate the control of bypass transition on a zero-pressure-gradient boundary layer. In order to delay bypass transition, a nonlinear optimal control algorithm is developed using the direct-adjoint approach. Using the Lagrange variational method, the blowing/suction control velocity is found by solving iteratively the non-linear Navier-Stokes and the adjoint equations in a forward/backward loop. The optimisation is performed in a finite time horizon. Large values of optimisation horizon result in instability of the adjoint equations. The results show that the control is able to reduce the energy of the flow in the region where the objective function is defined. Examination of joint probability density function shows that the control velocity is correlated with the stream-wise velocity. There is evidence of opposition control action i.e. the control velocity acts in a direction opposite to the wall-normal velocity close to the wall. The span-wise averaged instantaneous stream-wise velocity is significantly reduced in the near-wall region.

The accurate and efficient numerical simulation of general fluid-structure interaction

Y. X. Wang, P.K. Jimack and M.A. Walkley

School of Computing, University of Leeds

The accurate and efficient numerical simulation of general fluid-structure interactions (FSI) is a significant computational challenge because of the strong nonlinearities. The current gold standard for robust and accurate simulation of such problems is to use a monolithic method that strongly couples the fluid and solid models and discretizes them into a single nonlinear system at each time step. Such schemes are exceedingly computationally expensive however, and require sophisticated numerical schemes to ensure convergence of the nonlinear solver at each time step.

In this talk, we present a new finite element method for simulation of FSI which has the same generality and robustness of monolithic methods but is semi-explicit and therefore significantly more computationally efficient and easier to implement. Our proposed approach has similarities with classical immersed finite element methods, by approximating a single velocity field in the entire domain on a single mesh, but differs by treating the corrections due to the solid deformation on the left-hand side of the modified fluid flow equations.

In addition to motivating the derivation of our numerical scheme, we will provide a description of its implementation within an adaptive finite element code and a wide range of computational examples will be presented in order to validate the method across a wide range of flows, solids and interactions.

An LES-PBE-PDF approach with PBE-grid adaptivity for modelling particle formation in turbulent reacting flows

F. Sewerin and S. Rigopoulos

Dept. of Mechanical Engineering, Imperial College London

Turbulent reacting flows with particle formation appear in many engineering applications such as the formation of soot in hydrocarbon flames or the precipitation of nanoparticles from aqueous solutions. In this work, we consider an Eulerian description of the particulate phase in terms of its particle size distribution and present a comprehensive model and numerical solution scheme for predicting the evolution of the particle size distribution in a turbulent carrier flow. Here, the particle size distribution is governed by the population balance equation (PBE) which we combine with the LES turbulence model. In order to be able to accommodate both the fluid and particle phase kinetics without approximation, we augment the existing joint-scalars transported PDF-model for the turbulence-chemistry interaction by the particle size distribution and devise an efficient numerical solution scheme in the context of the stochastic fields method. This scheme includes, for instance, an adaptive discretization of the stochastic fields in particle size space by which sharp features of the particle size distribution can be accurately resolved. As an application, we consider the precipitation of $BaSO_4$ particles in a coaxial pipe mixer and assess the influence of mixing on the particle formation processes and the final particle size distribution.

A cut-cell based overset meshing approach for incompressible viscous flow

J. Mackenzie and L. Qian

School of Computing, Mathematics and Digital Technology, Manchester Metropolitan University

An overset grid approach is an effective method of simulating fluid flow involving multiple moving bodies. It consists of minor meshes representing solid objects, which overlap a Cartesian background grid, allowing bodies to move arbitrarily whilst retaining communication between grids. However, a hole beneath each overlapping mesh must be cut from the background grid, leaving a small overlap. Current hole-cutting methods tend to be complex with some requiring extensive user knowledge and input. Since the hole must be re-cut regularly for moving body problems, it can become very time-consuming.

A novel approach for performing a hole-cut has been implemented by employing the Cartesian cut-cell method. This method would ordinarily be used to cut the boundary of a solid object from a single Cartesian grid, as an alternative to the overset grids approach. It is fully automatic and since it cuts though cells, it performs a far simpler cut to a typical hole-cutting method, which must cut around cells.

This approach has been applied to an incompressible Navier-Stokes solver for viscous flow. For an improved performance with complex geometries, unstructured, triangular minor meshes are used. The solver has been validated using benchmark tests, including flow past an oscillating and rotating cylinder.

Multilevel solution algorithms for a numerical model of thin film flow

M. Al-Johani, M.A. Walkley and P.K. Jimack

School of Computing, University of Leeds

The focus of this research is to develop efficient multigrid solvers for the thin film flows. In order to solve this nonlinear PDE system, the computational approach uses the finite difference method (FDM) in space and a fully implicit scheme in time. At each time step the resulting discrete nonlinear algebraic system must be solved efficiently. The aim of this presentation is to contrast two different solution schemes: one based upon a nonlinear multigrid scheme (FAS) and the other based upon Newton-Multigrid methods. Both approaches will be demonstrated to have optimal efficiency however we will discuss the potential advantages of each.

The talk will introduce the governing PDE system and briefly describe the implicit discretization scheme. The FAS multigrid algorithm will be introduced, to solve the nonlinear system directly, followed by a description of the Newton-MG method. The Newton algorithm requires the solution of a sparse linear equation system at each nonlinear iteration, which is solved using a linear multigrid scheme. The talk will conclude with some comparative numerical results and an outlook for the next steps in our research. The significance will become most apparent when considering 3D flows, where the size of the algebraic system can become very large.

Ecient iterative solution algorithms for numerical models of multiphase flow

A. Alrehaili, M.A. Walkley and P.K. Jimack

School of Computing, University of Leeds

In this work we consider ecient numerical methods for the simulation of vascular tumour growth based upon the multiphase fluid model introduced by Hubbard and Byrne [1]. The talk will present a brief overview of the multiphase model, which involves the flow and interaction of four dierent, but coupled, phases. We will then describe the discretization schemes used to solve the time-dependent system of partial dierential equations (PDEs). This involves a finite volume scheme to approximate mass conservation and a conforming nite element scheme to approximate momentum conservation and reaction-diusion equation for the background nutrient concentration. The momentum conservation system is represented as a Stokes-like flow of each phase, with source terms that reflect the phase interactions. It will be demonstrated that the solution of these coupled momentum equations, approximated using a Taylor-Hood nite element method in two dimensions, is the most computationally intensive component of the solution algorithm.

The final part of the presentation will provide a detailed description of the structure of these discrete PDEs, and an assessment of the pre-conditioning strategy for the resulting algebraic system. Results will contrast the preconditioners considered and we will conclude with a discussion of these techniques.

References

[1] M. E. Hubbard and H. M. Byrne, *Multiphase modelling of vascular tumour growth in two spatial dimensions*, Journal of Theoretical Biology, 316 (2013), 70-89.

Fitted ale scheme for two-phase Navier-Stokes flow

M. Agnese

Dept. of Mathematics, Imperial College London

We present a novel fitted ALE scheme for two-phase Navier-Stokes flow problems that uses piecewise linear finite elements to approximate the moving interface. The meshes describing the discrete interface in general do not deteriorate in time, which means that in numerical simulations a smoothing or a remeshing of the interface mesh is not necessary.

Modelling of corium spreading in a core catcher

H. Perrier[1], M. Eaton[1], V. Badalassi[1,2] and B. G. M. van Wachem[1]

[1]Department of Mechanical Engineering, Imperial College London, [2]Rolls-Royce plc, Derbyshire

In the event of a nuclear accident, reactor core meltdown can lead to failure of the reactor containment building and radioactivity releases to the public and the environment. This can be prevented by using so-called core catchers to spread the molten material (Corium) over a large area. The Corium can then be cooled efficiently and the accident progression can be stopped. Numerical simulation of Corium spreading flows is very challenging as it involves solidification, free surface flow, and strongly temperature dependent fluid/solid properties.

In the present contribution a Volume of Fluid methodology is derived and discussed to capture the free surface evolution of the Corium interface. The complexity arising from the combination of discontinuities at the interface and temperature dependent viscosity is discussed and numerical methods to deal with this complexity are proposed.

These methods are validated on a simple configuration to ensure a perfect match between corium volume fraction advection and energy advection. The final methodology is applied to a Corium spreading situation, and a comparison is made with experimental data.

Structure build-up and evolution in the drying of sessile blood droplets

A. Uppal[1], O.K. Matar[1], R.V. Craster[2]

[1]Dept. of Chemical Engineering, Imperial College London, [2]Dept. of Mathematics, Imperial College London

Blood exhibits a range of complex phenomenon during the drying process. Experimental observations have recorded blood undergoing a sol-gel transition during the drying/evaporation process. The gel front begins at the contact line when a critical concentration is reached. Thus, the rheology becomes non-uniform throughout the droplet and exhibits transitional complex phenomena that we must capture if we wish to accurately model the evaporative/cracking process. We propose a model where thixotropy is introduced to capture the evolving rheology as evaporation occurs. Thixotropy is often used to describe fluids which exhibit a decrease in viscosity due to flow and subsequent slow recovery of viscosity after the cessation of the flow. We introduce an additional parameter to describe the internal structure of the fluid at each point and consider a droplet in the limit of the lubrication approximation. Additional assumptions must be made to simplify the depth dependence of the new structure parameter, which in turn are compared to results from the full two-dimensional problem.

Mechanosensitive signalling pathways detection during TCFA formation

R.M. Pedrigi, T. Homma, L. Towhidi, Z. Kis, H. Sant'Ana Pereira, N. Maimari, N. Zogaib, D. Khodadadi, M. Ahmed, A. Post, E. Karadza, M. Dent, V. Mehta, A. Seneviratne, R. de Silva and R. Krams

National Heart and Lung Institute, Imperial College London and Dept. of Bioengineering, Imperial College London

A surge of recent studies have highlighted the role of blood flow in determining plaque growth and plaque composition, both in animal studies and recent clinical trials. Despite these compelling data the underlying mechanism for flow-induced plaque formation is currently unknown.

Blood flow is sensed by endothelial cells, and these cells react to blood flow by changing their gene signature. Initial studies showed that approximately 2000 genes are regulated by blood flow, distributed over 40 signalling pathways regulating 15 transcription factors and their changes during plaque development are currently unknown. In this presentation, we report the development of a novel platform that enables to study the adaptation of endothelial gene networks in vascular biology.

This platform consists of state-of-the art ultra-high resolution, small animal imaging (μCT, μMRI and US) coupled to finite element methods to determine the endothelial shear stress and strain fields during the development of the vulnerable plaque. These maps will subsequently drive a robot-driven laser-capture machine (Zeiss, PALM microbeam) which will isolate groups of endothelial cells exposed to a certain shear and strain field during plaque development. Next, RNA will be isolated, amplified with linear amplification kits, and libraries are prepared for deep RNA sequencing (Illumina Hiseq 2500). After this, in-house developed bioinformatics tools will be applied to decipher gene networks and gene modules. Entire gene networks are subsequently tested with a newly in-house developed siRNA microfluidics system for cardiovascular studies, which consists of a programmable robot-dispenser and a bespoke, in-house developed combined flow-cell and microporator. In parallel to this, live cell imaging tools are developed to determine network dynamics from single cell measurements, which will be coupled to the siRNA microfluidics system to steer synthetic network design. In a first series of studies we have developed a synthetic gene network to determine the activity of a GPCR-dependent mechanosensor and we are coupling this novel technology to a bespoke microfluidic chamber to test a library of small drugs to decipher a novel treatment of flow-induced TCFA.

Swirling transition in *Drosophila* oocytes

G. De Canio, E. Lauga and R.E. Goldstein

Dept. of Applied Mathematics and Theoretical Physics, University of Cambridge

The biological world includes a broad range of phenomena in which transport in a fluid plays a central role. Among these is the fundamental issue of cell polarity arising during development, studied historically using the model organism *Drosophila melanogaster*. The polarity of the oocyte is known to be induced by the translocation of mRNAs by molecular motor proteins along a dense microtubule cytoskeleton, a process which also induces cytoplasmic streaming. Recent experimental observations have revealed the remarkable fluid-structure interactions that occur as the streaming flows back-react on the microtubules. In this work we use a combination of theory and simulations to address the interplay between the fluid flow and the configuration of cytoskeletal filaments leading to the directed motion inside the oocyte. We show that the mechanical coupling between the fluid motion and the orientation of the microtubules can lead to a transition to coherent, unidirectional motion within the oocyte, as observed experimentally.

Moreover, we unveil and fully characterise the physical mechanism governing the onset of the observed transition.

Squirming in a droplet

S. Y. Reigh[1], L. Zhu[2], F. Gallaire[2] and E. Lauga[1]

[1]Dept. of Applied Mathematics and Theoretical Physics, University of Cambridge, [2]EPFL Lausanne

Active micro- and nano-particles have potential applications as transporters such as drug delivery to targeted cells in vivo. They may be encapsulated by a droplet to achieve their tasks, for example, in order to contain specific chemicals. We investigate the locomotion of micro-swimmers in a droplet and the resulting droplet dynamics in a viscous fluid. We use the squirmer model as general model of micro- and nano-swimmers and allow for two different fluid viscosities inside and outside the droplet. Analytical solutions for the fluid flow fields are obtained by solving the incompressible Stokes equations and compared with numerical simulations using the finite element method. Swimming velocity and fluid flow fields inside and outside the droplet for pusher, puller and neutral swimmers are obtained and quantitatively compared between theory and simulations, with excellent agreement.

Flows around bacterial swarms

J. Dauparas

Dept. of Applied Mathematics and Theoretical Physics, University of Cambridge

Flagellated bacteria on nutrient-rich substrates can differentiate into a swarming state and move in dense swarms across surfaces. A recent experiment (HC Berg, Harvard University) measured the flow in the fluid around the swarm. A systematic chiral flow was observed in the clockwise direction (when viewed from above) ahead of a *E.coli* swarm with flow speeds of about 10 μm/s, about 3 times greater than the velocity at the edge of the swarm. The working hypothesis is that this flow is due to the flagella of cells stalled at the edge of a colony which extend their flagellar filaments outwards, moving fluid over the virgin agar. In this talk we quantitatively test his hypothesis. We first build an analytical model of the flow induced by a single flagellum in a thin film and then use the model, and its extension to multiple flagella, to compare with experimental measurements.

Viscoelastic synchronisation

M. Lisicki and E. Lauga

Dept. of Applied Mathematics and Theoretical Physics, University of Cambridge

Synchronisation of biological filaments is vital for microbial propulsion and micro-scale fluid transport. To understand the route towards synchrony, we analyse a model system of colloidal oscillators. Beads are subject to one-dimensional driving potentials which are coupled to the configuration of the system. In a class of "geometric switch" models, the driving force witches when a certain limiting position is reached. In the presence of a second driving potential which is periodic in time (the "clock"), the beads exhibit periodic trajectories. In our work, we address the question of synchronisation of a single active colloidal bead moving in a viscoelastic fluid. In contrast to the Newtonian case, the dynamics is determined by the natural frequency of the driving force, and the two characteristic fluid retardation and relaxation times. Depending on the relation between them, various dynamical regimes can be found. By analysing the theoretical model, we determine trajectory of the particle for a given set of fluid and forcing characteristics which leads to the determination of the phase diagram for the synchronised and asynchronous states. Further on, we extend our analysis to include the effects of thermal noise and discuss the effect of noise on the phase diagram.

Optimising the deployment of a time dependent polymer to minimise growth of viscous fingers

T.H. Beeson-Jones and A.W. Woods

BP Institute, University of Cambridge

We investigate the growth of the Saffman-Taylor instability in the case that the viscosity of the displacing fluid gradually increases over time as may occur with a slowly reacting polymer solution. For injection of a finite volume of the displacing fluid over a finite time in a one-dimensional geometry, the amplitude of the most unstable linear mode depends on the injection rate as a function of time. Using variational calculus, we illustrate that when the gelling time is comparable to the injection time the optimal solution involves a gradual increase in the flow rate with time, and can lead to a substantial suppression of the instability compared to the reference constant injection case. In contrast, for both relatively slow or fast gelling compared to the injection time, use of the optimal injection rate leads to relatively little suppression of the instability as compared to the reference constant injection case. We extend this to an axisymmetric geometry and consider the impact of such results for minimising viscous fingering.

Shear-thinning effects on a coalescing drop

V. Voulgaropoulos, M. Chinaud and P. Angeli

Dept. of Chemical Engineering, University College London

Coalescence between two bodies of the same fluid is important in nature (during cloud formation and rain) and in engineering applications such as inkjet printing and the oil and gas industry. The growth of the meniscus that forms immediately after the interface between the two bodies breaks is controlled in the beginning by a balance between the interfacial tension forces driving the coalescence and the viscous forces resisting it.

In this work, an experimental study of an aqueous drop coalescing with an organic-aqueous interface is conducted inside a confined quasi-two dimensional Hele-Shaw cell. The liquids used (water/glycerol mixture and Exxsol D80) have their refractive indices matched to minimise diffractions at the interface, while different concentrations of xanthan gum are added in the aqueous phase. The shear rate range during the coalescence is found within the shear-thinning region of the rheological curves of the xanthan gum solutions. High-speed imaging is implemented to track the neck expansion evolution, while advanced time resolved PIV measurements (with the addition of particles in the aqueous phase) are conducted to acquire 2D velocity fields and vorticity measurements during the coalescence process.

Plug formation of non-Newtonian liquids in microchannels

E. Roumpea, M. Chinaud and P. Angeli

Dept. of Chemical Engineering, University College London

The present work investigates the dynamics of plug formation during the flow of a non-Newtonian shear-thinning aqueous solution and a Newtonian organic fluid in a circular microchannel using a two-colour micro-PIV system. Two aqueous glycerol solutions containing xanthan gum (1000 and 2000 ppm) are used as the non-Newtonian fluids, while silicone oils (Sigma-Aldrich) with different viscosities, 5 or 155 cSt, are the Newtonian phases. All runs are carried out in a glass microchannel (Dolomite® microfluidics) with ID = 200 μm at different combinations of flow rates of the two phases (0.01-0.1 mL/min). Using the two-colour μ-PIV, averaged velocity profiles were obtained in each phase for different locations of the plug tip inside the main channel. The velocity fields (Fig.1) reveal that the continuous phase resists the formation of the dispersed phase plug. This behaviour is affected by the rheology of the aqeuous phase.

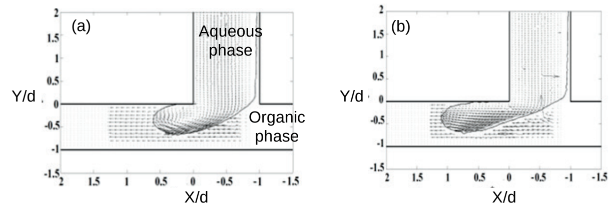


Fig. 1: Velocity fields in both phases for plug tip position (a) 0.75 (b) 1.25 (Newtonian case).

Turbulent combustion and auto-ignition of alternative engine fuels

I. Gorbatenko[1], M. Lawes[2], D. Bradley[2], A. Tomlin[3] and R. Cracknell[4]

[1]CDT in Fluid Dynamics, University of Leeds, [2]School of Mechanical Engineering, University of Leeds, [3]School of Chemical and Process Engineering, University of Leeds, [4]Shell Global Solutions

The current trend of downsizing and turbo-charging engines to reduce fuel consumption increases the possibility of engine knock and therefore requires fuels which are more resistant to auto-ignition. The addition of bio-fuels such as bio-ethanol and bio-butanol to conventional gasoline can reduce greenhouse gas emissions and probably offer a better knock resistance than traditional fossil fuels, thereby improving engine performance. However, it is debatable which bio-fuel offers the most benefits in terms of overall potential.

This study will investigate the effects of such alcohol fuel blends with gasoline and Toluene Reference Fuel (TRF) mixtures on ignition delay times and excitation times. Ignition delay times are in the range of milliseconds and will be obtained through experimental work on a Rapid Compression Machine (RCM). However, excitation times, during which the heat release occurs, are of the order of microseconds and can only be obtained through numerical simulations.

Both ignition delay times and excitation times are incorporated in the theoretical detonation peninsula and provide a more fundamental approach to the prevention of engine knock than use of Octane Numbers.

An artificial cilium

T.A. Spelman and E. Lauga

Dept. of Applied Mathematics and Theoretical Physics, University of Cambridge

Experimentalists have constructed a cylindrical column containing aligned magnetic material which move under the influence of a rotating magnetic field. Over a 360 degree rotation of the magnetic field this artificial cilium undergoes an effective and recovery stroke similar to that seen in natural cilium. Such artificial cilia are used for mixing and transportation in lab-on-chip devices.

We model the motion of this cilium using a theoretical energy balance model for the effective stroke and a force balance model for the recovery stroke. We show that the cilium is in dynamic equilibrium for the effective stroke provided the angular velocity of the magnetic field is small enough. The recovery stroke only occurs when the ratio of magnetic energy to cilium elastic energy is large enough and takes place at a specific direction of the magnetic field, dependent on this ratio, which we determine.

The Stokes-flow parachute of the dandelion fruit

C. Cummins[1,2], I. Maria Viola[1] and N.Nakayama[2]

[1]Institute for Energy Systems, [2]Institute of Molecular Plant Science, University of Edinburgh

The fluid mechanics principles that allow a passenger jet to lift off the ground are not applicable to the flight of small insects or plant fruit. The reason for this is scaling: human flight requires very large Reynolds numbers, while small insects or fruit have comparatively small Reynolds numbers. At this small scale, there are a variety of flight modes available to fruit: from parachuting to gilding and autorotation. Our research focuses on the aerodynamics of small, plumed fruit that utilise the parachuting mode.

When plumed fruits are picked up by the breeze, they can be carried over formidable distances. Incredibly, their parachutes are mostly empty space, making them extremely efficient fliers. Moreover, the fruit can "sense" whether or not its local environment is a favourable habitat, and can change its flight capacity to impede or aid its dispersal. The aim of this research is to uncover the novel engineering principles of these fruit using a combination of theoretical and physical modelling. In this talk, we present results from our mathematical modelling and laboratory tests, which are helping to reveal the fruit's ingenious flight mechanism.

Stability of active fluids with application to wound healing

D. Nesbitt[1], G. Pruessner[2] and C.F. Lee[1]

[1]Dept. of Bioengineering, [2]Dept. of Mathematics, Imperial College London

The ability for tissues to self-regenerate is key to biological life. Laboratory experiments on scratched monolayers of cells heal mainly through cell migration, due to the pressure generated from cell division within the bulk of the tissue, as well as the cells' individual motility forces as they crawl on the substrate. Velocity plots derived from experimental data show that motion is not localised to the interface [1]. Moreover the velocity patterns visualised are reminiscent of fluid mechanics, featuring currents and vortices. These observations motivate treating the tissue as an active fluid.

A common characteristic observed in healing monolayers is that an initially straight wound does not heal uniformly. Instead finger-like protrusions develop [1], similar to those occurring in the Saffman Taylor instability. We perform a linear stability analysis on an incompressible active fluid model representing the leading edge of closing wound, with the aim of detecting this instability. Although we obtain instability for one of the cases examined, our study suggests that the compressibility effects are crucial for finger development [2].

References:

1. L. Petitjean et al. *Biophysical Journal 98 (2010) 1790–180*
2. D. Nesbitt, G. Pruessner, C. F. Lee. In Preparation

The onset of double-diffusive convection with evolving background gradients

O.S. Kerr

Dept. of Mathematics, City, University of London

When a deep body of fluid with a stable salinity gradient is heated from below a destabilizing temperature gradient develops. As the heat diffuses into the fluid the effective thermal Rayleigh number based on the diffusion scale grows and is proportional to $t^{3/2}$, and so one may expect the fluid to be stable initially, and only becoming unstable after a finite time. In such circumstances the evolution of the background gradients can be important. Here we look at the onset of linear instabilities taking into account these changing background gradients, and find the optimal instabilities. This optimization requires an appropriate choice of a measure for the magnitude of the instabilities. We will compare these results to other more conventional results that make use of, say, the quasi-static assumption.

Low order models of layer formation in oscillatory double-diffusive convection.

T. Goodfellow, S.D. Griffiths, D.W. Hughes and P.K. Jimack

CDT in Fluid Dynamics, University of Leeds

Fluid dynamical systems with the tendency to form patterns have been the subject of much scientific interest over the years. One of the most interesting patterns is that of layers or “staircases”, which appear to form from a disordered turbulent state. Thermohaline staircases for instance are found in the oceans as mixed, almost neutrally stratified layers of salinity and temperature with steep gradient interfaces.

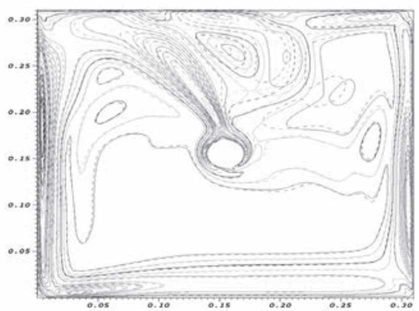
A minimal representation can be constructed to study the weakly non-linear behaviour of such double-diffusive systems. A heavily truncated Fourier series solution is employed to convert the governing system of PDEs into a system of ODEs. This system is numerically integrated using a 4th order Runge-Kutta scheme to determine the minimum number of modes required for the formation of layers, and to gain an insight into their interactions.

A reduced order modelling methodology for natural convection in a differentially heated enclosure with heated cylinder using the proper orthogonal decomposition while bypassing a Galerkin projection

G.R. McKenna

School of Mechanical and Aerospace Engineering, Queens University Belfast

The novelty in this work comes from the development of a reduced order model (ROM) for the modelling of natural convection while bypassing a Galerkin projection. By exploiting the governing equations the Galerkin projection is ideal for the modelling of complex flow regimes beyond the scope of forced convection. Nonetheless, the Galerkin has been continually identified as a computationally inefficient and unreliable means to evaluate the appropriate weighting coefficients. In this paper the proper orthogonal decomposition with orthogonal complements (PODc) was used to form a ROM of a complex parametric system in the natural convection regime. This was achieved by introducing a profile correlation procedure (PCP) alongside 2 sub-system ROMs that were incorporated into the one global ROM. The Grashof ratio θ^* is the flow parameter that best quantifies variations in the flow structure of the case. In the offline stage the ROM was constructed over the parameter range $7.07e-5 \leq \theta^*_{obs} \leq 7.92e-4$. Error analysis was conducted for 2 new sets of thermal boundary conditions corresponding to $\theta^* = 3.7e-4$ and $\theta^* = 7.66e-4$. Consequently, the final ROM was tested on 2 flow structures that differ considerably from one another with regard to the influence of buoyancy effects. All thermal approximations are shown to be in excellent agreement and share an L_2 *norm* error < 1%. Both velocity approximations share an error ≈ 10%. However, for the high buoyancy case a manual selection of arbitrary control surfaces based upon the experience of the researcher was required to evaluate the appropriate weighting coefficients.



ROM (solid line) and CFD (dashed line) velocity contour comparison $\theta^* = 7.66e-4$ L_2*norm error* 9.456%

Computational Fluid Dynamic Modelling of Benzene Abatement using Cryogenic Condensation

J. Hendry[1], J. Lee[1] and M. Battrum[2]

[1]Biopharmaceutical and Bioprocessing Technology Centre, Newcastle University, [2]Aesica Pharmaceuticals, Northumberland

Benzene is a carcinogen and chemical toxicant. Exposure to benzene has both acute and chronic effects on human health. Environmental regulations on industrial emission of benzene control vent concentrations to the parts-per-million level, to prevent the exposure of employees and the environment to the contaminant.

The two conventional alternatives to attain this level of emission control are adsorption and thermal oxidation. In order to reach this dilute concentration reliably adsorption strategies may feature periodic disposal of the adsorbent. This is undesirable as it produces a solid waste stream with associated disposal costs. Similarly thermal oxidation requires significant energy input in order to accomplish benzene combustion at the dilute concentrations necessary.

Cryogenic condensation is an alternative abatement technique that uses a cryogen (liquid nitrogen) to cool and condense contaminants from industrial vent streams. However unlike other volatile organic compounds benzene's high freezing point of +5.5°C implies it will desublimate rather than condense under the conditions being assessed. High temperature gradients inside the unit could produce a fine particulate that becomes entrained preventing separation from the gas. This work reports on a computational fluid dynamics model applied to assess the applicability of cryogenic condensation to benzene abatement.

Nonlinear acoustics in a viscothermal boundary over an acoustic lining

O. Petrie and E. Brambley
University of Cambridge

Acoustic linings are used in aircraft engines and exhausts to attempt to reduce the noise they produce. The acoustic lining can be modelled by a wall impedance, which is commonly implemented as a boundary condition by assuming the fluid to be inviscid and the mean flow to slip over the wall. It has been shown that this is not very accurate, nor well posed, and that the effect of a viscous boundary layer near the wall should be considered. Doing so has been found to introduce an amplification of the sound within the boundary layer. Here I will extend this linear work to look at louder perturbations by including nonlinear effects. An asymptotic solution within the boundary layer will be presented which is matched to the acoustics in the outer inviscid uniform mean flow. The consequences of this work for aircraft engine noise will also be discussed.

An analytical model for the acoustics of short circular holes

D. Yang and A.S. Morgans
Dept. of Mechanical Engineering, Imperial College London

The acoustic response of short but finite length circular holes with mean flow passing through them is highly relevant to Helmholtz resonators, fuel injectors, perforated plates and many other engineering applications. Widely used analytical models which assume an infinitesimally short hole were recently shown to be insufficient for predicting the impedance of holes with a finite length. In the present work, an analytical model based on the Green's function method is developed which takes the hole length into consideration. For "long" holes, the most important effect is shown to be the acoustic non-compactness of the hole. For "short" holes, the importance of capturing the modified vortex noise accurately is shown. The vortex sheet shed at the hole inlet edge is convected to the hole outlet and further downstream. It couples with the acoustic waves and this coupling has the potential to generate as well as absorb acoustic energy in the low frequency region. When a good model for the shed vortex path is used, the impedance predictions agree well with previous experimental and CFD results, for example predicting the potential for generation of acoustic energy at higher frequencies.

A two-dimensional model for three-dimensional symmetric flows

B. Font Garcia[1], G.D. Weymouth[2] and O.R.Tutty[1]

[1]Aerodynamics and Flight Mechanics Group, University of Southampton, [2]Fluid Structure Interactions Group, University of Southampton

A two-dimensional model for three-dimensional symmetric laminar flows is described. This model is derived from the incompressible Navier-Stokes equations using the velocity-pressure formulation. By locating the origin of the three-dimensional structures on the symmetry plane and applying an appropriate treatment of the three-dimensional term remaining in the derived equations an accurate solution of the three-dimensional flow at the symmetry plane can be achieved. The backward-facing step numerical test case is used to test the performance and accuracy of the derived model. Above $Re = 400$, three-dimensional structures arise [1] leading to different primary reattachment lengths for the two-dimensional and the three-dimensional cases. These structures are located close to the separation point. We show that using a two-dimensional transport equation for the responsible three-dimensional term would result into a reattachment length close to the three- dimensional solution. The direct benefit of this work is a significant reduction of the computational time required to achieve the three-dimensional solution of symmetric laminar flows. As a future work, a two-dimensional model of three-dimensional terms will be explored in the field of turbulence for spatially periodic flows.

References

[1] B.F. Armaly, F. Durst, J.C.F. Pereira and B. Schönung, *Experimental and theoretical investigation of backward-facing step flow*, Journal of Fluid Mechanics, vol. 127, pp. 473{496, 1983

Modelling internal wave focusing

W. Booker, O. Bokhove and M.A. Walkley

CDT in Fluid Dynamics, University of Leeds

Confined internal wave systems with oblique walls can lead to the evolution of a spatial singularity called an internal wave attractor. The existence of this singularity is the result of wave energy focusing, that is the wave energy becomes localised on small spatial scales. It has been observed that ocean ridges, such as the Luzon strait, can support such energy localisation and a study of this mechanism will lead to a better understanding of ocean mixing in these areas. Simulations of wave attractors have been shown to match laboratory observations, despite being derived from an inviscid, ideal fluid model.

We develop a structure preserving finite element method for internal gravity waves to study these flows. We consider the linearised, inviscid Euler-Boussinesq model and its associated Hamiltonian structure. We derive a discontinuous Galerkin finite element method that preserves this structure. This leads to a non-dissipative numerical method which accurately models the evolution of these wave attractors over long time periods. We use harmonic wave solutions to demonstrate that the method preserves the Hamiltonian structure and the discrete energy of the internal wave system.

Modelling the motion of waves in a "slice" of beach

E. Moulopoulou
University of Leeds

One dimensional non-linear shallow water equations have been used to model the waves observed in a Hele-Shaw tank in the absence of any bottom topography. The tank consists of two glass plates, which are 2 mm apart e.g[1],and waves are generated using a set of aquarium pumps, which work underwater whilst operated by a motor controller and programmed using an arduino board and lap-top.

The effect of momentum damping has been added in the model and the set of equations has been solved using a finite volume scheme. The point of interest is the volume of water that enters the cell at the location of the left wall called volume flux. Image processing techniques have been used to analyse consecutive snapshots of the cell. Processed snapshots returned data that led to the acquisition of a fitted function for the flux. The model has been "adjusted" so as to approximate the experiment more realistically, by using the fitted value of the flux at each time step as an input. Comparisons between the simulations with and without experimental data as an input have been carried out. The next step is to modify the existing model so as to include the bottom topography.

References

[1] O. Bokhove, A. J. van der Horn, D. van der Meer, A. R. Thornton, and W. Zweers. On wave-driven "shingle" beach dynamics in a table{top Hele{Shaw cell. In *Proceedings of 34th Conference on Coastal Engineering, Seoul*, volume 24, pages 118{173, 2014.

UK Fluids Conference: Poster presentations

1. **Jenny Wong (Leeds)**
   Two phase flow in the Earth's core
2. **Pranav Chandramouli (INRIA, Rennes)**
   Assessment of sub-grid scale tensors in the context of very Large Eddy Simulations
3. **Shian Gao (Leicester)**
   Numerical simulations of turbulent flow and heat transfer in ladle degassing
4. **Rachel Philip (Oxford)**
   Modelling oil droplet formation in turbulent oil plumes
5. **Steven Böing (Leeds)**
   Fluid dynamics of cumulus convection at the cloud-scale
6. **Abrar Ali (City)**
   The interaction of buoyant magnetic structures with convective plumes
7. **Maryam Argungu (ICL)**
   Mathematical modelling of blood and interstitial fluid in a poroelastic model of the liver
8. **Hugo Castillo (UCL)**
   Instabilities in shear-thinning fluids
9. **Hannah Kreczak (Leeds)**
   Rates of chaotic mixing in models of fluid devices
10. **Godwin Madho (Leeds)**
    Using data assimilation techniques to validate fluid flow models
11. **Elnaz Naghibi (QMUL)**
    Spinning the Earth's pole with the oceanic turbulence: the effect of a double-gyre on Chandler wobble
12. **Natalie Gilkeson (Leeds)**
    Parametric study on personalised ventilation systems using computational fluid dynamics
13. **Evaldas Greiciunas (Leeds)**
    Modelling novel heat exchangers for aircraft thermal management
14. **Oliver Halliday (Leeds)**
    Convectively forced gravity waves and their sensitivity to heating profile and atmospheric structure
15. **Jacob Van Alwon (Leeds)**
    Experimental and numerical modelling of aerated flows over stepped spillways
16. **Julieth Figueroa (Leeds)**
    Computational fluid dynamics modelling of liquid-liquid slug flow in capillaries
17. **José Aguilar (Cambrid**ge)
    Adjoint based sensitivity analysis in low order thermoacoustic networks
18. **Srinivasa Rao (Osmania, India)**
    Behavior of turbulence models in low Reynolds number flows and near wall regions
19. **Chris Knight (ICL)**
    CFD simulations of flow in polydisperse granular media with application to internal erosion
20. **Steven Taggart (Strathclyde)**
    CFD-experimental comparison of a Broady 3500 series safety relief valve
21. **Yongxin Chen (Southampton)**
    An immersed boundary method for turbulent flow over blocks
22. **Rohan Ramasamy (First Light Fusion)**
    Hytrac: a front-tracking code for the simulation of multi-fluid systems
23. **John Chalke (Southampton)**
    Simulation of aircraft wing hitting turbulence during heaving
24. **Anwar Haque (IIUM, Malaysia)**
    Effect of boundary layer of balance platform on an under floor external balance data of a half model
25. **Andrejs Tucs (Greenwich)**
    MHD model for liquid metal battery description
26. **Susanne Höllbacher (Goethe, Frankfurt)**
    Bi-directional coupling of particulate flow - a generalized framework for the physically motivated choice of discrete spaces of a bilinear operator
27. **Jonathan Heffer (Dyson)**
    Flapping of hair fibres in a high aspect ratio jets

Two phase flow in the earth's core

J. Wong, C. Davies, C. Jones, P. Livermore and J. Mound

CDT in Fluid Dynamics, University of Leeds

Seismic observations have revealed the existence of a stably-stratified layer located above the inner core boundary (ICB). The density of this 'F-layer' increases with depth, in contrast to what is expected for a well-mixed core. How can such a layer be maintained? The liquid outer core is composed of an alloy, predominantly iron and a lighter component. Overall the Earth is cooling, therefore liquid iron in the outer core solidifies onto the inner core and lighter material remains in the liquid. The presence of buoyant lighter material would disturb the stably-stratified layer: how is this compatible with observations? Consequently, the project aim is to develop a self-consistent, fluid dynamical model of the convective process occurring in the F-layer that can explain the observations. The model will be built using a systematic approach, where thermal convection will be considered first. This will then be extended to simulate thermochemical convection. Previous thermal and thermochemical convection models have failed to explain how light material is transported out without disturbing the stable stratification. This project considers slurry layer as a possible scenario. Here, heavy iron-rich particles precipitate downwards from the liquid and freeze onto the inner core, displacing the lighter material upwards.

Assessment of sub-grid scale tensors in the context of very large eddy simulations

P. Chandramouli[1], D. Heitz[1,2], S. Laizet[3], E. Memin[1]

[1]INRIA, Rennes, [2]Irstea, Rennes, [3]Dept. of Aeronautics, Imperial College London

Large Eddy Simulations (LES) are effectively used as an alternative to Direct Numerical Simulations (DNS). However, computationally even LES are quite expensive to perform for complex flows. This brings into the forefront the concept of Very Large Eddy Simulations (VLES) involving coarser meshes and hence cheaper computations. An associated limitation with VLES is the accuracy and stability of the sub-grid scale (SGS) model to be used. This is the focus of this poster wherein several SGS models have been compared for the simple case of channel flow at coarse resolution. The development of LES SGS models has been an area of scientific research for many decades starting with Smagorinsky [1]. Recent developments in this field have produced many modern SGS models which have been shown to work better than classical models [2][3]. In this study, the classical models namely, classic Smagorinsky as well as its dynamic version along with the WALE model, have been compared with the contemporary uncertainty based models developed by [2] and the implicit LES model developed by [3] in the context of VLES. The accuracy of the statistics have been analysed by comparison with the channel flow data of [4].

Reference

1. Smagorinsky, J. "General circulation experiments with the primitive equations. i. the basic experiment." Mon. Weather Rev. 91, 99 (1963).
2. Memin E. "Fluid flow dynamics under location uncertainty." Geophysical & Astrophysical Fluid Dynamics, 108, 119-146 (2014).
3. Eric Lamballais, Veronique Fortune, and Sylvain Laizet. "Straightforward high-order numerical dissipation via the viscous term for direct and large eddy simulation." Journal of Computational Physics, 230(9):3270-3275 (2011).
4. R. D. Moser, J. Kim, and N. N. Mansour. "Direct numerical simulation of turbulent channel flow up to Re=590." Physics of Fluids, 11(4):943-5, 04 (1999).

Numerical simulations of turbulent flow and heat transfer in ladle degassing

J. Daiabai and S. Gao

Dept. of Engineering, University of Leicester

A three-phase model (Steel/Slag/Gas) based on the fundamental transport equations has been developed to study the secondary steel refining process. The current project is a continuation of a research carried out by Al-Harbi et al (2007) in University of Leicester. In the previous research, the Computational Fluid Dynamics (CFD) analysis using Fluent was carried out to predict the flow and thermal fields of the gas-stirred ladle system, and the thermodynamic analysis package MTDATA from the National Physical Laboratory (NPL) was dynamically linked to the CFD model to predict the mass transaction at the steel/slag interface during the refining process. The three-phase flow structures in the ladle had been analyzed and were found consistent with the published data. However, the previous study employed the traditional time-averaged k-e model to represent the turbulence features of the flow, which cannot reveal the instantaneous flow structure details. In the current project, a more advanced approach using the Large Eddy Simulation (LES) techniques is used to predict the transient flow dynamics of the system. Some selected LES results will be presented at the conference, and their dynamics and typical features will be analyzed in details.

Modelling oil droplet formation in turbulent oil plumes

R. Philip

University of Oxford

In a deep-sea oil spill, a broken pipe hundreds of metres below the sea surface releases a highly turbulent jet of oil into the water. It is advantageous to use the jet's vigorous turbulent mixing to break up the oil drops in the jet and produce smaller drops. These smaller oil droplets can then be eaten by microbes in the sea, and thereby are removed from the water column before reaching the surface.

Modelling oil droplet breakup involves examining and unifying disparate macroscopic and microscopic scales. On the large scale, we model the turbulent jet. On the smaller scale, we consider the droplets which are deformed and broken up by eddies of different sizes. These scales are connected by breakup parameters, which are used in the microscopic droplet breakup models and depend on macroscopic flow properties. By combining these macroscopic and microscopic models, we produce droplet population distributions. We use these models to provide insights into the factors affecting droplet breakup.

Fluid dynamics of cumulus convection at the cloud-scale

S. Boeing[1], D. Parker[1], A. Blyth[2], A. Stirling[3] and D. Dritschel[4]

[1]University of Leeds, [2]National Centre for Atmospheric Science, University of Leeds, [3]UK Met Office, [4]University of St Andrews

Climate Models are highly sensitive to the representation of cumulus clouds. These models depend on a statistical representation (parametrisation) of cumulus convection, but the fundamental behaviour of the clouds is still poorly known. In this work, a hierarchy of Large Eddy Simulations is used to explore the processes which govern the dynamics of individual clouds. These simulations include buoyant moist bubbles in saturated and unsaturated environments with varying stratification.

We compare the Large Eddy Simulations to integral models of moist thermals and plumes, and explore the role of phase transitions and pressure drag. A key difference with dry convection is that mixing between the updraughts and their environment generates evaporation and hence negatively buoyant air, which needs to be accounted for. Updraught size depends on a subtle balance between entrainment processes and the detrainment of this negatively buoyant air. Besides these comparisons to integral models, we also discuss ongoing work to improve the representation of small-scale flow in LES and Numerical Weather Prediction models. This is achieved by using Lagrangian particles to represent part of the transport of buoyant and moist air.

The Interaction of buoyant magnetic structures with convective plumes

A. Ali

City, University of London

Motivated by a desire to greater understand dynamics inside the Sun, I have considered a quasi 2D model to examine how magnetic structures in the tachocline interact with descending convectively driven plumes. Here I present some of the results from my calculations as well as discuss the implications of the findings.

Mathematical modelling of blood and interstitial fluid in a poroelastic model of the liver

M. Argungu and J. H. Tweedy

Dept. of Bioengineering, Imperial College London

The liver has a natural ability to regenerate its tissue, except in the case of severe or repeated damage often caused by liver diseases. Liver diseases can also cause an increase in portal blood pressure and change the liver tissue permeability, resulting in an increased outflow of excess interstitial fluid across the surface of the liver and into the surrounding peritoneal cavity. The abnormal accumulation of fluid in the peritoneal cavity is known as ascites and is characterized by large abdominal girth, abdominal pain and discomfort.

The aim of our research is to mathematically model the microcirculation of blood and interstitial fluid in the liver, so as to investigate the changes in vasculature that lead to the formation of ascites. We have developed a dual-porosity, dual-permeability deformable model of the liver. Using Darcy's law and mass conservation, we obtained equations for blood and interstitial fluid flow, as well as the rate of lymph formation in the liver. This gave the pressure distribution and magnitude of flow velocity across the liver tissue, in both healthy and unhealthy cases, as well as an estimate of the total outflow of fluid through the liver surface.

Instabilities in shear-thinning fluids

H. A. Castillo and H. J. Wilson

Dept. of Mathematics, University College London

We consider the linear stability of channel flow of a shear-thinning viscoelastic fluid, motivated by recent experimental work in which an instability is seen at a reproducible critical flow rate.

A previous study [Wilson & Loridan, Linear instability of a highly shear-thinning fluid in channel flow, JNNFM, 2015] investigated this system using a modified UCM model in which the shear modulus and relaxation time were both empirical power-law functions of the instantaneous shear rate. An instability was found, showing a partial match to experiments.

A natural question is whether the mechanism of this instability is truly elastic or principally a result of the strong shear-thinning. To address this, we have added a solvent viscosity which shear-thins at the same rate as the UCM viscosity, thereby introducing one new dimensionless parameter, beta: the ratio between solvent viscosity and polymer viscosity.

The previous work (beta=0) found a minimum in the critical Weissenberg number when the shear modulus is independent of shear-rate (m=n). In our new model, although we observed a similar minimum, as beta increases there is a displacement in the minimum value to parameters where the modulus is shear thickening (n<m), and the flow is more unstable than for beta=0.

Rates of chaotic mixing in models of fluid devices

H. Kreczak, R. Sturman and M. Wilson

CDT in Fluid Dynamics, University of Leeds

In non turbulent fluid flows the mixing of a passive scalar is accelerated under the action of chaotic advection. The exponential growth of fluid elements in chaotic flow fields makes such problems difficult to study using grid-based CFD methods, hence a dynamical systems approach is used. The use of mathematical models is employed to study how the underlying dynamics of stirring effects the rate of mixing, for example, the presence of boundaries have been shown to slow the expected exponential mixing rate of chaotic behaviour to algebraic. However, when considering the process of advection and diffusion these models are very abstract; such as 1D lamellar models or special cases of toral diffeomorphisms, and extending to more realistic models of fluid mixing devices increases the complexity.

Through a combination of mathematical analysis and numerical simulation, this project aims to investigate the interplay between stirring and diffusion in more accurate models of fluid mixing device. By first studying the advantages and limitations of current methods in modelling and measuring mixing, an extension of methods to more complicated mixer designs or the development of new approaches, will lead to the establishment of criteria for optimising mixing under certain conditions.

Using data assimilation techniques to validate fluid flow models

G. Madho, S. Tobias[1], C. Jones[1], S. van Loo[2] and W. Arter[3]

[1]Dept. of Applied Mathematics, University of Leeds, [2]School of Physics and Astronomy, University of Leeds, [3]Culham Centre for Fusion Energy, University of Leeds

Many studies run high resolution simulations to compare results against experimental data. In our study we aim to keep the accuracy of high resolution runs while reducing computational cost. This can be achieved using a data assimilation method which combines observations and numerical results to obtain new initial conditions. Here we use an Ensemble Kalman Filter (EnKF) to combine low resolution runs with experimental data. The Lorenz Model has been used to cross-validate our version of EnKF written in Python against the Matlab version available online at http://enkf.nersc.no/, which was originally developed for oceanography. We aim to use EnKF to assimilate rotating annulus data for experiments done at the Atmospheric, Oceanic and Planetary Physics (AOPP) department of the University of Oxford using the MORALS code created at AOPP. The aim of the project is to test the cost-effectiveness and the speed of the technique, as well as its predictive power in experiments exhibiting increasingly complex flows. This work will be then considered for application to plasma physics at Culham Centre for Fusion Energy (CCFE).

Spinning the Earth's pole with the oceanic turbulence: the effect of a double-gyre on Chandler wobble

S. E. Naghibi[1], S. A. Karabasov[1] and M. A. Jalali[2]

[1]Queen Mary, University of London, [2]University of California, Berkeley

One of the mechanisms of the Earth's Chandler wobble excitation is associated with the ocean dynamics. In this work, a hierarchy of mathematical models for simulating the effect of a mesoscale oceanic current corresponding to Gulf Stream and its northern extension towards Europe through the angular momentum transfer to the Chandler wobble is considered. The models range from a highly idealised depth-averaged semi-analytical ocean circulation model to a multi-layer quasi-geostrophic ocean model in the eddy-resolving regime. A new multiscale depth-averaged semi-analytical model is suggested which is based on the idea of space-time scale separation to capture both the integral angular momentum and the kinetic energy of the reference multi-layer quasi-geostrophic simulation. Using the new multiscale semi-analytical model, a feedback of the Chandler wobble onto the ocean dynamics is taken into account. The computed Chandler wobble excitation functions with and without the feedback effect are compared with the Oceanic Angular Momentum (OAM) observations for polar motion excitation around Chandler frequency.

Parametric study on personalised ventilation systems using computational fluid dynamics

N. Gilkeson, C. Noakes, A. Khan and M. F. King

University of Leeds

There is a complex relationship between indoor air quality, health, personal comfort and building energy use. A personalised ventilation (PV) system creates a micro-climate around an individual, and has the potential for improving personal comfort, indoor air quality and productivity of building occupants whilst simultaneously lowering energy consumption.

An extended computational fluid dynamics (CFD) parametric study of PV using ANSYS Fluent is conducted, with simulations of a nozzle delivering clean air to a seated computational thermal manikin (CTM) in a mechanically ventilated chamber. The study considers parameters such as location, distance and angle of the nozzle, ventilation rate and temperature.

The PV nozzle is aimed towards the CTM breathing zone with initial test locations chosen for comparison with literature. Additional simulation locations are chosen using a modified Latin hypercube Design of Experiments. The resulting parametric design domain is explorable with a moving least squares metamodel.

Results are presented in terms of thermal comfort (PMV) and dissatisfaction levels (PPD), with the mean age of air and air quality measures calculated in the CTM breathing zone. In addition, the effects of the PV flows on the CTM convective boundary layer (CBL) are explored.

Modelling novel heat exchangers for aircraft thermal management

E. Greiciunas, D. Borman and J. Summers

University of Leeds

Future aircrafts face an appreciable technical challenge to design thermal management systems (TMS) of adequate capacity within installation constraints (e.g. size, efficiency, thermal signatures, etc.). Various heat exchangers (HE) are key components within aircraft TMS, and the achieved heat transfer performance for a given size, and flexibility of physical shaping, have a strong determining effect on overall system performance and feasibility. Novel manufacturing techniques (e.g. ALM) will be used to provide exciting opportunities for innovative HE geometry. Numerical approaches are used to develop validated models of existing and conceptual HE designs using experimental data. This study explores the physical aspects affecting the HE performance at all ranges of design feature levels. Current focus of the project is to learn how to predict the flow within the HE when it enters transitional Reynolds regime. The project involves working alongside BAE Systems and a specialist HE supplier.

Convectively forced gravity waves and their sensitivity to heating profile and atmospheric structure

O. Halliday[1], D. Parker[1], S. Griffiths[2], S. Vosper[3] and A. Stirling[3]

[1]Institute for Climate and Atmospheric Science, University of Leeds, [2]Dept. of Applied Mathematics, University of Leeds, [3]UK Met Office

It is observed that atmospheric deep convection is highly organised on the mesoscale (O(100kms)), and widely accepted that gravity waves provide a mechanism for the aggregation of cumulonimbus storms. Despite this, the radiation of gravity waves in macro-scale models, which are typically forced at the grid-scale by mesoscale parameterization schemes, is not well understood. We present here theoretical work directed toward improving our fundamental understanding of convectively forced gravity wave effects at the mesoscale, in order to begin to address this problem. Starting with hydrostatic, non-rotating, 2D, equations for a deep atmosphere, we find a radiating, analytical solution to prescribed sensible heat forcing for both the vertical velocity and potential temperature response. Both steady and pulsed heating with adjustable horizontal structure are considered. From these solutions we construct a simple model capable of interrogating the spatial and temporal sensitivity to prescribed heating functions of the remote forced response. We find that the macro-scale response to convection is highly dependent on the radiation characteristics of gravity waves, which are in turn dependent upon the temporal and spatial structure of the source, and upper boundary condition of the domain.

Experimental and numerical modelling of aerated flows over stepped spillways

J. van Alwon

University of Leeds

Stepped spillways have been shown to be efficient overflow structures for reservoirs. They dissipate large amounts of energy, which prevents scour at the foot of the dam, reducing the size of the required stilling basin. They also entrain large amounts of air into the flow which prevents cavitation and plucking damage.

Current practice is to use physical scale models to predict flow characteristics over stepped spillways. While they have limitations, reservoir engineers prefer physical to numerical models as these limitations are well understood. Numerical modelling has the potential to predict flows over stepped spillways more accurately and at a lower cost than physical models. However, the accuracy of these models must be proven before they are used for the design and maintenance of stepped spillways.

This project will aim to evaluate the ability of several numerical modelling techniques to predict free surface aeration over stepped spillways. The models will be validated using free surface profile and pressure measurements from experimentation. The experiments will be designed so that the steps are close to prototype scale with an air entrainment device placed upstream of the steps. The flow will be either aerated or non-aerated so the numerical modelling results and pressure distributions can be compared.

Computational fluid dynamics modelling of liquid-liquid slug flow in capillaries

J.A. Figueroa Rosette[1], A.D. Bursn[1] and B. Bennett[2]

[1]School of Chemical and Process Engineering, University of Leeds, [2]School of Computing, University of Leeds

Microchannel reactors or microreactors have been introduced as a promising alternative to conventional reactors, allowing for higher mass transfer rates and product yields in multiphase flow systems. Many studies have focused on the analysis and modelling of gas-liquid flow in capillaries or microchannels, but very few on liquid-liquid flow systems. The flow patterns and mass transfer limitations of liquid-liquid reactions represent a challenge for both experimental and numerical studies, due to the complex phenomena occurring along the process. Detailed understanding of the interfacial transport phenomena is essential for the design and optimisation of such systems.

Computational fluid dynamics (CFD) is used to study the flow behaviour and mass transfer of liquid-liquid slug flow in a circular capillary with a view towards the application in monolithic and packed bed reactors. The role of the interfacial forces in liquid-liquid slug flow is investigated under the limits of high and low viscosity ratios in low Reynolds number pressure driven flow. The velocity profiles and shear stress profiles across the fluid-fluid interface are calculated for the case of elongated droplets dispersed in a continuous phase in a capillary. The results are compared with experimental data from the literature and are also compared with similar theoretical approaches. The CFD model developed in this work will be applied in the mixing and reacting stage of bio-fuel technologies, namely in biodiesel production for process optimisation and scale- up purposes.

Adjoint based sensitivity analysis in low order thermoacoustic networks

J.G. Aguilar and M.P. Juniper

Dept. of Engineering, University of Cambridge

Strict pollutant emission regulations are pushing gas turbine manufacturers to develop devices that operate under conditions that encourage the appearance of combustion instabilities. Although methods to predict and control unstable modes inside combustion chambers have been actively developed in the last decades, in some cases they are computationally expensive. Sensitivity analysis aided by adjoint methods gives valuable gradient information at a very low computational cost. This study introduces the adjoint methods and their application in wave-based low order network models to predict and control thermoacoustic oscillations. For a configuration that considers a steady base flow, a nonlinear eigenvalue problem is derived, and adjoint methods are used to obtain the sensitivities of the system to small perturbations. These perturbations are considered to be in both steady base flow equations and the time varying perturbation equations. Hence, base flow sensitivities, base state sensitivities (to internal parameters) and sensitivities of the eigenvalue to feedback mechanisms are studied. Coupling between base flow sensitivities and the sensitivities to feedback mechanisms is performed to analyze the overall effect of mass injection, drag devices or heat addition into the system.

Behavior of turbulence models in low Reynolds number flows and near wall regions

D. Govardhan[1] and S. Rao[2]

[1]Dept. of Mechanical Engineering, Osmania University, [2]CurlVee TechnoLabs, India

At low Reynolds numbers the fluid flow near the walls is a complex phenomenon as the flow is viscosity affected and boundary layered poses both modelling and numerical challenges. The Viscous effects on the turbulence need to be accounted for often by including ‘near-wall damping’ terms and other source terms in the modeled transport equations. Since steep gradients of velocity and turbulence statistics occur across the viscous layer, very fine grid is needed to provide adequate numerical resolution to understand the physics involved. In the present paper it is consider that Low-Reynolds-number modelling, where the viscous layer may be is resolved numerically, and viscous effects can be included in the turbulence model. 2-D zero pressure gradient flow over a flat plate and 2-D wedge flow are considered from subsonic to supersonic Mach numbers and 3-D subsonic flow over bump is investigated and results are presented. The models exhibit a range over which they behave as if the flow is in transition, in the sense that the flow is neither laminar nor fully turbulent. The SST model well defined transition location, whereas the SA model does not. The models are predisposed to delayed activation of turbulence with increasing free stream Mach number.

The models can be made to achieve earlier activation of turbulence by increasing their free stream levels, but too high a level can disturb the turbulent solution behavior. Both SST and SA models are established to be incapable of predicting re-laminarization. Eddy viscosities remain weakly turbulent in accelerating or laterally- strained boundary layers for which experiment and direct simulations indicate turbulence suppression. Large eddy simulation is also performed and the main conclusion is that these models are intended for fully turbulent high Reynolds number computations. The near-wall asymptotic behavior and turbulence kinetic energy were discussed with respect to the Reynolds-number dependence and an influence of the computational size. It is concluded that low Reynolds number k–ε turbulence model is more accurate than the standard k–ε turbulence model, and LES model can demonstrate and predict the near wall effects of low Reynolds numbers.

CFD simulations of flow in polydisperse granular media with application to internal erosion

C. Knight[1], C. O'Sullivan[2], B.G.M. van Wachem[3] and D. Dini[3]

[1]CDT in Theory and Simulation of Materials, Imperial College London, [2]Dept. of Civil and Environmental Engineering, Imperial College London, [3]Dept. of Mechanical Engineering, Imperial College London

Internal erosion is an important failure mechanism of large dams accounting for around half of all failures of embankment dams (Foster et al. 2000). Internal erosion occurs in polydisperse granular media when the fine particles migrate under the influence of a hydraulic gradient ultimately resulting in loss of strength and failure of the material. Knowledge of the momentum coupling between the fluid and solid phases in the type of densely packed materials that are found in zoned embankment dams comes largely from empirical correlations of fluidised bed experiments (Ergun 1952, Di Felice 1994) which are not sensitive to the particle size distribution. Computational fluid dynamics (CFD) simulations using the Immersed Boundary Method (IBM) following the direct-forcing scheme of Uhlmann (2005) have been carried out in order to resolve pore scale flow conditions and allow investigation of the drag forces acting on individual particles. Initial validation of the IBM in the low Reynolds number and high solids volume fraction regime was carried out with simulations of flow through ordered arrays of spheres which are compared with the well known results of Zick and Homsy (1982). Particle packings have been prepared at a confining pressure of 100 KPa with linearly graded particle size distributions with a range of coefficient of uniformity values (Cu = 1.01, 1.2, 1.5, 2.0 and 3.0) in order to investigate the influence of Cu on drag and permeability. Comparison is made between the results of these simulations and predictions of the drag forces and permeabilities from empirical correlations widely used in industry and research.

Foster, M., Fell, R., & Spannagle, M. (2000). The statistics of embankment dam failures and accidents. Canadian Geotechnical Journal, 37(1992), 1000–1024. http://doi.org/10.1139/t00-0301932294900116

Ergun, S. (1952). Fluid flow through packed columns. Chemical Engineering Progress, 48(2), 89–94. http://doi.org/10.1029/JB088iS01p0B353

Felice, R. Di. (1994). The voidage function for fluid-particle interaction systems. International Journal of Multiphase Flow, 20(l), 153–159. Retrieved from http://www.sciencedirect.com/science/article/pii/0301932294900116

Uhlmann, M. (2005). An immersed boundary method with direct forcing for the simulation of particulate flows. Journal of Computational Physics, 209(2), 448–476. http://doi.org/10.1016/j.jcp.2005.03.017

Zick, A. A., & Homsy, G. M. (1982). Stokes flow through periodic arrays of spheres. Journal of Fluid Mechanics. http://doi.org/10.1017/S0022112082000627

CFD-Experimental comparison of a Broady 3500 series safety relief valve.

S. Taggart[1], A. Mowforth[2] and W. Dempster[1]

[1]Strathclyde University, [2]Broady Flow Control, Hull

Safety relief valves are self-actuating devices used to protect pressurised systems from the dangers of over-pressure. They can be found throughout most industries in some form or another and they are vital to ensure the safe working of pressurised systems. Due to the self-actuating nature of a safety relief valve, accurate design of their internals is crucial to ensure they function properly. To gain a deeper understanding of the fluid dynamics occurring within them, advanced fluid modelling techniques may be used to simulate the fluid flow path throughout these valves. Utilising computational fluid dynamics (CFD) tools should help in gaining a greater understanding of how these valves operate.

It is intended to use CFD simulations to investigate the fluid behaviour inside a safety relief valve supplied by Broady Flow Control. In the design and analysis stage of a safety relief valve, two of the main characteristics of interest are the mass flow rate and disc forces. These two parameters will be calculated using CFD software and compared to physical values obtained through experiment. The outcome of this study should determine whether CFD is a capable design and development tool for use on a Broady Flow Control safety relief valve.

An immersed boundary method for turbulent flow over blocks

Y.X. Chen, K. Djidjeli and Z.T. Xie

Aerodynamics and Flight Dynamics Group, University of Southampton

Large Eddy Simulation (LES) is applied to calculate incompressible turbulent flow over blocks in a channel. A 6m*6m*6m domain is used. Inside the domain, 3 (streamwise) * 2 (lateral) building-like blocks are evenly distributed where periodical boundary conditions are specified in streamwise and spanwise directions. Immersed Boundary (IB) method with further simplifications to represent the wall conditions of the blocks is implemented at a moderate Reynolds number (around 1000) which based on maximum free stream velocity and block height.

An in-house Fortran code is used to develop IB method and it is validated and compared with conventional LES results and wind tunnel measurements. Compared to conventional methods, it is a challenge to use IB method to predict accurate flow field in the vicinity of the solid wall. This will be tackled in the research. The developed method can be applied in a large range of areas, such as the prediction of flow and turbulence around over an array of buildings, oscillating bridges, moving high speed train, heaving/pitching wings and rotating wind turbine blades.

Hytrac: a front-tracking code for the simulation of multi-fluid systems

R. Ramasamy and B. Tully

First Light Fusion, Oxford

The poster describes a front-tracking code that is used to simulate compressible fluid systems containing multiple components. Front-tracking represents the interface between two fluids by an explicit boundary, implemented using a Lagrangian grid that is advected around a higher dimensional Eulerian domain. The interface separates the system into distinct regions, each containing a single fluid with a unique equation of state. Numerical codes of this nature perform well in the simulation of compressible flow instabilities, such as the Richtmyer-Meshkov instability. The code implementation makes use of the versatility of C++ to focus on architectural concepts that maintain modularity in algorithms and data structures. This poster aims to outline the benefits and potential pitfalls of this approach, in contrast to more common strategies that boost performance at the cost of additional development time and code modularity. Simulation results of Richtmyer-Meshkov instabilities will be discussed in the context of these methods.

Simulation of aircraft wing hitting turbulence during heaving

J. Chalke and Z.T. Xie

Aerodynamics and Flight Dynamics Group, University of Southampton

To minimise the risk of aircraft structural failure, an in-depth knowledge of the loading and fatigue process is necessary. The loading of a wing can be very unpredictable due to erratic changes in airflow during turbulence and cross-flow conditions. Aircraft wings are usually subject to accelerated life cycle tests which include erratic changes in loading, this process would be far more effective if the effects of erratic wing loading were better understood. As a result, much research is now being undertaken into wing aerodynamics in extreme conditions. This includes high levels of turbulence, large erratic manoeuvres and pitching/heaving; producing highly unsteady flow and transient effects such as vortex shedding.

During this project the aerodynamic performance parameters of a wing subject to large turbulent eddies while periodically heaving will be examined and quantified to support the research being conducted into erratic dynamic loading. Large eddy simulation will be used to investigate the aerodynamic effects of a homogenous wing undergoing a prescribed oscillatory heaving motion with Reynold's number ranging from Re =50,000 to Re =300,000 at moderate angle of attack. Validation of the adopted methods will be undertaken on a static case to confirm the credibility of the results obtained. The phenomenon of leading edge vortex shedding and dynamic stall will be examined with respect to performance parameters. Finally, the effects of implementing a synthetically generated turbulence on the heaving wing will also be quantified.

Effect of boundary layer of balance platform on an under floor external balance data of a half model

A.U. Haque and W. Asrar

International Islamic University Malaysia

While using an under floor external balance to measure the forces acting on a semi span wing model, there is always a requirement to keep some gap between the model and the balance platform of subsonic wind tunnel. There seems to be no universal school of thought to define that gap/spacer height for half models attached with the external balance's platform. Moreover, no hypothesis is there which drives the difference in the results obtained from the balance. Moreover, the balance measurement data at low speed is missing for semi span model of a wing in which the said gap is equal to average value of the estimated boundary layer thickness. The present study focuses on the quantitative comparison of the balance data by defining physical gap between the model and the balance platform. Based on this comparison study at different Reynolds numbers for an array of positive as well as negative angle of attacks, it was found that the said gap has a significant effect on the balance data at moderate and high angle of attacks; specially on the windward side of the wing. It has also been found that the change in velocity has also significant contribution towards overall change in the aerodynamic and static longitudinal stability coefficient.

MHD model for liquid metal battery description

V. Bojarevics, A. Tucs and K. Pericleous

University of Greenwich

Liquid metal batteries are a new concept for grid-scale energy storage. The three density-segregated liquid layer structure of the battery provides a variety of advantages in comparison to existing battery technologies: fast kinetics, long life-time, large current densities, easy recycling beneficial for energy storage of fluctuating renewable sources (wind, solar, tidal, etc.). The concept of liquid metal batteries and corresponding MHD effects are in a close similarity to aluminium electrolytic production. The aim of this research is to develop a numerical model based on a spectral method approach for the three density-stratified electrically conductive liquid layers using 3d and shallow layer approximation taking into account specific MHD effects during periods of battery charge/discharge. The results for the 3d electric current distribution, the mixing velocities for shallow liquid layers and the interface dynamics will be presented.

Bi-directional coupling of particulate flow – a generalized framework for the physically motivated choice of discrete spaces of a bilinear operator

S. Höllbacher and G. Wittum

Goethe-Center for Scientific Computing, Goethe-Universität Frankfurt

In the biological system of a neuronal cell, tiny vesicles are supposed to move as rigid “membrane-balls” within the surrounding cytosol. Therefore, a bi-directional coupled method was derived for the direct numerical simulation (DNS) of particles in a fluid.

Since most methods introduce external forces to describe the interaction between fluid and particles, an interative solving procedure is necessary. The use of a nite volume discretisation of the navier stokes equations and its theoretical treatment as a so-called Petrov-Galerkin method enabled us to develop a consistent and bi-directional coupled model, naturally incorporating the particles into the equations. The resulting discrete scheme can even be prooved to be stable. Consequently, the solving procedure is conducted in one step and the bi-directional coupling therefore preserved. Second order convergence in space was reached.

Inspired by the key ideas of our model, further theoretical work was done on the underlying, mathematical concepts: With a generalized view on the trial space of a bilinear operator a physically motivated criterium for the choice of discrete spaces was formulated, depending on the particular application. This framework even serves as a tool to compare a nite element with a finite volume method.

Flapping of hair fibres in a high aspect ratio jets

J. Heffer

Dyson Technologies Ltd.

During hair styling heated jets are used to dry and heat hair to produce a desirable end style. Hair fibres in these jets can flap resulting in tangling and a reduction in alignment of fibres in the end style, resulting in a poorer styling outcome for the user. We aim to further understand this flapping motion. The literature relating to fluid structure interactions relevant to long cylinders was reviewed, De Langre et al 2007 show that the fibres may be susceptible to divergence and flutter type instabilities. We also developed a simple finite element model for the force response of a hair fibre in a 2D jet with synthetic turbulence. This is compared to experimental measurements of the movement of a single fibre in a high aspect ratio jet. We compare the movement predicted by the model and the experiment, see fig. 1 and 2. The model requires a higher turbulence intensity to achieve similar motion as seen in the experiment. This may be due to the role of fluid structure instability or simplifications in the model.

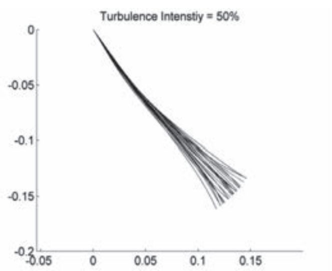


Fig 1 Overlaid images of single hair flapping in a 2D jet from the model.

Fig 2 Experimental images of single hair flapping in jet overlaid.

Notes