UK Fluids Conference

Contents

Page 4-8: Conference schedule

Page 10-13: Plenary speakers

Page 14-77: Sessions

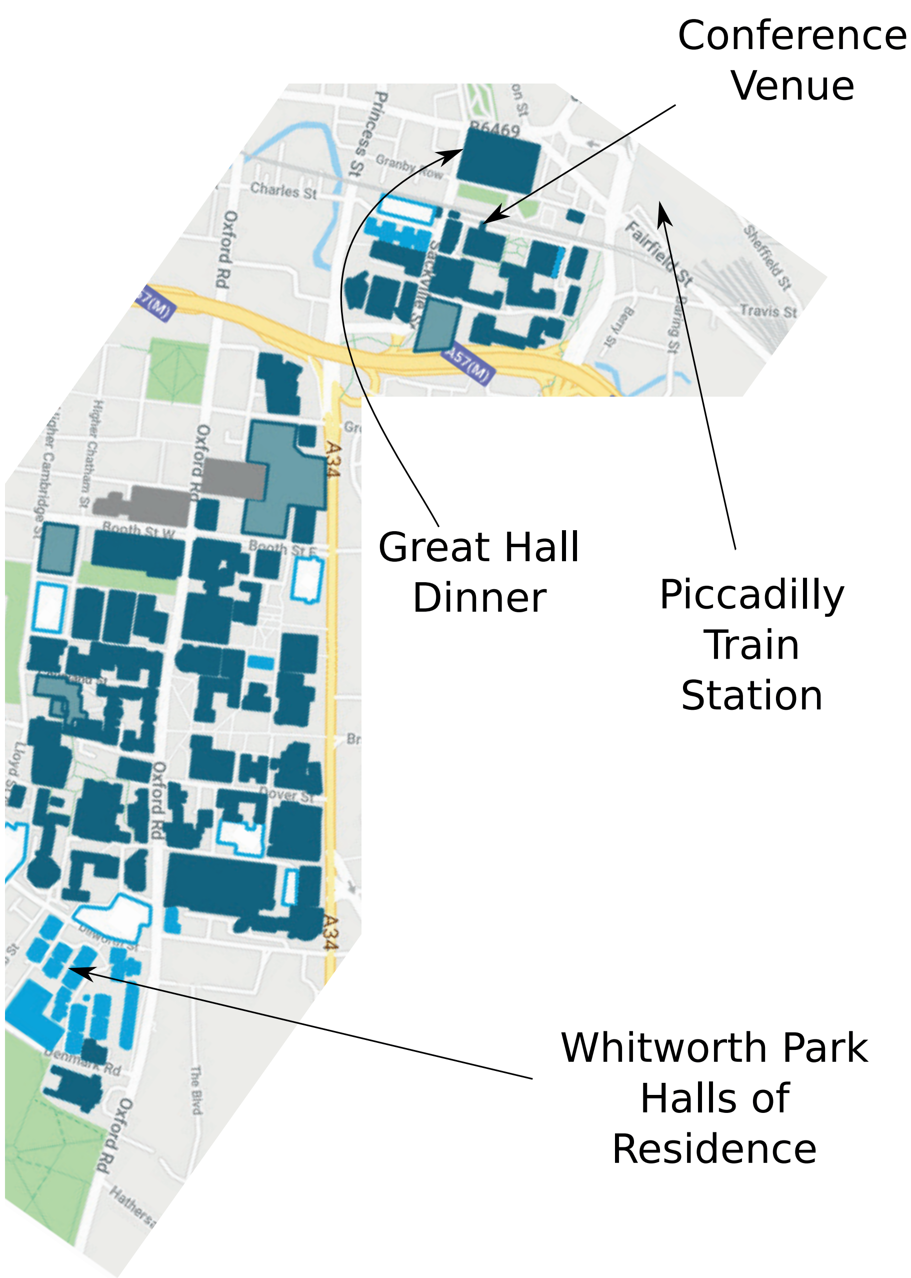
Page 78-104: Posters

Page 106-107: Oral presentation index

The cover image was acquired by Landsat 7's Enhanced Thematic Mapper plus (ETM+) sensor and provided by the USGS EROS Data Center Satellite Systems Branch as part of the Earth as Art II image series.

University of Manchester

Campus map



Tuesday 4 September

Venue: C2, Renold Building		Time	Venue: C9, Renold Building	
Lunch (meet the UK Fluids Network SIG leaders)		12:00 - 13:30	Lunch (meet the UK Fluids Network SIG leaders)	
Welcome		13:30 - 13:40		
PLENARY SESSION 1				
Olivier Pouliquen	An active granular liquid at the origin of the sensitivity of plants to gravity	13:40 - 14:25		
SESSION S1	Aerodynamics 1		SESSION S2	Waves & free surfaces 1
Quan (Manchester)	Attenuation of flow separation over a double ramp using herringbone riblets at Mach 5	14:30	Deakin (Manchester)	Optimal coordinate transformations for the perfectly matched layer method to model acoustic wave propagation in fluids
Gillespie (Southampton)	Simulations of strong shock wave / turbulent boundary-layer Interactions	14:42	Oloo (Keele)	Sub-surface waves on the ocean under strong winds
Taylor (Imperial)	Effects of surface roughness on boundary layer transition on a blunt body in a hypersonic flow	14:54	Booker (Leeds)	Wave attractors in stratified fluids
Fisher (Manchester)	Sensitivity of background-oriented Schlieren (BOS) applied to hypersonic flow	15:06	Parsons (MUN)	Acoustic-gravity waves generated by impulsive sources at the ocean surface - a field study
Rees (Imperial)	Hypersonic flow-fields around faceted shapes	15:18	Nethercote (Manchester)	High contrast approximation to penetrable wedge problems
Jankee (Southampton)	Impact of orifice lip ratio on the formation and evolution of synthetic jets	15:30	Wilson (Strathclyde)	The fluid dynamics of anti-surfactant solutions
Kedward (Bristol)	Gradient-limiting shape control for efficient aerodynamic optimisation	15:42	Sykes (Leeds)	Internal flows during the coalescence of a free and a sessile droplet
Break		15:54 - 16:30	Break	
SESSION S3	Aerodynamics 2		SESSION S4	Interfacial flows
Baddoo (Cambridge)	Analytical models for propulsion strategies	16:30	Wang (Northumbria)	Biaxially shaping droplets with localized elastic pattern bifurcation
Dai (Edinburgh)	Oscillating foils at moderate Reynolds numbers	16:42	Schofield (Strathclyde)	The influence of the thermal properties of the system on the lifetime of an evaporating droplet
Zurman Nasution (Southampton)	3D influence on propulsive flapping foil	16:54	Uppal (Imperial)	Modelling the evolution of fully-gelled sessile drops
Mahfoze (Imperial)	Bayesian optimisation for skin-friction drag reduction of a spatially evolving turbulent boundary layer using wall blowing	17:06	Nazarzadeh (Glasgow)	Controlling the size of acoustically nebulised droplets by pinning surface waves for precise delivery of aerosolised medicine
Xu (Manchester)	Boundary layer development and riblet wavelength effect of laminar flow over convergent-divergent riblets	17:18	Kahouadji (Imperial)	Numerical simulations of drop impact on interfaces
Williams (Manchester)	Stability of short spanwise scale injection flows	17:30	Schirrmann (Manchester)	Grouping in 3-phase systems in microfluidic devices
Lloyd (Leeds)	A numerical and experimental investigation of the effects of sharkskin denticle geometry on a flat plate boundary layer	17:42	Bhagat (Cambridge)	On the origin of hydraulic jump
Rabey (Imperial)	Experimental 3D mapping of reflected shock - boundary layer interaction unsteadiness	17:54	Dallaston (Coventry)	Computation of discrete self-similar branches of thin film rupture solutions
Poster reception		18:10 - 20:00		

Wednesday 5 September

Venue: C2, Renold Building		Time	Venue: C9, Renold Building	
PLENARY SESSION 2				
Angela Busse	Influence of roughness topography on wall-bounded turbulence	9:00 - 9:45		
SESSION S5	Turbulence 1		SESSION S6	Granular flows
Hetherington (Leeds)	Freestream turbulent inflow for CFD	9:48	Edwards (Manchester)	Applying particle-size segregation theory to the erosion-deposition dynamics of granular avalanches
Agrawal (Liverpool)	Instantaneous velocity and wall shear stress measurements during low-drag events in turbulent channel flow	10:00	Barker (Manchester)	Well-posed continuum modelling of granular flows
van Alwon (Leeds)	Experimental and numerical modelling of aerated flows over stepped spillwayss	10:12	Rocha (Manchester)	Viscosity sets the width of self-channelised granular flows
Dallas (Oxford)	Spontaneous mirror-symmetry breaking due to inertial waves in rotating turbulence	10:24	Ozel (Heriot-Watt)	Gas-solid flows with tribocharging
Shin (Edinburgh)	Characteristics of fluid residence time in turbulent steady and unsteady round jets	10:36	Russell (Manchester)	Retrogressive failure of a static granular layer on an inclined plane
Golov (Manchester)	Coexistence of quantum and classical flows in quantum turbulence in the T = 0 Limit	10:48	Chalk (Leeds)	Numerical modelling of landslides: a dual particle smoothed particle hydrodynamics approach
	Tea/Coffee	11.00-11.30		Tea/Coffee
SESSION S7	Acoustics		SESSION S8	Complex fluids
Rona (Leicester)	Affordable detached eddy simulations of dual-stream jet structures related to shock cell noise	11:30	Vitasari (Aberystwyth)	Prediction of bubble dynamics during foam flow through a constricted microfluidic channel
Moschou (Manchester)	Understanding the noise generated by a jet engine: a study of the 3D diffraction by a quarter-plane	11:42	Münch (Oxford)	On yield stress of concentrated suspensions
Baker (Cambridge)	Silent owl flight: effect of boundary-layer on trailing-edge noise	11:54	Barlow (Durham)	Instability of pressure driven channel flows of shear-thinning viscoelastic fluids
Higgens (Southampton)	Investigation into the tip gap flow and its influence on ducted propeller tip gap noise using acoustic analogies	12:06	Escott (UCL)	Rheological properties of a solid sphere suspension in second order fluid using a single cell model
Petrie (Cambridge)	Acoustics in a lined duct with non-parallel boundary layer flow	12:18	Ng (Liverpool)	Capillary breakup extensional rheometry (CaBER) of graphene oxide suspensions
Garita (Cambridge)	Predicting and controlling thermoacoustic oscillations in a Rijke tube through data assimilation: an ensemble Kalman filter approach	12:30	Azahar (Leeds)	Numerical modelling of polymer blends using OPENFOAM
Priddin (Cambridge)	Vortex sound models for active flow control	12:42	Alahmadi (Keele)	Non-Newtonian models for a vertical draining free liquid film
	Lunch	12:54 - 13:45		Lunch

Wednesday 5 September continued

Venue: C2, Renold Building		Time	Venue: C9, Renold Building	
PLENARY SESSION 3				
Shahrokh Shahpar	Role of fluid simulation and optimisation in turbomachinery design	13:45 - 14.30		
SESSION S9	Turbulence 2		SESSION S10	Biological flows 1
Doohan (Imperial)	The state space of near-wall turbulence at infinite friction Reynolds number	14:36	Dvoriashyna (Genoa)	Aqueous flow in the anterior chamber of the rotating eye: a mathematical model
Wada (Imperial)	Turbulent dissipation and entrainment in planar jets	14:48	Dyson (Birmingham)	Lockhart with a twist: The effect of variations in mechanical properties through the plant cell wall thickness on growth and twisting
Sewerin (Braunschweig)	On the accuracy and convergence properties of the stochastic field method	15:00	Erlich (Manchester)	Physical and geometrical determinants of transport in feto-placental microvascular networks
Abdullah (Sheffield)	Re-building turbulent fields using 4D variational data assimilation: reproduction of instantaneous structures	15:12	Williams (Oxford)	Mathematical modelling of ureteroscope irrigation
Yu (Cambridge)	Data assimilation and parameter estimation of thermoacoustic instabilities in a ducted premixed flame	15:24	Kelly (Bath)	Transition to turbulence is delayed in shear-thinning blood analogs in contrast to Newtonian analogs
Ioannou (Imperial)	Entrainment and mixing enhancement in plasma-controlled turbulent jets	15:36	Padgett (Leeds)	An Eulerian-Lagrangian agent method model to predict fish responses to hydrodynamic cues
Duguid (Leeds)	Interaction between turbulent convection and tidal flows in stars and planets	15:48	Etzold (Cambridge)	Poroelastic response of soft hydrogels connected to a water reservoir with an evaporation surface
Break		16:00 - 16:30	Break	
SESSION S11	Flow simulation		SESSION S12	Biological flows 2
Kreczak (Leeds)	Rates of mixing by cutting and shuffling	16:30	Smith (Birmingham)	Accurate, efficient and easy modelling of biological Stokes flow
Matharu (Manchester)	The direct computation of time-periodic solutions of PDEs & applications to fluid dynamics	16:42	Bearon (Liverpool)	Enhanced sedimentation of elongated plankton in simple flow
Marino (Strathclyde)	An investigation of the effect of biomimetic tubercles on the drag of a flat plate	16:54	Katsamba (Cambridge)	Hydrodynamics of bacteriophages
Bartholomew (Imperial)	Simulating variable density flows in the low-Mach number limit with Incompact3D	17:06	Cupples (Birmingham)	Viscous propulsion in active transversely-isotropic media
Madho (Leeds)	Predicting chaotic behaviour in rotating fluids using data assimilation	17:18	Tyrrell (Birmingham)	Vesicle transport and cytoplasmic streaming in the pollen tube
Rhodes (Nottingham)	Extremum seeking control for drag reduction of a truck trailer	17:30	Cara Neal (Birmingham)	A finite element approach to spring network models of micro-swimmers
Harwood (Manchester)	Real-time, interactive CFD on heterogeneous mobile clusters	17:42	Gallagher (Birmingham)	Model-based image analysis of a tethered Brownian fibre for shear stress sensing
Padula (Coventry)	Experimental and numerical analysis of a swirling flow in a pipe with annular cross section	17:54	Fung (Imperial)	Stability of upflowing and downflowing gyrotactic microorganism suspensions in a three-dimensional vertical cylindrical pipe
Dinner		18:45 - late	Dinner	

Thursday 6 September

Venue: C2, Renold Building		Time	Venue: C9, Renold Building	
SESSION S13	Industrial applications		SESSION S14	Thin film & interfacial flows
Song (Strathclyde)	Predicting the effect of barnacle fouling on ship resistance and powering using CFD	09:00	Hazel (Manchester)	Fluid deposition and spreading on topography
Niotis (Strathclyde)	Numerical simulation of progressive flooding of a ship after damage	09:12	Morrow(Queensland)	A numerical scheme for controlling instabilities in Hele-Shaw flow
Viola (Edinburgh)	Tidal turbine hydrodynamics	09:24	Batchvarov (Imperial)	Numerical simulations of annular flows in the presence of surfactants
Zahari (Durham)	Modal analysis of wake flow behind a wind turbine in a weakly stable stratified atmosphere	09:36	Pihler-Puzovic (Manchester)	Viscous fingering in a radial elastic-walled Hele-Shaw cell
Faccetti (Leeds)	CFD modelling of mixing and dispersion from marine outfall discharges	09:48	Algwauish (Keele)	The spreading and stability of a cooling drop on an inclined pre-wetted substrate
Souppez (Solent)	On highly cambered thin circular arcs at low Reynolds numbers	10:00	Juel (Manchester)	Oscillatory fingering in an elasto-rigid Hele-Shaw channel
Greiciunas (Leeds)	Numerical analysis of a novel heat exchanger utilising additive layer manufacturing for aerospace applications	10:12	Benilov (Limerick)	Oblique liquid curtains
Gokstorp (Cambridge)	Shape optimisation of a gas exchange valve for the production of carbon nanotubes	10:24	Keeler (Manchester)	Transient dynamics of a bubble in a perturbed Hele-Shaw cell
Gorbatenko (Leeds)	The investigation of combustion characteristics of n-butanol/ gasoline blends.	10:36	Landel (Manchester)	Low-order models for the drag reduction of surfactant-contaminated superhydrophobic surfaces
Break		10:48 - 11:24	Break	
SESSION S15	Waves & free surfaces 2		SESSION S16	Geophysical flows
Jamshidi (UCL)	Coastal outflows into buoyant layers of arbitrary depth	11:24	Parkinson (Oxford)	Numerical simulations of convection in mushy layers
Park (Dundee)	How to generate very long waves	11:36	Wong (Leeds)	A slurry model of the F-layer at the base of Earth's outer core
Lane-Serff (Manchester)	Drag on pairs of square section obstacles in free-surface flows	11:48	Wilczynski (Leeds)	Stability of scrape-off layer plasma: a modified Rayleigh-Benard problem
Agrawal (Northumbria)	Leidenfrost engine: dynamics of rotating disks on turbine-like surfaces	12:00	Allen (Leeds)	Pulse propagation in quasi-laminar gravity currents
Launay (Northumbria)	Self-propelled droplet motion on gradient slippery liquid infused porous surfaces	12:12	Ashcroft (Leeds)	Predictability of tropical cyclones: a comparison of typhoon Haiyan (2013) and typhoon Hagupit (2014) using convection-permitting ensemble forecasts
McHale (Northumbria)	Manipulating droplets on slippery lubricant impregnated surfaces	12:24	Marshall (Leeds)	Turbulence in the body of pseudo-steady gravity currents
Gonzalez Hernandez (Imperial)	Receptivity of supersonic boundary layers over smooth and wavy surfaces to impinging slow acoustic waves	12:36	Harris (Manchester)	Rate-independent and viscoplasticity models for granular materials
Hussain (Manchester Met)	On new insights into the three-dimensional boundary-layer instability over broad rotating cones	12:48	McCorquodale (Reading)	Using 3D-printed analogues to understand the sedimentation of complex ice particle
Lunch		13:00 - 13:48	Lunch	

Thursday 6 September continued

Venue: C2, Renold Building		Time	Venue: C9, Renold Building	
SESSION S17	Experimental flows		SESSION S18	UKFN Thesis Prize Finalists
Davies Wykes (Cambridge)	Buoyancy effects on cross-ventilation	13:48	Brackston (Imperial)	Stochastic modelling and feedback control for efficient bluff body drag reduction
Gramola (Imperial)	Photogrammetric deformation measurement of adaptive shock control bumps	14:00		
De Domenico (Cambridge)	Unsteady temperature and composition measurements with LITGS	14:12		
Frank (Cambridge)	Two-phase plumes in a rotating environment	14:24	Chen (Liverpool)	Drop impact phenomena with complex fluids on heated and soft surfaces
Lagopoulos (Southampton)	The impact of aspect ratio and swept angle on the propulsive performance of bio-inspired flapping foils in a tandem configuration	14:36		
Silva-Leon (Manchester)	Aerodynamics of flexible filaments hanging in cross flow	14:48		
PLENARY SESSION 4				
Emily Shuckburgh	Using fluid dynamics to explore our changing atmosphere and oceans	15:05 - 15:50		
Prizes and closing remarks		15:50 - 16:00		

An active granular liquid at the origin of the sensitivity of plants to gravity

Olivier Pouliquen

Abstract

A plant accidentally put in a horizontal position bends and deforms to recover a vertical position. A crucial step in this gravisensing occurs in specific cells, the statocytes, which contain small grains of starch. The grains being denser than the surrounding intracellular fluid, they sediment at the bottom of the cell and form miniature granular piles at the bottom of the gravisensing cells. How such a sensor works and can detect inclination is unclear, as granular materials like sand are known to display flow threshold and finite avalanche angle due to friction and interparticle jamming. Here, we address this issue by combining direct visualization of statoliths avalanches in plant cells and experiments in biomimetic cells made of microcavites filled with a suspension of heavy Brownian particles. We show that, despite their granular nature, statoliths move and respond to the weakest angle, as a liquid clinometer would do. Comparison between the biological and biomimetic systems reveals that this liquid-like behavior comes from the cytoskeleton activity, which agitates statoliths with an apparent temperature one order of magnitude larger than actual temperature. Our results shed light on the key unlocking role of these active fluctuations for explaining the remarkable sensitivity of plants to inclination.

Co-authors: A. Berut, H. Chauvet, V. Legue, B. Moulia, Y. Forterre

Biography

Olivier Pouliquen is CNRS Research Director at IUSTI, Aix-Marseille University. His research concerns the physics of granular materials and dense suspensions. Since a few years, he has started to study plant gravitropism both at the macroscopic and cellular level. In 2017, he has been awarded the prestigious Mergier-Bourdeix prize.

Influence of roughness topography on wall-bounded turbulence

Angela Busse

Abstract

While the standard assumption in fluid dynamics is that walls are smooth, in reality most surfaces encountered in the real world show at least some degree of roughness. This is a consequence of the many different processes that lead to surface roughness, ranging from surface pits caused by corrosion to the role played by forests and urban roughness in atmospheric dynamics.

Surface roughness affects near-wall turbulence if its height is significant compared to the viscous length scale of the flow. The roughness effect increases with the height of the roughness, but the influence of the roughness topography, i.e. the height distribution of the rough surface and its spatial correlation, on the near-wall flow is not yet fully understood.

Historically, investigations on the fundamental fluid dynamic effects of surface roughness have mainly focused on simple, regular forms of roughness, such as transverse bars or roughness elements arranged in a regular pattern. More recently, irregular forms of roughness have received increased attention as they are more representative of forms of surface roughness found in applications, such as roughness caused by erosion or fouling.

Using direct numerical simulations, results have been obtained for a wide range of irregular rough surfaces that are either based on surface scans or that have been generated using a surface generation algorithm adapted from tribology. Artificial surface generation enables the control of the statistical properties of an irregular surface allowing a systematic probing of the surface topography parameter space. The effect of the height distribution on the roughness, characterised by skewness and flatness, and the influence of the spatial correlation of the surface features on the fluid dynamic properties, such as friction factor, roughness function and near wall turbulence statistics, will be discussed.

Biography

Dr Angela Busse is a lecturer within the Aerospace Sciences Division of the School of Engineering, University of Glasgow. Her main research area is the effect of rough and superhydrophobic surfaces on wall-bounded turbulence. This work is currently supported by two EPSRC grants, “Fluid dynamic properties of irregular, multi-scale rough surfaces” and “Surface-specific Moody diagram: A new paradigm to predict drag penalty of realistic rough surfaces with applications to maritime transport”. In addition to wall-bounded turbulence, she has research interests in Lagrangian statistics of turbulence, magnetohydrodynamic turbulence, aerodynamics of plants, and high performance computing. She recently joined EPSRC’s Research Infrastructure e-Infrastructure Strategic Advisory Team.

Angela completed her undergraduate studies in physics at the University of Bayreuth, Germany. During her doctorate, she worked at the Max-Planck-Institute for Plasma Physics investigating the universal properties of hydrodynamic and magnetohydrodynamic turbulence from a Lagrangian viewpoint. After obtaining the doctorate in Physics she worked for several years as a research fellow in the Aerodynamics and Flight Mechanics research group at the University of Southampton, where she started to investigate fluid flow phenomena from an engineering perspective.

Role of fluid simulation and optimisation in turbomachinery design

Shahrokh Shahpar

Abstract

Aerothermal design of turbomachinery components is quite challenging, involving a large number of design parameters, often conflicting multiple objective functions and a number of near equality constraints. Using high fidelity CFD simulation also requires large computing resources. Under stringent time scales, advanced optimisation techniques are needed to achieve a practical design. This presentation will discuss the above issues and the strategies used to achieve a useful application of automatic, Simulation-based optimisation to real-world problems. A number of novel methodologies has been included and demonstrated as part of the Rolls-Royce SOPHY system (SOFT-PADRAM-HYDRA). This will be described in details with examples emphasising the MDO (multi-disciplinary) applications for a modern high-bypass ratio aero-engine.

Biography

Shahrokh is a Rolls-Royce Associate Fellow in Aerothermal Design Systems and the Manager of the 3DGEMO team, which is part of the CFD Methods within the DSE (Design System Engineering).

His focus is on producing design systems and associated optimisation tools for aero-engine components. He has a national and international reputation in the field of CFD meshing and automatic aerothermal design optimisation of turbomachinery components and systems, published more than 110 internal and external technical papers and holds four patents. His current research is on uncertainty quantification and robust design. Some of his work can be found here.

He has a First Class Honours degree and PhD on Hypersonics Flows including Real Gas Effects from the University of Manchester. He is a Fellow of the RAeS (Royal Aeronautical Society) and an Associate Fellow of AIAA (American Institute of Aeronautics and Astronautics). He was appointed Visiting Professor at the Institute of Engineering System & Design, Faculty of Engineering Leeds University in 2010 and at the Aeronautic department of Imperial College London in 2016. Shahrokh also acts as Associate Editor of the Royal Aeronautical Engineering Journal (Aerospace) since 2005 and has been a previous winner of the team and individual entry for Sir Henry Royce Award for Technical Innovation in the year 2002 and 2006 respectively.

Using fluid dynamics to explore our changing atmosphere and oceans

Emily Shuckburgh

Abstract

The atmosphere and oceans are thin films of fluid on the spherical Earth under the influence of heating by the Sun, gravity, and the Earth's rotation. Thus fluid dynamics is central to many of the processes governing our climate system. For example, if we are to predict the future trajectory of our changing climate we need to understand the fate of the carbon dioxide we emit. A key aspect of that is examining the processes associated with the uptake of carbon from the atmosphere by the oceans, a sink which currently accounts for around a quarter of our total emissions. In this talk I will describe the fluid dynamics associated with this, the work we have been doing to understand these processes and how they might change in the future, and the broader implications of our findings.

Biography

Dr Emily Shuckburgh is a climate scientist based at the British Antarctic Survey. There she is deputy-head of the Polar Oceans Team, which is focused on understanding the role of the polar oceans in the global climate system. Her personal research concerns investigating the dynamics of the atmosphere, oceans and climate using theoretical approaches, observational studies and numerical modelling. She is the principal investigator of national capability project funded by the Natural Environment Research Council, "Ocean Regulation of Climate by Heat and Carbon Sequestration and Transports (ORCHESTRA)".

She holds a number of positions at the University of Cambridge. She is a fellow of Darwin College, a member of the Faculty of Mathematics and an associate fellow of the Centre for Science and Policy. She is also a fellow of the Cambridge Institute for Sustainability Leadership, which is dedicated to working with leaders from business, government and civil society on the critical global challenges of the 21st century such as climate change, water scarcity and food security.

She completed her undergraduate studies in mathematics at the University of Oxford and a PhD in applied mathematics at the University of Cambridge. She then conducted post-doctoral studies in atmosphere and ocean dynamics at École Normal Superiéure in Paris and at MIT. She is a fellow of the Royal Meteorological Society, co-Chair of their Climate Science Communications Group and a former Chair of their Scientific Publications Committee. She is also a trustee of the Campaign for Science and Engineering. She has acted as an advisor to the UK Government on behalf of the Natural Environment Research Council.

In 2016 she was awarded an OBE for services to science and the public communication of science.

She is co-author with HRH The Prince of Wales and Tony Juniper of the Ladybird Book on Climate Change.

Attenuation of flow separation over a double ramp using herringbone riblets at Mach 5

Pengcheng Quan, Shan Zhong and Qiang Liu

University of Manchester

In this paper, the effect of herringbone riblets as a means of separation control is investigated in a Mach 5 flow. An array of ribleted strips is applied behind the leading edge of a double ramp model. High-speed Schlieren technique is used to examine the shock pattern and boundary layer developing over the surface of the model. Both pressure-sensitive paints and temperature-sensitive paints are employed to study the impact of riblets on the interaction between the shockwave and the boundary layer. This experiment provides convincing evidence that herringbone riblets are capable of delaying shock-induced flow separation in high-speed flows. Furthermore, it is also revealed that these micro-scale riblets induce large-scale streamwise vortical structures within the boundary layer as found in incompressible flows. It is believed that these vortices promote momentum transfer within the boundary layer hence providing the dominant mechanism for supressing flow separation.

Simulations of strong shock wave / turbulent boundary-layer Interactions

Alex Gillespie and Neil Sandham

University of Southampton

Implicit Large Eddy Simulations (LES) are performed on M = 2.7 flow in a 3D duct with an oblique shock wave generated with a ramp on the upper boundary. The resulting shock wave / turbulent boundary-layer interaction on the lower boundary is investigated. The simulations are performed on OpenSBLI, an in-house finite difference solver. A method for generating the turbulent inflow is considered by comparing it to more well-established techniques. The turbulent boundary layer is generated by (1) applying a digital filter method to a field of random disturbances (2) scaling the disturbances with prescribed Reynolds stress data (3) combining with an analytically-determined time-averaged profile. The suitability of the implicit LES model with in the Targeted ENO (TENO) scheme is assessed by re-running the simulation parameters from an explicit LES test case conducted in a previous study. For the main aspect of the work, a higher ramp angle is used to increase the strength of the shock interaction, and various aspects of the flow physics are explored and analysed. This comprises: the influence of the sidewalls on the main flow structure, the upstream propagation of disturbances through subsonic regions, and the underlying sources of unsteadiness.

Effects of surface roughness on boundary layer transition on a blunt body in a hypersonic flow

Oliver Taylor and Paul Bruce

Imperial College London

Experiments using the Imperial College Hypersonic Gun Tunnel have been undertaken to assess the effect of different configurations of pairs of isolated roughness elements on boundary layer transition on a blunt body in a hypersonic flow. The structure of the flow around the configurations of roughness elements has been investigated using oil flow visualisation in the Supersonic Wind Tunnel and computationally using Eilmer v4 to explore the changes to the boundary layer transition.

The inter-element spacing has an effect: the transition of the boundary layer behind configurations with spacings of 4-6 element widths appears more advanced, configurations with no spacing lead to longer, more developed turbulent features in the flow, whilst configurations with intermediate spacing (0.5-2 widths) lead to a boundary layer where transition appears suppressed.

The structure of the flow around the roughness elements also changes with inter-element spacing. The centre of the wake was found to move across the geometric centre-line of the roughness elements depending on their spacing. Configurations with intermediate spacings, which appeared to have boundary layers where the transition was less advanced, exhibit smaller streak velocities than other configurations.

Sensitivity of background-oriented Schlieren (BOS) applied to hypersonic flow

Tom Fisher, Mark Quinn and Kate Smith

University of Manchester

Evaluation of density gradients in compressible flows is typically achieved through the schlieren technique, which provides a qualitative view of salient flow features including shock waves, expansion regions, and shear layers. Schlieren and other established optical techniques require accurate and expensive optics, cut off filters or knife edges and cameras to produce quality images. Developments in digital image processing allowed the tracking of distortions in a background pattern due to changes of refractive index to be processed as “synthetic schlieren” images.

Investigations have been performed at The University of Manchester to evaluate the sensitivity of the background-oriented schlieren (BOS) technique to changes in free stream density conditions. A flared cone model was used to generate a series of strong and weak density variations in Mach 5 flow. The freestream density is altered by an order of magnitude for constant total pressure but varying total temperature. The sensitivity is then evaluated through comparison between the established optical schlieren method and the results obtained from the BOS method.

Hypersonic flow-fields around faceted shapes

Thomas Rees[1], Paul Bruce[1] and James Merrifield[2]

[1]Imperial College London
[2]Fluid Gravity Engineering

To analyse the casualty risk associated with satellites undergoing atmospheric entry, a complex calculation is required. This simulation must incorporate knowledge of the satellite geometry, aerothermodynamic characteristics, and fragmentation mechanisms. It must also take into account the cross-sectional area of the surviving components and the population distribution on the ground. Therefore, in order to improve casualty risk calculations, it is necessary to improve our knowledge of the hypersonic aerodynamics of the complex faceted shapes typified by satellite geometries. We use laminar CFD and a preliminary experimental investigation to explore the hypersonic flow-fields around various simple faceted shapes. Results show that variations in Mach and Reynolds numbers give rise to a variety of different flow regimes. For the flow over a two-dimensional square, a separation bubble is formed over the top and bottom surfaces as the Reynolds number is increased. This significantly changes the heat transfer along these surfaces. The presence of flow phenomena such as separation is not accounted for by current destructive re-entry prediction methods, and therefore these tools are likely to predict higher heating rates than those that are encountered during re-entry.

Impact of orifice lip ratio on the formation and evolution of synthetic jets

Girish Jankee and Bharathram Ganapathisubramani

University of Southampton

The periodic ingestion and ejection of fluid through an orifice yield vortex rings forming an unsteady jet. The ability to impart momentum with a zero net mass flux makes synthetic jet actuators coveted components in flow control applications. Previous studies underlined the influence of geometrical parameters on synthetic jet evolution. This investigation introduces the orifice lip ratio as a novel concept and assesses the sensitivity of synthetic jet flow fields on this parameter. Both 2D and axisymmetric orifice shapes are considered. The flow fields are examined using hot-wire anemometry and planar PIV while pressure transducers monitor the pressure variation within the cavity over a range of operating conditions. Comparisons of test configurations establish the lip ratio as a pivotal parameter and indicate noticeable differences in the velocity profiles. The results further highlight the dependency of the jet formation criterion on the orifice lip ratio. The analysis conclusively demonstrates the significance of the orifice lip ratio in the design and optimisation of synthetic jet actuators.

Gradient-limiting shape control for efficient aerodynamic optimisation

Laurence Kedward, Christian Allen and Thomas Rendall

University of Bristol

The efficiency and robustness of aerodynamic shape optimisation are highly dependent on the shape control method since this acts as the interface between the search algorithm and the analysis code. The shape control method should ideally be able to represent a variety of shapes while maintaining smoothness and mesh regularity. Several methods for shape control exist (e.g. splines, shape functions, filtering) which address these characteristics either directly or indirectly; however, such methods often a priori constrain the design space at low fidelities and perform poorly when shape control is refined, due to insufficient regularisation or deteriorating conditioning of the numerical shape problem. In this work, it is demonstrated that these issues arise since the discrete shape problem is ill-posed by being insufficiently bounded; that is, it naturally includes geometries that are invalid in physicality (shape) and discretisation (mesh). A high-fidelity shape control method is developed here using local shape control methods to retain a well-conditioned parameterisation and shape gradient constraints are derived to control the smoothness of the surface during optimisation. The new method is tested on a standard two-dimensional transonic drag minimisation case and convergence is shown to be independent of shape control fidelity.

Optimal coordinate transformations for the perfectly matched layer method to model acoustic wave propagation in fluids

Jonathan Deakin[1], Andrew Hazel[1], Robert Harter[2] and Matthias Heil[1]

[1]University of Manchester
[2]Thales UK Limited

Many acoustic wave propagation problems are set in infinite domains and so require non-reflecting boundary conditions to be applied on the outer computational boundary. The widely used perfectly matched layer (PML) method applies a coordinate transformation in a layer surrounding the region of interest (the bulk) and in its exact form completely absorbs waves leaving the bulk without spurious reflections. However, when discretised, if the PML region is under-resolved, spurious reflections are created, limiting the achievable accuracy of the solution in the bulk. Conversely, if the PML region is over-resolved then computational effort is wasted. The optimal balance between bulk and PML refinement is problem dependent and difficult to find a priori.

To address this problem we propose a PML which is optimal in the sense that the transformation ensures the solution varies linearly through the PML, making it trivial to discretise. We present an algebraic method for finding this optimal transformation which utilises information about the solution. While this makes the problem non-linear, we show that we can converge to the exact solution by iterating, using information from the previous solution. We show that with this procedure, the numerical error is completely controlled by the refinement in the bulk.

Sub-surface waves on the ocean under strong winds

Joseph Oloo and Victor Shrira

Keele University

Under strong winds gravity waves on the ocean surface often break which results in entrainment of air bubbles into the water and creation of a multiphase flow of “bubbly” fluid adjacent to the surface. This flow is characterized by substantial density stratification confined to a relatively thin layer. Adjacent to this layer is the density uniform layer with a noticeable vertical shear which decreases with depth to almost zero within the oceanic mixed layer. The main question we are aiming at is what kind of waves can be supported by such a boundary layer and what mathematical models capture their dynamics

As a first step we consider a three-layer model with a thin stratified layer and no shear, a layer when the current drops from its surface value at the bottom of the stratified layer to zero and infinitely deep unstratified layer with no current below. A straightforward analysis of linearized dispersion relation shows that the boundary layer can support an infinite number of internal gravity modes modified by the vertical shear and vorticity mode modified by the density stratification. Since there are arguments that vorticity mode dynamics is most interesting and important we focus our attention on weakly nonlinear dynamics of the vorticity mode.

Wave attractors in stratified fluids

Will Booker[1], Onno Bokhove[2] and Mark Walkley[1]

[1]University of Leeds
[2]University of Leeds & University of Twente

A disturbance in a stratified fluid can generate motion that is resisted by gravity. This competition of forces leads to oscillations that form internal waves in the fluid and these propagate along wave beams that are inclined with respect to gravity. In an asymmetric domain successive reflections from solid boundaries can force the wave beams onto a limit orbit, known as a wave attractor, focusing the wave energy of the system onto small spatial scales.

In a geophysical context, internal waves can be considered confined between lateral boundaries and the ocean surface. Examples of such lateral boundaries include the Luzon Strait. The wave attractors formed in such domains provide a mechanism for an energy cascade, and provides insight into how energy transported to internal waves can induce mixing of the fluid.

We consider a computational model of stratified fluids using a non-dissipative Hamiltonian discontinuous finite element method. Three formulations are considered: compressible, incompressible and Euler-Boussinesq.

In each case internal waves are generated by applying a body force to the fluid at rest in an asymmetric domain and the resulting system is monitored for the presence of wave attractors. We compare our numerical results with analytical and experimental internal wave attractors.

Acoustic-gravity waves generated by impulsive sources at the ocean surface – a field study

Wade Parsons[1], Bruce Colbourne[2] and Usama Kadri[3]

[1]MUN, [2]Cardiff

Impulsive sources at the ocean surface, such as falling meteorites, sudden formation of rogue waves, or storm surges, generate propagating compression-type modes known as acoustic–gravity waves (AGWs) that travel in the water column at speeds near the speed of sound in water leaving a measurable pressure signature [1].

We carried out experiments in a water tank where neutrally buoyant spheres impacted the water surface, and the generated acoustic modes were recorded from a distance using a hydrophone. The shape of the pressure signature revealed three main regions that are associated with the impact, cavitation, and secondary wave formation. Employing these findings and solving the inverse problem allows remote sensing and prediction of the main source properties [2].

We repeat the experiment in the field where more real world conditions can be studied and both the water depth and the distance from source to hydrophone can be increased.

References
[1] Yamamoto, T. (1982), Gravity waves and acoustic waves generated by submarine earthquakes, Soil Dyn. Earthquake Eng., 1, 75–82.
[2] Kadri, U., Crivelli, D, Parsons, W., Colbourne, B. and Ryan, A. (2017), Rewinding the waves: tracking underwater signals to their source, Scientific Reports 7, Article number: 13949 (2017), doi:10.1038/s41598-017-14177-3.

High contrast approximation to penetrable wedge problems

Matthew Nethercote[1], Raphael Assier[1] and Ian David Abrahams[2]

[1] University of Manchester
[2]University of Cambridge

The problem of wave scattering by a penetrable wedge is one of the most important canonical problems in diffraction theory. Solutions to this problem will be useful for both acoustic and electromagnetic applications, including noise transmission in non-viscous fluids, electromagnetic propagation and crystal diffraction.

To this day there is no clear analytical solution for penetrable wedge diffraction, however there have been numerous attempts at computational and asymptotic solutions by authors such as A. Rawlins, M. Lyalinov, V. Daniele and A. Shanin.

In this talk, material parameters of the scatterer and the fluid host are used in order to create a high contrast asymptotic approximation. What follows is an iterative scheme where the complicated penetrable wedge problem is split into an infinite amount of 'easier' impenetrable wedge problems. The first of the 'easy' problems is well known and can be solved explicitly. Each following stage uses boundary data from the previous problem and is solved by the Wiener-Hopf technique.

The fluid dynamics of anti-surfactant solutions

Stephen Wilson[1], Justin Conn[1], David Pritchard[1], Brian Duffy[1] and Khellil Sefiane[2]

[1]University of Strathclyde
[2]University of Edinburgh

While there is a large and ever-expanding body of research on the fluid dynamics of surfactant solutions (in which the presence of the surfactant decreases the surface tension of the solution relative to that of the pure solvent), there has thus far been almost no work on the fluid dynamics of so-called “anti-surfactant” solutions, such as salt dissolved in water, in which the opposite effect occurs (i.e. the presence of the anti-surfactant increases the surface tension of the solution). In this talk we formulate and analyse a fluid-dynamical model for the flow of both surfactant and anti-surfactant solutions which reduces to the classical model for an surfactant in the appropriate limit. In particular, we use a combination of analytical and numerical techniques to analyse the stability of a uniform layer of anti-surfactant and the nonlinear dynamics of a thin film of anti-surfactant.

Conn, J.J.A., Duffy, B.R., Pritchard, D., Wilson, S.K., Halling, P.J., Sefiane, K., Fluid-dynamical model for anti-surfactants, Phys. Rev. E 93 043121 (2016)

Conn, J.J.A., Duffy, B.R., Pritchard, D., Wilson, S.K., Sefiane, K., Simple waves and shocks in a thin film of a perfectly soluble anti-surfactant solution, J. Eng. Math. 107 167-178 (2017)

Internal flows during the coalescence of a free and a sessile droplet

Thomas C. Sykes[1], Mark C. T. Wilson[2], David Harbottle[3], Zinedine Khatir[1] and Harvey M. Thompson[1]

[1]EPSRC Centre for Doctoral Training in Fluid Dynamics, University of Leeds
[2]University of Leeds
[3]School of Chemical and Process Engineering, University of Leeds

Liquid droplets coalescing on solid surfaces are found in many applications, from inkjet printing to microfluidic devices. Whilst most work on droplet coalescence focuses only on the external dynamics, there is a growing interest in the internal flows and mixing between the fluids contained within each of the coalescing droplets. Adequate mixing is crucial in emerging printing technologies, but the process of mixing within coalescing droplets is currently not well understood, especially when the coalescence occurs on a solid surface, and at small length scales.

The coalescence of a free and a sessile droplet is investigated both experimentally and through numerical simulation using OpenFOAM. The initial configuration of the droplets ensures that their velocities at the onset of coalescence are negligible. Due to the dominant surface tension force, the free droplet is injected into the sessile droplet which drives the internal flow. The factors affecting the internal dynamics are considered, including the droplet volume ratio and surface wettability. The presence of the solid surface is shown to promote good mixing, even for modest droplet volume ratios.

Analytical models for propulsion strategies

Peter Baddoo and Lorna Ayton

University of Cambridge

The propulsive efficiency of wings in two configurations (close to the ground, called “ground effect”, and proximate to another wing, called “formation flight”) are investigated using a variety of analytical techniques. Firstly, the steady, mean flow past these configurations for thin aerofoils is solved as a Riemann-Hilbert problem. Enhanced lift is found in both steady situations. Secondly, we model the unsteady vortex dynamics by modelling the wings as flat plates and placing a free point vortex in the flow. The trajectories of the vortices are calculated using conformal mappings and the Hamiltonian formulation of the system. The trajectory of a Brown-Michael vortex is calculated to model vortex shedding by the sharp trailing edge. Finally, we allow the flat plates to oscillate, or flap, by establishing a novel conformal map from a canonical strip domain using a modified version of the Schwarz-Christoffel formula. This is used to develop an unsteady point vortex model for the coupled fluid-solid problem, where vortices are shed at each period of oscillation of the plate. Results on the propulsive efficiency of each configuration are discussed for a variety of Strouhal numbers and are compared to numerical and experimental results.

Oscillating foils at moderate Reynolds numbers

Weidong Dai and Ignazio Maria Viola

School of Engineering,
Institute for Energy Systems,
University of Edinburgh

We modelled with RANS and LES different pitching foil experiments at moderate Reynolds numbers (Re). We studied numerically the vortex street produced by a 2 degree sinusoidally pitching foil at Re = 12 000, for a wide range of reduced frequencies. We found that transition occurs in the wake and has a negligible effect on the unsteady forces. However, further increase in the pitch amplitude triggers transition near the leading edge even at moderate Re conditions. We study a pitch-up, hold, pitch-down motion for a flat plate at Re = 10 000. The plated was pitched around leading edge linearly to 25 degree, then maintained constant for a short time and then linearly pitched down to zero. Onset of separation of flow at the leading edge occurs early on the upstroke, generating a large leading-edge vortex convecting downstream, leading to large load fluctuations. Transition occurs within the leading-edge vortex and patches of turbulent flow convected downstream along the foil. We noticed that the correct prediction of laminar-turbulent transition is essential to correctly compute the unsteady forces and, hence, this complex flow regime is a major challenge for numerical modelling.

3D influence on propulsive flapping foil

Andhini Novrita Zurman Nasution, Gabriel D. Weymouth and Bharathram Ganapathisubramani

University of Southampton

Many of the 2D approximations used for flapping studies at low Reynolds number are overpredicting power. In this work, the 3D fluid effects of oscillating foils are studied through the comparison of thrust force, power and propulsive efficiency. The objective of this study is to find out which characteristics nature gives as the best propulsion.

In this work, we simulate the unsteady flow with the Boundary Data Immersion Methods, which is documented to accurately predict low Re stationary and dynamic foils up to Re 100000. As the first step, a single flapping foil of pure heaving, pitching, and heaving-pitching combination are studied to see where the 3D starts to be important. This work shows that as the amplitude is decreased (reducing Strouhal number), the 3D fluid effects become more critical to the foil performance. The same condition as the amplitude increased, the unsteadiness of the flow makes the 3D effects also important.

The following step is to study finite wing span geometries by varying the aspect ratio and the vertical plane kinematics. The change in 3D flow features over the variation of aspect ratio and the change from heaving to rolling motion is presented, as well as the propulsive efficiency.

Bayesian optimisation for skin-friction drag reduction of a spatially evolving turbulent boundary layer using wall blowing

Omar Mahfoze, Andrew Wynn and Sylvain Laizet

Department of Aeronautics, Imperial College London

Continuous flow injection from the wall in a turbulent boundary layer (TBL) is a very effective method to reduce shear stress and skin-friction drag (up to 70% drag-reduction have been reported in the literature). However, the energy expenditure can be high, leading to net energy saving as low as 5%. Interestingly, the effect of drag reduction can extend for a long distance downstream of the blowing region. This behaviour indicates the possibility of using intermittent blowing to obtain comparable drag reduction with less power than for a large continuous blowing region. Bayesian optimisation – a global optimisation technique requiring only a few simulations, and is yet to be exploited in the field of turbulence – will be used to obtain the streamwise lengths of the blowing regions, blowing intensity and frequency to achieve high net-energy saving compared to a continuous blowing approach. The study is based on Direct Numerical Simulations (DNS) of a spatially evolving zero pressure gradient TBL and is performed using the high-order flow solver Incompact3d. This solver is based on finite-difference sixth-order schemes on a Cartesian mesh, a semi-implicit time advancement, a spectral solver for the Poisson equation and a powerful 2D domain decomposition for supercomputing.

Boundary layer development and riblet wavelength effect of laminar flow over convergent-divergent riblets

Fang Xu, Shan Zhong and Shanying Zhang

University of Manchester

Surface topography of convergent-divergent riblets, as a novel nature-inspired passive flow control method, has broad application prospects in drag reduction and separation control. In this work, the development of a laminar boundary layer over convergent-divergent riblets with different spanwise length scales (riblet wavelengths) is investigated in detail using mono and stereoscopic particle image velocimetry and dye visualisation in a water flume. The flow topology and the flow mechanism over this highly directional spanwise roughness are proposed, and the relation between riblet wavelength and vortical structures is revealed for the first time. By dye visualisation, the helicoidal path of the fluid inside riblet valley and the overflow above riblet crest are discovered. Two large-scale co-rotating vortical structures are observed within half wavelength in the cross-stream plane, and the formation mechanism of the secondary flow is proposed. The streamwise development of the boundary layer over convergent-divergent riblets is divided into a developing stage and a developed stage. In the developing stage, the shape factor, the amplitude of the induced velocity and vorticity over the converging line increase continually, whereas in the developed stage the values of these parameters remain constant.

Stability of short spanwise scale injection flows

Anthony Williams and Richard Hewitt

University of Manchester

The stability of a streamwise aligned streak within a steady boundary layer is investigated. The streak is created by the injection of fluid through a slot, in the surface of a flat plate, into a laminar boundary layer. The injection slot exists at all downstream locations and has a finite spanwise width that is a fixed ratio of the local boundary-layer thickness. The stability of the streak is examined by introducing a travelling wave, which propagates downstream, that has a wavelength of comparable size to the boundary-layer thickness. Two-dimensional generalisations of the usual stability equations are obtained resulting in an eigenvalue problem for the wavespeed which is solved for various injection rates and wavenumbers. The interaction between the streak/vortex system and the travelling wave is of significant interest as such interactions are relevant to turbulent shear flows. By examining the stability of the streak/vortex system it is hoped that a fully interactive self-sustaining solution can be obtained where the streak is driven by the wave and vice versa.

A numerical and experimental investigation of the effects of sharkskin denticle geometry on a flat plate boundary layer

Charlie Lloyd[1], Jeffrey Peakall[1], Alan Burns[1], Gareth Keevil[1] and Robert Dorrell[2]

[1]University of Leeds
[2]University of Hull

The skin of sharks, unlike that of most other fishes, is comprised of small tooth-like structures called dermal denticles. The hydrodynamic function of sharkskin has been of interest for several decades, with most attention being focussed on the riblet features which are often present on the denticle crown. Simplified, two-dimensional, riblets have been successfully replicated and applied to engineering boundary layer flows, typically achieving a maximum drag reduction of 10%, with excellent agreement between literature data. The same cannot be said for real, three-dimensional sharkskin denticles; the few studies that have been carried out are often conflicting, even for flat plate boundary layer flows. This is largely due to the variability of denticle geometry between different shark species, and locations on the shark body. In order to investigate the effects of denticle geometry on a boundary layer flow, we adopt Reynolds Averaged Navier-Stokes (RANS) methods to simulate the flow over sharkskin denticles. The implications of using such methods to model these flows are discussed, and the results are compared against an experimental data set, obtained using Laser Doppler Anemometry (LDA) to measure the boundary layer over 3D printed denticle surfaces.

Experimental 3D mapping of reflected shock - boundary layer interaction unsteadiness

Paige Rabey and Paul Bruce

Imperial College London

An experimental study into oblique shock – boundary layer interaction (SBLI) unsteadiness has been conducted. The phenomenon is an important consideration when designing supersonic engine inlets, particularly as the shock unsteadiness exhibits a low frequency motion which can cause fatigue and lead to engine unstart.

SBLIs are invariably assumed to be two-dimensional and often only measurements at the wind tunnel centreline are obtained in experiments. Similarly, the vast majority of numerical studies employ periodic or reflected span-wise boundary conditions on the assumption that interactions should exhibit span-wise similarity.

At Imperial, we are in the process of obtaining wall pressure fluctuation measurements in a supersonic wind tunnel facility to produce the first detailed 3D map of low frequency unsteadiness under a reflected SBLI. We hope to identify trends across different configurations that could be harnessed to advance our understanding of SBLI unsteadiness in real applications and ultimately improve engine inlet design. Further, we will examine the cross-correlations of the simultaneously obtained data to seek sources of low frequency unsteadiness from three-dimensional effects and compare to a recent LES study with sidewalls by Jammy and Sandham (TSFP10, 2017).

Biaxially shaping droplets with localized elastic pattern bifurcation

Ding Wang[1], Fei Chen[2], Yifan Li[1], Glen McHale[1], Gary Wells[1], Ciro Semprebon[1], David Wood[1], Michael Cooke[3] and Ben Xu[1]

[1]Northumbria University
[2]School of Chemical Engineering and Technology, Xi'an Jiaotong University
[3]Durham University

Non-uniform distributions of surface energy can cause anisotropic wetting and droplet deformation, while asymmetric chemical or physical patterns on a material surface can cause directional wettability. On an elastic wrinkled groove surface, as the maximum compressive strain is approached, a droplet can start imbibing into the grooves leading to an eventual filling of entire grooves. To achieve highly controllable instabilities and a bi-axial switching droplet shape, we created a patterned elastic surface able to initialize localized surface instabilities and induce reversible surface morphology changes. At equilibrium, our topographic surface consists of a set of circular voids distribute din an equilibrium manner. By using plasma treatment and mechanical stimuli, we investigated sinusoidal secondary nano/micro structure, which form under mechanical stimuli and redistribute the surface energy. A droplet placed on our surface is pinned by the topographic features and deforms as the circular shapes elongate to elliptical shapes. The static, advancing and receding contact angles were measured before and after plasma treatment, showing the enhancement of the surface wettability due to changes in the surface chemistry, morphology and roughness. This finding opens a window to create the robust wetting state surface with potential applications in microfluidics, bio-engineering and soft robots.

The influence of the thermal properties of the system on the lifetime of an evaporating droplet

Feargus Schofield[1], Stephen Wilson[1], David Pritchard[1] and Khellil Sefiane[2]

[1]University of Strathclyde
[2]University of Edinburgh

The evaporation of a sessile liquid droplet on a solid substrate is a fundamental problem in fluid mechanics linked to many industrial processes, such as ink-jet printing, coating, and spray cooling.

Previous work has shown that the mode in which a droplet evaporates is a key factor in determining its evolution. However, much of this work uses the standard diffusion-limited model for droplet evaporation. The standard model does not account for various other key factors in droplet evaporation, such as the thermal properties of the droplet, substrate or atmosphere. In the present work we investigate the combined influences of the thermal properties of the system and the mode of evaporation on droplet evaporation, in order to predict the evolution, and hence the lifetime, of a droplet. In particular we investigate droplets evaporating on substrates with a relatively low thermal conductivity compared to that of the liquid, such as methanol on aerogel, in each of the constant radius (CA), constant angle (CA), stick-slide (SS) and stick-jump (SJ) modes of evaporation.

Modelling the evolution of fully-gelled sessile drops

Arandeep S. Uppal[1], Matthew G. Hennessy[2], Richard V. Craster[1] and Omar K. Matar[1]

[1]Imperial College London
[2]Centre de Recerca Matemàtica

Gelation occurs during the evaporation of many complex biological fluids, further evaporation of the gel phase leads to the formation of complex crack morphologies. During evaporation, flow drives solid particles to the contact line, where compaction leads to the formation of a gel front which propagates into the bulk. A variety of complex phenomena is involved in the formation and fracture of the gel front, however the dynamics of fully-gelled drops is not understood. Hence, we consider the problem of a drop with an already present solid network undergoing evaporation and eventual fracture.

A model is derived for the evaporation of a fully-gelled drop from the governing equations of nonlinear poroelasticity alongside the lubrication approximation. Concurrently, a phase-field model for fracture is derived for thin two-dimensional elastic films. Poroelastic free surface profiles show good agreement with experiments. The parameter evolution can be obtained from the fluid dynamics and fed into the fracture model to simulate a variety of crack morphologies also seen in experiments.

The authors would like to acknowledge funding from the EPSRC CDT for fluid dynamics across scales, EP/L016230/1.

Controlling the size of acoustically nebulised droplets by pinning surface waves for precise delivery of aerosolised medicine

Elijah Nazarzadeh, Xi King, Julien Reboud, Rab Wilson and Jonathan M. Cooper

University of Glasgow

Abstract: The effective delivery of medication to the lung via inhalation of aerosols has been shown to be crucially dependent upon the droplet size distribution within the aerosol. An effective pulmonary drug delivery requires a distribution with diameters between 1 and 5 µm.

Here we investigate formation of aerosols using surface acoustic wave (SAW), and, further, how the process can enable control of the droplet size by coupling the waves through a microfabricated array of apertures, using the liquid sample to be nebulised as the coupling agent.

SAWs were generated by interdigitated transducers on a piezoelectric substrate (lithium-niobate). The nebulisation process was monitored through a high-speed camera (600,000 fps) and the sizes measured using a Spraytec (Malvern Panalytical) and an Anderson Cascade Impactor (Copley Scientific).

Results show a correlation between capillary waves on the surface of liquid and ejected droplet size. We then show that physical confinement within the microstructure acts as a low-pass filter, controlling the capillary wavelength and the ejected droplet size, enabling us to obtain a mean droplet size within the optimum range for drug delivery (<5µm). We validated the applicability of the technique by forming and nebulising drug-loaded nanoparticles in range of 300nm, directly on the device.

Numerical simulations of drop impact on interfaces

Lyes Kahouadji, Assen Batchvarov, Richard Craster and Omar Matar

Imperial College London

We perform direct numerical simulations of complex, drop splash configurations using a hybrid front-tracking/level-set CFD solver. The first configuration we consider involves oblique drop impact on a 'deep' pool whose depth exceeds significantly the drop diameter. The second configuration involves the vertical impact of two identical drops on a deep pool. In the former case, we simulate the details of crater and crown formation, as well as the ejection of droplets from the crown. In the latter, we use the simulation results to examine the intriguing phenomena that accompany the formation and subsequent interaction of the two craters that follow the impact of the individual drops on the surface of the pool. For both configurations, we provide qualitative and quantitative comparisons with the experimental studies.

We would like to acknowledge funding from the EPSRC Programme Grant MEMPHIS through grant number EP/K003976/1. We also acknowledge the contribution of Drs Damir Juric and Jalel Chergui (both from LIMSI, CNRS, France), and Dr Seungwon Shin (Hongkik University, South Korea).

Grouping in 3-phase systems in microfluidic devices

Kerstin Schirrmann, Gabriel Caceres and Anne Juel

University of Manchester

Regular droplet trains of one or more fluids are formed and processed in many microfluidic lab-on-the-chip applications to study biological and chemical processes in very small probe volumes. Usually, droplet trains are manipulated using channel bifurcations and simple networks or drop-on-demand systems in order to separate individual droplets or segregate them into packets. We investigate step changes in channel cross section and show that they enable the re-organisation of a regular droplet train into different patterns depending on flow rates. We have used a 3-phase system with a dyed glycerine-water-mixture, perfluorinated oil with surfactant and silicone oil. The perfluorinated oil surrounds the other two phases, which form a regular pattern of alternating glycerine-water and silicone oil droplets, initially. Typical channels are 300μm wide and 250μm deep and have a step where the cross section increases by 50%. We find that the alternating pattern transforms into groups of 2 to 10 glycerine-water droplets separated by a droplet of silicone oil at the expansion. We explore the parameter range of this grouping phenomenon in terms of flow rates and step size in order to construct a phase diagram.

On the origin of hydraulic jump

Rajesh Kumar Bhagat, Narsing Jha, Paul Linden and Ian Wilson

University of Cambridge

When a liquid jet impinges normally on a surface, the liquid spreads radially outwards in a thin film until a point where the film thickness changes abruptly, forming a hydraulic jump. The pervading view has been that hydraulic jumps are created by gravity. We have found that thin-film hydraulic jumps are not induced by gravity. When a jump initially forms on a horizontal plate, the jump remains at the same location until the thicker, downstream film reaches the plate edge. At this point gravity contributes: the film thickness increases and the jump radius moves inwards. For circular jumps produced by a jet impinging on a solid surface, we found that a jump is initially formed when surface tension and viscous forces balance the momentum in the film. This study explores the origin of thin-film hydraulic jumps. Experiments show no dependence on the orientation of the surface: a scaling relation balancing viscous and surface tension collapses the experimental data. Including the interfacial energy associated with creating the film is the previously neglected, critical ingredient in these flows. Capillary waves play the role of gravity waves in a traditional jump, in demarcating the transition from supercritical to subcritical regimes.

Computation of discrete self-similar branches of thin film rupture solutions

Michael Dallaston[1], Dmitri Tseluiko[2], Serafim Kalliadasis[3] and Marco Fontelos[4]

[1]Coventry University
[2]Loughborough University
[3]Imperial College London
[4]Instituto de Ciencias Matemáticas

The partial differential equation that describes thin liquid film dynamics can exhibit either finite-time rupture (thickness going to zero in finite time), or finite time blow-up (thickness going to infinity). The particular behaviour depends crucially on parameters, particularly the exponents in nonlinear coefficients.

The transition from one between these two behaviours is still not completely understood. For some parameters, rupture does not occur in a straightforward self-similar way, but exhibits a cascade of geometrically shrinking structures that corresponds to periodic behaviour in logarithmic time. Computation of these scaled-periodic solutions presents some significant challenges. In this talk I will describe some such methods, such as numerical continuation and dynamic estimation of the rupture location and time.

Freestream turbulent inflow for CFD

Rory Hetherington, Robert Dorrell, Jeff Peakall, Nikolaos Nikitas and Andrew Sleigh

University of Leeds

Increased computing power and developments in turbulence modelling have led to a growing interest in large-eddy simulation (LES) for various engineering applications, e.g. aerodynamics, noise prediction, and turbo-machinery. Freestream turbulence plays a pivotal role in such flows, therefore it is imperative to control the characteristics of inflow turbulence. Although LES solutions are known to be sensitive to inlet condition, there exists no consensus on an optimal method for generating turbulent inflow.

There appears to be no such confusion in experimental studies: inflow is passed through a grid to generate pseudo-homogeneous turbulence, a process thoroughly documented over the last few decades. However, with the advent of fractal grids, grid-generated turbulence has led to a reformulation of the classical dissipation scaling law. Consequently, grid turbulence is valuable for two distinct reasons: generating freestream turbulence, and developing turbulence theory.

A simple method of generating grid turbulence in LES is considered. Grid geometries from the aforementioned experiments are projected onto the inlet patch in a similar fashion to the 'gridInlet' method developed by Blackmore et al. (Int. J. Comput. Fluid Dyn 2013). Well established results from experiments are reproduced, and an extension towards generating synthetic turbulence inflow is briefly discussed.

Instantaneous velocity and wall shear stress measurements during low-drag events in turbulent channel flow

Rishav Agrawal[1], Henry Ng[1], Richard Whalley[2], David Dennis[1] and Robert Poole[1]

[1]University of Liverpool
[2]Newcastle University

Recent Direct Numerical Simulations in channel flow shows low drag events for Newtonian turbulence during which the mean velocity profiles approach the Maximum Drag Reduction (MDR) profile, something more usually associated with drag-reducing additives. These moments of low drag are termed "hibernating turbulence" and are said to occur when the instantaneous wall shear stress drops to 90% of the mean and stays there for at least non-dimensional time (t* = t * friction velocity/half-height) of 3. In this project, experimental verification of this phenomena is investigated in a large-scale channel flow facility using Newtonian fluids. Simultaneous measurements of velocity and wall shear stress are carried out using Laser Doppler Velocimetry and Hot-film Velocimetry. Experiments are conducted for friction Reynolds numbers of 67, 85 and 103 and it is observed that the mean velocity profiles conditioned over hibernating events approach the MDR profile. Increasing Reynolds number shows a negligible change in the ensemble-averaged wall shear stress during hibernating events, with only the frequency of these events decreasing. The effect of varying the criteria for a hibernating event (minimum hibernation duration and threshold) is also studied and it is found that varying these two criteria changes the number of hibernating events observed.

Experimental and numerical modelling of aerated flows over stepped spillways

Jacob van Alwon, Duncan Borman, Andrew Sleigh and Nikil Kapur

University of Leeds

The Volume of Fluid (VOF) and Eulerian multiphase models are used to simulate flows over two experimental stepped spillways using a range of turbulence models. The VOF model does not predict air entrainment, whereas the Eulerian model does. Pressures, velocities, air concentrations and flow depths are all predicted well, with different models performing more accurately at different locations in the spillway.

Physical modelling is conducted in an experimental spillway of 150mm width and a step height and length of 80mm. The pressures acting on the spillway and the free surface position are measured. These experiments show that, in addition to the stream-wise vortices which are expected to occur in the step cavities, further vortices occur which circulate in the span-wise direction. At each step there are two vortices observed which circulate in opposite directions, meeting at the midpoint of the channel and changing direction at each subsequent step. Both the VOF and Eulerian models predict these vortices, however when the spillway width is increased the models predict different results. A parametric study of various spillway geometries is presented in order to investigate the conditions which cause this cross-stream vorticity.

Spontaneous mirror-symmetry breaking due to inertial waves in rotating turbulence

Vassilios Dallas[1] and Steve Tobias[2]

[1]University of Oxford
[2]University of Leeds

The effects of large scale mechanical forcing on the dynamics of rotating turbulent flows are studied by means of direct numerical simulations (DNS), varying systematically the nature of the mechanical force in time. In this talk we will show that rapidly enough rotating flows with a time-independent forcing bifurcate from a non-helical state to a helical state despite the fact that the forcing is non-helical. The random superposition of inertial waves exhibits a mirror-symmetry breaking only when the mechanical forcing is on a preferred direction in the propagation of the waves with respect to the axis of rotation. Otherwise, the random superposition of inertial waves in equal proportions by our random-in-time forcing gives zero net helicity in agreement with H. K. Moffatt, J. Fluid Mech., 44: 705-719 (1970). We will further demonstrate that the nature of the mechanical force in time and the emergence of helicity have direct implications on the cascade dynamics of these flows, affecting the anisotropy in the flow, the energy condensation at large scales and the power-law energy spectra.

Characteristics of fluid residence time in turbulent steady and unsteady round jets

Dong-Hyuk Shin[1], Andrew Aspden[4] and Edward Richardson[3]

[1]School of Engineering, University of Edinburgh
[2]Newcastle University
Faculty of Engineering and the [3]Environment, University of Southampton

Residence time or fluid age is a key concept in the understanding and design of chemically reacting flows. For multi-species flows, Ghirelli and Leckner (Chem. Eng. Sci., 2004) derived transport equations for residence times of each species. Direct Numerical Simulations (DNS) of turbulent round jets each with a jet Reynolds number of 7,290 have been conducted to investigate spatial and temporal statistics of the species residence time for both steady-state and unsteady conditions.

For the steady-state condition, mass-weighted species age (MWSA), a product of the species residence time and its mass fraction, gives a quantity that has stationary and self-similar statistics for a turbulent jet. The transport equation for the species residence time is used to derive a model describing a self-similar profile and the axial evolution for the mean MWSA.

For the unsteady case, a decelerating jet is created by suddenly stopping the inlet flow from the steady-state condition. After some transient from the stopping, another self-similar profile of the MWSA, which is lower from that of the steady-state one, is observed. Also, the transport equation is employed to develop models for both axial and temporal evolution of the centreline MWSA.

Coexistence of quantum and classical flows in quantum turbulence in the T = 0 Limit

Andrei Golov and Paul Walmsley

University of Manchester

Turbulence in pure superfluid helium is made of a tangle of vortex lines, all with the same circulation $K=h/m=0.10\text{mm}^2/\text{s}$. The “quantum” energy spectrum at short length scales is related to the total length of vortex lines, while the extent of the “quasi-classical” part at long length scales depends on the degree of alignment (polarization) of these lines and has the Kolmogorov spectrum and dynamics. We measured the vortex line length L(t) during a free decay of tangles of different initial polarizations. For small polarization, the excess random component of L first decays as $L \propto t^{-1}$ until it becomes comparable with the structured component responsible for the classical velocity field, and the decay changes to $L \propto t^{-3/2}$. The latter regime always ultimately prevails, provided the classical description of large-scale eddies holds. Our quantitative model of coexisting cascades of quantum and classical energies describes all regimes of the decay. The rate of the flux of energy per unit mass towards the dissipative processes (emission of sound and small vortex rings) at short length scales is described by the phenomenological relation $dE/dt=cK^3L^2$ with c=0.10. Walmsley & Golov, Phys. Rev. Lett. 118, 134501 (2017).

Applying particle-size segregation theory to the erosion-deposition dynamics of granular avalanches

Andrew Edwards[1], Sylvain Viroulet[2] and Nico Gray[3]

[1]University of Manchester
[2]Institut de physique du globe de Paris
[3]University of Manchester

Almost all granular geophysical events, e.g. debris flows, snow avalanches and pyroclastic flows, exhibit a continuous exchange of particles between a depositional flow and an erodible substrate. The balance between erosion and deposition influences the flow duration and runout distance, which are important for hazard risk assessment and mitigation. A perfect balance is even possible in certain conditions (Edwards, Viroulet, Kokelaar & Gray, 2017), resulting in a flow that propagates steadily downslope. By releasing a small amount of yellow sand onto an erodible bed of the same material, but coloured red, it is shown that the erosion-deposition process results in an avalanche consisting entirely of particles from the substrate layer. The experiments are simulated using a depth-averaged avalanche model and a friction law that allows dynamic, static and intermediate flow regimes (Edwards et al., 2018). This model is augmented with the large-particle transport equation (Gray & Kokelaar, 2010) for the evolution of an inversely graded shock interface between an instantaneously, sharply segregated layer of large particles above a layer of small particles. The inclusion of a segregation equation here allows for tracking of the interface between the different coloured sands. The key experimental features are qualitatively reproduced by the numerics.

Well-posed continuum modelling of granular flows

Thomas Barker[1], Nico Gray[1], David Schaeffer[2] and Michael Shearer[3]

[1]School of Mathematics, University of Manchester
[2]Duke University
[3]North Carolina State University

Flowing granular materials may be solid-, gas- or liquid-like depending on the dominant mechanism of energy dissipation. It is postulated that the transition between each of these phases may be quantified by the inertial number I. This is a non-dimensional strain-rate that describes the rate of grain rearrangements and thus the frequency of frictional contacts. The $\mu(I)$-rheology adds dependence on I to the classical Coulomb friction model, through modification of the bulk friction coefficient μ. The resultant incompressible equations resemble the Navier–Stokes equations except that the viscosity is strain-rate and pressure dependent. Unfortunately, the $\mu(I)$ equations are ill-posed when I is too low or too high. For flows in these ranges, numerical solutions exhibit grid-dependent behaviour where spontaneous perturbations grow ever faster as the grid spacing is decreased. In order to overcome these issues, two distinct strategies have been developed. Firstly, it is found that modifying the functional form of the $\mu(I)$ curve allows for the construction of a partially-regularised rheology which gives well-posed equations for all values of I below an extreme maximal value. Well-posed equations can also be formed by allowing compression so that the dissipation depends on the packing of grains as well as their rearrangements.

Viscosity sets the width of self-channelised granular flows

Chico Rocha, Chris Johnson and Nico Gray

Manchester Centre for Nonlinear Dynamics & School of Mathematics, University of Manchester

Granular flows are observed in a wide range of natural phenomena, from debris flows, snow avalanches and dense pyroclastic flows, to small scale chute flows that are used extensively in industry. Independently of the scale, these flows tend to spontaneously self-channelise on shallower slopes leading to longer run-out. In experiments with monodisperse sand, stable levees are formed and the width of the channel increases linearly with the mass flux, while the flow thickness stays approximately constant. Explaining these observations is still a significant challenge despite major recent advances in modelling the constitutive behaviour of granular flows using the μ(I)-rheology and nonlocal models. In part this is because it is necessary to incorporate frictional hysteresis to allow static and flowing regions to coexist. In this talk we propose a depth-averaged approach that provides a simple quantitative explanation of the steady-state experimental observations without any fitting parameters. Moreover, numerical simulations are able to capture the levee formation process and produce channels of the same width as that predicted by the steady-state theory, provided the flux is sufficiently high. At low fluxes, when there are no steady-state solutions the two-dimensional simulations show that a sequence of avalanches are triggered, as observed in experiments.

Gas-solid flows with tribocharging

Ali Ozel[1], Jari Kolehmainen[2] and Sankaran Sundaresan[2]

[1]Heriot-Watt University
[2]Princeton University

In this study, we seek to shed light on dynamics of tribo-electrically charged particles in granular materials through a combination of computational modelling and experiments. Tribocharging is a process by which two materials exchange electric charge upon mechanical contact; it has been observed in many industrial applications such as fluidised beds, pneumatic conveying pipes and silo flows. To study how tribocharging affects hydrodynamics of granular materials, we develop Computational Fluid Dynamics-Discrete Element Method (CFD-DEM) simulation platform, where the locally-averaged equations of motion for the fluid phase are solved in a Eulerian framework and the particles are tracked in a Lagrangian fashion by solving Newton's laws of motion. In this approach, we use a finite-volume based particle-particle and particle-mesh type Poisson solver for the electric field and a tribocharging model that accounts for the charging tendency of particles captured by effective work function difference between the contacting surfaces and the electrical field at contact. We then undertake fluidized bed experiments and measure average charge on polyethylene particles at different controlled humidity conditions. Subsequently, we perform CFD-DEM simulations of the same flow configuration and show that the predicted charge values are in a good agreement with experimental data.

Retrogressive failure of a static granular layer on an inclined plane

Aaron Russell, Nico Gray and Chris Johnson

University of Manchester

The flow of granular materials down an inclined plane is closely related to many natural hazards, such as landslides and avalanches, which can cause serious damage to life and property. Avalanches can be triggered by many different factors, such as human activity, material accumulation, wind or earthquakes. If a static layer of granular material is disturbed, it is commonly known that material downstream of this disturbance will be dislodged. However, sometimes material upslope of the disturbance also collapses, through an erosion wave that propagates upwards through the layer. This 'retrogressive failure' separates the regions of flowing and static material and is a common geophysical phenomenon that is observed on mountains and sand dunes. Retrogressive failure is critically dependent on physics beyond the μ(I)-rheology and, despite being one of the basic waves in granular flow, has not been modelled in detail before. Retrogressive failure increases the mass of material in an avalanche, and so understanding this process is crucial for improving predictions of runout distances and flow paths, and consequently reducing the associated risks. We use small scale lab experiments, novel theory and numerical simulations to model retrogressive failure, and apply our results to both geophysical and industrial contexts.

Numerical modelling of landslides: a dual particle smoothed particle hydrodynamics approach

Caitlin Chalk[1], Raul Fuentes[1], Duncan Borman[1], Bill Murphy[1], Manuel Pastor[2], Jeff Peakall[1] and Andrew Sleigh[1]

[1]University of Leeds
[2]Universidad Politécnica de Madrid

The behaviour of landslides is complex and governed by both solid and fluid dynamics. The forces that dominate vary depending on the stage of the landslide, from initiation to deposition. In addition to finding an appropriate constitutive model, a major challenge lies in how to implement the model numerically. While meshless methods have been widely accepted as a suitable numerical tool for modelling large deformation problems, they exhibit some major instabilities that are particularly relevant to landslide behaviour. Here a dual particle, Smoothed Particle Hydrodynamics (SPH) method is presented that is able to combat the instabilities known as rank deficiency and tensile instability. A rate-dependent, viscoplastic constitutive law has been implemented within this method and applied to both landslide initiation and propagation. The results are compared to those obtained with a standard SPH model, where the numerical instabilities are highlighted. The dual particle method solves these instabilities without the need for any parameter tuning. The method is also used to predict an experimental granular-water dam break flow, where velocity profiles have been obtained with Particle Image Velocimetry. The velocity and free-surface profiles of the numerical model predict the experimental results well.

Affordable detached eddy simulations of dual-stream jet structures related to shock cell noise

Aldo Rona, Alessandro Mancini and Danilo Di Stefano

University of Leicester

Pursuing the sustainable aviation targets of the Advisory Council for Aviation Research and Innovation in Europe (ACARE) in Flightpath 2050 and, equivalently, the N+2 and N+3 goals by NASA, aircraft engine manufacturers are developing Ultra High Bypass Ratio (UHBR) turbofans, such as the PW1500G and the UltraFan, using technology demonstrators like in the Clean Sky 2 Joint Undertaking (JU). By increasing the bypass ratio, a greater portion of the engine thrust and of its associated jet noise comes from the bypass stream which, with a fan operated near its design pressure ratio of 1.4 (reduced from 1.6 of earlier engines), still generates a cold under-expanded jet and unwanted shock-cell noise at cruise. Modern mission-optimized engines require affordable numerical methods able to capture the essential flow physics for performing multi-point optimizations of the propulsion system, including predictions of thrust and jet noise. In this work, the authors apply Detached Eddy Simulations to a dual-stream nozzle representative of a UHBR configuration. The large-scales, whilst coarse-grained compared to LES, are shown by acoustic analogy be sufficient for providing physically sound broad-band shock cell noise spectra.

Understanding the noise generated by a jet engine: a study of the 3D diffraction by a quarter-plane

Marianthi Moschou, Raphael Assier and William Parnell

University of Manchester

Living or working near an airport can be quite annoying. One of the reasons is the noise generated by aeroengines. Hence, the understanding of noise generated by a jet engine, which is linked with the theory of diffraction of acoustic waves, is of great importance.

Canonical diffraction theory investigates the diffraction of an incoming wave by an obstacle with simple geometry, but with certain characteristics, such as edges or corners. Most of the analytical methods in aeroacoustics are based on blade-gust interactions, where the blade is often modelled as a half-plane. The corresponding canonical problem is the Sommerfeld problem. Our goal is to understand the effect being caused by blade tips; hence we first need to study the three-dimensional diffraction by a quarter-plane, a still unsolved canonical problem in diffraction theory.

A lot of progress has been recently made about the quarter-plane problem with simple boundary conditions, the widely known Dirichlet and Neumann boundary conditions thanks to the method of embedding formulae. This is a new technique used in diffraction theory that enables to fully describe the far-field in a scattering problem.

In this talk, we will focus on extending this method to more realistic boundary conditions of impedance type.

Silent owl flight: effect of boundary-layer on trailing-edge noise

David Baker and Nigel Peake

University of Cambridge

When looking at ways of reducing the noise of an aircraft wing or a turbine blade, it is possible to take inspiration from nature: the silent flight of owls. Larger owls have evolved to make essentially no noise when they fly, through a combination of unique wing features (including rigid feathers at the trailing-edge and a soft, downy coating) and their low gliding speed, so as to both hear their prey with precision and, in turn, not be heard. Inspired by this soft, downy coating on the upper side of owl wings, a variety of noise control devices have been created, including long, fence-like structures near the trailing-edge. These have been shown to have a large effect on the boundary layer near the trailing-edge, and the mechanism by which this might reduce noise is investigated.

Investigation into the tip gap flow and its influence on ducted propeller tip gap noise using acoustic analogies

Adam Higgens[1], Phillip Joseph[2] and Stephen Turnock[1]

[1]University of Southampton
[2]Institute of Sound and Vibration Research, University of Southampton

Ducted propellers are commonly used on naval vessels such as submarines and understanding the noise signature of these vessels is of importance from detection and concealment perspective. However, the detailed physics of how the tip of the propeller blade interacts with the boundary layer on the duct are not sufficiently well understood. Several simulations are presented in which tip vortices formed over finite-span lifting surfaces are investigated with the aim of better understanding tip-gap flow and its interaction with incident boundary layers on radiated noise. Attention is devoted to characterising the nature of the incipient complex flow features and their effect on the force coefficients, surface pressure fluctuations. Variations of force coefficients with tip-gap height may be used as an early indicator of tip vortex being suppressed as the gap is reduced. Analysis of the acoustic signatures computed using Ffowcs Williams-Hawkings acoustic analogy indicates high levels of directivity with majority of the noise being generated by the foil rather than the duct. Substantial amounts of vorticity in the flow also suggests that accounting for the non-linear quadrupole noise sources within the fluid might be necessary to fully describe the noise pattern developed.

Acoustics in a lined duct with non-parallel boundary layer flow

Owen Petrie[1] and Ed Brambley[2]

[1]University of Cambridge
[2]Mathematics Institute and WMG, University of Warwick

The problem of acoustics in ducts has important applications for aircraft engines. The wall of an aircraft engine is typically lined with an acoustic lining that acts as an array of Helmholtz resonators whose action can be described by a linear impedance relation between acoustic pressure and acoustic normal velocity. Historically the Myers boundary condition has been used to model the acoustics when a mean flow exists in the duct, however it has been shown to be ill-posed so better models are needed. Many improved boundary conditions involve integrals over the boundary layer flow that must be computed numerically. Here we consider a non-parallel boundary layer flow and use a three-layer asymptotic regime to derive an analytic boundary condition that is well-posed.

Predicting and controlling thermoacoustic oscillations in a Rijke tube through data assimilation: an ensemble Kalman filter approach

Francesco Garita, Hans Yu and Matthew Juniper

University of Cambridge

Prediction and control of thermoacoustic oscillations is one of the greatest challenges in the design of combustors of aircraft engines. This work shows how this can be achieved on a simple thermoacoustic system by making use of data assimilation. In the present study, an analytical model of an electrically-heated Rijke tube is derived in order to describe (i) the base flow, driven by natural convection, and (ii) the acoustic field, driven by heat release rate fluctuations at the hot wire, which are themselves driven by the acoustic velocity. The unknown parameters of the model are estimated from many thousand experimental measurements taken from a laboratory rig at several operation conditions. For this purpose, an ensemble Kalman filter is used, which accounts for both model and experimental errors. Knowledge of these parameters is essential to predict and control thermoacoustic oscillations, and at the same time gain physical insights to apply a similar methodology to more complex systems.

Vortex sound models for active flow control

Matthew Priddin

DAMTP

Reducing aerodynamic sound remains a significant concern as aircraft and wind turbine numbers rise and we become increasingly aware of the adverse environmental effects of noise pollution. Passive control mechanisms taking biological inspiration have attracted much recent interest, offering some promise in reducing airframe noise. Active flow control enables additional means to manipulate flows, as well as the versatility to adapt to its environment. Initial experimental and numerical studies have indicated that the use of blowing and suction could offer appreciable reductions in trailing edge noise. Our theoretical understanding is limited, with investigations facing the difficulty of accurately modelling the control device in addition to any underlying complexity of the flow considered. Vortex sound offers an intuitive 2D model for the interaction of a turbulent eddy with a body and could provide valuable insights by demonstrating the role of well-defined physical processes, so elucidating the significance of additional modelling complexities. Its simplicity allows a range of geometries and active flow scenarios to be considered, thus indicating optimal strategies for control to guide more expensive numerical and experimental work. More generally, analytic techniques used to study passive control mechanisms could provide a complementary viewpoint to further develop our understanding.

Prediction of bubble dynamics during foam flow through a constricted microfluidic channel

Denny Vitasari and Simon Cox

Aberystwyth University

Foam is used as a displacement fluid in improved oil recovery and soil remediation. Knowledge about the mechanisms of foam generation in porous media is therefore important in determining appropriate material parameters. The viscous froth model has been modified by adding the disjoining forces between bubble interfaces to simulate foam flows in a two-dimensional microfludic channel. The system consists of one layer of bubbles flowing though a gradual constriction, designed to resemble the shape of channels in porous media. Upstream the bubbles form a staircase structure; rearrangement of the bubbles occurs beyond the constriction. The simulation examines the effect of driving velocity, bubble size, constriction size and liquid fraction on the rearrangement of the bubbles as they pass through the constriction. The effect of liquid fraction is examined by varying the disjoining force to give various thicknesses of the films separating the bubbles in the foam. Bubble pinch-off occurs for certain combinations of the parameters, and a phase diagram relating the driving velocity, the bubble size, the constriction size and the likelihood of pinch-off will be presented.

On yield stress of concentrated suspensions

Andreas Münch[1] and Barbara Wagner[2]

[1]University of Oxford
[2]Weierstraß-Institut für Angewandte Analysis und Stochastik

A two-phase model for concentrated suspensions that incorporates a constitutive law combining the rheology for non-Brownian suspensions and granular flow is derived from the underlying microscopic balance laws. The resulting model exhibits a yield-stress behavior for the solid phase as a function of the collision pressure. This property is investigated for simple geometries such as plane Poiseuille flow, where jammed zones of finite width arise. Phase-space methods and asymptotic analysis are used to discuss the evolution to these stationary states. The stability properties of these solutions and their relation to classical Bingham-type flows are discussed.

Instability of pressure driven channel flows of shear-thinning viscoelastic fluids

Hugh Barlow[1], Ewan Hemingway[1], Andrew Clarke[2] and Suzanne Fielding[1]

[1]Durham University
[2]Schlumberger

Viscoelastic fluids are a common feature of industrial and natural processes. In this study we examine the stability of shear-thinning viscoelastic fluids under pressure driven channel flow conditions. Motivated by recent experimental findings and previous theoretical work, we demonstrate, using linear stability analysis and full non-linear simulations, the instability to be rather generic across models of shear thinning viscoelastic polymer solutions. This finding is consistent with experimental findings (Bodiguel et al. PRL 2015, Poole PRF 2016, Picaut et al. PRF 2017), as well as previous theoretical work (Wilson and Rallison JNNFM 1999, Wislson and Loridon JNNFM 2015, Castillo and Wilson JNNFM 2017). We consider the way in which the onset of the instability depends on the variation in the flow properties across the channel. This results from the stress gradient inherent to channel flows, combined with the shear thinning properties of the fluid and the normal stress properties common in viscoelastic polymeric materials. We study these factors by comprehensively varying across simulation runs the values of the model parameters and imposed pressure drop.

Rheological properties of a solid sphere suspension in second order fluid using a single cell model

Liam Escott and Helen Wilson

University College London

Suspensions described by the inclusion of solid spherical particles into a solvent fluid phase has been a topic of interest for rheology and mathematics for the last half a century. More recently, investigations into the material functions for a dilute suspension with a weakly linear background flow profile have been conducted, in particular where the suspending medium is a second order fluid. In our current work, we have employed a single cell model to describe the complex fluid at higher solid volume fractions, and we also demonstrate the validity of our results under both the dilute limit and the sphere packing limit. We explore the effective viscosity and normal stress differences in both shear and extensional flow, providing the relationship between these functions and their solvent counterparts, while also endeavouring to show a non-linear dependence on concentration, made possible by our cell model proxy.

Capillary breakup extensional rheometry (CaBER) of graphene oxide suspensions

Henry Ng[1], Andrew Corker[1], Eduardo Saiz[2], David Dennis[1], Esther Garcia-Tunon[1] and Robert Poole[1]

[1]University of Liverpool
[2]Imperial College London

Graphene oxide (GO) colloids are known to be shear thinning (with their viscosity/viscoelastic properties known to be highly concentration dependent), forming networks at low concentrations (2 – 4vol%) that in the absence of any other additives can be 3D printed using extrusion based processes; recovering to predominantly solid-like behavior (G′>10kPa) when the shear is removed. Remarkably, the addition of GO at lower concentrations (1vol% depending on the system) facilitates the formation of pastes to build free-standing 3D printed objects using materials such as; polymer solutions (PVA) and particulate systems (steel microspheres, SiC, and Al_3O_2 platelets) which, in the absence of additives are unsuitable for 3D printing.

In the current work we explore the extensional properties of GO using capillary breakup extensional rheometry (CaBER). In these experiments an unstable fluid filament held between two parallel plates undergoes capillary-thinning and breakup. Extensional properties of the fluid are then determined from the time evolution of the filament's minimum radius which we capture using a high-speed camera. Our results indicate that even small amounts of GO flakes (0.05–0.3vol%) can significantly alter the pinch-off dynamics of a liquid bridge in comparison to Newtonian fluids of equivalent viscosity.

Numerical modelling of polymer blends using OPENFOAM

Adila Aida Azahar, Oliver Harlen and Mark Walkley

University of Leeds

Modelling of polymer flow has largely focused on monodisperse materials where rheological properties can be predicted from molecular structure. Industrial polymers are a blend of molecules of different molecular weights. Understanding blend rheology involves modelling the interaction between different length polymer chains, so that predictions of the behaviour of the blended material can be made from its molecular weight distribution.

We consider a mathematical model for bidisperse polymer blends based on the Rolie-Poly constitutive equation[Boudara,2016]. This incorporates the interaction of short and long chain polymers. These equations are coupled with the flow as an additional, polymeric stress.

The numerical model is developed in OpenFoam using the rheoTool package[Pimenta,2017], which allows viscoelastic constitutive laws to be simulated. We extend rheoTool to include the Rolie-Poly model and further to blends.

Simulations are carried out for a hyperbolic contraction geometry that produces regions of both shear and extensional flow. 2D and 3D simulations span extension rates from below the inverse reptation time to rates greater than the inverse Rouse time to capture the rheological behaviour of the fluid. Both blended and non-blended models are compared to observe the effect of the coupling in the blended model and prediction of the extensional viscosity.

Non-Newtonian models for a vertical draining free liquid film

Hani Alahmadi and Shailesh Naire

Keele University

The evolution and stability of thin liquid films are important in variety of applications such as in foam networks relevant to the manufacture metal foams, food industry and processing in the petro-chemical industry and the biological and life sciences. In a liquid foam there are gas bubbles separated by thin liquid lamella. If one is interested in predicting the lifetime of a foam or its overall stability then, as a starting point, understanding the drainage within the lamella is important.

Motivated by the above, we consider theoretically a model system of a vertical free film suspended between wire frames and draining due to gravity. The liquid in the film is considered to be non-Newtonian. Lubrication theory is employed derive coupled nonlinear evolution equations for the film thickness and the extensional flow speed. We use the power-law, Bingham and Herschel-Bulkley constitutive laws to describe the flow rheology. The equations are solved numerically and similarity solutions are derived. The similarity solutions reveal power-law behaviour in the thinning rate of the middle section of the film. Comparisons are made with the Newtonian case.

This work is relevant to addressing fundamental questions regarding the influence of non-Newtonian effects on liquid film and foam drainage.

The state space of near-wall turbulence at infinite friction Reynolds number

Patrick Doohan[1], Ashley P. Willis[2] and Yongyun Hwang[1]

[1]Imperial College London
[2]University of Sheffield

There has been a growing body of evidence that there exists self-similar coherent structures in the form of Townsend's attached eddies. Each of these structures bears a self-sustaining process remarkably similar to that in the near-wall region. To model this universal feature of wall-bounded turbulence, we have designed a shear flow model of near-wall turbulence applicable to various parallel shear flows at infinite friction Reynolds number. As a first step, we consider the minimal unit of near-wall turbulence: the governing equations are rescaled in inner units, while a constant shear stress is imposed as the top boundary condition of the domain located at y+=90. The model is validated against Couette flow, the near-wall region of which is independent of Reynolds number, and there is excellent agreement between velocity statistics and spectra for y+<70. Thirteen relative equilibrium solutions are presented, the first discovered for this flow configuration. Through continuation in the spanwise width Lz+, the bifurcation behaviour of the equilibria over domain size is examined. The physical properties of the equilibria are explored through state space projection. Finally, the asymptotic behaviour of the equilibria is studied and three lower-branch solutions are found to scale consistently with vortex-wave interaction theory.

Turbulent dissipation and entrainment in planar jets

Tai Wada and Christos Vassilicos

Imperial College London

Zhou & Vassilicos (JFM 2017) showed that the turbulent cascade's impact on the turbulence dissipation's scaling has a defining influence on the scalings of the turbulent/non-turbulent (TNT) interface velocity relative to the flow in a turbulent wake. In this work we demonstrate both theoretically and computationally that this effect, which relates the turbulence cascade to entrainment, is also present in a turbulent planar jet. Direct Numerical Simulations (DNS) of such jets have been carried out and confirm the local entrainment predictions which follow from non-equilibrium turbulent cascades and which are different from the classical predictions of Corrsin & Kistler (1955).

On the accuracy and convergence properties of the stochastic field method

Fabian Sewerin

Technical University of Braunschweig

In large eddy simulation (LES) of turbulent reacting flows, we are confronted with the turbulence-chemistry interaction closure problem, relating to the evaluation of LES-filtered source terms for fluid phase scalars. This closure problem can be circumvented with the aid of an LES-transported PDF formulation that is based on an evolution equation for the LES-filtered probability density function (PDF) associated with a single realization of the fluid composition. In the present contribution, we analyze in detail the accuracy and convergence properties of a stochastic scheme, the method of Eulerian stochastic fields, for solving the PDF transport equation.

To this end, we confine the attention to a single reactive scalar that evolves in a turbulent jet flow. Here, the PDF transport equation takes on the form of a four-dimensional convection-diffusion-reaction equation. Since this equation may be solved using a direct discretization approach, the single scalar problem allows us to obtain reference solutions with which solutions from the stochastic field method may be compared. Our analysis focusses on the statistical convergence properties of the PDF moments both on an instantaneous basis and for time averages. On the side, we address the problem of bound violation for bounded scalars such as species mass fractions.

Re-building turbulent fields using 4D variational data assimilation: reproduction of instantaneous structures

Naseer Abdullah and Yi Li

School of Mathematics and Statistics, University of Sheffield

We look into a turbulent system where incomplete velocity measurement data over a time period are available. Assuming that the dynamical evolution of the system is governed by the Navier-Stokes equation, we use 4D variational data assimilation, which is a valuable tool for recovering missing information about a system from what is directly measurable to re-build the initial state of the system, such that the evolution of the system matches the measurement data.

We formulate the problem as an optimization problem, where the initial field is taken as the control variable. The goal is to find the optimal control variable to minimize the difference between the measurement and the velocity field evolved from the re-constructed initial field, subject to the constraint imposed by the Navier-Stokes equation.

The re-built fields are compared with direct numerical simulation (DNS) data. We look into the difference between the instantaneous variations of the reconstructed and the DNS fields. The examples of the averaged geometries show that the differences decrease with time within the optimization horizon. Also, the instantaneous high vorticity structures for the re-built fields are reproduced with good agreement with DNS field.

Data assimilation and parameter estimation of thermoacoustic instabilities in a ducted premixed flame

Hans Yu, Francesco Garita, Matthew Juniper and Luca Magri

University of Cambridge

Thermoacoustic instabilities are a persistent challenge in the design of jet and rocket engines. If acoustic pressure fluctuations inside the combustor are sufficiently in phase with the unsteady heat released by the flame, thermoacoustic oscillations may arise. These oscillations can cause structural damage, vibrations and fatigue to the jet or rocket engine.

The time-accurate calculation of thermoacoustic instabilities is challenging due to the presence of both aleatoric and epistemic uncertainties as well as the extreme sensitivity to small changes in the parameters of the system. We propose a method to time-accurately calculate thermoacoustic instabilities and the flame dynamics by augmenting lower-order models with data from experiments and high-fidelity simulations using data assimilation and parameter estimation with the ensemble Kalman filter. Data assimilation gives an optimal estimate of the true state of a system. Parameter estimation uses the data to find a maximum-likelihood set of parameters for the model. The thermoacoustic system under investigation is the vertical Rijke tube with a premixed flame modelled with a G-equation. The ensemble Kalman filter gives an improved estimate of the location of the flame front as well as the parameters in the G-equation. Additionally, the uncertainties in the estimates are quantified.

Entrainment and mixing enhancement in plasma-controlled turbulent jets

Vasilis Ioannou and Sylvain Laizet

Imperial College London

Large Eddy Simulations are employed to investigate the effect of Dielectric Barrier Discharge (DBD) plasma actuators on the mixing and entrainment properties of a turbulent jet. These simulations are carried out using the high-order, finite-difference, flow solver Incompact3d (www.incompact3d.com), which solves the incompressible Navier-Stokes equations. Several radially distributed DBD plasma actuators are placed just before the exit of a convergent nozzle in order to manipulate the initiation, evolution and growth of the jet main instabilities. Different configurations of plasma actuators are studied, including different azimuthal modes, number of actuators, as well as their location inside the nozzle. We will show that DBD plasma actuators can significantly change the near-field structure of the jet, modifying the large-scale coherent structures, potential core, jet spreading rate thereby affecting the entrainment rate of the ambient fluid into the jet. As a consequence, a better mixing can be observed for plasma-controlled turbulent jets, especially when the plasma actuators are based on a pulsating motion at a frequency corresponding to the jet preferred frequency.

Interaction between turbulent convection and tidal flows in stars and planets

Craig Duguid and Adrian Barker

University of Leeds

Since the discovery of the first exoplanet orbiting a main sequence star in 1995 a large, and still growing, number of planets of approximately Jupiter mass but with orbital periods shorter than 10 Earth days have been discovered. These planets are now classified as Hot Jupiters (HJ) and there is great interest in understanding how these giants found themselves in such close proximity to their host stars. Gravitational tidal interactions can play an important role in the evolution of their orbits and spins, but the fluid dynamical mechanisms responsible for tidal dissipation are poorly understood. Our approach is to look at the interaction between an oscillatory background shear flow, which represents a large-scale tidal flow, and the convecting fluid inside a planar layer of a star or planet. We simulate Rayleigh–Bénard convection and explore the how the effective viscosity of the turbulence depends on the tidal (shear) frequency. A brief background into the subject will be presented followed by preliminary results on the effects of the background tidal shear in laminar and weakly turbulent non-rotating cases.

Aqueous flow in the anterior chamber of the rotating eye: a mathematical model

Mariia Dvoriashyna[1], Rodolfo Repetto[1] and Jennifer H. Tweedy[2]

[1]University of Genoa
[2]Imperial College London

The anterior chamber (AC) of the eye is the region between the iris and the cornea. It contains the aqueous humour, a transparent fluid, which flows from the posterior to the AC at an approximately constant rate. Understanding aqueous flow dynamics is important for several reasons, among which are mixing and transport of nutrients and generation of stresses on the cornea. Previous numerical studies suggest that flow due to eye rotations plays an important role in these processes, and we develop and solve a theoretical model to investigate their roles. We model the AC as a domain sitting on the surface of a sphere that rotates harmonically about a fixed axis, and the aqueous as a Newtonian and incompressible fluid. The AC is relatively narrow in the anterior—posterior direction, which allows us to simplify the problem using lubrication theory and find a semi-analytical solution. Our model predicts that the flow in the AC has a highly three-dimensional structure, which was not observed in previous numerical works. Eye rotations cause steady streaming that contributes to mixing in the AC, and our results suggest it is the main mechanism for aqueous mixing during sleep.

Lockhart with a twist: The effect of variations in mechanical properties through the plant cell wall thickness on growth and twisting

Rosemary Dyson[1] and Jeevanjyoti Chakraborty[2]

[1]University of Birmingham
[2]IIT Kharagpur

Understanding plant growth and development is key to ensuring a secure food source. Plant cell expansion is governed via manipulation and rearrangement of oriented polymers including aligned cellulose microfibres within the cell wall via either passive reorientation as the wall expands or active biological control. In particular, rapid cell expansion must be accompanied by new material deposition to prevent rupture of the cell wall. These newly-incorporated components, combined with passive motion of material through the thickness of the cell wall, can lead to local variations in the effective mechanical properties of the composite wall material. We derive, analyse and interpret a mathematical model, treating the cell wall material as a composite fibre-reinforced fluid, which takes these variations into account, in particular considering the mechanical anisotropy given by oriented cellulose microfibres (and their angle at deposition) and the effect of time-dependent modification in the pectin matrix properties. We present our findings, and discuss their consequences with a particular focus on the resultant growth and twist of the cell.

Physical and geometrical determinants of transport in feto-placental microvascular networks

Alexander Erlich[1], Philip Pearce[2], Romina Plitman Mayo[3], Oliver E. Jensen[1] and Igor L. Chernyavsky[1]

[1]University of Manchester
[2]Massachusetts Institute of Technology
[3]University of Cambridge

What governs the supply of a human fetus with oxygen from the mother? We address this question with a model of the human feto-placental microvasculature. The physical setup in the human placenta is unique: the maternal and fetal blood supplies are separated by a thin layer of villous tissue (the syncytiotrophoblast), through which oxygen is exchanged by diffusion. Compared to other vasculatures, the capillaries are unusually loopy and bulged. Oxygen transfer from mother to fetus can be estimated using 3D finite-element simulations on realistic geometries. However, owing to the computational expense of such calculations, it is infeasible to perform them on entire feto-placental capillary networks. Instead, we introduce a reduced 1D network model to simulate blood flow and oxygen transfer in feto-placental capillary networks efficiently. We validate the reduced model against full 3D computational fluid dynamics simulations on three-dimensional feto-placental geometries, obtained by confocal microscopy. The reduced model is used to study how network topology and vessel geometry affect oxygen transfer to the fetus. This reduced 1D flow and transport model of the feto-placental microvasculature may contribute to multiscale models of the placenta and other biological systems.

Mathematical modelling of ureteroscope irrigation

Jessica Williams, Sarah Waters, Derek Moulton and Ben Turney

University of Oxford

Ureteroscopy is a procedure to examine the urinary tract, often used for kidney stone removal surgery. The flexible ureteroscope has a deflectable tip, along with a central lumen for working tools (laser fibres, stone collecting baskets, guidewires). Successful ureteroscopy relies on a clear intrarenal view which is achieved by an irrigation process (driving fluid through the lumen via an applied pressure drop). The relationship between the fluid flow and the applied pressure drop depends on the geometric properties of the ureteroscope and working tools.

We constructed a mathematical framework, based on systematic reductions of the Navier Stokes equations, to relate pressures and volumetric flow rates during ureteroscopic procedures. Our theoretical predictions were validated via wet-lab experiments.

The model for flow rate in ureteroscopes demonstrated an excellent correlation with the wet-lab findings. The ureteroscope containing a working tool can be accurately modelled via two coaxial, non-concentric cylinders of uniform cross-section. We explore how geometrical modifications to the annular fluid domain, either via varying the position of the tool, or by adjusting the cross-sectional shape of the lumen affect the flow rate. This allows us to identify geometrical configurations which maximise flow rate for a given pressure drop.

Transition to turbulence is delayed in shear-thinning blood analogs in contrast to Newtonian analogs

Nathaniel Kelly, H. S. Gill, Andrew Cookson and Katharine Fraser

University of Bath

Both multiphase and non-Newtonian, blood is a complex fluid. In blood-contacting medical devices the flow becomes turbulent and simplification as a Newtonian fluid can result in incorrect predictions of flow behaviour, which could in turn lead to misconceptions about biological processes and design and performance errors. This work aimed to calculate the velocity fields over a backward facing step for two blood analogs, Newtonian and shear-thinning, finding the transition in both.

The backward facing step was modelled using open source finite volume software OpenFOAM. Reynolds number was $50<Re<7000$ to encompass the full transition-to-turbulence. The Newtonian model assumed a constant viscosity while the shear-thinning used a Carreau rheological model.

Comparisons between the recirculation zone lengths (RZLs) of the blood analogs showed that the shear-thinning fluid reduced the length by 12.3% at Re=389. The RZLs for Newtonian analog were in good agreement (<8% difference) with existing experimental data. Velocity fluctuations over time showed that transition to turbulence in the shear-thinning analog was delayed from Re=1200 to Re=1800.

This work will allow a clearer depiction of the damaging shear stresses present in transitional blood flow. Future work will explore this behaviour experimentally with multiphase blood analogs and whole blood.

An Eulerian-Lagrangian agent method model to predict fish responses to hydrodynamic cues

Thomas Padgett[1], Robert Thomas[2] and David Mould[3]

[1]University of Leeds
[2]University of Hull
[3]JBA Consulting

A huge number of fish migrate through river systems, and must overcome an increasing number of man-made barriers such as weirs. These barriers are mitigated through the installation of fish passes designed to facilitate fish passage. However, the efficiency of such structures is low and little has been done to determine their optimal geometries, angles and placement relative to the channel centreline and banks. An Eulerian-Lagrangian Agent Method (ELAM) model is described that simulates movements of a group of individual fish through a river environment based on their reactions to hydrodynamic and environmental cues. Hydrodynamic cues are determined using a 3D RANS methodology. A Lagrangian framework is used to determine the sensory perception and subsequent movement of each individual, using a sensory ovoid which defines an individual's sensory range. Each individual makes a movement decision through a ruleset-based approach that considers hydrodynamic and environmental drivers with some stochasticity. The model structure is presented along with preliminary, calculated fish trajectories. This is the first step in building an open-source tool which can be used in the design of future fish pass facilities.

Poroelastic response of soft hydrogels connected to a water reservoir with an evaporation surface

Merlin A. Etzold, Paul F. Linden and M. Grae Worster

University of Cambridge

We consider covalent hydrogel beads of spherical geometry (when fully saturated) whose lower part is connected to a water reservoir, the upper part is exposed to open air and evaporation. The pressure below the hydrogel is below ambient. The beads deform and shrink; over time an equilibrium is reached in which the bead size also depends on the relative humidity of the surrounding air. The bead size responds to changes in pressure by shrinkage (increase) and swelling (decrease).

We argue that the key to understanding the response of the hydrogel is the osmotic pressure within the bead, which balances the entropic stress from the polymer when fully saturated. The osmotic pressure is a function of the local deformation of the hydrogel. Neglecting evaporation, the connection to the reservoir decreases the pore pressure and causes the bead to shrink. In equilibrium, evaporation on the bead surface is balanced by Darcy-type flow, which is driven by a gradient in osmotic pressure. We will present a quantitative one-dimensional model for this problem and compare to the experiments.

Our motivation to study this problem lays in understanding how trees can lift water beyond the cavitation limit (~10 m) by evaporation from the leaves alone.

Rates of mixing by cutting and shuffling

Hannah Kreczak, Rob Sturman and Mark C.T. Wilson

University of Leeds

The theory of dynamical systems has been successful in improving understanding of the underlying features of fluid mixing and how smooth stirring fields, coherent structures and boundaries effect mixing rates. The main mechanism of mixing in fluids is the stretching and folding of fluid elements, although this is not the only mechanism to achieve complicated dynamics. Mixing by cutting and shuffling occurs in many situations; such as card shuffling, granular mixing or cut-and-recombine stirring devices, however, the dynamics of this mixing mechanism are subtle and not well understood. Analytical results on the mixing rates of cutting and shuffling systems focus on asymptotic time and are sensitive to infinitely small tuning of parameters. Such results would not be of interest in finite time considerations or apply in the presence of process error in practical applications. Hence, a computational study addressing the rates of finite time mixing in systems in which cutting and shuffling are the dominating stirring mechanism is presented. The study considers the effect of stirring protocol parameters, initial conditions and the effect of molecular diffusion.

The direct computation of time-periodic solutions of PDEs & applications to fluid dynamics

Puneet Matharu, Andrew Hazel and Matthias Heil

University of Manchester

At sufficiently large Reynolds numbers the flow around a stationary cylinder results in the formation of the famous von Karman vortex street – a time-periodic flow in which vortices are shed, alternately on either side of the cylinder. When the cylinder performs forced oscillations transverse to the flow direction, the vortex shedding pattern becomes significantly more complex, leading to the formation of so-called “exotic wakes” whose character is controlled by the Reynolds number as well as the frequency and amplitude of the cylinder’s motion (see Williamson and Roshko 1988).

The ultimate aim of our project is to numerically analyse the dependence of the time-periodic “exotic wake” flow regimes on the system’s control parameters. The determination of such solutions by direct time-integration can be very inefficient as transients may take a long time to decay. Furthermore, this approach cannot capture unstable periodic solutions, making it impossible to determine the system’s full bifurcation structure underlying the transition between the different flow regimes.

In this talk we present a finite-element based space-time approach that allows the direct computation of time-periodic flows. We also discuss ongoing work on the development of efficient solvers/preconditioners for the large, structured system of equations arising from the discretisation.

An investigation of the effect of biomimetic tubercles on the drag of a flat plate

Alessandro Marino, Mehmet Atlar and Yigit Kemal Demirel

University of Strathclyde

This work describes a CFD study of the effect of biomimetic tubercles on a flat plate. Tubercles are inspired by those observed on the head and pectoral fins of humpback whales (Megaptera Novaeangliae). Recently, extensive research has been carried out on the effect of sinusoidal tubercles on the leading edge of wing profiles and marine foils, showing a general improvement in the post-stall performance and in terms of lift-to-drag ratio.

The authors investigated the effect of similar sinusoidal tubercles on a flat, smooth plate. Various arrangements of the tubercle were positioned at different points along the plate's length, and the change in its drag characteristics is investigated. The drag of the plate with tubercles and that of a smooth flat plate of identical dimensions are compared; the flow quality is described, in particular in terms of pressure distribution and flow speed close to the plate.

Future implementation of this research will include a systematic variation of the geometry and distribution of the tubercles, and physical tests performed with the Fully Turbulent Flow Channel that is being built at the University of Strathclyde. Therefore, a short reporting on the design and commissioning of this facility will be included in the paper.

Simulating variable density flows in the low-Mach number limit with Incompact3D

Paul Bartholomew and Sylvain Laizet

Imperial College London

Incompact3D is a widely used flow solver providing highly scalable, high (6th) order accurate capabilities for simulating incompressible turbulent flows, an efficient spectral Poisson solver providing the divergence-free condition up to machine accuracy. The QuasIncompact3D solver presented in this talk is a newly developed solver for the Low Mach Number (LMN) approximation of low speed variable density flows.

In implementing the LMN approximation equations, two approaches for the Poisson equation are possible: one yielding a constant-coefficient Poisson equation, similar to the incompressible case; and one yielding a variable-coefficient Poisson equation. The variable-coefficient Poisson equation has theoretical advantages, guaranteeing the satisfaction of the LMN approximation equations, however spectral Poisson solvers can only solve constant-coefficient Poisson equations, hence an iterative approach is required. In this work, both constant-coefficient and variable-coefficient Poisson solvers are implemented and a new density interpolation is proposed to improve convergence rates, potentially saving significant computational effort.

Results for strong and weak scaling tests on both EPCC's ARCHER, and CINECA's Marconi machines will be presented alongside investigations of variable-density, low-speed flows: a non-isothermal mixing layer, a turbulent buoyant jet and a non-Boussinesq gravity current.

Predicting chaotic behaviour in rotating fluids using data assimilation

Godwin Madho and Steven Tobias

University of Leeds

All computer models are imperfect. This problem is intensified further when studying chaotic behaviour. It is important to reconcile the differences in the results produced by modelling and observations. Many studies have used data assimilation to overcome this issue, where the model is "corrected" from time to time using the observation. This method has been used in various fields such as weather prediction and oceanography successfully.

Thermally rotating annulus experiments have been used for a long time to study the general circulation of the atmosphere. And although data assimilation work has been done on these systems, there have always been problems when the flow becomes too chaotic/turbulent. In this study we present results using the Ensemble Kalman Filter (EnKF) data assimilation method and look at how the EnKF system copes with different flow regimes that are usually observed in rotating annulus studies. We show how different tuning parameters in EnKF such as the number of ensembles affect our ability to accurately "correct" our model. We present results using observation obtained from synthetic data and data obtained from experiments.

Extremum seeking control for drag reduction of a truck trailer

Peter Rhodes, Mark Jabbal, Kwing-So Choi and Harald Pfifer

University of Nottingham

The aim of this paper is to present a novel approach to reducing aerodynamic drag on truck trailers. Overall net gains with flow control in an open loop setting are currently too small to justify application. Hence, the present paper will exploit the use of feedback control to adapt to changes in the flow. Extremum seeking control has been widely applied to the problem of drag reductions in the aerospace field, e.g., in formation flight, but also in the context of flow control. It is a gradient based optimization method that does not rely on an explicit model formulation. The goal of the closed loop system will be to minimize the steady state drag of the truck trailer using minimal control power.

While the CFD model to be presented will not be a high fidelity model of a truck trailer, it will cover the necessary characteristics to show the applicability of the method for drag reduction. The model will use a pre-defined jet actuator located at the rear of the body to reduce the extent of separated flow. Validation results for the CFD model, including wake analysis, pressure recovery and power savings made by the control scheme, will be presented.

Real-time, interactive CFD on heterogeneous mobile clusters

Adrian Harwood

University of Manchester

Real-time, interactive CFD tools in virtual engineering require accelerated fluid simulation; the calculation advances at a rate fast enough for instantaneous rendering of flow information. Perceiving the movement of flow in 'real-time', users may interact with the running simulation, changing geometrical or physical parameters. However, each real-time simulation requires a suitable compromise between accuracy and speed. Furthermore, the computational effort needed must be delivered by one or more GPUs or by large CPU clusters which are costly solutions. Mobile devices are an affordable, pervasive computing resource with the potential for offering significant computing power if used as a networked cluster. This work explores the use of a peer-to-peer (P2P) network of heterogeneous mobile devices for real-time, interactive CFD. 2D CFD simulations are performed on mobile GPUs using an implementation of the D2Q9 lattice-Boltzmann Method. The performance, accuracy and scalability of the P2P-distributed calculation is established for a network of heterogeneous Android-powered mobile devices.

Experimental and numerical analysis of a swirling flow in a pipe with annular cross section

Gianluca Padula, Svetlana Aleksandrova, Humberto Medina and Stephen Benjamin

Coventry University

Swirling flows are present in many engineering applications (e.g. turbomachinery, vortex generators, combustion and heat exchangers) as well as in natural flows.

Although extensively studied numerically and experimentally, they still present many challenges in terms of modelling and understanding the mechanisms governing the flow.

This project is dedicated to the evolution of a swirling flow in an annular conduit with a diameter ratio of 0.44 for a range of mass flow rates and swirl levels. The static pressure distribution on the outer wall of the annular pipe and the mean velocity at the pipe inlet and outlet sections are measured and used for validation of the CFD model implemented in StarCCM+. The effect of different turbulence models, inlet and outlet conditions on the model predictive capabilities is investigated. The tuned model is then used to carry out a parametric study of the effect of swirl number and mass flow rate on flow pattern. In particular, the parameters governing the flow evolution from axial flow to highly swirling flow featuring wall jet pattern are investigated.

A comparison between the numerical and the experimental results is carried out to identify the limitations of the RANS models used to correctly predict swirling flows.

Accurate, efficient and easy modelling of biological Stokes flow

David Smith

University of Birmingham

Biological Stokes flow includes examples such as flagellar motility of sperm, algae and bacteria, and cilia-driven transport in reproduction and development. A key feature unifying these problems is that the geometry is characterised by geometrically complicated moving boundaries. Regularized stokeslet methods have received major interest in the last decade in this field due to their ease of implementation and numerical robustness when compared with boundary integral or finite element methods. However, the standard (Nystrom) discretization results in excessively large dense linear systems, limiting the reach of the method for problems involving multiple cells or cilia. We present a novel discretization method based on nearest neighbour interpolation; the method is available in openly-accessible Matlab libraries. The potential of the technique is demonstrated on problems from chemistry, sperm motility and embryonic nodal flow: we find an order-of-magnitude improvement in efficiency and accuracy, with only mildly increased implementational complication.

Enhanced sedimentation of elongated plankton in simple flow

Rachel Bearon[1], William Clifton[1] and Martin Bees[2]

[1]University of Liverpool
[2]University of York

Negatively buoyant phytoplankton play an important role in the sequestration of carbon dioxide from the atmosphere. However, there is still much to discover on transport mechanisms from the upper photosynthetic regions to the deep ocean.

In contrast to intuitive expectations that mixing increases plankton residence time in light-rich regions, recent experimental and computational evidence suggests that turbulence can actually enhance sedimentation of negatively buoyant diatoms. We here explore the role that shape has on the sedimentation mechanisms in simple flows. For example, in vertical shear, preferential flow alignment and aggregation in down-welling regions both increase sedimentation, whereas horizontal shear reduces the sedimentation due only to alignment. In simple vertical Kolmogorov flow elongated particles also have an enhanced sedimentation speed as they spend more time in down-welling regions of the flow with vertically aligned orientation, an effect that increases with the magnitude of shear. In horizontal Kolomogorov flow, the impact of shear-dependent sedimentation speed is to cause aggregation in regions of high-shear where the sedimentation speed is minimum. In cellular flow, there is an increase in mean sedimentation speed with aspect ratio and shear strength associated with aggregation in down-welling regions.

Hydrodynamics of bacteriophages

Panayiota Katsamba and Eric Lauga

University of Cambridge

Bacteriophage viruses, one of the most abundant entities in our planet, lack the ability to move independently. Instead, they crowd fluid environments in anticipation of a random encounter with bacteria. Once they 'land' on their victim's surface, they eject their genetic material inside the host cell. A big fraction of phage species, however, first attach to the flagella of bacteria. Being immotile, these so-called flagellotropic phages still manage to reach the cell body for infection, and the process by which they move up the flagellum has intrigued the scientific community for over four decades. In 1973 Berg and Anderson proposed the nut-and-bolt mechanism in which, just like a nut being rotated moves along a bolt, the phage wraps itself around a flagellum possessing helical grooves (due to the helical rows of flagellin molecules) and exploits the rotation of the flagellum in order to passively travel along it. We provide here a first-principle theoretical model for this nut-and-bolt mechanism and show that it is able to predict experiment observations.

Viscous propulsion in active transversely-isotropic media

Gemma Cupples, Rosemary Dyson and David Smith

University of Leeds

Taylor's swimming sheet is a classical model of microscale propulsion and pumping. Many biological fluids and substances are fibrous, having a preferred direction in their microstructure; for example cervical mucus is formed of polymer molecules which create an oriented fibrous network. Moreover, suspensions of elongated motile cells produce a form of active oriented matter. To understand how these effects modify viscous propulsion, we extend Taylor's classical model of small-amplitude zero-Reynolds-number propulsion of a 'swimming sheet' via the transversely-isotropic fluid model of Ericksen, which is linear in strain rate and possesses a distinguished direction. We find that the energetic costs of swimming are significantly altered by all rheological parameters and the initial fibre angle; we discuss the impact of a range of these parameters on the speed and efficiency of microscopic propulsion.

Vesicle transport and cytoplasmic streaming in the pollen tube

James Tyrrell, Rosemary Dyson and David Smith

University of Birmingham

The rapid elongation of the pollen tube in seed plants cannot occur without the transport of sufficient cell wall and membrane material to the growing apex. The movement of this material, delivered via exocytic secretory vesicles, can be categorised under two regimes: ‘long distance’ movement in the shank (via active transport along actin filaments), and ‘short distance’ movement in the apex (where vesicles diffuse and advect freely). Many current models of vesicle transport focus on diffusion in the apical region alone, neglecting advective effects as well as the resulting distribution profile in the pollen tube shank. Using the method of regularised Stokeslets with an adjustment made for axisymmetry, we produce a complete advective velocity profile for cytosolic flow in the tube based on the drag induced by the active transport of vesicles along actin. We go on to derive a pair of advection-diffusion equations for exocytic and endocytic vesicle motion in the tube, coupled by conditions at the growing boundary under the assumption of steady growth. We show that it is possible to obtain the desired increased apical vesicle concentration (the so-called ‘inverted vesicle cone’) using this regime, with advection being essential to the shape of the distribution.

A finite element approach to spring network models of micro-swimmers

Cara Neal

University of Birmingham

The study of micro-swimmers has a variety of different biological and clinical applications. For example, studying the swimming of human spermatozoa is an increasingly important problem when dealing with the diagnosis and treatment of male infertility. In this talk, we start by modelling a simple Najafi-Golestanian swimmer consisting of three spheres, linked by two springs. Altering the configuration of the spring network periodically in a nonreciprocal fashion allows for one-dimensional swimming at low Reynolds numbers. This so called ‘bead spring network’ model combines the method of regularised stokeslets and the finite element method to capture the motion of a swimmer in three dimensions. This method can then be extended to study more complicated micro-swimmers, such as the beating flagella of spermatozoa, in a variety of different domain geometries.

Model-based image analysis of a tethered Brownian fibre for shear stress sensing

Meurig Gallagher, Cara Neal, Kenton Arkill and David Smith

University of Birmingham

The measurement of fluid dynamic shear stress acting on a biologically relevant surface is a challenging problem, particularly in the complex environment of the vasculature. While an experimental method for the direct detection of wall shear stress via the imaging of a synthetic biology nanorod has recently been developed, the data interpretation so far has been limited to phenomenological random walk modelling, small angle approximation, and image analysis techniques which do not take into account the production of an image from a three-dimensional subject. In this talk I will present a mathematical and statistical framework to estimate shear stress from rapid imaging sequences based firstly on stochastic modelling of the dynamics of a tethered Brownian fibre in shear flow, and secondly on a novel model-based image analysis, which reconstructs fibre positions by solving the inverse problem of image formation. I will present the testing of this framework on experimental data, providing the first mechanistically rational analysis of the novel assay. This work further develops the established theory for an untethered particle in a semi-dilute suspension, which is of relevance to, for example, the study of Brownian nanowires without flow, and presents new ideas in the field of multidisciplinary image analysis.

Stability of upflowing and downflowing gyrotactic microorganism suspensions in a three-dimensional vertical cylindrical pipe

Lloyd Fung and Yongyun Hwang

Imperial College London

Experiments by Kessler (1986, J. Fluid Mech. 173:191–205) have shown that in a suspension of downflowing gyrotactic swimming microalgae, the cells form a beam-like structure. Such a structure is prone to an axisymmetric blip instability. To better understand the blip instability, a three-dimensional stability analysis is performed in a circular pipe flow by extending the previous analysis of two-dimensional channel flow (Hwang & Pedley (2014, J. Fluid Mech. 749:750-777) with the aim to provide a direct comparison between a theoretical analysis and Kessler's early experiment. Further to this, upflowing scenario of the suspension is considered, for which little has been understood, except the general expectation that the swimming cells would tend to swim towards the pipe surface. The present analysis could provide some important physical insights into the mechanism and the flow structures in a pipe flow of swimming microalgae suspension, a popular flow setting in design of modern photobioreactors.

Predicting the effect of barnacle fouling on ship resistance and powering using CFD

Soonseok Song, Yigit Demirel and Mehmet Atlar

University of Strathclyde

To reduce the fuel consumption and green-house gas emissions of ships, understanding ship resistance is important. In this context, understanding the effect of surface roughness on the frictional resistance is of particular importance since the skin friction, which often takes a large portion in ship drag, increases with surface roughness.

In this study, a Computational Fluid Dynamics (CFD) model was developed to predict the effect of barnacle fouling on ship resistance and effective power of the full-scale KRISO container ship (KCS) hull. A roughness function model was employed in the wall-function of the CFD software to represent the different barnacle fouling conditions, and the simulations of flat plate representing KCS and 3D KCS hull were conducted at the design speed and a slow streaming speed to predict the roughness effect of different fouling conditions on the ship resistance and the effective power. The resulting changes in the frictional resistance, residuary resistance and effective power due to different fouling conditions were compared with each other. Finally the frictional resistance values obtained by CFD and the similarity law analysis were compared.

Numerical simulation of progressive flooding of a ship after damage

Athanasios Niotis, Dracos Vassalos and Evangelos Boulougouris

Maritime Safety Research Center - University of Strathclyde

The survivability of a ship, in case of flooding following damage, is a quest which occupies the maritime community for more than one century now. The flux of flood water in ship's complex internal compartmentation is influenced by various factors and has a substantial impact on ship motion. Scientists and engineers, trying to understand and to take practical decisions during the design process, have developed simplified models with questionable assumptions and simplifications of this complex hydrodynamic problem. One commonly used approach for the evaluation of the survivability performance of a vessel against damage is the hydraulic analysis of the water inflow and propagation. This research investigates the flooding of the ITTC benchmark model, a box shape barge with a simplified internal geometry. For the numerical simulations are used a commercial high fidelity computational fluid dynamics tool and a simplified hydraulic model. The aim of this work is the understanding of the flooding process and the ensuing flooding moments, as well as the qualification and quantification of the impact of the various physical assumptions that the existing models incorporate. The parameters under investigation are the size of the damage, air compressibility, flow characteristics and internal arrangement.

Tidal turbine hydrodynamics

Gabriel Thomas Scarlett[1], Brian Sellar[1], Ton van den Bremer[2] and Ignazio Maria Viola[1]

[1]School of Engineering, Institute for Energy Systems, University of Edinburgh
[2]Department of Engineering, University of Oxford

Tidal turbines operate in a very unsteady environment, which results in large-amplitude fluctuations in the thrust and torque acting on the rotor. Large load peaks and repeated load fluctuations can result in dynamic and fatigue failures. Hence, it is critical that unsteady loads are accurately predicted. We developed a low-order model of the unsteady hydrodynamics of the rotor, and we considered field measurements of the onset flow. The code is freely available on GitHub. We found that when a rotor operates in large, yet realistic wave conditions, the loads are governed by the waves rather than by the turbulence or by the shear of the mean current. The power and blade bending moments oscillate by more than half of their mean values. While the flow remains attached near the tip of the blade, dynamic stall and leading-edge vortex shedding occurs near the root. This, however, has a negligible effect on the total forces and moments due to the low arm and low relative flow velocity near the root. However, when the rotor operates below the optimal rotational speed, dynamic stall occurs over most of the blade and maximum peak loads can be twice as those predicted with a quasi-steady approximation.

Modal analysis of wake flow behind a wind turbine in a weakly stable stratified atmosphere

Nor Mazlin Zahari[1], Lian Gan[1] and Xuerui Mao[2]

[1]Durham University
[2]University of Nottingham

The power production in wind farms and the mechanical loads on wind turbines are strongly impacted by the wake of the upstream wind turbine. Thus, there is a need for understanding and modelling the turbine wake dynamic in the wind farm and the layout to optimize the output. Having a good wake model is important in predicting plant performance and understanding fatigue loads. In this paper, two modal analysis techniques which are Dynamic Mode Decomposition (DMD) and Proper Orthogonal Decomposition (POD) were employed to the simulation data generated by a Direct Numerical Simulation (DNS) of flow around a turbine in a weakly stable stratified atmosphere. The DMD and POD analyses were able to tell us the dynamics of the wake behind a wind turbine, e.g. the dominant mode of the helical flow structures and the corresponding frequency spectrum. As the result, the identification of the coherent structures and dynamics of the helical vortices behind wind turbine in stratified atmosphere will be presented with the discussion on how the deformation and breaking down of the helical wake structure were influenced by the stratified temperature field.

CFD modelling of mixing and dispersion from marine outfall discharges

Claudia Fernanda Castro Faccetti, P Andrew Sleigh, Duncan Borman and Amirul Khan

University of Leeds

Municipal and industrial wastewater is commonly discharged into coastal waters. Pollutants present in wastewater can affect the water quality of the receiving environment; therefore, accurate prediction of mixing and dispersion is important. In the near field, mixing is governed by the momentum and buoyancy of the discharge. Such buoyancy results from temperature and salinity differences between the effluent and the receiving waters. On the other hand, in the far field, dispersion is driven by the wider ocean hydrodynamics. Traditional modelling approaches predict dilution in the near field based on empirical or entrainment models, whereas the far field is simulated using coastal circulation models. CFD approaches have been scarcely used to model both regions, despite their great potential. This study presents the validation of near field simulations from single and multiple port discharges using RANS standard and realizable k-ε turbulence models. Water density was defined as a function of temperature and salinity using the UNESCO seawater state equation. To overcome the difficulties arising from the different length scales of the discharge ports and the receiving water body, mesh adaptation was implemented. Predictions of flow trajectory, velocity and dilution show good agreement with experimental data, providing higher accuracy than conventional modelling approaches.

On highly cambered thin circular arcs at low Reynolds numbers

Jean-Baptiste R. G. Souppez[1] and Ignazio Maria Viola[2]

[1]Warsash School of Maritime Science and Engineering, Solent University
[2]School of Engineering, Institute for Energy Systems, University of Edinburgh

Flow diagnostics and force measurements have been undertaken at low Reynolds number (circa 50k to 370k) on a highly cambered thin circular arc with a sharp leading edge in order to investigate the laminar to turbulent transition of modern asymmetric spinnakers. Evidence of a critical Reynolds number inducing transition has been provided, as well as the existence of a velocity dependent critical angle of attack triggering transition at Reynolds numbers lower than the critical. Moreover, the ideal angle of attack for the tested geometry has been assessed, challenging the assumed value in the literature. The results also lead to reconsider current knowledge and enable to explain discrepancies previously notice in the wind tunnel testing of yacht sails, thus allowing for a new interpretation of published research. In addition, it will allow to refine the setup of future experimental testing of model spinnakers to ensure the appropriate flow regime, representative of the full-scale sail, will be replicated.

Numerical analysis of a novel heat exchanger utilising additive layer manufacturing for aerospace applications

Evaldas Greiciunas
University of Leeds

Next generation aircraft require efficient thermal management systems which have challenging design constraints (e.g. size, geometry, thermal signatures). Heat Exchangers (HE) are key components of a thermal management system in determining the overall performance and feasibility of a given system. Aerospace HE are typically manufactured using a stacked-plate approach from a large number of repeating layered networks of flow channels, designed to efficiently extract heat from one flow to another whilst minimising pressure losses. A recent development in Additive Layer Manufacturing (ALM) of metals allows a much greater potential design freedom for HE and opportunities to create designs that overcome challenges of flow maldistribution. There are challenges in how to best exploit the technology for HE as modelling the detailed thermal behaviour within the HE channels is highly complex and computationally intensive. This research presents the Computational Fluid Dynamics (CFD) simulations of a novel HE geometry which that is based on a validated approach. The performance of the novel HE is also compared to the conventional designs.

Shape optimisation of a gas exchange valve for the production of carbon nanotubes

Filip Gokstorp and Matthew Juniper
University of Cambridge

In the production of carbon nanotubes (CNTs), the reactor gases used are a safety risk when released to the environment. It is therefore beneficial to separate the CNT products from the reactor gases with a gas valve. This is done by replacing the reactor gases with a safe gas. The optimal shape of the gas valve minimises the concentration of the reactor gases at the exhaust outlet of the gas valve, while leaving the flow of the CNT product undisrupted. A secondary objective of the gas valve is to recover concentrated reactor gases to allow them to be reused.

A gas valve formed of several flush flows separated by circular baffles was investigated. The flush flows are organised in outlet and inlet pairs. The outlet extracts gas from the valve, while the inlet injects nitrogen gas into the gas valve. This design space has been explored and designs that both fulfil both objectives have been found.

This presentation will focus on this optimisation, and the extension work completed. The optimisation method has been extended using adjoint methods for shape-gradient driven optimisation of the geometry, and the CNT model has been extended using flow-structure interaction techniques to model the CNT product.

The investigation of combustion characteristics of n-butanol/gasoline blends

Inna Gorbatenko

University of Leeds

Bio-derived liquid fuels are among the principal contenders to replace fossil fuels within the transport sector owing to their compatibility with conventional powertrain designs and existing fuel distribution technologies. Among those n-butanol is considered as a feasible alternative. In spark ignition (SI) engines, one of the important characteristics of fuel is its anti-knock quality. Knock is an abnormal combustion phenomenon which limits the efficiency of SI engines. Knock intensity is controlled by the development of the pressure wave set off by the initial auto-ignition and the intensity of the heat release rate, which could also manifest in super-knock. Since the knock in SI engines widely depends on the chemical kinetics of fuel oxidation, it is paramount to gain improved understanding of chemical kinetics and the effects of n-butanol blending with gasoline to enable operation and benefit optimisation of these technologies.

Therefore, this work attempts through the experimental and modelling framework to investigate the important fuel characteristics such as ignition delay times and excitation times for n-butanol blends with gasoline and representative surrogate mixture to gain a better understanding of anti-knock properties of these fuels.

Fluid deposition and spreading on topography

Pallav Kant, Andrew Hazel, Alice Thompson and Anne Juel

University of Manchester

Microdroplet deposition is a technology that spans applications from tissue engineering to microelectronics. Our high-speed imaging measurements reveal how sequential linear deposition of overlapping droplets on flat uniform substrates leads to striking non-uniform morphologies for moderate contact angles. We develop a simple physical model, which for the first time captures the post-impact dynamics drop-by-drop: surface-tension drives liquid redistribution, contact-angle hysteresis underlies initial non-uniformity, while viscous effects cause subsequent periodic variations. Motivated by applications to the manufacture of POLED displays, we turn to the spreading of a single droplet within a recessed stadium-shaped pixel. We find that the sloping side wall of the pixel can either locally enhance or hinder spreading depending on whether the topography gradient ahead of the contact line is positive or negative. Locally enhanced spreading occurs via the formation of thin pointed rivulets along the side walls of the pixel through a mechanism similar to capillary rise in sharp corners. We demonstrate that a thin-film model combined with an experimentally measured spreading law, which relates the speed of the contact line to the contact angle, provides excellent predictions of the evolving liquid morphologies of multiple-droplet deposits on topography.

A numerical scheme for controlling instabilities in Hele-Shaw flow

Liam Morrow

Queensland University of Technology

One of the well-studied problems in fluid dynamics involves the motion of two immiscible fluids in a Hele-Shaw cell, which is an experimental device made up of two plates separated by a narrow gap. The problem has received significant attention largely due to the interfacial patterns which form due to the Saffman-Taylor instability. In recent years, there has been increased interest in investigating how manipulating the geometry of the experimental apparatus can influence the development of these instabilities. This in turn has increased the need for robust numerical schemes to investigate the nonlinear dynamics of the problem. While numerical methods for solving the standard Hele-Shaw problem are well-established, often they cannot be extended to incorporate these changes in geometry. Here we present a scheme based on the level set method capable of modelling each of these variations to the standard Hele-Shaw model, and show that numerical solutions qualitatively compare well with experimental and analytical results.

Numerical simulations of annular flows in the presence of surfactants

Assen Batchvarov, Lyes Kahouadji, Aditya U Karnik, Lachlan Mason and Omar Matar

Imperial College London

Vertical annular counter-current air-water two-phase flows in the presence and absence of surfactants are investigated using numerical simulations. The computations are carried out using Blue, a hybrid front-tracking/level-set CFD solver. Preliminary validation against experimental data of velocity profile development and wave statistics against five experimental falling film scenarios shows good agreement. In the presence of a gas core velocity, the results show good qualitative agreement with experimental observations in terms of interfacial phenomena (i.e. three-dimensional, large-amplitude wave formation, the development of long ligaments, and droplet entrainment). The addition of surfactants to the flow gives rise to Marangoni stresses. At low gas and liquid flow rates, the Marangoni effect is found suppress wave formation. At higher gas/liquid flow rates, however, surfactant-induced effects promote droplet detachment and have a marked effect on the transition to ‘flooding’.

We would like to acknowledge funding from the EPSRC Programme Grant MEMPHIS through grant number EP/K003976/1. We also like to acknowledge the contribution of Drs Damir Juric and Jalel Chergui (both from LIMSI, CNRS, France), and Dr Seungwon Shin (Hongkik University, South Korea).

Viscous fingering in a radial elastic-walled Hele-Shaw cell

Draga Pihler-Puzovic[1], Gunnar Peng[2], John Lister[2], Matthias Heil[1] and Anne Juel[1]

[1]The University of Manchester
[2]University of Cambridge

We study the viscous fingering instability in a radial Hele-Shaw cell in which the top boundary has been replaced by a thin elastic sheet. The introduction of wall elasticity delays the onset of the fingering instability to much larger values of the injection flow rate. Furthermore, when the instability develops, the fingers that form on the expanding air-liquid interface are short and stubby, in contrast with the highly-branched patterns observed in rigid-walled cells (Pihler-Puzovi\'{c}, 2012).

We report the outcome of a comprehensive experimental study, which is compared to a theoretical model based on the lubrication equations coupled to the Foppl-von-Karman equations for the deformation of the elastic sheet.

We show that the theoretical predictions for the growth rate of small-amplitude perturbations to the time-evolving base flow are in agreement with experimental observations for injection flow rates that are slightly larger than the critical flow rate required for the onset of the instability. We also characterize the large-amplitude fingering patterns that develop at larger injection flow rates. We show that the wavenumber of these patterns is still well predicted by the linear stability analysis, and that the length of the fingers is set by the local geometry of the compliant cell.

The spreading and stability of a cooling drop on an inclined pre-wetted substrate

Ghanim Algwauish and Shailesh Naire

Keele University

Molten liquid flows that cool as they spread are important in a wide variety of contexts, e.g., lava domes in geophysical flows and coolant in nuclear reactors. The interplay between the flow and cooling can also give rise to a variety of intriguing flow features and fingering instabilities.

Motivated by the above, we consider theoretically a model system of a molten viscous drop extruding from a source and spreading over an inclined plane that is covered initially with a thin liquid film. Lubrication theory is employed to derive coupled nonlinear evolution equations for the film thickness and temperature. The coupling between flow and cooling is via constitutive relationship for the temperature-dependent viscosity. This model is parametrised by the heat transfer coefficients at the drop-air and drop-substrate interfaces, the Péclet number and the substrate inclination angle. A systematic exploration of the parameter space reveals a variety of solutions illustrating the dynamics of such flows.

We examine the stability of the one-dimensional solutions to small-amplitude variations in the thickness and temperature in the transverse direction. The existence of a fingering instability is revealed. A dispersion relationship is described using long-wavelength asymptotics and numerical simulations, which elucidates the mechanisms underlying this fingering instability.

Oscillatory fingering in an elasto-rigid Hele-Shaw channel

Callum Cuttle, Draga Pihler-Puzovic and Anne Juel

University of Manchester

We study the propagation of an air finger in a Hele-Shaw channel where the top boundary is an elastic membrane. The injection of air at a constant flow rate works against elastic, viscous and capillary forces to form an approximately steadily propagating blistering finger, which depends on the capillary number Ca – the ratio of viscous to surface tension forces – and the level of initial collapse of the channel cross-section. A stable Saffman-Taylor finger is formed when the membrane is initially flush with the side walls, and in the rigid channel this mode is linearly stable for all values of Ca. By contrast, increasing the initial deflation of the membrane leads to the broadening of the finger and the formation of small-scale fingering at its tip, which is advected around the finger as it advances. This form of propagation is oscillatory in the reference frame of the moving finger. We present evidence of symmetric and asymmetric unsteady solutions, which are associated with two distinct modes of oscillatory fingering. One resembles the tip-splitting instabilities observed to occur subcritically in rigid channels above a critical value of Ca and the other is linked to stubby finger formation in tapered channels.

Oblique liquid curtains

Eugene Benilov

University of Limerick

This paper examines liquid curtains emitted downwards, at an angle to the vertical. The analysis is based on the assumption that the Froude number, involving the curtain's thickness, is large. Naturally, the trajectories of the liquid particles are parabolic in this case – whereas the curtain's cross-sectional structure is determined by a parabolic equation reflecting a balance between viscosity, surface tension, and kinematic effects associated with the curvature of the particles' trajectories.

Transient dynamics of a bubble in a perturbed Hele-Shaw cell

Jack Keeler, Andrew Hazel, Alice Thompson and Anne Juel

University of Manchester

Taylor & Saffman (1958) demonstrated that for a finite air bubble in a Hele-Shaw cell there are an infinite number of steady propagation modes in the absence of surface tension. For modest surface tension, later work revealed that there is a single, stable, steadily propagating family of symmetric solutions, but a countably infinite number of disconnected unstable solutions. Recent experimental and theoretical work at MCND in geometrically-perturbed cells has exposed a wide range of phenomena, including symmetry breaking, tip-splitting and Hopf bifurcations when the flow rate is small. These phenomena all arise from changes to the stability and solution structure of the system caused by the geometric perturbation. Here, we present results for higher flow rates where there is increased sensitivity to noise in the system. A rich bifurcation structure is revealed, indicating the presence of bi-stable asymmetric steady states and periodic solutions that emanate from multiple Hopf bifurcations. The transient behaviour of these states will also be discussed where a non-trivial exploration of the solution space is observed.

Low-order models for the drag reduction of surfactant-contaminated superhydrophobic surfaces

Julien R. Landel[1], François J. Peaudecerf[2], Fernando Temprano-Coleto[3], Frédéric Gibou[3] and Paolo Luzzatto-Fegiz[3]

[1]University of Manchester
[2]ETH Zurich
[3]University of California Santa Barbara

In a recent study, Peaudecerf et al. (2017) PNAS, 114, 28:7254–59 showed that trace amounts of surfactant, below typical environmental concentrations, can severely limit the drag-reduction performance of superhydrophobic surfaces (SHS). As SHSs could have a large impact in reducing energy utilisation for many applications, it is important to understand how surfactant-Marangoni stresses affect the flow over these coatings. However, modeling surfactant-laden flow is challenging, as it is governed by seven coupled partial differential equations comprising nine dimensionless groups.

We present a semi-analytical model solving 2D Stokes flow over an SHS, coupled with surfactant transport in the bulk and at the interface using modified Frumkin kinetics. We consider a Poiseuille flow in a channel with a one-sided periodic array of gas-water interfaces. At steady state, we obtain a prediction for the Marangoni shear stress due to the concentration gradient of the surfactant adsorbed onto the plastron, and thereby calculate drag and slip length. We compare our model with highly resolved numerical simulations of the full transport problem. Good agreement is found for a large range of geometries, flow conditions and surfactant properties.

Coastal outflows into buoyant layers of arbitrary depth

Sean Jamshidi and Edward Johnson

University College London

The influx of river water into the oceans brings nutrients, sediments and pollutants, as well as driving coastal currents, and as such is an important area of study. Previous works have largely explored the role that density differences play in these systems. However, large scale currents can also be driven by jumps in potential vorticity, and there are far fewer studies that investigate the impact that this has on river outflows. The present work isolates the effect of potential vorticity, looking in particular at the semi-geostrophic (long-wave) equations. Although the system is fully nonlinear, it allows theoretical predictions to be made, which are then compared with numerical experiments.

Two mechanisms drive the flow: a nonlinear Kelvin wave, and flow driven by the jump in potential vorticity – the latter of which can drive fluid in either direction. The relative strengths of these two effects determine the qualitative behaviour of the outflow: flows can feature coastal currents, anticyclonic gyres and plumes that expand offshore, all of which have been observed in laboratory experiments and the oceans, and a theory is presented that divides up the parameter space according to these behaviours.

How to generate very long waves

Yong Sung Park

University of Dundee

While solitary waves have been widely used to simulate tsunamis, it is also well-known that the solitary waves may not reflect important properties of tsunamis observed in nature. In particular, solitary waves are orders-of-magnitude shorter when compared to field observation with the same wave height-to-water depth ratio. In this talk, we will demonstrate that a simple wave maker formed of a bottom flip can generate very long waves. Using analytic solution for the linear theory, numerical solutions of the nonlinear theory as well as the experimental data, the long waves are characterised in terms of some parameters related to wave generation. Then, a couple of applications will be presented: (i) wave run-up on a plane beach; and (ii) breaking of leading-depression waves. Finally, it will be discussed how this simple idea can be developed into a full 3-dimensional extreme wave generator that can be used for coastal inundation studies.

Drag on pairs of square section obstacles in free-surface flows

Gregory Lane-Serff[1] and Francis Robertson[2]

[1]University of Manchester
[2]University of Birmingham

The drag on pairs of square obstacles (side D) in open channel flow (width B) is measured in laboratory experiments to simulate conditions in natural river or flood flows past vegetation and buildings. The conditions are $Fr < 0.59$, $Re = 4800$ to 21900 and turbulent intensity $TI \sim 10\%$. Standard tandem and side-by-side arrangements are studied, followed by a full range of relative positions covering $5 \leq sx/D \leq 20$ (downstream) and $0 \leq sy/D \leq 7.0$ (cross-stream). The lowest drag coefficients (defined relative to the average flow speed) are observed on the downstream obstacle when shielded directly behind the upstream obstacle giving negative CD. The largest drag coefficients are observed for nearly side-by-side arrangements, with the peak values found to be for the slightly upstream obstacle of a pair. Much of the variation in CD on an individual obstacle can be accounted for by the change in the flow speed induced by the other obstacle, except within about 4D. The blockage ratio (D/B) is found to be an important factor. For $D/B = 12.7\%$ the largest drag coefficient is $CD = 3.82$ for blockage ratio $D/B = 12.7\%$ and 2.85 for $D/B = 6.3\%$.

Leidenfrost engine: dynamics of rotating disks on turbine-like surfaces

Prashant Agrawal[1], Gary Wells[1], Rodrigo Ledesma-Aguilar[1], Glen McHale[1], Khellil Sefiane[2], Anthony Walton[2], Adam Stokes[2] and Anthony Buchoux[2]

[1]Northumbria University
[2]University of Edinburgh

When a liquid droplet is placed on a surface that has a temperature significantly higher than the liquid's boiling point, the droplet levitates on a cushion of its own vapour. This phenomenon is called the Leidenfrost effect. The cushion of vapour provides a virtually frictionless surface on which the droplet can move. Directed motion of liquid droplets (and sublimating solids) can also be achieved by asymmetrically texturing the substrates and forcing a preferential vapour flow direction. Here we investigate the rotation of a liquid pool, supporting glass disks, on heated turbine-like asymmetrically textured substrates. The transparent disk allows us to visualise the liquid distribution over the turbine, which informs our assessment of the torque losses and rotation stability of the supported disk. We also demonstrate that by replenishing the liquid fuel, sustained rotation can be achieved. Experimental observations are supported by an analytical model. The dynamic analysis of the rotation of such Leidenfrost liquid (and solid) rotors paves way for developing mm and sub-mm scale heat engines. The concepts from this analysis can be extrapolated to alternative liquid and solid fuels to develop extreme environment engines for applications in deep mining and planetary exploration.

Self-propelled droplet motion on gradient slippery liquid infused porous surfaces

Gaby Launay[1], Gary Wells[1], Rodrigo Ledesma Aguilar[1], Halim Kusumaatmaja[2] and Glen McHale[1]

[1]Northumbria university
[2]Durham University

Slippery Liquid-Infused Porous Surfaces (SLIPS) are super-hydrophobic surfaces inspired by nepenthes pitcher plants. These surfaces allow very low contact angle hysteresis, and consequently, high droplet mobility. Based on this property, we have developed SLIPS capable of inducing and controlling the motion of droplets. This has many practical applications in microfluidics, including: (i) sorting droplets depending on their size and/or fluid type, (ii) accurately positioning droplets and (iii) merging and splitting droplets.

The low contact angle hysteresis exhibited by droplets on SLIPS implies that a small force is sufficient to induce its motion. We propose here to generate such force with a small imbalance in the underlying surface geometry. To do so, flat surfaces are patterned with macro-structures with an inherent roughness gradient. These surfaces are then made into SLIPS using a coating of nanoparticles imbibed with oil. When a droplet is deposited on the resulting surface, the force generated by the roughness imbalance, allied to the high droplet mobility, induces a motion in a specific, controlled direction.

We investigate here the drop motion and velocity experimentally for different oils, drop fluids, and roughness gradients. Results allow to gain a better understanding of the physical mechanisms behind the drop motion.

Manipulating droplets on slippery lubricant impregnated surfaces

Glen McHale[1], James H. Guan[2], Gary G. Wells[1], Rodrigo Ledesma-Aguilar[1], Bethnay Orme[1], Élfego Ruiz-Gutiérrez[1], Ciro Semprebon[1] and Ben B. Xu[1]

[1]Northumbria University
[2]University of Oxford

Lubricant impregnated surfaces, also known as slippery liquid-infused porous surfaces (SLIPS), remove the contact line pinning normally associated with sessile droplets. On such a super-slippery surface are completely mobile and this leads to the challenges of i) stabilising the position of a droplet and ii) directing motion. Here, we show that droplet position can be controlled through a SLIPS Cheerios effect induced by the overlap of the droplet-lubricant meniscus and the lubricant meniscus arising from structuring the underlying surface. We find that overlapping menisci can be used to direct and control motion of a droplet relative to surface features or adjacent droplets. These ideas are exemplified droplets on a super-slippery surface fabricated to have a shallow vertical walled V-shaped groove. Beyond the study of droplets on a single super-slippery surface we consider droplets confined between two super-slippery surfaces forming a wedge. In such a situation, we find droplets can be continuously translated in an energy invariant manner by reconfiguring the boundaries. Comparison between the experimental data and a hydrodynamic model shows the dissipation involved in this motion is dominated by the bulk contribution.

Receptivity of supersonic boundary layers over smooth and wavy surfaces to impinging slow acoustic waves

Carlos Gonzalez Hernandez

Imperial College London

In this talk, we address a new receptivity mechanism and revisit an old one, in a supersonic boundary layer due to slow acoustic waves: the impingment of two acoustic waves at certain angles on a smooth wall and the interaction between a single acoustic wave and a wavy wall. Both mechanisms are similar in that the interaction involved (sound-sound or sound-distributed roughness) generates a forcing in resonance with a nearly neutral Tollmien-Schlichting wave. The latter is thus excited near the lower branch of the neutral curve and subsequently undergoes exponential amplication. A triple-deck formalism is adopted to calculate the coupling coefficient. Supersonic boundary layer transition is of great technological interest, given the role of its prediction in the correct characterization of the flowfield around high-speed vehicles. The sound-sound receptivity mechanism detailed in this work is similar to that due to the interaction between acoustic and vortical disturbances studied in Wu 1999, but a difference is that both acoustic waves penetrate into the lower deck where they interact to generate a forcing able to excite a T-S wave. Matching with the upstream response is carried out and an expression for the coupling coeffcient is given.

On new insights into the three-dimensional boundary-layer instability over broad rotating cones

Zahir Hussain

Manchester Metropolitan University

In this study, a new solution is applied to the problem of boundary-layer flow over a rotating cone in still fluid. The mean flow is perturbed leading to disturbance equations that are solved via a spectral method involving Chebyshev polynomials, both of which are compared with previous numerical and analytical approaches. Importantly, we observe favourable comparison with existing experiments in the literature. Meanwhile, further comparisons will be drawn with experiments currently in the pipeline.

Physically, the problem represents airflow over rotating machinery components at the leading edge of a turbofan. Laminar-turbulent transition within the boundary layer can lead to significant increases in drag, resulting in negative implications for fuel efficiency, energy consumption and noise generation. Consequently, delaying transition is seen as beneficial, and controlling the primary instability may be one route to achieving stable flow characteristics.

Importantly, broad-angled rotating cones are susceptible to a crossflow instability, whereas slender rotating cones have transition characteristics governed by a centrifugal instability. We investigate both parameter regimes in this study and comment on the accuracy of the new solution compared with previous methods.

Numerical simulations of convection in mushy layers

James Parkinson[1], Andrew Wells[1], Daniel Martin[3] and Richard Katz[1]

[1]University of Oxford
[2]Lawrence Berkeley National Laboratory

Sea ice is a reactive porous material composed of ice crystals and an interstitial saline fluid (or 'brine'); a so-called mushy layer. Dense brine tends to sink through the ice, driving convection. Downwelling at the edge of convective cells leads to the development of narrow, entirely liquid channels, through which saline brine is efficiently rejected into the underlying ocean.

We consider numerical simulations of ice growth and convective brine transport. This natural convection during sea ice formation leads to patterns of varying porosity and fluid flow across multiple scales, with brine channels occupying just a few percent of the sea ice volume and evolving in time. Hence previous attempts to numerically simulate convective brine drainage have struggled to resolve the narrow brine channels at acceptable computational cost. To overcome this limitation, we use the Chombo software framework to implement a computational mesh which adapts to provide additional resolution near brine channels.

Using these techniques, we calculate the ice-ocean salt flux during steady-state growth and find a sublinear dependence of the flux on the porous medium Rayleigh number. Transient simulations reveal complex behavior; neighboring brine channels interact with each other, whilst their spatial density tends to decrease over time.

A slurry model of the F-layer at the base of Earth's outer core

Jenny Wong, Chris Davies and Chris Jones

University of Leeds

Seismic observations suggest that a stably-stratified layer, known as the F-layer, 150–300 km thick exists at the base of Earth's liquid outer core. This contrasts with the density inferred from the Preliminary Reference Earth Model, which assumes a well-mixed and adiabatic outer core. Current models do not provide a sufficient dynamical description of how light element is transported across the stable F–layer into the overlying outer core. We propose the F-layer is explained by a slurry on the liquidus, whereby solid particles of iron crystallise from the liquid alloy throughout. We make two key assumptions: 1. fast-melting, where the timescale of freezing is considered short compared to other processes, 2. a binary alloy, where the light element is oxygen that is expelled into the liquid during freezing. A steady state one-dimensional box model of a slurry is formulated in a frame moving at the speed of inner core growth. We ascertain temperature, light element concentration and solid flux profiles by varying the layer thickness, inner core heat flux and thermal conductivity, since these estimates are uncertain. We demonstrate the steady state slurry satisfies geophysical constraints on the density jump across the layer and the core-mantle boundary heat flux.

Stability of scrape-off layer plasma: a modified Rayleigh-Benard problem

Fryderyk Wilczynski, David Hughes and Sven Van Loo

University of Leeds

In magnetic confinement devices, the boundary turbulence is characterised by intermittent ejection of coherent filamentary structures. These filaments transport plasma from the well-confined core region, through the Scrape-Off Layer (SOL), towards the material surfaces. This results in increased plasma-wall interaction, which has the potential to damage plasma-facing components and shorten the lifetime of the device. It is therefore essential to develop full understanding of the mechanisms behind the transport in the edge of the plasma.

We consider an established model of two-dimensional SOL plasma, for an insight into the instabilities present at the edge of magnetic confinement devices. The model equations are based on the Braginskii fluid equations under the assumption of drift ordering and electrostatic plasma. The model also employs the common slab geometry approximation, whereby the magnetic field is assumed constant and straight and the effects of curvature are reintroduced as effective gravitational terms.

In this limit, the SOL plasma equations can be reduced to a thermal convection problem with additional effects. These new effects include a non-uniform basic state gradient, linear damping terms, and additional advective terms. We analyse the linear stability of the plasma problem and relate the results to the established theory of Rayleigh-Benard convection.

Pulse propagation in quasi-laminar gravity currents

Paul Allen[1], Robert Dorrell[2], Oliver Harlen[1], Robert Thomas[2] and William McCaffrey[1]

[1]University of Leeds
[2]University of Hull

Gravity currents are flows that are driven by a density difference and exist in many significant geophysical flows. For example, the downdraughts of thunderstorms, turbidity currents, pyroclastic flows, and avalanches. Pulses are an inherent feature of gravity currents and may result from flow instabilities, variable supply of dense material or topography. We explore pulse evolution through theoretical shallow-water modelling validated against laminar, lock-exchange generated gravity currents comprising of diluted glycerol.

Numerical simulations of the lock-release problem were undertaken with a Lax-Wendroff finite-difference scheme based on the depth-averaged, shallow-water equations. Fluid at constant depth is release at $t=0$ from the first lock box. At a specified time $t=t^*$, a second lock box is released creating a pulse that travels through to the front of the current. The shallow water assumption breaks down at the head where significant vertical gradients and dissipation are present. The strength of dissipation is captured by fixing the Froude number Fr at the head. Investigation of the (Fr, t^*) parameter space revealed that qualitatively different solutions exist within it. These were distinguished by whether or not the finite length of the lock-box affects pulse propagation and the varying structure of the fluid released from the first lock-box.

Predictability of tropical cyclones: a comparison of typhoon Haiyan (2013) and typhoon Hagupit (2014) using convection-permitting ensemble forecasts

John Ashcroft[1], Juliane Schwendike[1], Andrew Ross[1], Stephen Griffiths[1] and Christopher Short[2]

[1]University of Leeds
[2]Met Office

Although forecasting the intensity of tropical cyclones (TCs) remains challenging, track forecasts have improved considerably over the past few decades. However, there remain cases where numerical weather prediction models are unable to accurately forecast the motion of a TC. Typhoon Haiyan (2013) and Typhoon Hagupit (2014) are examples of two storms where, despite similarities in the track and intensity, the predictability of the storms differed greatly. Both storms made landfall over the Philippines having followed a similar track across the Pacific and both reached intensities in excess of 60 m/s. Operational ensemble forecasts showed large uncertainty in the track of Hagupit, whereas the ensemble spread for Haiyan was considerably less.

Using the Met Office's Unified Model, cutting-edge 4.4-km, limited-area, convection-permitting, 5-day ensemble forecasts were produced for both storms along with 5-day global ensemble forecasts. Each ensemble includes 12 members and forecasts are initialised 12 hours apart over a 6 day period. Consistent with the operational forecasts produced at the time of the storms, the spread of tracks is greater in the forecasts produced for Hagupit than Haiyan. Analysis of the simulations shows the important role Hagupit's environment plays in the predictability of its motion.

Turbulence in the body of pseudo-steady gravity currents

Caroline Marshall[1], Gareth Keevil[1], Robert Dorrell[2], Steven Tobias[1] and Jeff Peakall[1]

[1]University of Leeds
[2]University of Hull

Despite the prevalence of gravity currents in both nature and industry, and resulting extensive research into their structure and mechanics, the turbulence structure within the body of gravity currents remains poorly understood. Using a combination of state-of-the-art experiments and direct numerical simulation (DNS), we here address this knowledge gap.

Using the high-order spectral element solver NEK5000, three-dimensional DNS was carried out for solute-driven constant-flux gravity currents. This was done to establish the effect of varying Reynolds and Schmidt numbers on the turbulent structures within the body of the current, and thereby identify their impact on flow dynamics. These simulations were compared to planar and volumetric particle image velocimetry (PIV) measurements of velocity fields within similar laboratory-based flows.

This research is advancing our understanding of mixing processes in gravity currents. The combination of PIV and DNS constitutes the first high-resolution measurements of turbulence structure within the body of pseudo-steady gravity currents. In particular, flow scale turbulent structures within the body are quantified for the first time and their effect on flow dynamics established.

Rate-independent and viscoplasticity models for granular materials

David Harris

University of Manchester

We present a class of well-posed plasticity models incorporating both rate-independent and rate-dependent behaviour. The models can be used in the context of both classical and Cosserat continua. A classical continuum is sufficient to describe some aspects of the flow of granular materials. The physical properties that may thus be described by the models include pressure-dependent yield and dilatancy. Coaxial/non-coaxial behaviour between the Cauchy stress and deformation-rate is permitted. The classical continuum is inadequate to describe the flow in the interior of shear-bands and in the immediate vicinity of external boundaries. Such effects may be included by using a Cosserat continuum which incorporates the mean grain spin in a representative volume element thereby enabling flow in such regions to be considered. The kinematic quantities associated with grain spin are (i) the difference between the anti-symmetric part of the velocity gradient tensor and the anti-symmetric dual tensor to the intrinsic spin vector and (ii) the spatial gradient of the intrinsic spin. These kinematic quantities are associated with the anti-symmetric part of a non-symmetric Cauchy stress and a couple stress tensor.

In this talk we describe the models, discuss some of their more important properties and consider a one dimensional application.

Using 3D-printed analogues to understand the sedimentation of complex ice particle

Mark McCorquodale and Chris Westbrook

University of Reading

The terminal velocity of ice crystals in the atmosphere is a function of their shape, size and mass. Various parameterisations have been proposed to approximate this function. However, the accuracy with which these parameterisations are able to predict the terminal velocities of natural ice particles, where there is enormous variability in both shape and Reynolds number, remains unclear.

In addition to the speeds of the falling particles, preferred orientations and onset of unsteady motions are important characteristics of the sedimentation process, and are crucial to the interpretation of meteorological data acquired through remote-sensing, as well as our understanding of optical effects like halos and arcs.

Some of the challenges faced during field studies of natural snowflakes can be avoided in experimental studies of “analogues” falling through liquids. Here, we report on new experimental work that revives this idea by using 3D-printed snowflake analogues to investigate the aerodynamics of relatively complex ice particle shapes. The snowflake analogues are allowed to sediment through a 1.8 metre tall column of liquid during which their orientation and projected area are imaged, and settling velocity measured. Subsequently, drag coefficients and Reynolds numbers can be derived, allowing fall speed parameterisations to be tested against accurate data.

Buoyancy effects on cross-ventilation

Megan Davies Wykes, Jean-Etienne Debay, Nouhaila Fahdi, Elkhansaa Chahour and Paul Linden

University of Leeds

Cross-ventilation occurs when there are openings on opposite walls of a room, at the same height. This talk will present the results of laboratory experiments using a water flume and a cross-ventilated model room at 1/10th scale. These experiments examine the effect of an initial indoor-outdoor temperature difference on the ventilation rate and the removal of contaminants from a cross-ventilated room. The results from experiments are also compared to numerical simulations, using the large eddy simulation code Fluidity.

Photogrammetric deformation measurement of adaptive shock control bumps

Michela Gramola, Paul Bruce and Matthew Santer

Imperial College London

Previous research on 2D adaptive shock control bumps at Imperial College has looked at their shock stabilisation and wave drag reduction potential. In all these studies, Schlieren Photography has been used to estimate the spanwise-averaged instantaneous bump shape. The presence of 3D effects and discrepancies between experimental and computational results have led to the need for a more accurate technique for measuring deformations of the flexible bump during experiments.

Photogrammetry has been applied to FSI experiments of adaptive shock control bumps in the Imperial College supersonic wind tunnel at Mach 1.4. The bumps are modelled as flexible aluminium alloy plates, deformed through actuation normal to the surface. Videos of the flexible plate, equipped with 80 coded targets, are recorded at 100 Hz by four Phantom high speed cameras, which allows accurate recovery of the deformed shapes.

Preliminary experiments with a solid plate used to calibrate and characterise the photogrammetry technique will be presented. In addition, photogrammetry results for passive and active 2D adaptive bumps will be shown and compared with Schlieren videos and static pressure measurements along the centreline of the flexible plate to explain the performance improvement associated with the bumps.

Unsteady temperature and composition measurements with LITGS

Francesca De Domenico and Simone Hochgreb

University of Cambridge

High and low frequency laser-induced thermal grating spectroscopy (LITGS) measurements of temperature and composition are performed in non-reacting flows. Biacetyl in trace amounts is used as an absorber, and the thermal grating is generated at 10 and 1000 Hz using two different pulsed pump lasers tuned to 355 nm. We demonstrate that LITGS can be used as a technique to detect instantaneous, unsteady temperature and density variations in a flow, requiring only limited optical access. However, we highlight that there is a the trade off between spatial resolution and thermalisation of the probe volume induced by the energy delivered from the pump laser. For a fixed pulse energy and crossing angle, good spatial resolution can be achieved by using small beam diameters at the crossing point. However, as the energy per unit area increases, the thermalisation of the probe volume induced by the laser pulse might lead to non-negligible local temperature increase. For larger beam diameters at the crossing point, the thermalisation effect becomes negligible, but the probe volume length increases substantially, decreasing the spatial resolution.

Two-phase plumes in a rotating environment

Daria Frank[1], Julien R. Landel[2], Stuart Dalziel[1] and Paul Linden[1]

[1]University of Cambridge
[2]University of Manchester

Motivated by the effects of the Earth's rotation on the dynamics of the oil plume after the Deepwater Horizon accident, we study the behaviour of two-phase plumes in a rotating system. We conducted small-scale laboratory experiments on bubble and mixed bubble-freshwater plumes which were released into a rotating saltwater environment. In particular, we focus on how the rotation modifies the plume characteristics such as its rise velocity and the temporal evolution of its width. Similar to single-phase plumes, two-phase plumes also exhibit anticyclonic precession of the plume axis in the near-source region. Another important question to address is how the profile of the plume just below the free-water surface differs from the non-rotating case - this is crucial for the lateral spreading of the plume on the water surface at later times. We also briefly discuss the formation of tornado-like plumes which were observed under specific experimental conditions.

The impact of aspect ratio and swept angle on the propulsive performance of bio-inspired flapping foils in a tandem configuration

Nikolaos Lagopoulos[1], Luke Muscutt[2] and Bharathram Ganapathisubramani[1]

[1]University of Southampton
[2]Imperial College London

Tandem flapping wings have been found in nature, such as dragonflies and extinct marine reptiles such as plesiosaurs. This type of configuration can be used in practical applications such as underwater vehicles where performance and manoeuvrability are of critical importance. Previous work has shown that this type of configuration can lead to higher thrust and better efficiency. However, these studies were limited to either 2D simulations or high aspect ratio wings. This study aims to experimentally investigate the impact of aspect ratio on the performance of tandem flapping wings.Load cell measurements and flow visualisation will be conducted for a combination of identical and mixed AR's of bio-inspired flippers in the recirculating flume tank at the University of Southampton. The inter foil space will be kept constant (3 chord lengths, as this was previously found to be suitable) and the experiments will be conducted for one flapping frequency (Strouhal number of 0.4), which falls within the range that is observed in nature St ~ [0.1, 0.5]. Our ultimate goal is to verify an optimal ratio of hind-to-fore flipper AR suitable for underwater unmanned vehicles.

Aerodynamics of flexible filaments hanging in cross flow

Jorge Silva-Leon, Andrea Cioncolini and Antonio Filippone

University of Manchester

The study of the dynamics of highly flexible structures interacting with fluid flow is relevant in connection to energy harvesting technologies and drag reduction applications. In particular, cantilever filaments hanging in crossflow is a benchmark configuration, useful for validating fluid-structure interaction simulation codes, but which has not been studied in detail. Under a steady wind load filaments remain in static equilibrium [1]. However, at certain wind speeds the filaments exhibit low-amplitude vibrations, and, depending on their length, may incur in two-dimensional large amplitude flutter motions. At higher wind speeds the motion becomes three-dimensional and complicated.

We employed hot-wire anemometry for studying the flow surrounding the filaments in order to establish the mechanism responsible for the onset of motion. The results obtained suggest that a vortex-induced vibration is not possible due to the difference of time scales between vortices and structural motion. However, the upstream turbulence length scale of the flow is comparable in size to the filament size and may thus be responsible for the excitation observed. Furthermore, the three-dimensional motion of the filaments was studied via nonlinear time-series analysis techniques. Moreover, it was established that the flapping dynamics is regulated by the Scruton number, related to the structural mass-damping.

Stochastic modelling and feedback control for efficient bluff body drag reduction

Rowan Brackston, Andrew Wynn and Jonathon Morrison

Imperial College LONDON

The wakes of bluff bodies, such as automotive vehicles, exhibit complex behaviour due to three-dimensionality and high Reynolds numbers, and are furthermore responsible for significant aerodynamic drag. There are significant environmental and economic incentives for reducing drag, however practicalities limit the extent to which this can be achieved through geometric changes, motivating the use of active feedback control methods to modify the flow. In this work we develop control strategies for the wake of the widely used Ahmed body, aiming to regulate specific flow features. Two such features understood to be related to the drag are the static symmetry breaking (SB) mode and the quasi-oscillatory vortex shedding, both of which may be strongly influenced via dynamic forcing flaps. In order to guide feedback controller design, we develop stochastic models for each of the flow features, describing their dynamics and response to forcing. Controllers are then implemented, achieving an efficient drag reduction of 2% when suppressing the asymmetry of the SB mode. Vortex shedding control proved ineffective at drag reduction, despite the suppression of measured fluctuations around the frequency at which coherent oscillations are observed. This is suggested to be due to unavoidable amplification of fluctuations at other frequencies.

Drop impact phenomena with complex fluids on heated and soft surfaces

Simeng Chen and Volfgango Bertola

University of Liverpool

Drop impact dynamics on solid surfaces is a classical subject of interfacial hydrodynamics. So far, most of the studies are restricted within Newtonian fluids and isothermal solid surfaces. There is very little research contribution which characterises the drop impact behaviours of non-Newtonian fluids on heated and soft surfaces. However these situations are relevant to various applications, such as spray cooling, 3D ink-jet printing, and manufacture processes of food, pharmaceutical and cosmetic industries. The talk summarises my four year experimental research in University of Liverpool, which aims at studying the impact of non-Newtonian droplets (i.e., droplets of complex fluids such as polymer solutions) on heated surfaces (i.e., surfaces with a temperature above the Leidenfrost point) and soft surfaces (i.e., surfaces that undergo temporary or permanent deformations upon drop impact) through high-speed imaging. The first part of the talk focuses on the Leidenfrost drop impact of different model fluids with matching flow curves. In the second part, the Leidenfrost drop impact experiment is extended to viscoplastic fluids. A systematic investigation on the impact of viscoplastic drops onto viscoplastic substrates is introduced in the third part. Lastly, the experimental study of drop impact on spherical elastic surfaces is addressed.

UK Fluids Conference: Poster presentations

Modelling bubble propagation in elasto-rigid Hele-Shaw channels

João Fontana, Anne Juel and Andrew Hazel

University of Manchester

We study a model of pulmonary airway reopening where air is driven at constant volume flux into a liquid-filled Hele-Shaw channel, with an upper compliant boundary. An equivalent rigid channel supports a stable, steadily propagating air finger and a variety of unstable solutions. In the compliant channel, however, initial collapse of the channel introduces additional cross-sectional depth gradients. The induced normal and transverse depth variations alter the finger morphology and promote a variety of instabilities from tip-splitting to small-scale fingering on the curved front. A depth-averaged model for the system is simulated numerically using the open-source finite-element library oomph-lib in order to explore the underlying mechanisms and the relative importance of the elastic, capillary and viscous effects. The model exhibits a complex solution structure and qualitatively similar instabilities to those observed experimentally. The complexity is a result of the large number of unstable solutions present in the rigid Hele-Shaw cells that can interact under very modest perturbations of the system. We focus on the stability of the steady modes of bubble propagation in this system and compare the solution structure with that of a rigid Hele-Shaw channel.

Erosion-deposition waves in granular flows

Callum Tregaskis

University of Manchester

Flows of granular material may exhibit both solid-like and liquid-like behaviour simultaneously, for example, in large amplitude surge waves that are a destructive feature of debris flows or in snow avalanches where the erosion of snow at the front is fundamental to the dynamics of the avalanche. The depth-averaged $\mu(I)$-rheology (Gray & Edwards, 2014) is a recent advancement in the continuum modelling of granular flow that alongside the extended friction law of Pouliquen and Forterre (2002) captures both the erosion and deposition of grains. The viscous dissipation combines with dynamic, intermediate and static friction regimes to generate finite-length waves with static and mobile regions (Edwards & Gray, 2015). Using a combination of mathematical analysis, numerical solutions and small-scale experiments, we can construct a detailed description of the phase-transition from static to moving grains. It is possible to develop an avalanche model that qualitatively explains many features of the avalanche with good agreement with experiments (Edwards, Viroulet, Kokelaar & Gray, 2017) and is therefore likely to have important practical implications in hazard assessment and risk mitigation.

Inertial and buoyancy effects on the horizontal flow of Taylor bubbles in a circular channel

Hannah Moran, Mirco Magnini and Omar Matar

Imperial College London

The effect of gravity on liquid film thickness around an elongated bubble travelling along a horizontal liquid-filled tube has been studied through numerical simulations. At small Reynolds (Re) and Bond (Bo) numbers buoyancy and inertial effects are negligible and the liquid film thickness at the tube wall is a function of only the capillary number (Ca). As the tube diameter reaches millimetre length scales the buoyancy forces become non-negligible. Two-dimensional simulations for Bond and capillary numbers in the ranges $0.05<Bo<0.42$ and $5x10^{-4}<Ca<0.12$ have been performed. These simulations capture the asymmetry of the liquid film thickness, where gravitational effects cause the liquid film to be thinner at the top of the tube than the bottom. The 2D simulations do not capture bubble inclination relative to the channel centreline, but this is in agreement with experimental studies on the effect of gravity on a Saffman-Taylor meniscus. Three-dimensional simulations can capture transverse flow, and thus film drainage from the top to the bottom of the tube and its effect on bubble inclination. 3D numerical simulations are systematically performed to span a wide range of capillary, Reynolds and Bond numbers and thus investigate how inertial forces impact the flow.

Simulating moving objects on a Cartesian mesh

Athanasios Emmanouil Giannenas and Laizet Sylvain

Imperial College London

Numerically representing the unsteady flow generated by the motion of a moving object is challenging, and various approaches are possible: moving immersed boundary method (MIBM), adaptive remeshing or overlapping meshes. When MIBMs are used, the solution can be contaminated by spurious force oscillations (SFOs). The two main sources of the SFOs are the pressure and velocity discontinuities when mesh nodes switch from a fluid region to a solid region and vice versa. It has been shown in the literature that an artificial flow inside the object can suppress SFOs. In this study, a new 1-dimensional (1D) cubic spline reconstruction strategy is proposed with interpolation of the velocity field inside the object to ensure a smooth solution while satisfying the no-slip condition at the wall of the object. The high-order incompressible flow solver Incompact3d will be used for this study. It is based on sixth-order finite-difference schemes on a Cartesian mesh and a spectral Poisson solver to ensure incompressibility. The proposed reconstruction is compatible with the 2D domain decomposition implemented in Incompact3d and thus is suitable for parallelisation. Benchmarks will be performed for various flows: a moving cylinder and a channel with an oscillating wall.

Fluidisation of a yield-stress material under vibration

Ashish Garg, Anne Juel and Matthias Heil

University of Manchester

The rheology of yield-stress fluids has been the subject of extensive research in recent years. However, the mechanics of fluidisation of such materials due to external forcing remains poorly understood. Using a combination of experiments and theory, we investigate the fluidisation of a sessile drop of microgel (carbopol) on a pre-existing layer of the same fluid due to vertical sinusoidal oscillations. The experiments show that the drop deforms harmonically with the driving frequency for low values of the driving acceleration. Above a critical value of the acceleration, transient axisymmetric spreading occurs until the drop reaches a new stable configuration where it deforms harmonically with the driving frequency. Informed by rheological measurements, we model carbopol as a yield-stress viscoelastic using an Oldroyd model with a Herschel-Bulkley constitutive relation for the viscous stresses. We derive a depth-averaged model for the dynamics of the drop in the limit of a slender axisymmetric drop and compute solutions using finite-difference methods. We find that the harmonic deformations of the drop are associated with elastic deformations and that the yield-stress controls the critical acceleration at which spreading is initiated. The extent of the spreading for given acceleration results from the interaction between elastic and viscous forces.

Modelling how corals apply the Goldilocks Principle to engineer habitat

Konstantinos Georgoulas

University of Edinburgh

The ecosystem services provided by coral reefs are worth over $100 billion annually and include coast line protection, tourism, food and medical derivatives. However, coral reefs (tropical and cold) are at threat from climate change. The proliferation of corals is reliant upon optimal current and light conditions, and we propose that the growth of reefs are determined by the ‘Goldilocks Zone’, where conditions are ‘just right’. By creating a model to understand coral growth under optimal conditions, it will be possible to simulate a variety of different future environments. Such a model would be a powerful tool for coral reef management.

The Smoothed Particle Hydrodynamics (SPH) method will be used for the model as its mesh-free Lagrangian nature is ideal for simulations where the examined object (i.e. a coral) is growing dynamically. The model will be written in C++ programming language and will be parallelized with the Open Multi-Processing (OpenMP) application programming interface to allow for time-effective high resolution simulations.

An emerging thermo-electric power generator driven by trampolining elastic gels on hot surface

Sreepathy Sridhar[1], Steven Wang[2], Yifan Li[1], Glen McHale[1] and Ben Xu[1]

[1]Northumbria University
[2]Newcastle University

n this research, we demonstrate an emerging thermo-electric generating technology based on Leidenfrost effect induced trampolining of elastic gel on hot surface1, which was occurred under a coupling effect from the elasticity of gel and Leidenfrost effect2. The thermal energy was transferred into electric energy through a Leidenfrost activated mechanical motion, supported by a self-constructed piezoelectric-hydrogel system. After carefully optimizing the system by selecting gel spheres with different solvent concentrations, sizes (4 mm – 25 mm), elasticity (0.5 kPa – 500 kPa), and studying other factors such as mass loss, impact dynamics, etc, we achieved a conversion unit to produce an optimum 3V and expected to get higher output (> 30V) by integrating a number of same units. This finding opens a window to create an energy-harvesting device that can generate electrical power from thermal sources and offer a potential solution for those regions with rich geo-thermal resources to acquire electric energy, such as Iceland or volcanic islands.

Can large oceanic eddies be stable?

Eugene Benilov

University of Limerick

Observations show that radii of oceanic eddies often exceed the Rossby radius of deformation, whereas theoretical studies suggest that such vortices should be unstable. The present paper resolves this paradox by presenting a wide class of large geostrophic vortices with a sign-definite gradient of potential vorticity (which makes them stable), in an ocean where the density gradient is mostly confined to a thin near-surface layer (which is indeed the case in the real ocean).

The condition of a thin ‘active’ layer is what makes the present work different from the previous theoretical studies, and is of utmost importance. It turns out that, without it, the joint requirement that a vortex be large and have a sign-definite potential vorticity gradient trivializes the problem by eliminating all vortices except nearly barotropic ones.

High performance racing yachts: an experimental comparison of the latest hydrofoil configurations

Jean-Baptiste R. G. Souppez

Solent University

Despite the omnipresence of hydrofoil-assisted monohulls and the inherent development phases to refine their design, very little scientific data has reached the public domain. Moreover, following the trend set by the racing yachts, the superyacht industry is now looking at the implementation of foils onto cruising superyachts, with several vessels already built. This poster presents a hydrodynamic comparison of three contemporary options, namely a Dynamic Stability System, a Dali Moustache and a Chistera foil, that have been towing tank tested on a 1:10 scale model of a 50ft hull. The hydrodynamic comparisons presented focus on the resistance, side force, heave and trim, as well as the induced drag factor and effective draft of each design, eventually allowing designers to decide on the most suitable configuration and providing the first experimental data relative to hydrofoils-assisted racing monohulls.

Towards green manufacturing of biologically inspired silica: effect of mixing and other operating parameters

Mauro Chiacchia and Siddharth V. Patwardhan

University of Sheffield

In the past decade, bioinspired silicification has shown to be a potential scalable method to manufacturing silica with highly desirable properties such as mesoporosity. This method has been extensively hailed as a feasible, more sustainable alternative to traditional synthesis of mesoporous silica; the latter not achievable with current large silica manufacturing methods. However, while very well characterised at laboratory small-scale, bioinspired silica has not achieved a translation to large-scale manufacturing. This is due to the lack of knowledge of the effect of scales on the bioinspired process chemistry, in particular mixing, which lead to the inability to match the commercially desired quantity and quality. In this work we experimentally studied the effect of mixing on the bioinspired silica process and the products. Results helped us to gain fundamental insight on how mixing affected the bioinspired silica reaction process and final silica product. Moreover, these results suggest that mixing parameters could be used to control some of the final product characteristics and must be taken into account when this process is transferred from the laboratory to the industrial scale.

Effect of oil-injection on twin screw compressor performance

Nausheen Basha
City University of London

Oil injected screw compressors are popular with oil and gas applications. Oil improves the compressor performance but excess oil leads to power losses as well as it would require a bigger downstream separator and heat exchanger. Therefore, it is necessary to find a balance between the least oil content and improved compressor efficiency. Comparison between the oil distribution patterns resulting from oil injection with varying nozzle diameters of 2 mm to 7 mm is studied in 98 mm air compressor. Also, the oil distribution patterns are correlated to the overall compressor performance of flow rate, indicated power, power loss due to oil, volumetric and adiabatic efficiency. Mesh required for the analysis is generated from SCORG for changing rotor positions and User Defined Functions (UDF) are written to update the mesh with varying time step in accordance with the SCORG files. Finally, a favourable parameter of oil mass flow and Mean Sauter Diameter (MSD) is established that would give a better performance and reduced losses as well as further improvements in the oil nozzle design are suggested.

Converging high-speed Schlieren for shock wave boundary layer interaction study

Zhengzhong Sun
City, University of London

Shock wave boundary layer interaction (SWBLI) is the cause of several aerodynamic problems including flow separation and flow unsteadiness. Plenty research efforts have been directed to the topic of SWBLI so as to understand this phenomenon. The present SWBLI is established at a compression ramp model with ramp angle ranging from 20 to 30 degrees at freestream Mach number of 2.0. The Reynolds number effect is also studied by using the different length of the flat plate upstream of the ramp. A new converging high-speed schlieren (CHSS) is proposed here to investigate SWBLI. The new CHSS technique is found being able to reveal the flow unsteadiness in a semi-quantitative way. Moreover, it can visualise the flow structures that are not immediately observable in the raw schlieren image, for example, the separation region and the location of the peak turbulent region. The CHSS is finally concluded as a powerful technique for SWBLI investigation with the advantages of easy-to-setup and high spatial resolution.

Dynamical scaling of spherical shell rotating convection

Robert Long, Jon Mound, Chris Davies and Steve Tobias

University of Leeds

Convection-driven flows under the influence of rotation exist in many natural systems, including Earth's liquid iron core. Flow in the core is turbulent and dominated by rotation; even state-of-the-art numerical models cannot run at the relevant control parameters. To connect numerical models and the Earth's core requires scaling laws for heat transfer and flow properties such as the characteristic velocity and length scale.

We undertake a systematic investigation of non-magnetic convection in a rotating spherical shell. The dynamics are determined by the Ekman number (E), a measure of the strength of the Coriolis force, the Rayleigh number (Ra), measuring the vigour of the thermal driving, and the Prandtl number (Pr), the ratio of the viscous and thermal diffusivities. We consider a suite of ~100 numerical models with Pr=1, E=1e-6 – 1e−3, and Ra up to 500 times the critical value for the onset of convection. Different dynamical regimes of rotating convection are identified through observed changes in the heat transfer and flow properties. We reach a region of parameter space at low E which is not viscously dominated; scaling laws based on simulations in this 'rapidly-rotating' regime are most suited for extrapolation to core-like dynamics.

Exact instantaneous solutions for inviscid dynamos

Colin Hardy, Phil Livermore and Jitse Niesen

University of Leeds

The geodynamo is driven by the motion of fluid within the Earth's outer core. In this motion, which is governed by the MHD equations, rotational forces are dominant over inertial and viscous forces, which means the Ekman and Rossby numbers are very small. Numerical models are used to simulate the geodynamo and have had some success, but are only possible in parameter regimes vastly different to that of the Earth, of Ekman and Rossby numbers being far too large. There is an alternative approach proposed by Taylor in 1963, of an inertia-free and viscous-free model as the asymptotic limit of Earth's dynamo. In this theoretical limit of a magnetostrophic balance, a necessary condition known as Taylor's constraint must hold.

The geostrophic flow is the component of the flow that evolves in such a manner as to ensure this constraint on the magnetic field is continuously satisfied through time.

We propose a method for determining the instantaneous geostrophic flow in a fully 3D geometry; amending Taylor's original analysis to consider a generalised magnetic state. We present a variety of examples implementing this method.

This work now facilitates the development of a time-stepping method to find 3D solutions of the magnetostrophic equations.

Parallel in time integration of the kinematic dynamo problem

Andrew Clarke, Daniel Ruprecht, Steve Tobias and Chris Davies

University of Leeds

The precise mechanisms responsible for the natural dynamos in the Earth and Sun are still not fully understood. Numerical simulations of natural dynamos are extremely computationally intensive, and are carried out in parameter spaces many orders of magnitude away from real conditions.

Parallelization in space is a common strategy to speed up simulations on high performance computers, but eventually hits a scaling limit. Additional directions of parallelization are desirable to utilise the high number of processor cores now available. Parallel-in-time methods can deliver speed-up in addition to that offered by spatial partitioning.

We will investigate Parareal's ability to speed up simulations of the kinematic dynamo, where the velocity field is a prescribed field in the induction equation.

We will describe an implementation of Parareal into the open-source Python software Dedalus, which implements spectral methods with implicit-explicit (IMEX) time-stepping. Good performance in depends on an efficient coarse propagator, which should be faster than the fine propagator, whilst still giving adequate accuracy for good convergence. In this case, reduced spatial resolution is used for the coarse solver.

Convergence properties, speed-up, and efficiency of the Parareal algorithm for the Roberts flow and other types of dynamos will be presented.

Drop formation in inkjet printing

Evangelia Antonopoulou

University of Leeds

In drop-on-demand inkjet printing individual drops are ejected from the nozzle by a pressure impulse applied to the printhead. Initially the drop remains connected to the printhead by a ligament, which subsequently thins and breaks off due to the surface-tension driven Rayleigh-Plateau instability. This ligament may either be absorbed into the main drop or break up into one or more small satellite drops. As a consequence, print quality, is critically dependant on both the waveform of the pressure impulse and on fluid properties such as viscosity and surface tension.

Here, we present results of simulations where we look at how the fluid properties, viscosity and surface tension, affect the drop formation and identify the parameter space for producing single drops at a prescribed speed. We compare the simulation results with experimental observations of drop formation from a commercial drop-on-demand printhead.

The role of environmental wind-shear in deep moist convection in the West African Sahel

Megan Bickle, John Marsham, Andrew Ross and Stephen Griffiths

University of Leeds

Organised mesoscale convective systems (MCS) within the West African monsoon deliver the vast majority of the Sahel region's annual rainfall. This rainfall is essential to agriculture and the livelihoods of the inhabitants of this vulnerable region.

It is generally agreed that environmental wind-shear plays a crucial role in allowing organised deep moist convection in these MCSs. Although there are several theories explaining how this occurs, their relative importance in controlling different aspects of convection is unclear. Further, global weather and climate models exclude the effects of wind-shear in their parametrisations of convection, possibly explaining the substantial errors in forecasting that occur for this region.

Squall lines have been simulated in an idealised high-resolution cloud resolving model based on average environmental profiles produced from observations made in the 2006 African Monsoon Multidisciplinary Analysis campaign. Sensitivity tests are being conducted to investigate the importance of various contributing factors, including the diurnal changes in wind-shear, temperature and humidity, to the nature of squall lines in West Africa. This work will help to identify which of the theories best explains the relationship between wind shear and organised squall lines, and hence provide guidance for convection parametrisation schemes used in global weather and climate models.

Eulerian indicators, blinking protocols, and mixing in multiple vortex systems

Tianqi Li and Andrew Cookson

University of Bath

In this study the classical blinking vortex system introduced by Aref [1] is extended to blinking systems with three, four and five point vortices, which are equally spaced on a circle of radius, R, within a circular boundary of radius = 2. The resultant mixing performance is investigated by tracking coloured, passive particles through the domain and quantified using an intensity of segregation measure.

Eulerian indicators, such as streamline transversality [2], were also calculated for each configuration, with the mixing ranking predicted by these indicators compared with that obtained via intensity of segregation. The parameters under investigation included radial position, R, blinking period, and the blinking sequence e.g. clockwise versus anti-clockwise and criss-crossing sequences.

The results show that for R = 0.5 - 1 the blinking sequence has a strong effect on the resultant mixing, but a weaker effect when R is close to either 0 or to 2. This is not reflected in the Eulerian indicators, which classify many of the different blinking sequences as identical. These indicators will therefore require further investigation for use on multi-blinking-vortex systems of this type.

References:
1. J. Fluid Mech. 143 (1984) pp. 1-12
2. New Journal of Physics 11 (2009) 075031

Examining the effect of cellulose reorientation in plant cell walls via a continuum mechanical approach

Euan Smithers
University of Birmingham

Understanding plants growth is essential for a future sustainable world. There are many unknowns to how rearrangement occurs, and how that regulates cell growth. Growth in plants is driven by high internal turgor pressure causing viscous stretching of the cell wall, with embedded cellulose microfibrils providing the wall with anisotropic properties. Evidence shows that cellulose reorients in cell walls as the cell expands. We investigate what the role of these are, and the potential mechanism of growth retardation when cells exit the expansion zone.

We derive and analyse a model for the effective mechanical properties of the evolving cell wall network by assuming the cell wall as a fluid. We inflict a flow on the cell wall to represent the growth and use this in a PDE to calculate the reorientation of the cellulose fibres. Using this flow in a fibre bond density equation we assume an elastic response for each component and calculate the total stress resultant. We perform asymptotic expansion to analyse the most important aspects. Initial results show the stress resultant value rises above the yield threshold through cellulose reorientation, hinting at a possible mechanism for growth retardation and that enzyme action is required to continue creep.

Miscible displacements in layered porous media

Japinder Nijjer, Duncan Hewitt and Jerome Neufeld
University of Cambridge

Miscible displacement processes, where a viscous fluid is injected into a porous medium initially saturated with another miscible fluid, are relevant in a number of physical situations. These include enhanced oil recovery, carbon capture and storage and subsurface contaminant transport. In many cases the goal is to control the displacement front and the amount of mixing that occurs between the two fluids, which can be controlled by a number of different factors. In this work, we explore the combined effects of permeability heterogeneities and viscosity variations on the mixing of the ambient and injected fluids, using high-resolution numerical simulations.

Specifically, we consider an idealized porous medium consisting of alternating layers of high and low permeability into which a fluid that is either equally-viscous, less-viscous or more-viscous than the ambient fluid is injected. We find that at intermediate times the dynamics depend on the viscosity variations, but eventually the flow enters a shear-enhanced dispersion regime, such that ultimately the mixing depends only on the permeability variations. We investigate the different regimes that arise depending on the relative magnitude of the permeability variations and viscosity variations, and develop simplified models that describe the evolution of the concentration field in each case.

Numerical modelling of gas-oil-water flow phenomena in horizontal pipelines

Jiri Polansky[1] and Mi Wang[2]

[1]Czech Technical University in Prague
[2]University of Leeds

Multiphase flows and associated numerical modelling are omnipresent in industrial and environmental processes, typically, the transportation of petroleum products, the operation of absorbers, gas lift pumps and many mores. It is known that different input parameters for the modelling would induce topologically different flow patterns with different flow behaviour. The present study concentrates to adiabatic incompressible three-phase flow in horizontal pipeline with separated characters of gas speed < 10 m/s and liquids speed < 0.2 m/s, such as stratified wavy, plug and slug flow regimes including typical Kelvin-Helmholtz instability. The Proper Orthogonal Decomposition (POD) was used to extract synthetic information to understand and model flow phenomena. In this study, POD is used to identify flow structure in the horizontal pipeline specially under transient of separated flow regimes.

To better understand the fluid dynamic nature of gas-liquids flows, this paper focuses on flow regimes simulation and pressure drop identification using approaches based on CFD and POD techniques under the set-up of gas-oil-water flow, which demonstrates the capability and limitation of numerical simulation by using OpenFoam. The InterFoam solver captures the interfaces and includes surface-tension and contact-angle effects for each phase. Defined flow parameters corresponding to typical gas-oil-water transportation properties are demonstrated.

Support from the Project No CZ.2.1.16/3/1.00/21569 Centre 3D Volumetric Anemometry is gratefully acknowledged.

Suppression of viscous fingering in a Hele-Shaw cell by changing geometry

Christian Vaquero-Stainer, Matthias Heil, Anne Juel and Draga Pihler-Puzovic

University of Manchester

Displacement of a viscous fluid by a less viscous fluid, such as air, in a narrow gap between two parallel plates (a Hele-Shaw cell) produces the well-known Saffman-Taylor interfacial instability. In recent literature, a number of possibilities for suppressing the instability by alternating the geometry of the cell have been explored theoretically and experimentally. Here, we focus on two particular cases; a static, tapered geometry and a time-dependent, power-law plate separation, $b(t) = b_1 t^{1/7}$, where $b(t)$ is the gap thickness between parallel plates. We present the results of direct numerical simulations, which employ a finite-element method to solve the depth-averaged lubrication equations. We confirm that both strategies are capable of stabilizing the air-liquid interface. We explore regimes beyond the validity of the linear stability analysis to study nonlinear mechanisms for pattern selection.

Sediment dispersal from hydrothermal plumes generated by volcanic eruptions at mid-ocean ridges

Daniel Ward[1], David Ferguson[1], Sam Pegler[1] and Rob Dorrell[2]

[1]University of Leeds
[2]University of Hull

It has recently been discovered that volcanic eruptions at mid-ocean ridges routinely involve the generation and dispersal of pyroclastic material up to distances of five kilometres from the eruptive vent. The dispersal of particles generated at seafloor volcanic eruptions at mid-ocean ridges is a problem limited by the understanding of the dynamics of volcanic plumes on the seafloor, and how they control the shapes, sizes, and lateral dispersal of volcanic particles. Establishing new theoretical frameworks for the dispersal of submarine volcanic material will aid interpretations of seafloor volcanic activity, resulting in the advancement of knowledge of present and paleoenvironments and improving predictions for the future. In this research there is a focus on understanding the dispersal dynamics of heated seawater at an eruption site and subsequently towards incorporating particle dispersal.

This project will undertake both numerical and mathematical modelling of particle-laden thermal plumes generated during seafloor eruptions. A mass-momentum conservation model for a thermal density driven plume, incorporating ocean currents, reversing plume buoyancy, and sediment deposition is the main objective. Initially, research has been focused on the heating boundary condition on the seafloor and the transition of the plume between laminar and turbulent regimes.

Numerical modelling of advanced aircraft fuel systems

Connor Rohmann-Shaw, Duncan Borman and Mark Wilson

University of Leeds

Advanced-aircraft fuel-systems have multiple requirements, Aside from necessity to provide fuel to the engine safely and efficiently, the fuel and tank distribution play a critical role in aerodynamic stability. CoG needs to be well known and appropriately controlled which is challenging during dynamic flight manoeuvres, especially those involving large accelerations which can result in complex sloshing within the tanks. In addition, fuel is increasingly being used as a heat-sink for other system components within aircraft. As fuel takes-up heat, which is then subsequently recirculated back into ‘cold’ fuel tanks, the thermal mixing process of sloshing fluid needs to be well understood such that new designs can be evaluated effectively. A model that considers the detailed fluid flow along with the coupled thermal response is needed to support the design process of new fuel systems. This work reports on a project to use computational fluids dynamics to provide this improved understanding. Results will be presented for numerous test fuel tank scenarios that are simulated using a multi-phase VOF approach, where sloshing fluid interacts with fuel entering the tank at a raised temperatures. An aspect of the work involves a series of thermal-sloshing experiments that will enable models to be appropriately validated.

A homogenization approximation for solute transport in the placenta

George Price and Oliver Jensen

University of Manchester

In the human placenta, intricate fetal capillary networks are confined within so-called villous trees, while maternal blood occupies the intervillous space outside the trees. Solutes are exchanged between maternal and fetal blood by diffusive transport across the walls of the terminal villous branches. Solute transport in the intervillous space can be modelled crudely as a unidirectional flow past a 1D array of point sinks that are randomly distributed along a line, which remove solute via first order kinetics. Each sink represents a terminal villus. When advection dominates diffusion between sinks, concentration boundary layers form upstream of each sink. To describe the effect of disorder in sink locations on the overall solute distribution, we use a Green's function method that reveals the multiscale structure of concentration profiles. We compare this method with a traditional homogenization approximation. Our approach is helpful in characterising the uncertainty in net solute transfer associated with the inherent spatial disorder in the placenta.

Mathematical model to investigate effect of hypertension on mass transport in arterial walls

Waruj Akaraworatikamporn

Imperial College London

Mass transport is found to be reduced in patients with hypertension and in arteries with an implanted stent. It has been hypothesised that an expansion of the arterial wall leads to a reduction in mass transport as a result of two main mechanisms: (1) reduces the gaps between tightly packed cells; and (2) deforms the vasa vasorum. To investigate this, a mathematical model is developed. The arterial wall is treated as a homogeneous poroelastic medium and the vasa vasorum is included as a circular pressurised lacuna in the tissue. The results presented in this study contradicts hypothesis (1) and provides support for hypothesis (2) at sufficiently elevated arterial pressures. Thus, at slightly elevated pressures other mechanisms must be involved in the observed increased morbidity. In current research, we are adding a model of the network of vessels in the vasa vasorum which would enable us to calculate the total wall perfusion. In addition, the nonlinear elastic property of the wall and the effect of arterial wave propagation could also be added at later stage. By implementing these improvements, the transport phenomena in the arterial wall could be further investigated as they are important keys to understand the development of atherosclerosis.

Experimental measurements in transitional partially-filled pipe flows using stereoscopic particle image velocimetry

Thomas Thornhill, Henry Ng and David Dennis
University of Liverpool

Stereoscopic particle image velocimetry (S-PIV) has been used to obtain velocity measurements in transitional pipe flow in a partially-filled pipe. Generally, laminar flow in full pipes is seen to occur below the Reynolds number of Re ff 2100 and turbulent flow occurring at Re ff 4000, with transition occurring in between these limits. Transition has been studied both numerically and

experimentally in detail for the full pipe arrangement, but not for partially-filled pipes despite the fact that there are many applications for running a pipe partially full. The very large scale pipe flow facility in the University of Liverpool will operate under partially full conditions from laminar to fully turbulent Reynolds numbers and measurements were taken over a range of flow depths (44% to 80% full pipe). S-PIV was used to measure the instantaneous three component velocity field on the pipe cross-section and using Taylor's hypothesis a 3D flow field was reconstructed, allowing an in depth analysis of transitional flow in partially-filled pipes and the turbulent structures within the flow. The critical Reynolds number (Reynolds number at which transition first occurs) has been identified for flows with different fill depths.

Aerodynamic modelling of insect wings using Joukowski's transformation

Gonzalo L. Jiménez and Mostafa R. A. Nabawy
University of Manchester

A Leading-Edge Vortex (LEV) is known to form on the upper surface of flapping/revolving wings leading to noticeably higher lift coefficient values. Whilst there have been many experimental and numerical studies to investigate the LEV aerodynamics within insect flight, there remains a need to develop simple theoretical models that contribute towards understanding this interesting phenomenon. In this work, we developed a low order quasi-steady theoretical model to simulate the flow around insect-like wings. The proposed model is based on the well-known Joukowski transformation and takes into account the essential aerodynamic features of insect-like wings including the LEV, the downwash and the non-linearity of lift at high angles of attack. Results are compared to existing data in the literature allowing useful insights into the role of the LEV in lift augmentation. We propose that the LEV does not enhance lift by providing an additional circulation to the one generated by the wing. Instead, it reattaches the flow behind the LEV to the wing, allowing insects to avoid stall at high angles of attack.

High-order integration of particle motion for Particle-in-Cell Schemes using the Boris Algorithm with Spectral Deferred Corrections

Kristoffer Smedt

University of Leeds

Particle-in-cell (PIC) schemes are a type of numerical kinetic plasma modelling method for hot diffuse plasmas, which are not suitable for treatment using conventional numerical MHD. Plasma particles are tracked in continuous space while the electric and magnetic fields are discretised and solved on a grid, with charge /current interpolated from particles to grid and vice versa for the fields. Integrating the Lorentz force equation for the particles, 'particle pushing', has traditionally used the Boris algorithm, which is intrinsically second order accurate. A new method of tuneable order of accuracy called Boris-SDC was developed in a previous study, using the Boris algorithm as a base method for spectral deferred corrections used to converge on a collocation solution. The method was shown to produce the desired high order solutions for particles in a simple Penning trap with an imposed magnetic field and directly computed electric field. The current study is investigating the performance of the Boris-SDC algorithm integrated into a Particle in Cell (PIC) scheme, which allows for larger simulations than the directly computed field method used in the previous study. Success would result in a new class of PIC method capable of any order of accuracy in particle pushing.

On non-isothermal thin film rimming flows with partial slip

Jonathon Nicholson[1], Antonino La Rocca[1], Donald Giddings[1], Outi Tammisola[2] and Edward Kay[3]

[1]University of Nottingham
[2]KTH Royal Institute of Technology
[3]Norton Straw

Within aero-engine bearing chambers arise thin fluid films of oil that wholly coat the chamber surface for accomplishing heat extraction. The film is restored via continuously recycled oil, cooled externally and replenished by droplets impacting on the film. To circumvent oil degradation, it is important to study thermal and flow dynamics for predicting characteristics and pattern formations.

Simulation of thermal effects inside a fundamental chamber explores potential film formulations under a variety of fully coating flows driven by strong airflow incidental to adjacent rotational components. The film develops on the heated chamber wall, with a cooled airflow absent of droplets acting above the oil. Our model details a depth-averaged construct of the governing equations, incorporating partial slip applied at the fluid to solid interface. The slip condition is pertinent as thinner films appear and is explored to ascertain the impact on temperature distribution and heat transfer. Utilisation of moderate inertial effects incorporated when studying the thermal characteristics allow for flows from pool to smooth solution reproductions. Exploration into the series of thermal characteristics and their impact for a range of cases including those where recirculation zones of fluid entrapment exist in the system are also presented.

Bifurcation analysis of evaporating droplets on a smooth surface

Michael Ewetola and Marc Pradas

Open University

We investigate the behaviour of a droplet that is slowly evaporating on a solid surface due to mass diffusion. We consider a surface that has smooth periodic variations, and by making use of bifurcation theory we show that as the volume of the droplet is changing in time, a hierarchy of bifurcations are triggered by the underlying properties of the surface, including pitchfork and saddle-node bifurcations. The bifurcation points, which can be predicted theoretically, correspond to critical volumes at which the droplet may change location by shifting laterally. In addition, we observe the emergence of a cusp bifurcation as the amplitude of the surface smooth variation is changed continuously. By making use of a diffuse interface formulation we also analyse the dynamics of the droplet as it evaporates observing a sequence of events in which the droplet base radius, contact angle, and mid-point rapidly change, in agreement with the theoretically predicted bifurcation points.

An investigation of fluid structure interaction of marine propellers

Nicholas Mccaw[1], Stephen Turnock[1] and William Batten[2]

[1]University of Manchester
[2]QinetiQ

Numerous studies have been conducted to compute the deformation of propeller blades. Moreover, efforts have been made to study the natural frequency of foils and marine propellers. However there is a gap in literature for a comprehensive numeric study on flow induced vibration of a marine propeller and design against the flow induced vibrations near the fundamental frequencies of the propeller. The aim of the project is to establish a process for designing marine propellers against natural frequency excitation using open source software. This paper describes the initial investigations to achieve this aim. The first stage is to establish an appropriate CFD model of the marine propeller. This is done by using the PPTC with the CFD results compared quantitatively to experimental studies. The second stage is to investigate the natural frequencies of the propeller. This is initially done by using commercial software to determine the natural frequencies of a foil in air and in water. Initial results show that the CFD results match well with the experimental results, with significant refinement around the blade tips due to vortices.

Dewetting dynamics of liquid films

Glen McHale[1], Rodrigo Ledesma-Aguilar[1], Andrew M.J. Edwards[2], Michael I. Newton[2] and Carl V. Brown[2]

[1]Northumbria University
[2]Nottingham Trent University

The inherent difficulty of inducing a complete wetting film on a non-wetting surface usually inhibits the study of the dynamics of dewetting of a liquid into a single droplet. Here, we show how a non-uniform electric field can induce a dielectric liquid into a circular-shaped film, which can then be triggered by removal of a voltage to dewet into an equilibrium droplet. We observe that dewetting includes a constant speed rim with dimple stage followed by an exponential relaxation of a spherical cap shape. We show how the dewetting process of a liquid film changes as the displaced second immiscible fluid phase is changed from air to a viscous liquid enabling control both of viscosity ratio and final equilibrium contact angle. We also model the dynamics of dewetting using both analytical models and Lattice-Boltzmann simulations. Hydrodynamic theory explains the dewetting dynamics of the rim and dimple and the long time exponential approach to equilibrium of the spherical cap. The simulations and theory reveal a local dewetting mechanism driven by the equilibrium contact angle, where contact-line slip dominates the dewetting dynamics. Our results are relevant to a wide variety of dewetting situations, such as drop rebound, condensation and evaporation.

Mathematical models of vitreoschisis in the eye: derivation of an appropriate slip condition to model the resultant motion of the vitreous

Laura Bevis and Jennifer Tweedy

Imperial College London

Vitreoschisis occurs when aqueous parts of the vitreous humour migrate towards the circumference of the eye and become trapped between layers of the vitreous cortex. This condition is thought to increase the risk of retinal detachment, which affects about one in 300 people and can cause blindness if not treated quickly. In order to model the motion of the vitreous as a result of vitreoschisis, previous work has modelled a sphere filled with a fluid representing the vitreous, with a slip condition representing vitreoschisis over a small region of the boundary, and investigated the dynamics during periodic eye rotations (Isakova, 2016). Here we compute the correct mathematical form of the slip condition for a given vitreoschisis, by considering layers of different incompressible fluids above an oscillating flat plate representing the retina, with the intact vitreous layers treated as non-Newtonian and the aqueous layers as Newtonian. This allows us to derive a slip condition analytically, and it can be matched with the previous work to determine the full dynamics. Further work includes approximating the full analytical expressions to enable easier computation, and extending the current solution to layers with only finite extent and including the effect of retinal curvature.

Effect of viscosity and throw distance on nanoelectrospray printing

Rebecca Hill and Matthew Alexander

University of East Anglia

Inkjet is a versatile non-contact, drop-on-demand printing technique for applications, such as printed electronics, sensors and biological microarrays. However, several limitations exist including low print resolution and strict limitations to ink properties such as a viscosity of 1-25cP.

A new drop-on-demand technique under development, based on nanoelectrospray, has potential to improve print resolution (down to a few microns dot size) and broaden the range of printable ink properties, most notably in terms of viscosity.

Nanoelectrospray is where fluid ejection is controlled purely by the voltage applied to the nozzle. Once electric pressure overcomes surface tension, a droplet or spray is produced depending on parameters such as viscosity, surface tension, conductivity, throw distance and nozzle geometry.

In the present work, the main focus is the study of viscosity (1-500cP) and throw distance (nozzle to substrate distance) to determine the conditions required for single droplet deposition.

It has been determined that both viscosity and throw distance have a strong influence on print performance, in terms of print resolution and print speed, an important finding for optimisation of the novel printing technology. It has also been demonstrated that highly viscous inks (500cP) can be printed at high resolution, a key advantage over inkjet.

Assessing the hydrodynamic loads on a swimmer's arm

Nicholas Copsey, Joseph Banks, Martin Warner and Markus Heller

University of Southampton

Swimmers require the use of complex motions with multiple degrees of freedom to propel themselves effectively through the water. The majority of propulsive forces are generated by the arms. Understanding how these external forces relate to the internal forces acting across the joints will enable better advice to be given to both recreational and competitive swimmers to prevent injury and improve performance.

Directly measuring the local external forces from such motions is difficult and expensive, however computational fluid dynamics (CFD) modelling could provide this detailed understanding. To generate a CFD model accurate kinematics of the swimmer's upper limb motion are required. These motions were acquired using Qualysis underwater motion capture cameras where 21 markers were placed on the body of a swimmer allowing the arm segments position, orientation and joint centres to be calculated whilst swimming.

2D CFD simulations will be used in a preliminary study to assess the flow features experienced by a swimmer's arm in different positions along its span. The effect of the experimental uncertainty in arm position and orientation will be assessed with respect to the forces generated. This work will help inform a 3D CFD methodology which can assess swimming fluid loads in the future.

Dynamic full-field pressure reconstruction

Rene Kaufmann, Bharathram Ganapathisubramani and Fabrice Pierron

University of Southampton

This work comprises pressure reconstructions from an impinging jet experiment. A deflectometry setup was used to obtain data and the Virtual Fields Method (VFM) allowed processing data to yield pressure estimates. Deflectometry is an optical full-field measurement technique for surface slopes. The method's high sensitivity allowed detecting slopes of several mm/km on the investigated flat plate sample. The VFM allows extracting information from full-field data via the principle of virtual work using a suitable choice of virtual fields. It is non-iterative and computationally inexpensive.

The pressures investigated in the present work are low at several tens to hundreds of Pa. Normally, amplitudes of this magnitude are measured using arrays of pressure transducers. These can alter the flow and require a lot of preparation, like drilling holes into the sample and creating space for tubing and instrumentation. With the VFM, the entire test surface serves as load cell, yielding much higher spatial resolution.

A Finite Element Model and artificial grid deformation allowed an assessment of the accuracy of the pressure reconstructions. Pressure transducer measurements further validated the results. Dynamic Mode Decompositions allowed extracting information on low amplitude vortex rings forming upstream and impinging on the sample.

Modelling passive scalar dispersion within and above an urban canopy

Lewis Blunn[1], Omduth Coceal[2], Robert Plant[3], Janet Barlow[3], Sylvia Bohnenstengel[4] and Humphrey Lean[4]

[1]University of Reading
[2]NCAS
[3]University of Reading
[4]Met Office

It is important that urban air quality prediction models are accurate within the urban canopy since this is the height in the atmosphere at which humans are typically exposed to higher levels of pollution. Buildings within the urban canopy exert a drag force on the flow field. This modifies both the mean wind profile and turbulence characteristics and consequently affects mixing of pollutants in the canopy. Scalar concentration profiles are computed following an approach originally developed for vegetation canopies, in which the time- and spatially-averaged equations for momentum and scalar mass conservation are solved. The method is adapted for urban areas using modified parametrisations for the drag and turbulence, informed by results from direct numerical simulation data. We present solutions for scalar concentration both within and above the urban canopy, and an investigation of the sensitivity of flow speed and scalar concentration to the turbulence parametrisation.

Self-assembly of magneto-mechanical meta-materials

Mike Evans[1] and David Fairhurst[2]

[1]University of Leeds
[2]Nottingham Trent University

We report on preliminary results from a Short Research Visit, funded by the UK FLuids Network. The visit is in two parts, scheduled for 11-15 June and 17-19 September 2018.

We investigate the feasibility of using self-assembly in complex fluids to construct a novel metamaterial from simple colloidal ingredients. It will be a colloidal alloy of microgel particles, a proportion of which contain super-paramagnetic cores. We speculate that the material's acoustic and mechanical properties (e.g. stiffness, viscoelasticity) will be tunable via two routes: (i) varying osmotic pressure to swell the polymeric microgel; (ii) applying an external magnetic field inducing dipole-dipole interactions between the super-paramagnetic cores.

Inter-dependence between the metamaterial's nontrivial magnetic response and its sensitive structure, each delicately poised at the boundary between order and disorder will engender new phenomena, mediated by mixed magneto-mechanical waves.

Such materials' novelty will derive from their ease of fabrication by self-assembly, together with the interactions between their magnetic, structural and fluid properties. They would potentially be cheap and simple to produce in large quantities. The resulting metamaterials could find applications in fields as diverse as energy conservation (via thermal properties), nanomanufacturing, acoustics (tunable sound insulation), wearable technology and healthcare (tunable stiffness allowing bespoke appliances).

Towards slurry transport correlations for nuclear waste and decommissioning

Christopher Cunliffe[1], David Dennis[1] and Jonathan Dodds[2]

[1]University of Liverpool
[2]National Nuclear Laboratory

The safe and efficient clean-out and decommissioning of pipelines within nuclear facilities requires an understanding of the flow of nuclear slurries. Maximising the solids loading of slurries lowers waste production, reducing long-term storage requirements and minimising associated costs and environment consequences. The presence of nuclear slurries is common in nuclear reprocessing facilities and the transportation of this material is a paramount part of decommissioning operations. This project begins work on the development, modelling and validation of slurry transport correlations specifically for nuclear reprocessing purposes by analysing the flow regimes and transition behaviour of various types of sludges and emulsions in partially filled pipes. Electrical Resistance Tomography will be employed to examine the internal flow structure across the pipe cross-section leading to the identification of flow correlations in the transitory fluid. The Slurry Transportation Rig at the National Nuclear Laboratory will be used for the identification of key parameters that will feed into future modelling and help develop a strategy for the transportation of viscous non-Newtonian fluids for the nuclear industry.

Coupling hydrodynamics and biokinetic growth models for improvement of wastewater treatment

Andrew Oates

University of Leeds

Biokinetic models have been developed to describe the growth of biomass in wastewater treatment processes. These models describe the reaction rates for the growth and decay of biomass in wastewater bioreactors. They are a useful tool for understanding the biological processes occurring in these reactors which can be used to improve design and operating conditions to increase effectiveness and efficiency. However, a critical assumption made by these models is that the reactors are well-mixed such that the parameter concentrations are uniform throughout the reactor. This is valid if the reactors are perfectly mixed, which may be the case, but it can be extremely costly to do so. The ideal reactor design is one that uses the least amount of energy to mix while not hindering the desired reactions. By coupling the hydrodynamics and biokinetic growth models, we can observe and analyse the impact of the hydrodynamics on the biokinetic reaction rates. These enhanced predictions can aid in improved design and operating conditions of the bioreactors, leading to more efficient and effective processes which reduce overall cost of the systems.

Direct simulation of turbulent plumes in a crossflow

Owen H. Jordan and Maarten van Reeuwijk

Imperial College London

Direct Numerical Simulation of buoyant plumes in a uniform crossflow is performed with the aim of investigating the effect of the crossflow velocity on the dynamics of the plume evolution. A detailed comparison is made with the classical theory of bent-over plumes. The simulations indicate that the forcing associated with pressure differences across the cross-section of the plume is a leading order quantity, which is important since this term is generally discarded in the plume theory. We explore how this quantity can be parameterized with the help of DNS data. Furthermore, we investigate how the crosswind affects the evolution of the plume geometry.

Pore-scale simulation of flow through gas diffusion layers

Daniel Niblett, Vahid Niasar and Stuart Holmes

University of Manchester

Fuel cell Gas Diffusion Layers (GDL) are synthetic fibrous porous media that handle the transport of fluids through the void space system. Pore scale simulations of flow through porous media such as carbon paper GDL can uncover the mechanisms for the performance of these layers. We evaluating the permeability of the fluid phase defined by generated un-ordered (GDL) and ordered (cubic and hexagonal lattice) at different porosity levels. Simulation of the pressure potential is solved in the void space using CFD software OpenFOAM®. The 3D models allow the transport in these layers to be evaluated and characterised, unveiling the mechanism for resistances of pore scale flow in GDL. The ordered structures allow for understanding of how intersecting fibres affect the flow in the void space. The ordered structures represent an isotropic (cubic) and an anisotropic (hexagonal) system. Results show that the structured microstructure permeability is higher than both in-plane and through-plane directions. The decrease in permeability with porosity can be attributed to the mass transport resistances. Application of these changes to morphology into fuel cells will reduce mass transport resistances due to an improved permeability.

Non-Fickian transport under two-phase flow conditions

Sharul Nizam Hasan and Vahid J. Niasar

University of Manchester

Enhanced oil recovery comprises injection of chemicals with water to improve the oil recovery from the oil reservoirs. To simulate the transport for enhanced oil recovery, usually the transport of the chemicals is characterized by single-phase transport coupled with two-phase flow which is inconsistent. In this study we develop a pore-network model to simulation advective-dispersive transport in a three-dimensional pore network for different saturation states. The simulations evaluate the origin of the non-Fickian transport under different degrees of saturations at different Peclet numbers. Our results highlight the non-monotonic trend of the stagnant saturation versus the total saturation, which highlights the mixing the chemicals with the resident brine. This impact is very important in the efficiency of low salinity water flooding where the mixing of the low salinity and high salinity with reduce the efficiency of the technology.

Inertial and elastic flow instabilities in a mixing-separating microfluidic device

Allysson F. Domingues, Mahdi Davoodi, Robert J. Poole, and David J. C. Dennis

University of Liverpool

Combining and separating fluids has many industrial and biomedical applications such as biomedical devices, mini or micro-scale mixers and micro-rheometry. In this work, we have been investigating, experimentally and numerically, both purely inertial and purely elastic flow instabilities in a so-called mixing-separating cell micro-geometry using Newtonian and viscoelastic fluids. Our microfluidic device consists of two straight square parallel channels with flow from opposite directions with a central gap that allows the streams to interact, mix or remain separate (often referred: 'H' geometry). For the Newtonian fluid, under creeping flow conditions (Reynolds number [Re ff 0]) the flow is steady, two-dimensional and produces a saddle point at the centre of the geometry. For Re>30, an inertial instability appears which leads to the generation of a central vortex. As Re increases the central vortex divides into two vortices, although the flow remains steady for the Reynolds number range tested ($1 <$ $Re < 70$). In contrast, the creeping flow of a highly viscoelastic and constant viscosity fluid through the geometry undergoes a purely-elastic flow transition to time-dependent flow depending on the Weissenberg number ($Wi > 2$). The results suggest that the effect observed can be exploited in biomedical or other applications.

Implementation of a dual Navier-Stokes / lattice Boltzmann method on a heterogeneous CPU / GPU architecture

Marta Camps Santasmasas, Ben Parslew, Adrian Harwood, Bill Crowther, Alistair Revell

University of Manchester

Each computational method developed for fluid dynamics problems has different features; thus combining two methods allows exploiting the features of each one. The Navier-Stokes (NS) and lattice Boltzmann (LB) models can both solve incompressible flow problems, however there are significant differences in their algorithms. The LB algorithm is local and simple and thus suitable for implementation on graphic processing units (GPU). Whereas, NS models have lower memory requirements, are more stable and have greater mesh flexibility; but their algorithm is complex and thus less suitable for GPU implementation. We present a dual NS / LB method for external aerodynamics that combines the mesh flexibility, stability and low memory requirements of the NS solver with the computing speed of the GPU-accelerated LB solver. The method exploits the widespread availability of CPU / GPU hardware on consumer devices. The simulated domain is divided in a Navier-Stokes sub-domain run on CPU and a lattice Boltzmann sub-domain run on GPU. The NS sub-domain models the flow in the outermost region and the LB sub-domain resolves the near-wall ow with more detail. We will describe the algorithm of the dual solver including the CPU / GPU implementation and will validate it against NS results.

Experimental and numerical investigation of flow control for refrigerated freight transport

Maxime Savoie, Alistair Revell, Mark Kenneth Quinn, Ben Parslew, Nicholas Bojdo

University of Manchester

The average Heavy Commercial Vehicle (HCV) travels over 130000 km/ year and has a fuel cost of 40000 Euro/year. Despite representing only 5% of the total fleet of European vehicles, HCVs are responsible for around 30% of the CO_2 emission of the European road transportation. In a context of global warming and rise of fuel price, improving their efficiency becomes essential.

In terms of losses, at motorway speeds, aerodynamic drag is the second largest contributor after the engine. A small reduction in aerodynamic drag could lead to huge savings when multiplied across the whole European fleet.

This project focuses on improving the aerodynamics of refrigerated freight transportation. A review of current literature has shown that despite a good general understanding of the aerodynamics, further studies of complex flow features need to be done. The flow around a simplified 3D model is studied using advanced wind tunnel techniques, supported by CFD results. A stereoscopic PIV system (2-dimension 3-component) is used to investigate the flow around the model and correlation with in-house experiment and literature review is performed using hybrid RANS-LES methods.

These results will be used to develop more efficient flow-control drag reduction devices.

Instability of elevator modes in oscillatory double-diffusive convection

Thomas Goodfellow, David Hughes, Stephen Griffiths and Peter Jimack

University of Leeds

Fluid dynamical systems consisting of two or more diffusive components exhibit rich and interesting behaviour. In the oceans, heat and salt gradients interact to fuel phenomena such as salt fingers, oscillatory instabilities and long-lived density staircases. The linear stability of double-diffusive systems to small perturbations is well-studied: a static basic state is unstable to 'elevator modes' (i.e. modes that are vertically invariant), which, in turn, are fully nonlinear solutions to the governing equations. In this study, we go beyond the linear theory for a static basic state, and instead investigate the stability of an elevator basic state.

We focus on the diffusive case in which salt – the slower diffusing component – provides a stabilising gradient, and in which the primary elevator modes are purely oscillatory in time. We use a double-Floquet expansion in time and space to determine the growth rates of secondary modes. The structure of the fastest-growing secondary mode is influenced by the amplitude of the primary elevator mode, leading to two distinct solutions: at moderate amplitudes, the dominant secondary mode is horizontally and vertically dependent; at large amplitudes, the secondary mode is horizontally and vertically dependent, but it now sits within the basic state.

Passive flow control for wind turbine airfoils

Marinos Manolesos[1] and Spyros Voutsinas[2]

[1]Swansea University
[2]National Technical University of Athens

Passive flow control remains attractive for wind turbine manufacturers because of its simplicity, robustness, ease of application and effectiveness. This investigation will present an overview of five years of experimental studies on passive separation control and drag reduction methods for wind turbine dedicated airfoils.

Vortex Generators (VGs) are flow separation control devices that generate streamwise vortices, which energize the local boundary layer. Their small size, at the order of the local boundary layer height, renders the study of the resulting flow particularly challenging both experimentally and numerically. The most detailed Stereo Particle Image Velocimetry measurements for VGs on airfoils to date will be presented and their use as benchmark for numerical models will be demonstrated.

Drag reduction devices are becoming increasingly popular for wind turbines as flatback airfoils are included in the blade design. Stereo PIV, pressure and hot wire measurements reveal the drag reduction mechanism of an innovative device, which performs 30% better than traditional options. All experiments concern Wind Turbine dedicated airfoils at Reynolds number $O(10^6)$. The results reveal the expertise and the possibilities of the new low speed Wind Tunnel at Swansea University.

Comparative investigation of multi-pass flow for air photovoltaic/thermal collectors

Moustafa Al-Damook[1], Mansour Al Qubeissi[2], Darron Dixon-Hardy[1], Peter Heggs[1] and Khaled Al-Ghaithi[1]

[1]University of Leeds
[2]University of Coventry

Recently, much emphasis has been placed on improving the efficiency of solar systems with co-generation as a particularly promising research area. This work focuses on the integration of photovoltaic (PV) and solar air heater systems with the aim to evaluate electrical and thermo-hydraulic performances. Five air PV/Thermal collector designs are investigated numerically. These designs are classified based on the direction of flow and number of passes. The first design model is a single duct single pass, in which air flows underneath the PV panel and the upper surface of PV subject to ambient conditions. The second is similar to the first one, but with an air gap added between the upper surface of the PV module and glass cover. In the third, air flows under and over the surfaces of the PV module in the same direction. The fourth is similar to the third one, but the flow pathways are in opposite direction. In the fifth, air flows between the glass cover and the PV module and reverses in the second pass between the PV panel and lower absorber plate, making a U-shape flow. The fifth model displays the best thermal performance whilst the third offers the highest electrical efficiency.

Importance of irrotational components of swimming flows on the stability of a suspension of weakly squirming microorganisms

Michael Gravatt, Alys Clark, Vinod Suresh and Richard Clarke

University of Auckland

Understanding the behaviour of a suspension of microorganisms swimming in a fluid environment has a range of industrial applications from improving cell health in bioreactors for food production to efficient production of biofuels. Active suspensions of microorganisms display interesting collective dynamics on a macroscale despite only being microscopic in size. This dynamical behaviour is likely the result of multiple mechanisms including cell-cell collisions, photo/chemotactic stimuli and cell-cell hydrodynamic interactions. We are specifically interested in the latter and the collective dynamics it induces.

When a microorganism swims in a fluid environment, it induces a stress on that fluid arising from the rotational components of the flow field that an individual creates and is traditionally added as a forcing term on Stokes equations. We consider a more explicit formulation of cell-cell hydrodynamic interactions that captures a more complex flow field of an individual. By including irrotational components and conducting a linear stability analysis we show that for weakly squirming microorganisms the critical wavelength of the instability changes and in certain cases the finite wavelength instability is removed entirely. This changes the prediction for the lengthscale of the collective motion and bulk fluid flow that is induced by cell-cell hydrodynamic interactions.

Oral presentation index