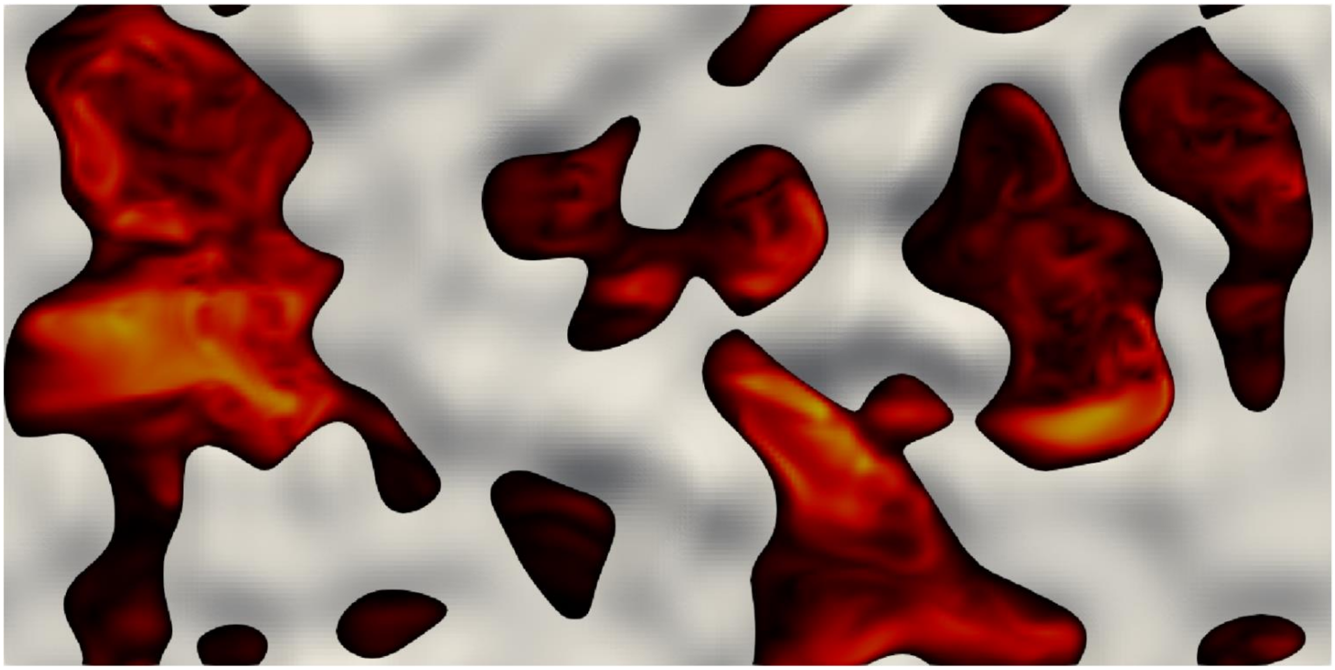

UK Fluids Conference
8th-10th September 2021
Book of Abstracts

UK Fluids Conference 8th-10th September 2021

Table of Contents

Welcome	3
Schedule overview	5
Invited talks	6
Programme for oral presentations	7
Poster presentations	19
Abstracts	22
Index of presenters	239

Organising Committee

Dr. Ralf Deiterding (University of Southampton)

Prof. Shuisheng He (University of Sheffield)

Dr. Davide Lasagna (University of Southampton)

Prof. Cath Noakes (University of Leeds)

Dr. George Papadakis (Imperial College London)

Dr. Ivo Peters (University of Southampton)

Prof. Neil Sandham (University of Southampton, Chair)

Dr. Claire Savy (University of Leeds)

Dr. Christina Vanderwel (University of Southampton)

Dr. Gabriel Weymouth (University of Southampton)

Welcome to the UK Fluids Conference 2021

Link to conference

You can access the conference site at this link: https://gather.town/invite?token=cSjYyniA

Instructions for all participants

If you haven't yet used Gather.Town, see https://gather.town/app/ZjoKsCgut1xMYH5L/Live%20Walkthrough or https://gather.town/app/p4B9DUqB8NAazd3t/DemoConference and the walk-through of our conference site on https://www.youtube.com/watch?v=gyP06pqQ-SQ. Make sure your browser works (Chrome or Firefox are the supported browsers, but not Safari) and you can navigate around the virtual world and share your camera and computer screen (this part is similar to Zoom/Teams etc). Any joining issues should be reported to ukfluids@soton.ac.uk (make sure you use exactly the e-mail you registered with). Note the use of 'x' to interact with content and 'g' to avoid blocking by other users. Note the speaking positions (orange spotlight squares) that you can move to and then heard by the whole room. Session chairs and speakers will use these during sessions.

Check presentation times in the final programme, since some have changed relative to the previous draft. Keep cameras and microphones **off** during plenary sessions on Teams and oral presentations in Gather.Town. Put cameras **on** during poster sessions and when interacting outside the oral sessions. Please leave the front row free in the lecture rooms for speakers. Ask questions after talks (orally via broadcast stations, or in the chat, noting the room letter A, B or C in the question, i.e. 'Room A, I have a question about …') and stay after sessions to meet authors of presentations in which you have a special interest. Say hello to the sponsors. If you get a red warning box about rooms being over the maximum, please put a note in the chat.

Prizes

You can vote for best student presentations and posters (highlighted in green in the schedule) via https://forms.office.com/r/M12W6YDE8T. Please do this before Thursday 18:00, to help the prizes committee make their choices.

Instructions for session chairs for parallel sessions (10 minutes talks, 2 minutes questions/changeover)

The role of session chairs is to keep the sessions to time, noting that there is no centrally managed clock. Chairs will provide short introduction of talks, provide a 1 minute-to-go warning, invite and moderate questions and getting the next speaker set up. Make sure you unmute before speaking. If background noise is coming through, Chairs should ask the audience to make sure they are muted and with cameras off. Chairs should be present from 5-10 minutes before the session to meet the session assistant and speakers. If it looks like there will more than 100 audience members, the chair/assistant should check with the organisers about whether to move that session to Teams. In the event that a speaker is absent, a gap should be left in the session so that the following talks start at the times stated in the programme.

Instructions for presenters in oral sessions

Prepare and practice the presentation so it fits in the time available. Check that you can share your screen and that any videos play properly while presenting, bearing in mind bandwidth limitations. Presenters should be at the front of the room from 5-10 minutes before the session to meet the session chair and assistant. Make sure you unmute before speaking. It is usually a better experience for the audience if speaker cameras are **on** for oral presentations, unless you get warnings about bandwidth issues. When you get the 1-minute warning move to your final slide and conclude. Presenters should remain in the room after the session to facilitate informal discussions.

Important note on invited (plenary) talks (35 minutes talks, 5 minutes questions)

In contrast to the parallel sessions, these will run as external calls from Gather.Town to Microsoft Teams. When you go to any of the lecture rooms A, B and C from 10 minutes before the start time, you will get the message 'Teams session in progress: **join by pressing 'x'**. Press 'x', follow the link and start MS Teams either in your own app, if you already have one, or else cancel the pop-up and select the option to join from your browser. Make sure your camera and microphone are off. You can then listen to the talk and post questions in the chat as in any normal Teams/Zoom meeting. At the end just close down your Teams window or tab and, back on the Gather tab of your browser, you will be able to click the green box to return to Gather.Town. The same method will be used for the initial welcome to the conference.

Instructions for poster sessions

Be ready with a short tour of your poster (1-2 minutes) when you get visitors to the private area. You have access to a pointer via Gather.Town to show items on the poster. You can also share a screen to show anything else (pptx, video etc.) If the area gets quiet for a while, try visiting your neighbours.

Instructions for sponsors

Try to be available during breaks, lunch hour etc. You can also share a screen to show anything else (pptx, video, websites, data sheets etc). You can also use the committee room for short presentations, which can be advertised in the chat.

Schedule overview

	Start	End	Lecture room A	Lecture room B	Lecture room C
Wed 8th Sept.	08:00	09:00	Site open (including help desk) for orientation and speaker checks		
	09:00	09:10	Welcome (access from any of the lecture rooms A, B or C)		
	09:15	10:15	1A: Bluff body wakes and plumes	1B: Geophysical flows 1	1C: Droplets 1
	10:30	11:30	2A: Aerodynamics & Roughness	2B: Geophysical flows 2	2C: Bubbles
	11:45	12:25	Invited talk I1: Aimee Morgans		
	13:15	13:55	Invited talk I2: Stephen Belcher		
	14:15	15:15	3A: Aerodynamics 1	3B: Biological flows 1	3C: Multiphase flows 1
	15:30	16:30	4A: Aerodynamics & turbulence	4B: Biological flows 2	4C: Porous media and surfactants
	17:00	19:00	Poster reception (Poster rooms A and B)		
Thursday 9th Sept.	09:00	09:40	Poster breakfast (Poster rooms A and B)		
	09:45	10:45	5A: High speed 1	5B: Data-driven modelling	5C: Thin film and interfacial flows
	11:00	12:00	6A: Aerodynamics and stratified flows	6B: Data /MHD	6C: Multiphase flows 2
	12:15	12:55	Invited talk I3: Emmanuel de Langre		
	13:45	14:45	7A: Aerodynamics/boundary layers	7B: Fluid-structure interactions	7C: Meshless/mesh-free methods for fluids
	15:00	16:00	8A: High speed 2	8B: Industrial flows	8C: Capillary flows/microfluids
	16:15	17:15	9A: Boundary layers	9B: Industrial and biological flows	9C: Porous media/microfluids
Friday 10th Sept.	09:00	10:00	10A: Aerodynamics 2	10B: Geophysical flows 3	10C: Droplets 2
	10:15	11:15	11A: Jets, wakes & turbulence	11B: Astrophysical flows	11C: Droplets 3
	11:30	12:30	12A: Aerodynamics 3	12B: Industrial flows	12C: Multiphase flows 3
	13:30	14:10	Invited talk I4: Rob Poole		
	14:15	14:30	Prizes		

Invited talks

I1 Sound, flames and aerodynamics: how they interact to generate thermoacoustic instability

Aimee Morgans, Imperial College

I2 Fluid mechanics for weather and climate research and operations

Stephen Belcher, Met Office

I3 Fluid-structure interaction with plants

Emmanuel de Langre, Ladhyx, Ecole Polytechnique

I4 Flow instabilities in the cross slot geometry: characterisation, exploitation and cure

Rob Poole, University of Liverpool

Parallel sessions 1: Wed 8th Sept 9:15-10:15

Session 1A Bluff body flows, wakes and plumes (Chair: Vanderwel)

9:15 (01A-1) Near-wake flow features of polygonal cylinders in different incidence angles, Esmaeel Masoudi, University of Durham

9:27 (01A-2) Attenuation of the unsteady loading on a high-rise building for different wind angles using feedback control, Xiao Hu, Imperial College London

9:39 (01A-3) Simulation of high Reynolds number sub-critical flow over a cylinder using a scale resolving hybrid RANS/LES closure model, Michael Mays, Imperial College London

9:51 (01A-4) Experimental modelling a buoyant pollutant source behind a backwards-facing step, Samuel Charlwood, University of Cambridge

10:03 (01A-5) The dynamics of a line plume confined by a single inclined boundary, Tom Newton, University of Cambridge

Session 1B Geophysical flows 1 (Chair: Tobias)

9:15 (01B-1) Channelized flow in the partially molten upper mantle, David W Rees Jones, University of St Andrews

9:27 (01B-2) Poroviscoelastic dynamics of mushy magmatic systems, Jennifer Castelino, University of Leeds

9:39 (01B-3) Investigation into the folding of layered viscous structures, Olivia Goulden, University of Leeds

9:51 (01B-4) Bounds for internally heated convection with fixed boundary heat flux, Ali Arslan, Imperial College London

10:03 (01B-5) Massively parallel Lattice Boltzmann Method models of dilute gravity currents, Damilola Adekanye, University of Leeds

Session 1C Droplets 1 (Chair: Peters)

9:15 (01C-1) Rim breakup dynamics in aerodynamic droplet breakup, Isaac M. Jackiw, University of Toronto

9:27 (01C-2) Atomisation of a capillary jet using a stacked-type electrofluidic actuator, Samy Lalloz, University of Coventry and CNRS

9:39 (01C-3) Numerical modelling of personal care products' spray efficiency, Liam Robert Gray, University of Leeds

9:51 (01C-4) Mathematical modelling and analysis of the evolution of and deposition from a sessile evaporating droplet, Hannah-May D'Ambrosio, University of Strathclyde

10:03 (01C-5) Governing equations and solution multiplicities for a static ridge of nematic liquid crystal, Joseph R. L. Cousins, University of Glasgow

Parallel sessions 2: Wed 8th Sept 10:30-11:30

Session 2A Aerodynamics & Roughness (Chair: Vanderwel)

10:30 (02A-1) Immersed boundary simulations on the effect of roughness: Flow over a flat plate, Jonathan Massey, University of Southampton

10:42 (02A-2) Effects of surface roughness in flapping foil performance, R. Vilumbrales Garcia, University of Southampton

10:54 (02A-3) Designing optimal roughness: A theory-led approach to roughness-based drag reduction, P.Y.A.G.S. Yapa, University of Warwick

11:06 (02A-4) Using flow control to create ideal platooning conditions for road vehicles, Craig Nolan, University of Glasgow

11:18 (02A-5) New insight into the increased turbulence due to non-uniform streamwise body forces in pipe flow, Stephen Jackson, University of Sheffield

Session 2B Geophysical flows 2 (Chair: Tobias)

10:30 (02B-1) A new Lagrangian drift mechanism in coastal oceans: role of current and seabed undulation interactions, Akanksha Gupta, University of Dundee

10:42 (02B-2) Internal wave triads in a bounded domain with mild-slope bathymetry, Saranraj Gururaj, University of Dundee

10:54 (02B-3) The interplay of symmetric, inertial, and baroclinic instabilities on the evolution of oceanic fronts, Aaron Wienkers, University of Cambridge

11:06 (02B-4) The effect of unbalanced exchange flow on the night cooling of buildings, Nick Wise, University of Cambridge

11:18 (02B-5) Validity of sound-proof approximations for magnetic buoyancy, John Moss, Newcastle University

Session 2C Bubbles (Chair: Peters)

10:30 (02C-1) Shape mode oscillation of a bubble in a viscous compressible liquid near to a rigid boundary, Callan Corbett, University of Birmingham

10:42 (02C-2) The effects of a Maxwell fluid on the evolution of a three-dimensional microbubble, Eoin O'Brien, University of Birmingham

10:54 (02C-3) Toroidal analysis of an oscillating sessile drop, Saksham Sharma, University of Cambridge

11:06 (02C-4) Influence of particle aggregation on evaporating and de-wetting droplets of complex fluids, Junzhe (James) Zhang, Loughborough University

11:18 (02C-5) Circular hydraulic jump in microgravity, Rajesh K. Bhagat, University of Cambridge

Parallel sessions 3: Wed 8th Sept 14:15-15:15

Session 3A Aerodynamics 1 (Chair: Lasagna)

14:15 (03A-1) Validation of analytical wind turbine wake models in turbulent base flows, Stefano Gambuzza, University of Southampton

14:27 (03A-2) Investigation of the validity of wall-modelling in simulations of rotating cavity flows, Tom Hickling, University of Oxford

14:39 (03A-3) Adaptive reduced order modelling of steady aerodynamic flows over ultra-high aspect ratio wings, Peter Nagy, University of Strathclyde

14:51 (03A-4) Modulation of leading-edge vortex shedding with chordwise deformation, Alfonso Martínez, University of Glasgow

15:03 (03A-5) Effect of thickness on wake dynamics of an airfoil pitching in uniform flow, Muhammed Najvan, IIT Kanpur

Session 3B Biological flows 1 (Chair: Weymouth)

14:15 (03B-1) On the coupling between back pressure and flow structure in airway bifurcations, Solène Cargill, Coventry University

14:27 (03B-2) 4D-Flow MRI based CFD to improve patient specific in-silico predictions of the post-surgical haemodynamics in the thoracic aorta, Molly Cherry, University of Leeds

14:39 (03B-3) Non-trivial transport of active Brownian particles (micro-swimmers) in shear, Lloyd Fung, Imperial College London

14:51 (03B-4) Is the shape of air particulate matter important? A study on the interactions of the two shapes of titanium dioxide nanoparticles with a model of pulmonary surfactant, Farzaneh Hajirasouliha, Northumbria University

15:03 (03B-5) Optimal turning gaits for undulators, Madeleine Hall, Imperial College London

Session 3C Multiphase flows 1 (Chair: Lawson)

14:15 (03C-1) Flow correlations and transport behaviour of turbulent slurries in a partially filled pipe, Christopher J. Cunliffe, University of Liverpool

14:27 (03C-2) A population balance approach for investigating the condensation of aluminum oxide droplets near a burning aluminum particle, Jannis Finke, University of Magdeburg

14:39 (03C-3) CFD simulation of particles-liquid turbulent pipe flow, Zhuangjian Yang, University of Birmingham

14:51 (03C-4) Improving multi phase measurement performance of a Coriolis Flowmeter with additional pressure drop correlations, Stephan Schäfer, University of Glasgow

15:03 (03C-5) Swelling driven autonomous soft gripper, Zhuofan Qin, Northumbria University

Parallel sessions 4: Wed 8th Sept 15:30-16:30

Session 4A Aerodynamics & turbulence (Chair: Lasagna)

15:30 (04A-1) New optimisation methods for scale-resolving turbulent flow simulation, Rhys Gilbert, University of Southampton

15:42 (04A-2) Optical MEMS sensors for wall-shear stress measurements in turbulent boundary layer flows, Nima Ebrahimzade, Newcastle University

15:54 (04A-3) The mechanism of lobes generation in the sound directivity patterns for gust-airfoil interaction, Shujie Jiang, University of Southampton

16:06 (04A-4) The turbulence production mechanism in the near-wall region under an adverse pressure gradient, Yuxin Jiao, Imperial College London

16:18 (04A-5) Dipole dynamics in the Point Vortex Model, Karl Lydon, Aston University

Session 4B Biological flows 2 (Chair: Weymouth)

15:30 (04B-1) Mathematical modelling of drug metabolism in a microfluidic system, Barnum Swannell, University of Oxford

15:42 (04B-2) A poroelastic model for loading induced damage and rehabilitation in tendons, Isabelle Scott, University Of Oxford

15:54 (04B-3) Reconstruction and segmentation of noisy flow images as an inverse Navier-Stokes problem, Alexandros Kontogiannis, University of Cambridge

16:06 (04B-4) Multi-objective optimisation of continuous flow polymerase chain reaction flow systems, Foteini Zagklavara, University of Leeds

Session 4C Porous media and surfactants (Chair: Peters)

15:30 (04C-1) Deformation-driven rupture of gas cavities within a soft porous medium, Oliver Paulin, University of Oxford

15:42 (04C-2) Using Dissipative Particle Dynamics for investigating surfactant solutions under shearing, Rachel Hendrikse, University of Leeds

15:54 (04C-3) Singularities in surfactant driven cavity flow, Richard Mcnair, University of Manchester

16:06 (04C-4) Hydrodynamic interactions of sedimenting squirmers, Albane Théry, University of Cambridge

Parallel sessions 5: Thur 9th Sept 9:45-10:45

Session 5A High speed 1 (Chair: Deiterding)

9:45 (05A-1) Influence of thermodynamic model on the shock interaction patterns of CO2 flows, Catarina Garbacz, University of Strathclyde

9:57 (05A-2) The UK's hypersonic multiphysics fluid simulation suite: OP2A, Thomas Greenslade, University of Southampton

10:09 (05A-3) Effect of rarefaction on the evolution of compressible Kelvin-Helmholtz instability, Vishnu Mohan, IIT Madras

10:21 (05A-4) Analysis of grid influence on the simulation of nonequilibrium flow over proximal bodies, Fábio Morgado, University of Strathclyde

10:33 (05A-5) Simulations of rotating detonation waves with adaptive mesh refinement, Han Peng, University of Southampton

Session 5B Data-driven modelling (Chair: Juniper)

9:45 (05B-1) Reconstruction of 3D large-scale structures inside a stirred tank from limited velocity measurements, Kirill Mikhaylov, Imperial College London

9:57 (05B-2) A minimal data-driven quasilinear approximation based approach for turbulent channel flow statistics, Jacob Holford, Imperial College London

10:09 (05B-3) CNN-aided flooding prognostic in packed column using Electrical Capacitance Tomography, Yuan Chen, University of Edinburgh

10:21 (05B-4) Ensemble simulation methods for in-silico trials of endovascular medical devices, Michael MacRaild, University of Leeds

10:33 (05B-5) Poisson CNN: Convolutional neural networks for the solution of the Poisson equation on a Cartesian mesh, Ali Girayhan Özbay, Imperial College London

Session 5C Thin film and interfacial flows (Chair: Hazel)

9:45 (05C-1) Experimental investigation of displacement instabilities in straight microchannels with pure viscoelastic fluids, Seng H. Hue, UCL

9:57 (05C-2) Reversible trapping of colloids in continuous flows past grooved microchannels by diffusiophoresis, Naval Singh, Loughborough University

10:09 (05C-3) Coating flow in the presence of an irrotational airflow with circulation, Andrew Mitchell, University of Strathclyde

10:21 (05C-4) Wetting and dewetting dynamics of a thin liquid film spreading on an immiscible liquid surface, Christophe J. Wilk, University of Manchester

10:33 (05C-5) Adjoint equation-based estimation of porous media permeability, Tao Zhang, University of Bath

Parallel sessions 6: Thur 9th Sept 11:00-12:00

Session 6A Aerodynamics and stratified flows (Chair: Deiterding)

11:00 (06A-1) The effect of large scale incoming turbulence on wind-turbine-blade, ThankGod E. Boye, University of Southampton

11:12 (06A-2) An experimental investigation of the flow past a very thick flatback airfoil suitable for wind turbine blades, Antonios Cene, Swansea University

11:24 (06A-3) Harmonic forcing of a laminar bluff body wake with rear pitching flaps, Athanasios E. Giannenas, Imperial College London

11:36 (06A-4) Fluid mechanics of sash windows, Gaël Kemp, University of Cambridge

11:48 (06A-5) A comparison of the development of spatially and temporally accelerating flows, Matthew Falcone, University of sheffield

Session 6B Data /MHD (Chair: Juniper)

11:00 (06B-1) Spectral Relaxation computation of heat transfer in aligned-field MHD Flow of nanofluid over a moving flat plate, Shahina Akter, University of Dhaka

11:12 (06B-2) Electrically driven Alfven Wave investigation, Samy Lalloz, University of Coventry and CNRS

11:24 (06B-3) Is machine learning robust for the prediction of chaotic flows? A. Racca, University of Cambridge (to be presented by Luca Magri)

11:36 (06B-4) Koopman with control for constitutive law identification, Emily Southern, Imperial College London

Session 6C Multiphase flows 2 (Chair: Hazel)

11:00 (06C-1) An Eulerian-Lagrangian approach to simulate particle-laden flows, Boyang Chen, University of Birmingham

11:12 (06C-2) Combining the method of characteristics with a reduced order modeling technique for economically solving the population balance equation, Fabian Sewerin, University of Magdeburg

11:24 (06C-3) A numerical study on suspended sediment transport in a partially vegetated channel flow, Mingyang Wang, Queen Mary University of London

11:36 (06C-4) A deep learning-CFD method for Reynolds-Averaged Navier-Stokes simulations of arterial flows, Konstantinos Lyras, King's College London

11:48 (06C-5) A family of non-monotonic toral mixing maps, Joe Myers Hill, University of Leeds

Parallel sessions 7: Thur 9th Sept 13:45-14:45

Session 7A Aerodynamics/boundary layers (Chair: Sandham)

13:45 (07A-1) Transpiration cooling of hypersonic flow past a flat plate with porous injection, Pushpender K. Sharma, University of Southampton

13:57 (07A-2) Direct numerical solution of compressible flow over rough surfaces, Raynold Tan, University of Southampton

14:09 (07A-3) Design and characterisation of a gust generator for aeroelastic wind tunnel testing, Davide Balatti, Swansea University

14:21 (07A-4) Travelling waves in the asymptotic suction boundary layer, Matthias Engel, University of Edinburgh

14:33 (07A-5) Comparison of Lagrangian curvature und torsion statistics in open and wall-bounded turbulent flows, Yasmin Hengster, University of Edinburgh

Session 7B Fluid-structure interactions (Chair: He)

13:45 (07B-1) Spatiotemporal dynamics of a vortex induced vibration system in presence of noise, M. S. Aswathy, IIT Madras

13:57 (07B-2) Numerical investigation of fluid-structure interaction of a surface mounted elastic plate in a cross flow, Meltem Cakir, Loughborough University

14:09 (07B-3) Energy harvesting potential of an Unimorph Flexible Beam in Uniform Flow, Rajanya Chatterjee, IIT Madras

14:21 (07B-4) Numerical investigation of pressure spaces in solving fluid-structure interactions using an ALE finite element method, Gregory Walton, University of Leeds

14:33 (07B-5) Computational study of vibration-based leak detection approach in 90-degree pipe elbow, Ahmed Abuhatira, University of Dundee

Session 7C Meshless/mesh-free methods for fluids (Chair: Rogers)

13:45 (07C-1) A pseudo-spectral high-order methodology for incompressible smoothed particle hydrodynamics, Georgios Fourtakas, University of Manchester

13:57 (07C-2) High Weissenberg number simulations with incompressible Smoothed Particle Hydrodynamics and the log-conformation formulation, Jack King, University of Manchester

14:09 (07C-3) High-order simulations using the Local Anisotropic Basis Function Method, Jack King, University of Manchester

14:21 (07C-4) Dealing with density discontinuities in planetary SPH simulations, S. Ruiz-Bonilla, Durham University

Parallel sessions 8: Thur 9th Sept 15:00-16:00

Session 8A High speed 2 (Chair: Sandham)

15:00 (08A-1) Effect of flow parameters on transonic buffet at moderate Reynolds numbers, Pradeep Moise, University of Southampton

15:12 (08A-2) Effect of boundary layer transition on shock buffet instability, Wei He, University of Liverpool

15:24 (08A-3) Numerical and experimental analysis of transonic buffet for different airfoil geometries at moderate Reynolds numbers, Markus Zauner, ONERA

15:36 (08A-4) Practical resolvent analysis applied to shock buffet on the common research model elastic wing, Jelle Houtman, University of Liverpool

15:48 (08A-5) Shock train response to back pressure forcing, Alex Gillespie, University of Southampton

Session 8B Industrial flows (Chair: He)

15:00 (08B-1) Indirect noise in multi-component nozzle flows with dissipation, Animesh Jain, University of Cambridge

15:12 (08B-2) Data assimilation of thermoacoustics, Andrea Nóvoa, University of Cambridge

15:24 (08B-3) Diffusiophoresis: a mechanism to produce small-on-top stratified coatings, Clare R. Rees-Zimmerman, University of Cambridge

15:36 (08B-4) Optimising anaerobic baffled reactors performance through combined numerical and experimental analysis, Riza Inanda Siregar, University of Birmingham

15:48 (08B-5) Energy-saving transportation of liquid in a Bend Tube: an endo-anodizing approach, Honghao Zhou, Northumbria University

Session 8C Capillary flows/microfluids (Chair: Sykes)

15:00 (08C-1) Detachment of a tilting plate suspended by a capillary bridge, Matthew Butler, University of Birmingham

15:12 (08C-2) Capillary-scale solid rebounds: experiments, modelling and simulations, Carlos A. Galeano-Rios, University of Birmingham

15:24 (08C-3) Substrate curvature affects droplet splashing, Thomas C. Sykes, University of Oxford

15:36 (08C-4) Bifurcations and control of bubbles in Hele-Shaw cells, Alice Thompson, University of Manchester

15:48 (08C-5) Colloidal particle focusing and sorting in microchannels via solute concentration gradients, Adnan Chakra, Loughborough University

Parallel sessions 9: Thur 9th Sept 16:15-17:15

Session 9A Boundary layers (Chair: Krishna)

16:15 (09A-1) Analysis of wall mass transfer in a turbulent pipe flow using extended proper orthogonal decomposition, Rasmus Korslund Schlander, Imperial College London

16:27 (09A-2) On the effects of in-plane solidity on the different regimes of canopy flows, S. Nicholas, City University

16:39 (09A-3) Linearised predictions of secondary currents in turbulent channels with topographical heterogeneity, Gerardo Zampino, University of Southampton

16:51 (09A-4) Heat transfer in next generation Gyroid Heat Exchangers, I. Qureshi, University of Leeds

17:03 (09A-5) To tread or not to tread? Swathi Krishna, EPFL

Session 9B Industrial and biological flows (Chair: He)

16:15 (09B-1) Redistribution of a passive tracer in a porous substrate under a surface washing flow at high Péclet number, Emily Butler, University of Manchester

16:27 (09B-2) Optimising tool channel design to improve coolant distribution and reduce cutting temperatures in machining processes, Eleanor Harvey, University of Leeds

16:39 (09B-3) Collective dynamics of active filaments on spherical surfaces, Timothy A. Westwood, Imperial College London

16:51 (09B-4) Shape, shear and diffusion effects on bacterial orientation distributions near walls, Smitha Maretvadakethope, University of Liverpool

17:03 (09B-5) Combined effects of wall transpiration and plate movement on self-similar biomagnetic flow, Sadia Anjum Jumana, University of Dhaka

Session 9C Porous media/microfluids (Chair: Sykes)

16:15 (09C-1) Scalar and momentum transport across patches of cylinders, D.D. Wangsawijaya, University of Southampton

16:27 (09C-2) Cell-scale haemodynamics and transport in canonical disordered porous media: microfluidic experiments and theoretical models, Kerstin Schirrmann, University of Cambridge

16:39 (09C-3) Pore-scale large-eddy simulations of turbulent flow in a composite porous-fluid system, Mohammad Jadidi, University of Manchester

16:51 (09C-4) Shock-induced collapse of cavitating surface nanobubbles, Duncan Dockar, University of Edinburgh

17:03 (09C-5) Dynamics of interfacial peeling, Sepideh Khodaparast, University of Leeds

Parallel sessions 10: Fri 10th Sept 9:00-10:00

Session 10A Aerodynamics 2 (Chair: Papadakis)

9:00 (10A-1) Hydrodynamics of flow over a flat plate with generic roughness elements, Melike Kurt, University of Southampton

9:12 (10A-2) Eigenmode decomposition as a foundation for exploring high-efficiency swimming, Amanda Smyth, University of Oxford

9:24 (10A-3) Recirculation regions in wakes with base bleed, Konstantinos Steiros, Imperial College London

9:36 (10A-4) PIV and LES investigation of the flow around a succulent-inspired cylinder, Oleksandr Zhdanov, University of Glasgow

9:48 (10A-5) Turbulent entrainment in large wind farms, N. Bempedelis, Imperial College London

Session 10B Geophysical flows 3 (Chair: Busse)

9:00 (10B-1) PIV investigation of flow inside a rotating tangent cylinder, Rishav Agrawal, Coventry University

9:12 (10B-2) Volcanic plumbing systems and the dynamics of magma ascent, Caitlin Chalk, University of Liverpool

9:24 (10B-3) Bubble curtains used as barriers across horizontal density stratifications, Daria Frank, University of Cambridge

9:36 (10B-4) Quasi-static magneto-hydrodynamic convection in a rotating cylinder, Anthony Rouquier, Coventry University

Session 10C Droplets 2 (Chair: McHale)

9:00 (10C-1) Evaporation of sessile droplets on slippery liquid-like surfaces and slippery liquid-infused porous surfaces (SLIPS), Steven Armstrong, University of Edinburgh

9:12 (10C-2) Kinetics of breath figure templating on photocurable polymer substrates, Francis Dent, University of Leeds

9:24 (10C-3) Dielectrowetting of low dielectric films in high dielectric liquids, Andrew M. J. Edwards, Nottingham Trent University

9:36 (10C-4) Multifaceted design optimization for superomniphobic surfaces, Halim Kusumaatmaja, Durham University

9:48 (10C-5) Droplet wetting and self-propulsion on liquid surfaces, Glen McHale, University of Edinburgh

Parallel sessions 11: Fri 10th Sept 10:15-11:15

Session 11A Jets, wakes & turbulence (Chair: Papadakis)

10:15 (11A-1) Effects of porosity on the flow structure in the outer region of turbulent boundary layers, Prateek Jaiswal, University of Southampton

10:27 (11A-2) Vortical structures of stratified shear layers in an inclined duct, Xianyang Jiang, University of Cambridge

10:39 (11A-3) Experimental study of vortex ring impingement: three-dimensional flow field, wall shear stress and heat transfer, Qianhui Li, City University

10:51 (11A-4) Experimental properties of continuously-forced, shear-driven, stratified turbulence, Adrien Lefauve, University of Cambridge

11:03 (11A-5) Generalised Quasilinear approximation of a two-dimensional Kolmogorov flow exhibiting spatially localised turbulent states, Hannah Kreczak, Newcastle University

Session 11B Astrophysical flows (Chair: Busse)

10:15 (11B-1) Shear driven magneto-buoyancy under the influence of rotation, Craig Duguid, Newcastle University

10:27 (11B-2) A new residual distribution solver for galaxy formation simulations, Ben Morton, University of Edinburgh

10:39 (11B-3) Diffusion and dispersion in anisotropic magnetohydrodynamic turbulence, Jane Pratt, Georgia State University

10:51 (11B-4) Magnetic layering in the solar radiative zone, Fryderyk Wilczynski, University of Leeds

11:03 (11B-5) Non-linear simulations of tidal flows in an adiabatic convective shell, Aurélie Astoul, University of Leeds

Session 11C Droplets 3 (Chair: Kusumaatmaja)

10:15 (11C-1) Controlling the breakup of toroidal liquid films on solid surfaces, Glen McHale, University of Edinburgh

10:27 (11C-2) Dynamics of respiratory saliva droplets, Avshalom Offner, The University of Edinburgh

10:39 (11C-3) Robust interpolation schemes for dispersed particle flows using the Full Lagrangian Approach, Chris Stafford, University of Brighton

10:51 (11C-4) Nanobubble nucleation from acoustothermal physics, Saikat Datta, University of Edinburgh

Parallel sessions 12: Fri 10th Sept 11:30-12:30

Session 12A Aerodynamics 3 (Chair: Papadakis)

11:30 (12A-1) Efficacy of turbulence modeling techniques in capturing dynamic stall at low Reynolds numbers, Chandan Bose, University of Liège

11:42 (12A-2) Far-field acoustic investigation surrounding a separated aerofoil using time-resolved PIV, Douglas W. Carter, University of Southampton

11:54 (12A-3) Modelling rough and porous media with an adaptive mesh refinement Lattice Boltzmann Method, Mikaël Grondeau, University of Southampton

12:06 (12A-4) Effects of fractal-like multiscale roughness on turbulent boundary layers, T.Medjnoun, University of Southampton

Session 12B Industrial flows (Chair: Busse)

11:30 (12B-1) A combustion instability mechanism in rocket engines produced by orifice whistlin, Philipp Brokof, Imperial College London

11:42 (12B-2) Particle image velocimetry of a jet impacting a tightly packed tube bundle, Thomas Charpentier, Polytechnique Montréal

11:54 (12B-3) Stability, receptivity and sensitivity of mixed baroclinic convection in a cavity, Abhishek Kumar, Coventry University

Session 12C Multiphase flows 3 (Chair: McHale)

11:30 (12C-1) Mass transfer from small spheroids suspended in a turbulent fluid, J. M. Lawson, University of Southampton

11:42 (12C-2) A phase-field model for capillary bulldozing, Liam Morrow, University of Oxford

11:54 (12C-3) Boiling flow dynamics in non-isothermal microchannels with conjugate heat transfer, Federico Municchi, University of Nottingham

Poster presentations A

PA01 Towards optimization of a wave-to-wire energy device in a breakwater contraction, Jonathan Bolton, University of Leeds

PA02 Withdrawn

PA03 A Bayes factor comparison of the Brownian and Langevin models of passive particle transport, Martin Brolly, University of Edinburgh

PA04 Predicting the statistics of chaotic systems using resolvent based modelling, Thomas Burton, University of Southampton

PA05 Real-time parameter inference in reduced-order flame models with heteroscedastic Bayesian neural network ensembles, Maximilian L. Croci, University of Cambridge

PA06 Turbulent flow field reconstruction from sparse measurements, Francis De Voogt, University of Southampton

PA07 DNS of incompressible Rayleigh--Taylor instabilities at low and medium Atwood numbers, Arash Hamzehloo, Imperial College London

PA08 Withdrawn

PA09 Do ambient shear and thermal stratification impact wind turbine wake breakdown? Amy Hodgkin, Imperial College London

PA10 Simulating atmospheric boundary layer flow in a recirculating water tunnel, Desmond Lim, University of Southampton

PA11 Simulating surface wave dynamics with Convolutional Neural Networks, Mario Lino, Imperial College London

PA12 Machine learning modelling of transonic aerodynamic loads for aeroelastic analysis of an airfoil, David Massegur, University of Southampton

PA13 Turbulent channel flow over streamwise-aligned ribs, Mattias Nilsson, University of Southampton

PA14 Swirling vortex rings, Rigoberto Ortega-Chavez, Durham University

PA15 Sensitivity analysis of flood prediction model performance, Saba Rabab, Heriot-Watt University

PA16 Recurrent Neural Network based surrogate modeling of unsteady forces acting on a plunging airfoil, Rahul Sundar, IIT Madras

PA17 Unsteady flow development in particulate filter channels due to oblique flow entry, Callum Samuels, Coventry University

PA18 Scale-space energy transfer pathways in inhomogeneous compressible turbulence, S. Arun, Imperial College London

PA19 Weakly nonlinear theory for deterministic wave forecasting, Raphael Stuhlmeier, University of Plymouth

PA20 Three-dimensional modification of Gurney flap to improve the lift-type vertical axis wind turbine performance, Taurista P. Syawitri, University of The West of England

PA21 Data assimilation for RANS using time-averaged PIV of the flow around a NACA0012 airfoil, Craig Thompson, University of Southampton

PA22 Simulation of turbulent axisymmetric bluff body wake with the effect of pulsed jet forcing, Taihanh Zhu, Imperial College London

PA23 Modelling airway mucus: Rayleigh-Plateau instability of an annular viscoplastic liquid film, James Shemilt, University of Manchester

PA24 Left ventricular remodelling: Integrating flow imaging and machine learning into patient-specific models, Fergus Shone, University of Leeds

Poster presentations B

PB01 Simulation of behavioural modification effects in multiphase flows, Jacob Anderson, University of Leeds

PB02 Cavity collapse near porous plates, Elijah Andrews, University of Southampton

PB03 Two-phase flow in the dynamic Earth: Reactive-infiltration instabilities, Danielle Bullamore, University of Leeds

PB04 Generation of bubbles by spilling breaking wave groups, Konstantinos Chasapis, UCL

PB05 A robust microfluidic device for fabricating deformable microcapsules based on water-oil-water double-emulsion templates, Qi Chen, University of Manchester

PB06 Surface-washing of contaminated porous substrates, Francesco P. Contò, University of Cambridge

PB07 Melt percolation in solid rocks: A numerical study of pattern formation in rock-melt mixtures, Giulia Fedrizzi, University of Leeds

PB08 Effects of shell thickness on cross-helicity generation in convection-driven spherical dynamos, Parag Gupta, University of Glasgow

PB09 Lateral stress effects on bubbles in Hele-Shaw channels, Jacob Harris, University of Manchester

PB10 Viscoelastic fluid flow in microporous media, Victor Ibezim, University of Liverpool

PB11 Fluid dynamics around freely falling ice-particle crystals: An experimental investigation using three-dimensional particle tracking velocimetry, M. H. Khan, University of Reading

PB12 Flexible sheets in turbulent flow, Marin Lauber, University of Southampton

PB13 Robotic inspection of pre-filled medical syringes, Hamza Liaquet, University of Leeds

PB14 A finite volume coupled level set and volume of fluid method with a mass conservation step for simulating two-phase flows, Konstantinos Lyras, King's College London

PB15 Particle patterning inside glass capillary tubes using thin film surface acoustic wave devices, Sadaf Maramizonouz, Northumbria University

PB16 The creation of a photoelastic force balance, Bradley McLaughlin, University of Southampton

PB17 Inactivation effect of human thermal plume and upper-room ultraviolet air disinfection on the COVID-19 transmission, Shuo Mi, Queen Mary University of London

PB18 Asymptotic framework for flood models comparison, Piotr Morawiecki, University of Bath

PB19 African Easterly Wave precursors to tropical cyclogenesis in a convection-permitting model, Fran Morris, University of Leeds

PB20 Withdrawn

PB21 Pattern formation in polymer droplets spreading on a smooth substrate, Ahmed Othman, University of Cambridge

PB22 Jet tilt and the understanding of North Atlantic jet variability and regimes, Jacob Perez, University of Leeds

PB23 Withdrawn

PB24 Cusps-filaments at receding viscoelastic contact line, Saksham Sharma, University of Cambridge

Abstracts

I1

Sound, flames and aerodynamics: how they interact to generate thermoacoustic instability

Aimee Morgans

Imperial College

Inside the combustors of aero-engines, rocket engines, gas turbines and boilers, sound waves, flames and aerodynamics can interact to cause thermoacoustic instability. The result is damaging, large amplitude oscillations. Thermoacoustic instability remains the key barrier to reducing gas turbine NOx emissions; it occurs in the presence of carbon-free fuels like hydrogen and ammonia, and the design of futuristic aero-propulsion engines needs to be highly alert to it. The feedback between sound (acoustic) waves and unsteady heat release rate underpins all thermoacoustic instability mechanisms. For example, sound waves hit the flame, leading to flame front oscillations. These generate new sound waves which propagate away, eventually reflecting from combustor boundaries to arrive back at the flame, completing the feedback loop. This talk will explore the role of acoustic-flow interactions in this feedback mechanism, in particular in damping/amplifying acoustic energy at area changes and combustor boundaries, this being key to the final oscillation amplitude. Certain types of boundary give rise to unusual acoustic-flow interactions, which will be discussed in more detail.

Fluid mechanics for weather and climate research and operations

Stephen Belcher

Met Office

Weather forecasting and climate prediction are fundamentally problems in computational fluid mechanics, and large computational models are used to simulate the motion of the atmosphere and oceans. There has been a revolution in our capability developed over the last ten years whereby simulations are now sufficiently high resolution to be convection permitting. For weather forecasting, this provides more reliable forecasts of convective weather events particularly for summertime flooding. This methodology is now also being used for climate projections as evidenced by the recently released extension of UKCP18 including climate projections for the UK at this convection-permitting scale. However at this point, the computational expense is so large that we can only afford to do this for limited-area weather and climate simulations. There is great future promise in extending these to a global extent. In this presentation, the power of these developments will be reviewed, and possible future opportunities will be discussed.

Fluid-structure interaction with plants

Emmanuel de Langre

Ladhyx, Ecole Polytechnique

Plants mechanically interact with fluids that surround them. These interactions can bring significant deformation, as for trees under high winds or kelp under waves, but the variety of configurations is actually immense, from buoyancy effects to light breeze flutter of leaves, or wind-induced waves on crop fields. I will first explore the variety of motivations for developing quantitative models of fluid-structure interactions effects for plants, from forestry to plant reproduction and from animal behavior to photosynthesis. Some of the specificities of plants in comparison with man-made structures, in terms of architecture and flexibility will be discussed. I will then focus on some recent results on three problems. The first involves the very general mechanism of drag reduction by flexibility in plants: with the use of large deformations plants under strong flow have a drag which is more linear than quadratic with the flow velocity. The second is the dynamics of a whole foliage under light wind: I will show that foliage motion is a combination of individual leaf flutter and branch motion. Finally, the use of artificial excitation by air as a tool for high-throughput phenotyping of plants will be discussed. Some perspective on bioinspiration in that domain will also be given.

Flow instabilities in the cross slot geometry: characterisation, exploitation and cure

Rob Poole

University of Liverpool

The so-called "cross-slot" geometry, created by two channels mutually intersecting at 90 degrees to each other, has been well studied due to its potential use as a rheometric device. Although in this regard it has been used to study elastic fluids or suspended particles/vesicles, or indeed individual DNA molecules, in extensional flow for a long time, it has only been relatively recently shown that the flow becomes unstable beyond a critical flowrate. Even for purely Newtonian fluids the base flow has been shown to undergo a steady symmetry-breaking bifurcation. In the 2D limit this has been shown numerically to occur at a Reynolds number (Re) on the order of 1500 and result in a lateral displacement of the stagnation point. In contrast, the imposition of side-walls has shown that a different form of symmetry breaking can be observed at Reynolds numbers as low as 30 for aspect ratios (ratio of channel height: width) close to one. We show how this purely-inertial instability can be used to enhance heat transfer at the microscale. The effect of weak-elasticity on this inertial instability, through polymer addition to the base solution making the flow inertio-elastic, is shown to be destabilising. In the absence of inertia, a still different form of steady symmetry-breaking is observed for viscoelastic fluids due to a purely-elastic instability. This purely-elastic instability, which remains in the 2D limit, has been studied in detail and we show how the channel aspect ratio can be used to change the instability from a steady symmetry-breaking bifurcation into a time dependent instability as the channel height is reduced towards the Hele Shaw limit. We also show how different viscoelastic model parameters effect the onset conditions for these instabilities. In contrast to the purely-inertial instability, where the addition of elasticity was found to be destabilising, in the purely-elastic case the addition of weak inertia is shown to be stabilising (elastic-inertio). Finally we show how a passive control device (a circular cylinder added at the geometric centre of the device) can be used in the cross-slot to delay the onset flowrate for the purely-elastic instability to significantly higher flowrates. This geometric configuration also provides insight into the underlying physical mechanism responsible for the instability.

Near-wake flow features of polygonal cylinders in different incidence angles

Esmaeel Masoudi, Lian Gan, David Sims-Williams

Department of Engineering, Durham university, Durham, 3H1 3LE, UK

In this study, large eddy simulation (LES) is used to simulate flow of incidence angle α around polygonal cylinders of side number N=5-8 at Reynolds number Re=10,000. Six incidence angles are studied on each cylinder, which covers the entire α spectrum from face to corner orientation. Time-mean aerodynamic forces including lift, drag and vortex shedding frequencies as well as the near wake flow features are discussed in detail. It is found that because of the intrinsic quasi-asymmetric shape of polygons in non-principle orientations, flow separation characteristics and hence the induced aerodynamic forces exhibit unique and complex dependence on both alpha and N. Even though the canonical inverse relation of drag coefficient and Strouhal number, which was previously proposed from experimental observations, still holds at arbitrary alpha, the variation of the two is found to be non-monotonic on both alpha and N. Furthermore, mean shear layer length measured from the cylinder centre is found to be a powerful scaling factor for all the quantities investigated, including lift and drag coefficients, Strouhal number and pressure coefficient. Furthermore, a flapping behavior is observed in the separated shear layer which could be responsible for flow reattachment that enforces double separation (instead of single separation) from top or bottom of the cylinders. This flapping behaviour found to be an essential factor that determines highest mean shear layer length in asymmetric incidence angles, which contributes to lowest drag and/or highest frequency of the vortex shedding.

Attenuation of the unsteady loading on a high-rise building for different wind angles using feedback control

Xiao Hu and Aimee S. Morgans

Department of Mechanical Engineering, Imperial College London, SW7 2AZ, UK

Unsteady wind loading on high-rise buildings has the potential to deteriorate their structural performance in terms of serviceability, habitability and occupant comfort. This work numerically investigates the flow features around a canonical high-rise building - the Commonwealth Advisory Aeronautical Council (CAARC) standard high-rise building - in an atmospheric boundary layer, for two different oncoming wind angles 20 deg and 45 deg, using Large Eddy Simulation (LES). The mean and unsteady forces and moments on the building at these two different angles are compared to experimental results available in the literature. Modal decomposition methods are applied to analyze coherent flow structures in the wake, and the influence of these structures on the unsteady wind loading is studied. Feedback control is then implemented to attempt to attenuate the unsteady loading. Actuation in the form of unsteady synthetic jets along the edges of the building is implemented and sensor signals which consist of the unsteady pressure force on one or more of building surfaces are used. The choice of sensor signals and the placement of actuation are numerically studied. The responses of different sensor signals to actuation are characterised using system identification. Linear feedback control strategies are then designed to alleviate the unsteady loading with the designed feedback controllers are implemented in simulations, and they are found to successfully attenuate the unsteady loading.

Simulation of high Reynolds number sub-critical flow over a cylinder using a scale resolving hybrid RANS/LES closure model

Michael Mays (1), Sylvain Laizet (1) and Sylvain Lardeau (2)

(1) Department of Aeronautics, Imperial College, London, UK (2) Siemens Industry Software GmbH, Nurnberg, Germany

The simulation of flow over a cylinder at Reynolds number 140,000 serves as an excellent test of a turbulence closure models ability to simulate highly separated turbulent flows. Accurate prediction of the shear layer turbulent transition in close proximity to the wall is especially sensitive to closure model behaviour. This can be a particular challenge for hybrid RANS-LES methods which often suffer from grey area effects in the near-wall region. In this study a variety of closure models is investigated, with varying levels of details, using the commercial software Star-CCM+, to simulate this canonical case. Of particular focus is a novel Scale Resolving Hybrid (SRH) model derived from a time averaging of the RANS equations. Investigation of the significance of resolution, time step and convective scheme in achieving optimal behaviour is considered in this numerical work. The simulations are judged by their ability to predict experimentally determined time-averaged drag, mean pressure coefficient distributions and wake profiles of velocity and Reynolds stresses, in addition to the correct breakdown behaviour in the shear layers. Results from the SRH model are promising, showing significantly improved agreement with experiment over RANS models at similar resolution and similar results to the WALES model at much higher resolution. The SRH model, therefore, provides a significant reduction in computational resource compared to LES methods, though the required time-averaging over many vortex cycles means the computation remains demanding. Adapting the model parameters and shielding to optimise the transition between the RANS and LES behaviours of the hybrid model will be discussed as well as potential for application to more complex geometries and industrial flows, such as the DrivAer model.

Experimental modelling a buoyant pollutant source behind a backwards-facing step

Samuel Charlwood, Megan Davies Wykes

University of Cambridge, Engineering department

An experimental model of flow over a backwards-facing step with a line 'pollutant' source at the base has been constructed and tested. Pollutant concentration profiles have been collected via dye attenuation for various exhaust buoyancies spanning from a regime of neutral buoyancy to buoyancy being the dominant force within the wake. The backwards-facing step has applicability as a stepping stone to determining the influence of buoyancy on the wakes of other more complicated geometries whilst also having direct relevance to physical scenarios such as road pollution along terrace housing and the dispersal of smoke ejected from buildings

The dynamics of a line plume confined by a single inclined boundary

Tom Newton & Gary R. Hunt

Department of Engineering, University of Cambridge

A buoyant turbulent line plume developing from a source on a horizontal plane boundary in an unconfined environment will rise vertically, spreading linearly with height as it entrains fluid from the surroundings. If the plume instead develops from a source on an inclined plane, it might be anticipated that the flow behaviour will remain largely unaffected unless the inclination of the plane is sufficiently steep that the plume perimeter impinges on the boundary. We will show, however, that the presence of such a boundary dramatically alters the flow dynamics at much shallower inclinations. Indeed, in this work two novel flow configurations with non-vertical plume trajectories have been identified, which thereby enable the parameter space of boundary inclination to be divided into three distinct regimes of plume behaviour. Our study consists of an experimental investigation utilising aqueous saline plumes, that makes use of both light-induced fluorescence flow visualisation and particle-image velocimetry (PIV) techniques. The PIV measurements we collect include the velocity field in the ambient environment, and this has offered us insights into how the presence of the boundary modifies the entrainment flow into the plume. In turn, such insights have allowed us to begin clarifying the physical mechanisms underlying the observed flow behaviours. The findings of this work have numerous potential applications across the engineered, and natural, world - for example in understanding the spread of a wildfire front on sloped terrain or designing a natural ventilation scheme for a steeply-raked lecture theatre.

Channelized flow in the partially molten upper mantle

David W Rees Jones (1), Hanwen Zhang (2) and Richard F Katz (2).

(1) University of St Andrews (2) University of Oxford

Melting of the upper mantle generates magma that rises buoyantly and erupts, for example, at mid-ocean ridges. From a dynamical point of view, the upper mantle is a compacting porous medium with material properties (permeability, shear and bulk viscosity) that depend on porosity. Various lines of evidence point to focused, channelized magma flow through this system. Here, we discuss the two main candidate instability mechanisms: (1) a chemical reaction-infiltration instability; (2) a shear-driven instability in a porosity-weakening medium. We present a combined linear analysis of the two instabilities and show that the combination of these mechanisms favours tabular porosity channels, consistent with geological observations. We derive a dimensionless parameter that governs the relative importance of the two mechanisms. Finally, we calculate the evolution of packets of porosity channels through the mid-ocean ridge system, assuming a separation of scales between the small-scale channels and the geological scale of the partially-molten region beneath the ridge. We show that reactive-channelization is the dominant mechanism through most of the system and discuss the geophysical implications. Reference: D W Rees Jones, H Zhang, R F Katz, Magmatic channelization by reactive and shear-driven instabilities at mid-ocean ridges: a combined analysis, Geophysical Journal International, Volume 226, Issue 1, July 2021, Pages 582–609, doi:10.1093/gji/ggab112

Poroviscoelastic dynamics of mushy magmatic systems

Jennifer Castelino, Susanna Ebmeier, Oliver Harlen, Sam Pegler

University of Leeds

More than one in ten of the world's population live within 100km of an active volcano and are exposed to threats to life and livelihood from eruptions. Understanding the mechanisms by which magma flows through the Earth's crust is of fundamental importance to volcanology, and to forecasting volcanic activity. Historically in volcanology, it was thought that a magmatic system consisted of a simple spherical liquid chamber of magma, surrounded by host rock. However, recent evidence suggests that large volumes of melt are disseminated in crystal mush regions, leading to large trans-crustal mushy-magmatic systems. The very presence of a crystal mush has many implications for the characteristics of surface displacements caused by magma movements, such as during an intrusion or eruption. A rich variety of solid- and fluid-mechanical processes may operate including flow through interconnected conduits, and structural deformation due to poroelastic and viscoelastic processes. Currently, there are no models that account for the behaviour of mush on a poroviscoelastic spectrum. We expand on existing poroelastic and viscoelastic models to produce an encompassing poroviscoelastic model that can predict the characteristics of measurable deformation at the Earth's surface. In order to do this, we first present a generalised model that describes the behaviour of a poroviscoelastic material. We then adapt this model to a relevant geometry to provide tools for inferring structural components of a given system and to provide insight into deformation signals produced by these systems.

Investigation into the folding of layered viscous structures

Olivia Goulden, Samuel Pegler, Sandra Piazolo, Oliver Harlen

University of Leeds

Viscous structures comprising layers of different viscosities arise widely throughout nature, with important examples including the Earth's ice sheets and lower crust. It is well documented that such structures can deform to produce folds and buckles when subject to compressive stresses. Observations of exposed rock and radar data of ice sheets have both indicated the widespread existence of large-scale folds and small-scale wrinkles of immersed layers. The relationships between the properties of folding and the properties of the material is key to the geological technique of inferring the properties of the lower mantle from the geometry of folds, and is thus of central importance in structural geology. The classical analysis of the problem of folding has focused on the case of a strongly viscous or solid beam compressed or extended laterally in an infinite domain. Motivated particularly by the geometry of an ice sheet, this work addresses new effects introduced by the presence of a horizontal rigid boundary, which is found to introduce new effects that can readily dominate the mechanics of folding.

Bounds for internally heated convection with fixed boundary heat flux

Ali Arslan (1), Giovanni Fantuzzi (1), John Craske (2) and Andrew Wynn (1)

(1) Department of Aeronautics, Imperial College London, (2) Department of Civil and Environmental Engineering, Imperial College London

Rigorously bounding emergent properties of turbulent flows provides a means for improving our mathematical understanding of turbulence, while yielding results with real world applications. Working in the framework of the auxiliary functional method and by utilising tools from convex optimisation, we prove bounds on the mean convective heat transport $< wT >$ in internally heated (IH) convection, with an insulating lower boundary and an upper boundary with fixed heat flux. Bounds for Rayleigh-Bénard convection have been extensively studied, yet an extension of the same analysis to IH convection is not complete. The change in mechanism driving convection presents a unique problem, which is of importance in geophysical and astrophysical applications such as convection in the mantle or the Venusian atmosphere. Bounds that depend explicitly on the Rayleigh (Ra) number have not yet been proven in IH convection. This talk will demonstrate that such Ra-dependent bounds on $< wT >$ can be obtained using quadratic auxiliary functions. Numerical optimisation suggests that the asymptotic value of the bound and the -1/3 exponent on the Ra scaling are optimal within our bounding framework.

Massively parallel Lattice Boltzmann Method models of dilute gravity currents

D. Adekanye (1), A. Khan (2), A. Burns (3), W. McCaffrey (4), M. Geier (5), M. Schönherr (5) , R. Dorrell (6)

(1) EPSRC Centre for Doctoral Training in Fluid Dynamics, University of Leeds, Leeds, LS2 9JT United Kingdom (2) Institute for Public Health and Environmental Engineering, University of Leeds, Leeds, United Kingdom (3) School of Chemical and Process Engineering, University of Leeds, Leeds, LS2 9JT, United Kingdom (4) School of Earth and Environment, University of Leeds, Leeds, UK, LS2 9JT, United Kingdom (5) Institute for Computational Modelling in Civil Engineering, TU Braunschweig, Pockelsstr. 3, 38106 Brunswick, Germany (6) Energy and Environment Institute, University of Hull, Hull HU6 7RX, United Kingdom

The computational expense of conventional depth-resolving numerical models limits the insights that can be gained into the dynamics of dilute gravity currents, as it becomes prohibitively expensive to extend the models to incorporate more of the full complexity of real-world environmental flows. Lattice Boltzmann method models offer a novel framework for the simulation of high Reynolds number gravity currents. The numerical algorithm is well suited to acceleration via implementation on massively parallel architectures, such as graphics processing units. Here we present two lattice Boltzmann method models of lock-exchange dilute gravity currents, in which the largest turbulent length scales are directly resolved. The three-dimensional simulations are accelerated by exporting computations to a graphics processing unit and are validated against experiments and high-resolution simulations for buoyancy Reynolds numbers up to 30,000. The lattice Boltzmann method models achieve equivalent accuracy to conventional large eddy simulation models in the prediction of key flow properties such as the front velocity within each phase of current propagation, and shear stress on the lower boundary of the channel. A conservative analysis of computational performance relative to conventional methods indicates that the presented framework reduces simulation times by at least an order of magnitude. Therefore, it can be used as a foundation for the development of depth-resolving models that capture more of the complexity of environmental gravity currents.

Rim breakup dynamics in aerodynamic droplet breakup

Isaac M. Jackiw and Nasser Ashgriz

Department of Mechanical and Industrial Engineering, University of Toronto, Toronto, ON M5S 3G8, Canada

The present work investigates the breakup mechanisms of rims formed in the late stages of low Weber number aerodynamic droplet breakup. High-speed shadowgraph videos of droplet breakup at side and end-on viewing angles are used to visualize and quantify the breakup of the rims. The rims formed in droplet breakup are known to fragment bimodally, with a large size resulting from large nodes that form around the rim and a smaller size resulting from the breakup of the thinned regions between the nodes; however, few studies have investigated the mechanisms leading to this behaviour. In this study, both the Rayleigh-Taylor and Rayleigh-Plateau instability theories are investigated as potential mechanisms of node formation and rim breakup to elucidate which mechanism is dominant. The theories are compared to the present experiments and to the literature in their ability to predict the number of large nodes formed and to estimate the two characteristic breakup sizes of the rim. The node and rim sizes are related geometrically to formulate a model to predict the relative breakup volumes of each mode using the instability theories. The suitability of each instability theory to model the breakup is discussed in terms of the appropriateness of the physics to the process, the efficacy of the required assumptions, and the sensitivity of the result to variations in the assumptions.

Atomisation of a capillary jet using a stacked-type electrofluidic actuator

S. Lalloz (1,2,3), L. Davoust (1)

(1) SIMAP - Science et Ingénierie des MAtériaux et Procédés, 1130 rue de la Piscine, Saint Martin d'Hères, France (2) Center for Fluids and Complex Systems, University of Coventry, Priory Street, Coventry CV1, UK (3) National Laboratory for High Magnetic Fields, 25 avenue des Martyrs, BP 166, 38042 Grenoble CEDEX 9, France

An alternative atomisation approach is proposed which differs significantly from usual piezoelectric1 or electrostatic techniques2. The objective consists in imposing a standing wave of electrostatic pressure along the surface of a capillary jet so as to deform it in a way as close as possible from most unstable RP modes. To this end, a stacked-type electrofluidic actuator is developed which generates a harmonic distribution of electrostatic pressure and stimulates the atomisation of a liquid jet. Apart from a theoretical study available in the literature3, this approach was surprisingly never investigated in full depth. To support our physical interpretations, some theoretical ingredients are given, based on a (linear) stability model ultimately invoked to calculate the breakup length of the capillary jet. After a presentation of the experimental device, a comparison with the stability model is carried out in order to clarify the link between the spatial periodicity of the electrode stack, the imposed frequency, the breakup length and the size distribution of the ejected drops which are obtained from fast imaging.

Numerical modelling of personal care products' spray efficiency

Liam Robert Gray, Nikil Kapur, Amirul Khan, Andrew Bayly

University of Leeds

The use of aerosols is a daily occurrence for many people using personal care products. Spray cans are one of the primary tools used as delivery vehicles for such products. The use of spray cans also extends to many other products, such as paints and lubricants. This is because they provide a convenient way of storing and delivering the material. Understanding the fate of droplets both during spraying and in impinging onto the target surface is important for quantifying the efficiency of droplet capture at the surface. The spray generation procedure is underpinned by complex physics, from the primary atomization of the contents, to the turbulent transport and impingement of the generated droplets. Generating a validated numerical model for the carrier jet is crucial for both calculating droplet trajectory and predicting aerosol cloud development. Here we present a computational fluid dynamics model that links the physics of jet formation with droplet behaviour within the spray. This extends to the motion of droplets around surfaces. The results show that the effect of standoff distance and surface topology play an important role in the ultimate capture of droplets. A description of the numerical approach used to characterise this behaviour is given, together with results of a parametric study.

Mathematical modelling and analysis of the evolution of and deposition from a sessile evaporating droplet

Hannah-May D'Ambrosio(1),Stephen K. Wilson(1), Brian R. Duffy(1), Alexander W. Wray(1),Teresa Colosimo(2), Lisong Yang(2), Colin D. Bain(2), and Daniel E. Walker(3)

(1) Department of Mathematics and Statistics, University of Strathclyde, 26 Richmond Street, Glasgow G1 1XH, United Kingdom (2) Department of Chemistry, Durham University, Lower Mountjoy, Stockton Road, Durham DH1 3LE, United Kingdom (3) Merck Chemicals Ltd, University Parkway, Southampton S016 7QD, United Kingdom

The evaporation of sessile droplets occurs in numerous physical contexts, with applications in nature, industry, and biology, including nano-fabrication, chemical decontamination, and ink-jet printing. As a consequence of the wide variety of everyday applications, the evolution of and deposition from an evaporating droplet has been subject to extensive investigation in recent years, with particular interest in different evaporation modes, the prediction of lifetimes, and the ring-like deposit (the “coffee-ring”) that often forms at the contact line of a pinned evaporating droplet. In this talk we review some recent developments in the study of evaporating droplets. We first formulate and analyse a mathematical model for the evolution and lifetime of an evaporating droplet in a well of rather general shape and validate the model by comparing the theoretical predictions with experimental results for the special case of cylindrical wells. We then investigate the effect of the spatial variation in the evaporative flux on the deposition from an evaporating droplet, determining the flow velocity, concentration of particles within the droplet, and the evolution of the deposit for a one-parameter family of evaporative fluxes. If time permits, we will also discuss very recent work exploring the effect of gravity and particle adsorption on the deposition process.

Governing equations and solution multiplicities for a static ridge of nematic liquid crystal

J. R. L. Cousins (1,2), B. R. Duffy (1), L. T. Corson (1), S. K. Wilson (1), N. J. Mottram (2)

(1) Department of Mathematics and Statistics, University of Strathclyde, Glasgow, United Kingdom, (2) School of Mathematics and Statistics, University of Glasgow, Glasgow, United Kingdom

Technological interest in nematic liquid crystal (nematics) thin films and nematic droplets includes their applications for liquid crystal display manufacturing [1,2] and new emerging technologies, such as microfluidics [3] and microelectronics [4]. Many of these complicated multiphase systems involve interfaces between the nematic, a solid substrate, and a passive gas and include nematic–solid–gas three-phase contact lines. Theoretical studies of these systems often use well-established theories of Newtonian droplets and films [5], which fail to account for the non-Newtonian nature of nematics. Motivated by a desire to increase understanding of nematic droplets and films, we use constrained energy minimisation of the nematic Oseen–Frank and Rapini–Papoular energy densities [1] to formulate and analyse the full governing equations for a static sessile two-dimensional ridge of nematic. We then consider the special case of a thin symmetric ridge, for which we obtain insight into the multiplicity of solutions for the free surface height and the nematic molecular orientation. Our findings allow us to construct parameter planes in terms of the nematic material parameters, which describe the wetting behaviour, the molecular orientation at the three-phase contact line and the possibility of multiple solutions for the free surface height. [1] I. W. Stewart, The Static and Dynamic Continuum Theory of Liquid Crystals, Taylor & Francis, 2004, [2] J. R. L. Cousins, S. K. Wilson, N. J. Mottram, D. Wilkes, L. Weegels, Phys. Rev. E. 102 (2020), 062703, [3] A. Sengupta, U. Tkalec, M. Ravnik, J. M. Yeomans, C. Bahr, S. Herminghaus, Phys. Rev. Lett. 110 (2013), 048303, [4] C. Zou, J. Wang, M. Wang, Y. Wu, K. Gu, Z. Shen, G. Xiong, H. Yang, L. Jiang, T. Ikeda, Small 14 (2018), e1800557, [5] P. G. de Gennes, F. Brochard-Wyart, D. Quéré, Capillarity and Wetting Phenomena, Springer, 2004

Immersed boundary simulations on the effect of roughness: Flow over a flat plate

Jonathan Massey, Melike Kurt, Rodrigo Vilumbrales-Garcia, Gabriel Weymouth, Bharathram Ganapathisubramani

University of Southampton

Sharkskin has impressive flow control properties, reducing friction drag and increasing their swimming e ciency. Recent studies have focused on using these properties to increase the e ciency of aerofoils. Restraining the energy output of our engineering systems is especially important in light of a report from World Energy Outlook IEA (2020) pronouncing the need to reduce our global energy output by 17% by 2030. We present initial work in understanding the role of roughness in turbulent, separated flows. Firstly, we simulate a foil at varying angles of attack and Re_c =120,000 using the in house developed boundary density immersion method Weymouth and Yue (2011). Comparison to an experiment using Particle Image Velocimetry (PIV) measurements allowed us to validate these. Next, we added hemispherical elements in a homogeneous arrangement to simulate the roughness on the flat plate surface under the same conditions. Finally, we were able to look at the flow physics and separation dynamics from the flat plate with a smooth, then rough surface. IEA. No Title. Technical report, World Energy Outlook 2020, Paris, 2020. URL https://www.iea.org/reports/world-energy-outlook-2020. G. D. Weymouth and Dick K.P. Yue. Boundary data immersion method for Cartesian-grid simulations of fluid-body interaction problems. Journal of Computational Physics, 230(16):6233{6247, 2011. ISSN 10902716. doi: 10.1016/j.jcp.2011.04.022. URL http://dx.doi.org/10.1016/j.jcp.2011.04.022.

Effects of surface roughness in flapping foil performance

R. Vilumbrales Garcia, M. Kurt, G. Weymouth, B. Ganapathisubramani

University of Southampton

Flapping-foil systems have been found to improve steady configurations in terms of propulsive efficiency and/or energy extraction. While the mentioned performance increase has been widely documented in past studies, other aspects such as the addition of surface roughness are not well understood. This could lead to new applications in the field of small underwater vehicles or in biomimetics developments such as the well-known use of 'shark-like' skins for drag reduction. In this study, three flat-plates with different surface roughness have been built using hemispherical bumpers, starting from zero rough, to mid-rough (35% coverage with the rough material), finishing with a rough foil (70% coverage). Experiments have been conducted in a water flume, under sinusoidal pure heaving motion and at a fixed St of 0.25, with a Re range of 15k-40k. The effects of the surface roughness on the flapping-foil performance – both efficiency and propulsive force – have been recorded in terms of forces and moments. PIV acquisition has been carried out to link together any potential force benefit to the flow characteristics. Preliminary results show that the roughness coverage can provide a performance augmentation at low Reynolds numbers while, for higher Re values, this benefit is diminished.

Designing optimal roughness: A theory-led approach to roughness-based drag reduction

P.Y.A.G.S. Yapa (1), S.J. Garrett (2), P.T. Griffiths (3) and P.J. Thomas (1)

(1) Fluid Dynamics Research Centre, School of Engineering, University of Warwick, Coventry, CV4 7Al, UK. (2) School of Engineering, University of Leicester, Leicester, LE1 7RH, UK. (3) School of Computing, Electronics and Mathematics, Faculty of Engineering, Environment and Computing, Coventry University, Coventry, CV1 5FB, UK.

A numerical study investigating the effects of both isotropic and anisotropic surface roughness on the convective stability of the von Kármán boundary-layer flow over a rough rotating disk is described. Surface roughness is modelled via the imposition of the partial-slip approach. The effect of the partial-slip parameters on the neutral characteristics for convective instability is presented. Linear stability analysis is performed to identify how roughness affects the stability characteristics of the inviscid Type I (or cross-flow) instability and the viscous Type II instability that arise in the rotating disk boundary layer. Stationary modes are studied and both anisotropic (concentric grooves and radial grooves) and isotropic roughness are shown to have a stabilizing effect on the Type I instability, where the results of Cooper et al. (2015, Physics of Fluids, 27(1):014107) are confirmed. We then show that a disk with radial grooves has a strongly destabilizing effect, whereas a disk with concentric grooves and isotropic roughness have a stabilizing effect for the Type II instability. The point of inflection in the mean flow profile is considered to be the direct cause of the cross-flow instability, and any changes in its location could affect the behaviour of the instability in turn. Hence an analysis of the variation of the spatial location of the point of inflection on the radial velocity profile is presented. In order to reconfirm the results of the linear stability analysis, drag or moment coefficients were calculated as a surface integral of shear stress. When the surface-roughness parameters increases drag reduction can be seen for a disk with radial grooves and isotropic roughness, whereas a disk with concentric grooves shows drag increase.

Using flow control to create ideal platooning conditions for road vehicles

Craig Nolan, Angela Busse, Kostas Kontis

James Watt School of Engineering, University of Glasgow

Road transport contributes about a fifth to the UK's greenhouse gas emissions. Whilst total greenhouse gas emissions dropped by 32% from 1990 to 2017, emissions associated with road transport rose by 6% over the same period [1]. The development of road vehicles of increasing autonomy provides new opportunities to reduce greenhouse gas emissions by exploiting the potential for cooperation between road vehicles. One such approach would be the widespread implementation of vehicle platooning on motorways. Platooning at short inter-vehicle spacing typically provides significant drag reductions for the rear vehicles that trail in the wake of lead vehicle due to the slipstreaming effect. However, platooning is not always beneficial for all vehicles in the platoon, for example, for some vehicle geometries a drag increase has been observed for the trailing vehicle over a range of spacings (see e.g. [2]). An ideal platoon would not only reduce the average drag for the platoon but would also provide drag reduction for each individual member of the platoon. In the current project, two-body platoons composed of simplified ground vehicle models are used to investigate the effectiveness of different flow control methods using both computational and experimental approaches. The aim is to eliminate adverse platooning scenarios and to optimise the wake of the lead vehicle to attain ideal platooning conditions for maximum drag reduction. [1] Office for National Statistics (2019), Road transport and air emissions https://www.ons.gov.uk/economy/environmentalaccounts/articles/roadtransportandairemissions/2019-09-16 [2] Watkins & Vino (2008), Journal Wind Engineering and Industrial Aerodynamics 96, 1232–1239.

New insight into the increased turbulence due to non-uniform streamwise body forces in pipe flow

Stephen Jackson, Shuisheng He

University of Sheffield

When a vertical turbulent pipe flow is subjected to an opposing non-uniform body force and the flow rate is maintained, it is well known that turbulence is increased. This is the dominant mechanism underpinning certain physical phenomena, for example the turbulent mixed convection in a heated downward flow influenced by buoyancy. We study the mechanisms of turbulence enhancement by defining a set of linear and step change idealised body forces to represent the buoyancy forces for subcritical and supercritical fluids respectively. Direct numerical simulations are then used to systematically vary the amplitude and coverage of the body force profiles acting against the main flow direction, and an overview of the resulting changes to turbulent characteristics is given. By using an unconventional equal pressure gradient framework and extending previous theory for body force aided flows, it is shown that, unlike under the equal flow rate framework, some key turbulence characteristics is mostly unchanged by the additional body force. Crucially the turbulent eddy viscosity and wall-normal and spanwise fluctuations are not changed which are significant in explaining the changes to heat transfer. This allows the principal turbulence characteristics of flows which were previously difficult to predict, to be easily predicted using an equal pressure gradient reference flow.

A new Lagrangian drift mechanism in coastal oceans: role of current and seabed undulation interactions

Akanksha Gupta (1),(2) and Anirban Guha (1)

(1) School of Science and Engineering, University of Dundee, DD1 4HN, U.K., (2) Department of Mechanical Engineering, Indian Institute of Technology, Kanpur, U.P. 208016, India

Surface gravity wave causes floating particles to undergo a slow drift in the direction of wave propagation. This forward drift, commonly known as the Stokes drift, plays a crucial role in transporting particles, from sediments to pollutants, in the marine ecosystem. In the classical analysis, the effect of mean current and seabed undulations are not considered while calculating the particle drift. In the nearshore region (shallow water region), the drift is non-trivially affected by the alongshore current and seabed undulations (topography). We theoretically show that the combined effect of the background current and seabed undulations introduces a time-independent (particular) solution, leading to additional terms in the governing pathline equations. This additional term breaks the inherent approximation used in the derivation of Stokes drift; hence we refer to the ensuing drift simply as a Lagrangian drift. We consider the order-wise analysis of the nonlinear effect of the surface wave and seabed undulations. Next, we numerically simulate wave-current-topography interactions in a 3D setting and compute the Lagrangian drift using the High-order Spectral method. We find that the resulting drift in the presence and absence of seabed undulations have significant differences. We consider parametric analysis to understand the role of the background current and the amplitude of bottom topography on the Lagrangian drift, and hence particle motion. Topographic interactions are inevitable in the nearshore region, and our study reveals that seabed corrugations can significantly affect the cross-shelf exchange of microplastics and other nearshore tracers like pathogens, contaminants, nutrients, larvae, and sediments.

Internal wave triads in a bounded domain with mild-slope bathymetry

Saranraj Gururaj & Anirban Guha

School of Science and Engineering, University of Dundee, DD1 4HN, Scotland

In oceans, internal gravity waves are often produced when the stably stratified ocean water is driven back and forth over submarine topography by tidal currents. Internal gravity waves can move through the ocean for very large distances (O(1000)km). Understanding the cascade of energy from the large-scale internal waves to small scales helps in predicting/quantifying deep ocean mixing. Weakly nonlinear wave-wave interaction is one of the mechanisms through which energy in large length scale internal waves, which travels in varying water depth, cascades to small scales that eventually leads to mixing and turbulence. Using multiple scale analysis, amplitude evolution equations for the waves constituting a triad as well as a self-interacting system are derived in the presence of slowly varying bathymetry, assuming the waves have a slowly varying amplitude and a rapidly varying phase in space and time. In the presence of a uniform stratification, when the constituent waves in a triad interact in a medium of varying water depth, the horizontal wavenumber condition of the waves in the triad is unaffected. The situation in non-uniform stratification is more complex, and is described as follows. Triads (and self-interaction systems) which consist of waves with different mode numbers may not satisfy the horizontal wavenumber condition as the water depth varies. Moreover, the non-linear coupling coefficients (NLC) do not decrease (increase) monotonically with increasing (decreasing) water depth. The NLC may also change by one order of magnitude for relatively small changes in the water depth, resulting in growth rates that are sensitive to the water depth. Additionally, the most unstable daughter wave combination for a given parent wave is also found to change with relatively small changes in the water depth.

The interplay of symmetric, inertial, and baroclinic instabilities on the evolution of oceanic fronts

Aaron Wienkers & John R. Taylor

DAMTP, University of Cambridge

Isolated fronts with large lateral density gradients in geostrophic and hydrostatic balance are common in the upper ocean. Such strong fronts may be the result of baroclinic frontogenesis or of sharp freshwater interfaces as are found in the northern Gulf of Mexico near the Mississippi-Atchafalaya river plume. These fronts may be unstable to symmetric or inertial instabilities which further enhance small-scale mixing and encourage vertical transport between the surface and the abyss. Here, we consider the problem of an initially balanced front of finite width and which is bounded by flat no-stress horizontal surfaces. We examine how the evolution and equilibration depends on the front strength and aspect ratio using nonlinear numerical simulations, and develop a model to predict the profile and effective width of the final equilibrated state in the absence of external forcing. While fronts with Ro > 2.6 collapse to a self-similar profile dependent only on the deformation radius, we find that for small enough Ro \lesssim 1, frontlets form as the front equilibrates. These frontlets increase both the kinetic and potential energy of the final balanced state, but are also found to interact with the boundaries if the front exhibits inertial oscillations.

The effect of unbalanced exchange flow on the night cooling of buildings

Nick Wise, Gary Hunt

Cambridge University Engineering Department

Night cooling, the practice of opening vents at night in order to purge a room of warm air that has accumulated during the day and thereby cool the thermal mass, is a key part of the ventilation strategy of many naturally ventilated buildings. A common flow pattern used for night cooling is displacement flow, where warm air leaves through a high-level opening and is replaced by cooler air entering through a low-level opening. The existing theory of displacement flow predicts that the flow will be maintained throughout the purging process, giving a finite time for all the warm air to be purged from the room. We show that displacement flow cannot be maintained throughout the purge; at some time, which we derive, the flow pattern must transition to unbalanced exchange flow. In unbalanced exchange flow, warm air exhausts through the high-level opening as before, but cooler air enters simultaneously through the low-level and high-level openings, cooling the layer of warm air. After this flow transition, displacement flow theory no longer applies and we develop a new theoretical model that includes the transition to unbalanced exchange flow, allowing the calculation of new ventilation flow rates and purging times. In particular, we find that for geometries where the high-level opening is much larger than the low-level opening, the time to purge most of the warm air from the room may be several times longer than calculated using displacement theory alone.

Validity of sound-proof approximations for magnetic buoyancy

John Moss, Toby Wood, Paul Bushby

Newcastle University

The presence of acoustic waves in compressible fluids makes the governing equations mathematically "stiff", which is a problem for computational models. Therefore, many models filter out these waves using a "sound-proof" approximation, such as the Boussinesq, anelastic or pseudo-incompressible models. We assess the accuracy of each of these approximations for describing magnetic buoyancy under a range of physical conditions relevant to the solar interior. By comparing the linearised equations for a general sound-proof model with those of the fully compressible model, we derive a number of constraints which must be satisfied for the sound-proof model to capture the leading-order behaviour of the fully compressible system. We discuss the physical significance of these constraints with reference to existing sound-proof models

Shape mode oscillation of a bubble in a viscous compressible liquid near to a rigid boundary

Callan Corbett, Qianxi Wang

University of Birmingham

With applications to cavitation erosion and ultrasonic cleaning, bubble dynamics near to a rigid boundary has been extensively studied. However, certain features regarding the shape mode oscillation of a bubble near to a rigid boundary remain unknown. In this talk, one such feature will be discussed. This is how the shear stress generated on a rigid boundary by an oscillating bubble differs between modes of oscillation. This is of particular interest to ultrasonic cleaning, as shape modes that generate higher shear stress can be targeted to aide in the removal of substances from a boundary. In order to simulate this behaviour, a weakly viscous, weakly compressible, axisymmetric model based on the boundary integral method (BIM) is employed. This allows for the accurate modelling of multiple cycles of bubble oscillation. Shape mode oscillation is stimulated by prescribing an initial velocity potential distribution to the bubble surface proportional to the Legendre polynomial of order n, where the integer n represents the desired shape mode number. The velocity potential at any point in the fluid domain can then be calculated via the BIM. As a boundary layer exists at the rigid boundary, the flow region with constant rate of shear must be located. The shear stress can then be directly computed using the velocity potential in this region. Two cases will be considered; one when the bubble is close to the boundary, and the other when the bubble is far from the boundary. The shear stress distribution is calculated for four different modes of oscillation in each case, and then compared.

The effects of a Maxwell fluid on the evolution of a three-dimensional microbubble

Eoin O'Brien, Mehdi Mahmud, Warren Smith, Qianxi Wang

School of Mathematics, University of Birmingham, B152TT, United Kingdom

Microbubble dynamics in non-Newtonian fluids have direct applications in drug delivery, ultrasonic cleaning and weaponry. The jet formation at the collapse of the bubble can lead to high pressure on the surrounding surfaces, resulting in large scale damage. Understanding the evolution of these microbubbles due to different geometries, high intensity ultrasonic waves, and the viscoelastic properties of the surrounding fluid is therefore of the utmost importance. The evolution of a three-dimensional bubble is modelled using potential flow theory, and solved numerically using the boundary integral method. The proposed model combines both the 3D bubble model used to simulate a microbubble near a rigid boundary subject to an acoustic wave (Q. X. Wang and K. Manmi), with the axisymmetric non-Newtonian model (S.J. Lind and T.N. Phillips) to simulate the effects of the relaxation time of the fluid on a three dimensional bubble. Maxwell's constitutive equation is used to model the effects of the surrounding viscoelastic fluid and solved at each time-step using a Euler method. The bubble is simulated both with and without the subjection of an acoustic wave in a semi-infinite fluid to show the response of the Kelvin impulse, bubble migration and the shape of the bubble jet at collapse due to an increase in the relaxation time. Results are presented on the radius history, oscillation period and maximum surface velocity for a bubble in an infinite fluid.

Toroidal analysis of an oscillating sessile drop

Saksham Sharma and Ian Wilson

Department of Chemical Engineering and Biotechnology, University of Cambridge, Philippa Fawcett Drive, Cambridge CB3 0AS, UK

Complicated physical domains in fluid mechanics are often tackled computationally. One such example is that of a sessile drop supported on a horizontal, flat substrate. In this talk, I will present a recipe for nding analytical solutions to simple problems de ned in complicated geometry by taking an example of an oscillating sessile drop. I start with describing the construction of toroidal coordinates - from cartesian coordinate system, using M obius transformation(conformal mapping in 2D) and rotation - which completely matches the symmetry of a sessile drop. We derive fully analytical solution for this problem when the sessile drop is of small Bond number (surface tension dominates gravity) and a fixed contact line [1]. Di erential geometry of toroidal system is formulated to express the governing equations in terms of the toroidal coordinate system, which yields solutions involving hypergeometric functions. Resonant frequencies are identified for zonal, sectoral and tesseral vibration modes. The predictions show excellent agreement with experimental data reported in the literature, particularly for flatter drops (lower theta_c, but not so low as to incur significant viscous dissipation) and higher modes of vibration. [1] Sharma, S., Wilson, D. I. (2021). On a toroidal method to solve the sessile-drop oscillation problem (J. Fluid Mech.) doi:10.1017/jfm.2021.419

Influence of particle aggregation on evaporating and de-wetting droplets of complex fluids

Junzhe (James) Zhang, Andrew Archer, Dmitri Tseluiko, David Sibley

Department of Mathematical Sciences, Loughborough University

We consider dynamic wetting and de-wetting processes of complex liquid droplets focusing on the case of colloidal suspensions, where the particle interactions cause agglomeration. This leads to interesting dynamics of the droplet surface. Incorporating concepts from thermodynamics, we construct a model consisting of a pair of coupled partial differential equations that represent the evolution of the droplet and the effective colloidal height profiles using the thin-film approximation. The model extends also to include mass transfer effects by allowing the solvent to evaporate. We determine the relevant phase behaviour of the uniform system, including finding associated binodal and spinodal curves helping to uncover how the emerging behaviour depends on the particle interactions. Performing a linear stability analysis of our system enables us to identify parameter regimes where agglomerates form, which we independently confirm through numerical simulation. We obtain various dynamics such as uniform colloidal profiles in an unstable situation evolving into agglomerates and thus elucidate the interplay between evaporation, de-wetting and particle aggregation in complex liquids on surfaces.

Circular hydraulic jump in microgravity

Rajesh K. Bhagat (1), Mehran Mohebbi (2), Kyle G. Phillips, Kerri Phillips (3) Tristan Wolfe (4) D.I. Wilson (5), Paul F Linden (1)

(1) Department of Applied Mathematics and Theoretical Physics, Wilberforce Road, Cambridge, CB3 0WA (2) Industrial Engineering Systems Group, Nijenborgh 4, 9747AG Groningen, Netherlands (3) The Johns Hopkins Applied Physics Laboratory, 11100Johns Hopkins Road, Laurel, MD, 20723 (4) Naval Surface Warfare Center Philadelphia Division, 5001 S Broad St, Philadelphia, PA 19112, USA (5) Department of Chemical Engineering and Biotechnology, Phillipa Fawcett Drive, Cambridge, CB3 0AS

When we run the tap on a kitchen sink, we observe a vertical jet impinging on the surface and spreading radially outwards as a thin film to a finite radius where the film thickness changes abruptly, forming a hydraulic jump. For more than a century, the consensus has been that the thin-film hydraulic jump that can be seen in kitchen sinks is created by gravity. Bhagat et al. (2018) reported that kitchen sink jumps are created by surface tension, and gravity does not play a significant role. We present experimental data for circular hydraulic jump experiments carried out aboard the NASA C-9 micro-gravity research aircraft at »2% of Earth's gravity in two separate expeditions (Painter et al. (2007); Phillips et al. (2008)). The existence of the kitchen sink scale hydraulic jump at approximately the same radius as observed in terrestrial experiments unequivocally confirms that gravity cannot play a significant role in the formation of these jumps. We compare experiments with Bhagat et al.'s theory, which gives a good prediction of the experimental data.

Validation of analytical wind turbine wake models in turbulent base flows

Stefano Gambuzza, Bharath Ganapathisubramani*

University of Southampton, Southampton, UK

Wind turbine wake models are powerful tools employed by wind farm designers to efficiently layout individual turbines in wind farms; determining the correct spacing and positioning of turbines is a crucial step of farm design, as the interaction of turbines with the wakes of upstream machines can result in consistent power losses (Barthelmie and Jensen, 2010). During the years, some analytical wake models have been developed to inexpensively predict a wind turbine wake based on simple quantities such as the turbine thrust, and thus to estimate the optimal placement of turbines in a farm; these are, however, not dependent on free-stream turbulence characteristics, which are instead known to affect the power harvesting mechanisms of wind turbines (Gambuzza and Ganapathisubramani, 2021) and the drag of turbine simulators (Blackmore et al., 2014). To gauge the robustness of different wake models under turbulent base flows, an experimental campaign has been dedicated to the measurement of the wake generated by a model wind turbine (rotor diameter of 180 mm) in turbulent conditions, with free-stream turbulence intensity ranging from 3% to 12% and integral length scale ranging from 0.1 to 10 times the turbine rotor diameter; the measurements have been acquired by means of planar PIV at a distance of 1 to 9 diameters downstream of the model wind turbine, distances comparable to the inter-turbine spacing in wind farms. Characteristics of the wake generated by the turbine (such as the wake width, the centreline velocity deficit and the added turbulence intensity) are then compared to the predictions of state-of-the-art analytical models. Results show that both turbulence intensity and scale considerably affect the wake recovery mechanisms, and that analytical models must take the free-stream turbulence content into account if a realistic result is desired.

Investigation of the validity of wall-modelling in simulations of rotating cavity flows

Tom Hickling, Li He

University of Oxford

Rotating cavities form a significant part of turbomachinery internal air systems and there is considerable interest in being able to predict the heat transfer within them. The buoyancy and Coriolis force interact within the unstably stratified annulus to form a characteristic large-scale coherent flow structure that consists of alternating regions of radial inflow and outflow. The flow in these cavities has distinct behaviour in different regions: the cavity sidewalls behave as Ekman layers, and small heat-transfer enhancing structures have been observed to form at the outer radii. Due to the unsteady nature of the flow and the difficulty of accounting for buoyancy and rotation in turbulence models, there is widespread acceptance in the turbomachinery community that it is necessary to use turbulence resolving approaches to simulate these flows. The cost of properly resolving the near-wall flow in a Large Eddy Simulation (LES) means that it is a common practice to use approaches such as Detached Eddy Simulation (DES) to model the effects of any near-wall turbulence with an eddy viscosity. However, when wall-modelling is used, the focus is usually on the reduced computational costs of wall-modelled simulations, and not on whether the behaviour of the near-wall flow is modelled accurately. To address this issue, we have carried out computational studies to contrast wall-resolved and wall-modelled simulations for a rotating cavity (Hickling and He, 2021). Some further simulations and analyses to examine the behaviour and sensitivity of the flow and heat transfer will be carried out and presented.

Adaptive reduced order modelling of steady aerodynamic flows over ultra-high aspect ratio wings

Peter Nagy, Giuseppe Fortunato, Marco Fossati

Aerospace Centre, University of Strathclyde, Glasgow, G1 1XJ, United Kingdom

In the development of new aerodynamic designs, the use of accurate but fast numerical methods is of paramount importance. Widely adopted computational fluid dynamics methods can lead to excessive costs, especially in areas such as design optimisation, where many high-fidelity solutions are required. Data-driven reduced order modelling offers an alternative allowing for significant reduction in complexity while maintaining accuracy. A low-dimensional model is constructed from a set of high-fidelity solutions, then predictions within the same parameter space can be obtained at a fraction of the cost. We proposed an adaptive reduced order modelling framework with the goal of maximising accuracy. With this approach a reduced order method is selected automatically among those available, so that a particular solution is reconstructed with the highest accuracy. The adaptive selection is driven by a measure of the reconstruction which must be obtained for the whole parameter space, in which solutions are computed. The aim of this work is to assess the use of the framework for steady flows over an ultra-high aspect ratio wing. This technology is being adopted in the design of next-generation aircraft which highlights the relevance of this test case. Furthermore, aeroelastic effects are significant compared to conventional wing geometries, therefore deformations must also be considered. Thus, the ability of the framework to reconstruct a deformed geometry along with the flow solution can be evaluated. Two test cases will be presented, a fully rigid and a statically aeroelastic wing in transonic flight conditions.

Modulation of leading-edge vortex shedding with chordwise deformation

Alfonso Martínez and Kiran Ramesh

Aerospace Sciences Division, School of Engineering, University of Glasgow

Leading-edge vortices on unsteady aerofoils can be altered by trailing-edge flexion. And although this fact has spurred numerical and experimental research aiming to affect leading-edge vortex characteristics (particularly in the last couple of decades), the theoretical approach to this question has received less attention. The nascent trend of designing unmanned flying machines inspired by natural flight, along with the deep-rooted interest in flow control within the aerospace community, could be a stepping stone towards the development of theoretical models to address this fundamental question. The broad aim of the work here presented is to develop mathematical relations between prescribed trailing-edge deformations and resultant flow behaviour, and to build a simple model to tackle the challenge of unsteady flow control at Low Reynolds numbers. To achieve this, thin-aerofoil theory is extended to account for large camber variations and is combined with a discrete-vortex method and a leading-edge vorticity shedding criterion. This provides a low-computational cost, physics-based tool capable of modelling unsteady flows around deformable bodies in the order of seconds. Computational fluid dynamics simulations are used to support the validation of the low-order model under different flow regimes (fully attached flow and dynamic stall scenarios are considered in this work), covering a broad spectrum of high reduced frequencies and amplitudes of trailing-edge deflection. The potential of chordwise deformation in altering unsteady vortical structures behaviour in dynamic stall situations is explored by variation of morphing-characterising parameters, such as the maximum camber and its chordwise position.

Effect of thickness on wake dynamics of an airfoil pitching in uniform flow

Muhammed Najvan (1), Chandan Bose (2), Sachin Y. Shinde (1)

(1) . Department of Mechanical Engineering, Indian Institute of Technology Kanpur, Kanpur 208016, India (2) Department of Aerospace and Mechanical Engineering, University of Liège, Liège 4000, Belgium

Transitional wake dynamics and the interesting wake patterns behind a pitching airfoil with increasing Strouhal number have been topics of continued research interest. These studies can reveal insightful physics of aerodynamic load generation by natural flapping flyers and swimmers, which can help in the efficient design of futuristic micro aerial vehicles (MAVs) and autonomous underwater vehicles (AUVs). Various numerical and experimental studies have been conducted in the recent past to investigate the effect of kinematic parameters of the pitching motion, such as frequency, amplitude, and forward speed, on the wake patterns and propulsive efficiency. On the other hand, the airfoil shape is also an important design parameter for futuristic flapping devices to optimize propulsive efficiency. Although the symmetric airfoils, such as NACA0012 and NACA0015, have been widely studied for the sake of simplicity, any change in thickness can significantly alter the flow, and thus, the aerodynamic load generation due to the changed frontal area. However, the existing literature reveals that the effect of thickness of flapping airfoils on its wake dynamics is least explored, especially for pure pitching kinematics. The present study aims to bridge this gap through high-fidelity Navier-Stokes simulations at Reynolds number, Re = 12000, using the open-source library OpenFOAM. To explore the effect of thickness, we study a set of six 2-D symmetric airfoils (NACA0006 to NACA0036) and a flat plate. Moreover, the focus of this study is the conventional, but least explored, drag-producing wake regime concerning a small amplitude-based Strouhal number. We use pure pitching motion with the pivot point located at the quarter-chord from the leading edge. The evolution and behaviour of unsteady vortex structures are compared qualitatively and quantitatively to identify the vital dynamical aspects of wakes. At a moderate foil thickness (in the case of NACA0024), we observed an interesting phenomenon that we term as "wake deflection/switching". In this study, we explore the underlying physics of wake deflection/switching. The present study would yield design parameters useful for MAVs and AUVs apart from providing insight into the effect of thickness present in the wings and fins of natural flyers and swimmers.

On the coupling between back pressure and flow structure in airway bifurcations

S. Cargill (1), S. Aleksandrova (1), J. Bernardino de la Serna (2), H. Medina (1), S. Benjamin (1)

(1) Research Centre for Fluid and Complex Systems, Coventry University; (2) National Heart and Lung Institute, Imperial College London

With lungs being crucial to the function of the human body, investigating human airway flow dynamics is important for multiple applications. Effective inhaler drug delivery, and ventilator oxygen delivery, rely on having a good understanding of airflow within the human airways. Airflow studies in the human airways also aid in predicting pollution effects on lung health and understanding respiratory disease development. The human airways consist of many parts with very different flow regimes. One of them, the conducting airways, is an asymmetric successively bifurcating network whose geometry gives rise to a range of flow patterns, such as secondary flow and flow separation, with turbulent flow present in the upper branches. Because of the large number of successively smaller flow passages, the so-called "resistance network" approach is often used, where one- and zero-dimensional pressure loss models represent each flow passage and their bifurcations. Existing studies have already demonstrated the significant effect bifurcation geometric properties, such as branching angle, have on airflow characteristics. However, there remains a lack of studies into the cumulative effect of multiple parameters; in particular, the effect of the downstream airway network on the flow distribution and associated flow losses in each bifurcation. In this paper, computational fluid dynamic techniques are used to establish the relationship between bifurcation geometry and flow properties in the human airways, with particular focus on the effect of back pressure from the downstream flow network on the flow structures, such as flow separation and secondary flow vortices. The results from this study are anticipated to provide insights that will aid in further airflow studies in human airways. For example, the effect of respiratory diseases on airway geometry and flow properties, and the importance of wall elasticity and airway obstruction on the flow distribution and gas exchange in the whole lungs.

4D-Flow MRI based CFD to improve patient specific in-silico predictions of the post-surgical haemodynamics in the thoracic aorta

Molly Cherry (1), Amirul Khan (1), Malenka Bissell (1), Zinedine Khatir (2)

University of Leeds (1), Birmingham City University (2)

Bicuspid aortic valve (BAV) is the most common congenital heart defect, affecting approximately 1.3% of all live births. It is the fusion of two of the aortic valve cusps which results in an off-centre jet of blood being ejected into the ascending aorta every cardiac cycle. This abnormal blood flow pattern causes highly eccentric and helical flow throughout the thoracic aorta, often leading to serious life-threatening complications and afflictions once the individual is past the age of 40, such as aortic dilation. It is commonly treated through an aortic valve replacement (AVR), either biological or mechanical. It has been hypothesised that alongside the current methods of treatment planning (based on aortic valve geometries), the flow eccentricity and helicity resulting from BAV could be used as a diagnostic tool, aiding risk stratification and patient-specific treatment planning. 4D-Flow Magnetic Resonance Imaging (4D-Flow MRI) is often referred to as the 'gold standard' of imaging, as it provides a non-invasive, non-ionising method of diagnosing and monitoring patients with BAV. On top of allowing the aortic geometry to be observed, it also determines a velocity flow field in the blood vessels of interest. This allows a series of key haemodynamic parameters to be calculated retrospectively such as WSS, oscillatory shear index (OSI), flow helicity, and vorticity. By combining this new imaging technique with computational fluid dynamics (CFD) techniques, it becomes possible to improve in-silico haemodynamic predictions for the individual patient. Through developing a methodology that combines 4D-Flow MRI techniques and CFD simulations, a tool that predicts patient-specific haemodynamics in the thoracic aorta can be created. By altering the inlet conditions, the post AVR haemodynamics can be predicted, aiding clinicians in decision-making regarding a patient-specific treatment plan, resulting in an improved outlook for patients with BAV.

Non-trivial transport of active Brownian particles (micro-swimmers) in shear

Lloyd Fung(1), Rachel Bearon(2), Yongyun Hwang(1)

(1) Department of Aeronautics, Imperial College London, (2) Department of Mathematical Sciences, University of Liverpool

Micro-swimmers such as motile algae can be modelled as active Brownian particles, which self-propel at a roughly constant speed and rotate randomly due to noise. A dilute suspension of active Brownian particles can be modelled with the Smoluchowski (a.k.a. Fokker-Planck) equation for the probability density function of swimmers' location and orientation. However, in most applications, we are only concerned with the locational distribution of swimmers. Therefore, there have been several models to coarse-grain the Smoluchowski equation into a transport equation for the swimmer density. The generalised Taylor dispersion theory (GTDT) has, by far, been shown to be the superior model among others. It was particularly successful in modelling the effective diffusion of swimmers due to the coupling of random rotation and motility of swimmers. However, its application can only be extended to pure shear flow. In this presentation, we will use the Smoluchowski equation to show how the transport of swimmers are more complex than what GTDT has predicted. The Smoluchowski equation can capture the shear-induced migration in an inhomogeneous shear flow while GTDT cannot. The equation can also show how swimmers' dispersion in shear is different from the diffusion GTDT has predicted. We will then present a novel way to derive a better transport equation via a transformation on the Smoluchowski equation. The resulting model is less restrictive and exact, which also implies that it is more accurate than GTDT.

Is the shape of air particulate matter important? A study on the interactions of the two shapes of titanium dioxide nanoparticles with a model of pulmonary surfactant

Farzaneh Hajirasouliha(1), Dominika Zabiegaj(1)

(1) Smart Materials and Surfaces Laboratory, Department of Mechanical and Construction Engineering, Faculty of Engineering and Environment, Northumbria University, NE1 8ST, Newcastle Upon Tyne, United Kingdom

Air quality indices are measured based on the concentration and size of the particles. But should the shape of these particles be considered as well? Upon inhalation, Air Particulate Matter with a size of less than or equal to 2.5µm (PM2.5) can reach the deepest areas of the respiratory system called alveolar region where the gas exchange with blood circulation happens. In this zone, the walls of almost 500 million tiny alveoli sacs are lined with a tiny layer of fluid which causes a high surface tension and instability in this large air-liquid interface. However, thanks to the lung surfactant, there is no collapse of this region in a healthy human. Therefore, the first barrier that PM2.5 encounter before reaching the blood circulation is the interface covered by a monolayer of lung surfactant molecules. The purpose of this research is to study the interfacial properties of these monolayers interacted with titanium (IV) oxide nanoparticles with two different shapes, spherical and irregular ones, and with the same average size of 20 nm. Dipalmitoyl phosphatidylcholine (DPPC) as one of the main constituents of the pulmonary surfactant was used as the synthetic model. Using Profile Analysis Tensiometry (PAT) as an automatic set-up working based on Young-Laplace equation, we measured the interfacial tension and surface dilatational viscoelasticity in the pendant drop mode. The temperature was constant at 37°C. Four different amplitudes, 1%, 2%, 5%, and 10%, for the volume change of drops were used. Moreover, various frequencies, 0.1, 0.125, 0.25, and 0.5 Hz, were applied as the representative of the breathing cycle at different ages for a healthy human. The dependence of the interfacial and mechanical properties of the bespoke monolayers on the shape of the nanoparticles can be considered as an important factor for the environmental authorities.

Optimal turning gaits for undulators

Madeleine Hall (1), Eric Keaveny (1), Andre Brown (2) (3)

(1) Department of Mathematics, Imperial College London, (2) Institute of Clinical Sciences, Imperial College London, (3) MRC London Institute of Medical Sciences

An organism's ability to efficiently traverse and search their surroundings can be important to its survival. This has inspired the study of optimal gaits and locomotion strategies, in particular for the case of undulatory movement of slender bodies. The primary focus has been on finding optimal waveforms for moving forwards along straight paths. However, the ability to turn and manoeuvre is also relevant to survival. We revisit this problem in the context of low Reynolds number hydrodynamics and obtain the optimal waveforms for undulators along curved trajectories. For shallow turning angles, we obtain small perturbations of a travelling wave as optimal. For larger turning angles, however, the optimal gait can be radically different, with the undulator abruptly curling and uncurling itself. We believe that these results can lend insight into the search behaviours of simple organisms, such as C. elegans, as well as be a tool for phenotyping their behaviour across mutant strains and under different environmental conditions.

Flow correlations and transport behaviour of turbulent slurries in a partially filled pipe

: Christopher J. Cunliffe1,2, Jonathan M. Dodds1, David J. C. Dennis2

(1) National Nuclear Laboratory, Workington CA14 3YQ, United Kingdom (2) School of Engineering, University of Liverpool, L69 3GH, United Kingdom

Transportation of fluids in partially filled pipes is common to many areas of industry (e.g. sewerage, mining, and nuclear), yet many aspects of the work remain under-developed, particularly for multiphase and non-Newtonian flows. A new experimental database had been developed and involves measurements of flow rate, flow depth and the observed settling regime for two different solids concentrations of two types of non-colloidal suspensions across a broad range of channel slopes, Reynolds numbers, and Froude numbers. The bulk flow behaviour is accurately predicted with a novel non-dimensional correlation. Only one parameter is required for this technique that is not related to the measurable physical properties of the pipeline or fluid. However, it is empirically shown to be well represented by a single, dimensionless constant for all fluids and pipelines in the dataset. The sedimentation behaviour of the slurries has been characterised by a single dimensionless parameter. Together this forms a powerful framework for the design and optimisation of pipeline operations by a methodology that can be used to make a priori predictions of critical sedimentation velocities in open channel ducts. This can inform future pipeline designs and operations to minimise the environmental and commercial costs of slurry transfers, including reducing the risk of pipe blockages. This work is being used to underpin current liquor transport operations typical to nuclear decommissioning which allows for a more refined operational envelope to be developed.

A population balance approach for investigating the condensation of aluminum oxide droplets near a burning aluminum particle

Jannis Finke, Fabian Sewerin

Emmy Noether Group for Dispersed Multiphase Flows, Chair of Mechanical Process Engineering, Otto-von-Guericke-Universität Magdeburg, Universitätsplatz 2, 39106 Magdeburg, Germany

In order to buffer the strong intermittency associated with renewable energy sources, different energy carriers are currently being developed and analyzed for safety and energy capacity. A recently proposed concept is based on metal powders whose oxidation products after combustion could potentially be recovered from exhaust fumes, recycled and reduced back to plain metals. In this regard, our objective is to predict the size distribution of metal oxide droplets produced during combustion along with the composition of the gas and of participating heterogeneous surfaces and to identify requirements on oxide recovery techniques. In this presentation, we adapt a population balance description to aluminum oxide precipitates considering nucleation, particle growth and coagulation. Since the kinetic description of these processes still contains open questions and challenges, we provide a detailed and tailored kinetic framework based on established physical principles that is applicable across the whole range from the free molecular to the continuum regime. The PBE is economically solved by means of an adaptive grid method and a tailored discrete formulation of the coagulation source terms. The chemical reactions in the gas phase and on the surface of an evaporating aluminum particle are described using recently reported detailed reaction mechanisms. Furthermore, dissociation of the condensed droplets is accounted for along with thermal gas phase and droplet radiation. The competition between the different physical processes shaping the particle size distribution is analyzed in the context of a perfectly stirred reactor. Finally, the PBE-based model is instrumented to predict the aluminum oxide size distributions in the vicinity of a burning aluminum particle and the efficacy of oxide deposition caused by advection, diffusion and thermophoresis is investigated. This constitutes a first step towards the analysis of oxide smoke and residue sizes in aluminum dust flames.

CFD simulation of particles-liquid turbulent pipe flow

Zhuangjian Yang, Chiya Savari, Mostafa Barigou

School of Chemical Engineering, University of Birmingham, Edgbaston, Birmingham B15 2TT, UK

A vital challenge for modern engineering is the modelling of the multiscale complex particle-liquid flows at the heart of numerous industrial and physiological processes. Industries dependent on such flows include food, chemicals, consumer goods, pharmaceuticals, oil, mining, river engineering, construction, power generation, biotechnology and medicine. Despite this large range of application areas, industrial practice and processes and clinical practice are neither efficient nor optimal because of a lack of fundamental understanding of the complex, multiscale phenomena involved. Flows may be turbulent or viscous and the carrier fluid may exhibit complex non-Newtonian rheology. Particles have various shapes, sizes, densities, bulk and surface properties. Our fundamental understanding has so far been restricted by huge practical difficulties in imaging such flows and measuring their local properties. Mixtures of practical interest are often concentrated and opaque so that optical flow visualization is impossible. We address this problem experimentally by using a technique of positron emission particle tracking (PEPT). In PEPT, radiolabelled particles are used as flow followers and tracked in 3D space and time through positron detection. Thus, each component in a multiphase particle-liquid flow can be labelled and its behaviour observed. Compared with leading optical laser techniques (e.g. LDV, PIV), PEPT has an enormous and unique advantage that it can image opaque fluids and fluids inside opaque apparatus with comparable accuracy. Computationally, we study the viscous pipe flow of dense (up to 40% v/v) particle suspensions in turbulence Newtonian regions using an Eulerian-Eulerian CFD model. The predicted flow fields of the continuous and discrete phases are successfully validated by accurate experimental measurements of velocity profiles obtained from PEPT. The effects of various flow parameters are investigated including particle size, density and concentration, and carrier fluid properties. Varying the particle concentration changes the behaviour of the flow velocity profiles of both phases. Moreover, particle size and concentration have a significant effect on the particle radial distribution

Improving multi phase measurement performance of a Coriolis Flowmeter with additional pressure drop correlations

Stephan Schäfer

Univ. of Glasgow

Coriolis Meters are one of the most accurate Flowmeters in the field and are widely used in oil & gas applications, chemistry, pharmaceutical industry, food & beverage and many more. Their measureing principle is based on the Coriolis force, which leads to a twist in their curved tubes. The time-shift between each of its two inner tubes is directly proportional to the mass flow rate. Under multi phase conditions however, they do not perform well, especially if the flow pattern is less homogeneous. These kind of flow conditions are quite present in the industries listed above, therefore compensation/correction models are in highly sought after. By adding a differential pressure sensor to the CMF, the equation system that the presence of two-phase flow left under-determined will be closed. In order to do so, multiple correlations must first be obtained and implemented in the form of complex models that cover a wide parameter range. The main focus of this presentation will be the physical and experimental approach to these correlations. An overview over the iterative algorithm of the solution will also be presented under the premise of some simplifications. A summary of already existing models containing multi phase pressure drop correlations will however be reduced to a minimum.

Swelling driven autonomous soft gripper

Zhuofan Qin(1), Yingzhi Liu(1,2), Ansu Sun(1), Sreepathy Sridhar(1), Haibao Lu(2) ,Xue Chen(1), Ben Bin Xu(1)

(1) Department of Mechanical and Construction Engineering, Faculty of Engineering and Environment, Northumbria University, Newcastle upon Tyne, NE1 8ST, UK (2) Science and Technology on Advanced Composites in Special Environments Laboratory, Harbin Institute of Technology, Harbin, Heilongjiang, 150001, P.R. China

Swelling in liquid is a common phenomenon for hydrogel material. The free swelling induced liquid/solid interaction with a predefined confinement of semi-cylinder structure, can yield an autonomous soft gripper. In the thickness direction, the decreasing gradient of the flow rate of absorbing water from the outer surface to the central area enables the local and global instability morphologies which are characterized by the wrinkles and creases at the axial edge and thin layer of circumferential surface respectively. Besides, the flow velocities of all surfaces are consistent but the area of asymmetric surfaces are different resulting in the different flow rate and uneven expansion (high flow rate means high expansion factor). By mentioned two effects, the se-mi cylinder exhibits the morphogenesis of a gripper during the expanding process. After that, as carrying on further swelling and balancing the internal distribution of flow rate, the wrinkles disappear and the gripper reopen spontaneously. Utilising the multi-stimuli responsive nature of the hydrogel, we recover the swollen gel part to its initial state, enabling reproducible and cyclic shape evolution. The described soft gel structure capable of shape transformation brings a variety of advantages such as easy to fabricate, large strain transformation, efficient actuation and high strength-to-weight ratio, which is anticipated to provide guidance for future applications in soft robotics, flexible electronics and off-shore engineering and healthcare products.

New optimisation methods for scale-resolving turbulent flow simulation

Rhys Gilbert, Davide Lasagna

University of Southampton

Steady-state sensitivity analysis methods have been widely utilised in the engineering community, e.g., for aerodynamic shape optimisation using Reynolds-Averaged-Navier-Stokes for flow simulation. However, these methods break down for long time averaged sensitivities when used with unsteady, eddy-resolving simulation. Due to the 'butterfly effect' associated to chaotic fluid motion, small changes in the initial conditions lead to large differences in solutions of the adjoint equations, producing unrealistic sensitivities. Therefore, new approaches are required. Some recent advances in the utilisation of the Shadowing Lemma have shown promise, such as the Least Squares Shadowing (LSS), the Multiple Shooting Shadowing (MSS) and the Non-Intrusive Least Squares Shadowing (NILSS). However, these methods can be expensive and often lead to poorly conditioned numerical problems. Hence, alternatives to the Shadowing Lemma have also been considered, whereby artificial viscosity is added to adjoint equations as a simple passive "control" mechanism to alter the production of adjoint energy and damp exponential growth. However, considering that exponential growth is associated to the unstable Covariant Lyapunov Vectors and these can have multi-scale spatio-temporal behaviour in turbulent flows, it is unlikely that such control strategy can be effective without strongly affecting sensitivity calculations. To develop a better understanding of how artificial viscosity techniques may work in practice and propose alternative control strategies, we propose here a new shadowing method inspired from control theory. Specifically, optimal control theory for linear time-varying systems is utilised to find the control that minimises the growth of the adjoint equation. To solve the optimal control problem, we utilise a highly parallel, multiple-shooting algorithm using specific preconditioners to accelerate the solution of the optimality conditions. The proposed method is then investigated on model problems such as the Lorenz system and the Kuramoto-Sivashinsky equation.

Optical MEMS sensors for wall-shear stress measurements in turbulent boundary layer flows

Nima Ebrahimzade (1), Jose Portoles (1), Jodi Reeve (1), Peter Cumpson (2), Richard Whalley(1)*

(1) School of Engineering, Newcastle University, Newcastle NE1 7RU, United Kingdom (2) Mark Wainwright Analytical Centre, University of New South Wales, Sydney, New South Wales, Australia

This study reports on the development and experimental characterisation of optical micro-electro-mechanical-systems (MEMS) sensors for wall-shear stress quantification in turbulent boundary layer flows. The MEMS sensors are developed to measure the instantaneous wall-shear stress directly via a miniature flush-mounted floating element, which is on the order of hundreds of microns square. The floating element is suspended flush to the wall by up to four specially designed micro-springs. Displacements as low as tens of nanometres are detected by a unique Moiré fringe transduction technique. These wall-shear stress sensors have a sensitivity range of 23 - 740 nm/Pa, an accuracy of 1.6%, repeatability of 0.9%, with a minimum detectable wall-shear stress of 41 μPa. In a series of wind tunnel experiments, the instantaneous wall-shear stress within the turbulent boundary-layer flow is measured simultaneously by the MEMS sensors and by either laser Doppler velocimetry or hot-wire anemometry using the near-wall velocity gradient technique. Excellent agreement is observed in the time series and statistics across these three independent measurement techniques.

The mechanism of lobes generation in the sound directivity patterns for gust-airfoil interaction

Shujie Jiang(1),(2), Xu Chen(1),(2), Zhiwei Hu(2)

1) Key lab of Aerodynamic Noise Control, China Aerodynamics Research and Development Center, Mianyang, Sichuan, 621000, People's Republic of China, 2) Faculty of Engineering and Physical Sciences, University of Southampton, Southampton SO17 1BJ, United Kingdom

The noise generated by the interaction of coming turbulence with airfoil is an important noise generation mechanism for helicopters, wind turbines, turbofan engines, and contra-rotating open rotor engines. In these applications, turbulence in the wake of upstream blades interacts with the leading edge of downstream blades. One way to study this is through modeling the wake as a superposition of harmonic gusts. When the acoustic wavelength, generated by gust-airfoil interaction, is much bigger than the chord length of the airfoil, the noise source on the airfoil can be treated as a compact source and giving dipole-like sound directivity patterns. However, when the noise source on the airfoil is not compact, the interaction between the coming gust and the airfoil surface becomes much more complicated and results in complex patterns in sound directivity. A high-resolution computational aeroacoustic simulation is performed based on a spectral/hp element method to obtain the acoustic field directly in this work. The noise source on the airfoil obtained by the numerical simulation is fed to the convective Helmholtz wave equation to calculate the sound in the field. Simulations were performed for Ma=0.5, 0.6, and gust wavenumber k=1, 4, 8. The difference in phases of these sound waves is the dominant mechanism of lobe generation for sound directivity patterns. When the difference between these phases bigger than about , another sound directivity lobe is generated. Interestingly, this 'switch value ' seems independent of acoustic wavelength. This study reveals the mechanism for the sound directivity lobe generation and its effect on different acoustic wavelengths.

The turbulence production mechanism in the near-wall region under an adverse pressure gradient

Yuxin Jiao, Yongyun Hwang, Sergei Chernyshenko

Department of Aeronautics, Imperial College, London, SW7 2AZ, UK

The effect of adverse pressure gradient (APG) on near-wall turbulence is studied, with a particular attention to turbulence production mechanism in the near-wall region. For simplification, we consider a turbulent Couette flow and apply constant pressure gradient, such that the lower channel is exposed to APG, whereas the upper channel is subject to favorable pressure gradient. By doing direct numerical simulation (DNS) in a large computational domain, it has been found that when a strong APG is applied the near-wall turbulence continues to exhibit strong cross-streamwise turbulent intensities despite the typical feature of the near-wall turbulence, the streamwise elongated streaks, being weakened with a slightly increased spanwise spacing in the local wall units. In addition, the large-scale motions are intensified by APG in accordance with previous studies. Further linear analysis reveals that as a result of the significant reduction of mean shear in the near-wall region, APG has a damping effect on the generation of near-wall streaks by the linear mechanism. A DNS in minimal flow units reveals that near the lower wall the streamwise elongated near-wall streaks indeed disappear under strong APG, the turbulence production is reduced in outer units and the resulting near-wall dynamics for turbulent production is found to be very different from the well-known self-sustaining process. However, according to the analysis based on conditional averaging, the turbulence near the lower wall persists being driven by the localized wall-normal velocity fluctuations coming from the upper near-wall region. As such, the turbulent production in the region near the lower wall under strong APG is activated via the Orr mechanism, unrelated to near-wall streamwise streaks.

Dipole dynamics in the Point Vortex Model

Karl Lydon (1), Jason Laurie (1), Sergey V. Nazareneko (2)

(1) Department of Mathematics, College of Engineering and Physical Sciences Aston University, Aston Triangle, Birmingham, B47 ET, UK (2) Institut de Physique de Nice, Universit e C^ote d'Azur CNRS { UMR 7010, Parc Valrose, 06108 Nice, France

The dynamics of turbulent flows can be viewed as the resulting velocity field prescribed by the complex interaction of many hydrodynamic vortices. Hence understanding vortex motion is fundamentally important to understanding problems arising in turbulence. Arguably, the most important system in which this analogy holds is in (quasi-) 2D flows where a turbulent inverse cascade of energy leads to turbulent states composed of many interacting quasi-stable vortices. The most basic description for 2D vortex dynamics is that given by the point vortex model arising from the consideration of infinitesimal points of constant vorticity, known as point vortices used extensively to model Coulomb gases, vortex dynamics of 2D flows, as well as quantum vortices arising in Bose-Einstein condensates and superfluid Helium. Following the pioneering work of Novikov, Aref, and Eckhardt we present new results on the evolution dynamics of dipole in the vortex scattering process between isolated vortices, dipoles, and a hierarchy of vortex cluster structures. We examine the evolution of the dipole size, as well as categorizing the type of scattering or collison process across a wide range of impact parameters. We show that for a majority of parameter ranges it is sufficient to model the dipole-cluster as an integrable three-vortex process leading to the possibility of new mathematical results.

Mathematical modelling of drug metabolism in a microfluidic system

Barnum Swannell (1), Nicolò Milani (2), Daniela Ortiz Franyuti (2) , James Oliver (1), Sarah Waters (1)

(1) Mathematical Institute, University of Oxford, Oxford, UK (2) F. Hoffmann-La Roche, Basel, Switzerland

Microfluidic devices play an important role in preclinical studies, including the estimation of drug metabolic clearance rates. We present two mathematical models for metabolism within a representative microfluidic system: a compartment based pharmacokinetic model and a more sophisticated reaction-advection-diffusion model which accounts for fluid flow and spatial structure. Our goal is to evaluate the agreement between the two models and determine under what conditions the simpler pharmacokinetic model is appropriate. Towards this aim, we investigate the importance of clearance rate, flow rates and system geometry, as well as different mechanisms of mixing and fluid recirculation. A quantitative estimate for the difference between the two models is obtained by considering a reduced one dimensional model and can be expressed in terms of a single error parameter which combines many of these effects. Comparison with numerical simulations show this estimate to be a reliable predictor at early times. We find that in practice the clearance rate is the key factor affecting agreement between the two models and that the pharmacokinetic model is appropriate for low to moderate clearance compounds. The full reaction-advection-diffusion model can offer insights into the design and usage of such systems however, by capturing features that have been observed in experiments and suggesting how higher clearance compounds can be studied.

A poroelastic model for loading induced damage and rehabilitation in tendons

Isabelle Scott (1), Derek Moulton (1), Sarah Waters (1), Peter Malliaras (2)

(1) Mathematical Institute, Radcliffe Observatory Quarter, University of Oxford, Oxford OX2 6GG, UK (2) Physiotherapy Department, School of Primary and Allied Health Care, Monash University, Frankston, Victoria 3199, Australia

Tendinopathy is a painful condition resulting from repetitive tendon loads that surpass the capacity of the tendon. Loading of the tendon generates tensional and compressive solid forces, as well as hydrostatic pressure changes, which stimulate tendon cells to modulate the tendons material properties. Certain forms of loading lead to maladaptive changes while others are utilised as a means of rehabilitation. How the tendon cells respond to loading is a complex interplay between loading factors such as the strain rate, magnitude and duration, as well as the current material properties of the tendon. We utilise a poroelastic model to investigate how different forms of loading over time can contribute to maladaptive or positive adaptive changes in the tendon. We model the tendon as a poroelastic structure with spatially varying stiffness and permeability. We investigate how tensional and compressive forces vary with different forms of loading. Tissue mechanics are coupled with changes in material properties over larger time, represented by collagen and proteoglycan production. This mechanistic model is paired with real patient tendon images taken pre and post rehabilitation, where computer vision approaches have been utilised to detect changes in structure. We simulate physiological tendon injury and recovery via imposing different loading scenarios and studying the impact of compressive stresses and hydrostatic pressure on tendon damage. Based off this work we suggest new loading regimes which may improve the material properties of the tendon and hence warrant further clinical research.

Reconstruction and segmentation of noisy flow images as an inverse Navier-Stokes problem

Alexandros Kontogiannis (presenting), Matthew P. Juniper

Engineering Department, University of Cambridge

Modern medicine heavily relies on the use of medical imaging techniques for health assessment. Among them, magnetic resonance velocimetry (MRV) has a particularly wide range of applications. One application of interest, in the field of biological fluid dynamics, is the study of anomalies in the cardiovascular system such as the development of aneurysms. MRV offers non-invasive medical diagnosis of cardiovascular diseases by providing measurements of velocity fields that are subsequently used to estimate the wall shear stress (WSS) on the blood vessel. Since the wall shear stress is closely related to the process of atherogenesis, correct estimation of its magnitude is crucial for valid prognosis. Although MRV presents many advantages, there are certain drawbacks. First, MRV suffers from partial volume effects near the vessel boundary, that do not allow precise measurement of the WSS. In addition, due to noise, multiple scans need to be averaged leading to long acquisition times and motion artefacts due to breathing. To tackle the above problems, i.e. improve WSS measurement accuracy and reduce the MRV acquisition time, we propose a method that combines MRV data of velocity fields and a model equation (Navier-Stokes), to infer uncertain quantities, such as the viscosity of the fluid and the shape of the domain. At the same time, we obtain a regularized solution of the velocity field as the MRV signal is filtered through the model equation. We show that our proposed method successfully reconstructs very noisy MRV images that were acquired within seconds, providing high-quality flow images and estimates for the hidden flow quantities.

Multi-objective optimisation of continuous flow polymerase chain reaction flow systems

Foteini Zagklavara (1), Peter K. Jimack (1), Nikil Kapur (2), Osvaldo M. Querin (2), Harvey M. Thompson (2)

(1) School of Computing, University of Leeds, Woodhouse Lane, Leeds LS2 9JT, United Kingdom, (2) School of Mechanical Engineering, University of Leeds, Woodhouse Lane, Leeds LS2 9JT, United Kingdom

The invention and development of the Polymerase Chain Reaction (PCR) technology has revolutionised molecular diagnostics. Optimising the performance of PCR technology can lead to reduction of total construction and operation costs. The present work focuses on developing an optimisation methodology that utilises CFD to study the competing objectives of DNA amplification efficiency and pressure on continuous-flow single-phase PCR devices with a serpentine structure channel. This is achieved by optimising the width and the height of the channel of the μPCR device by performing a series of simulations of its performance (Design of Experiments) using COMSOL Multiphysics®. The values of the objectives are recorded and response surfaces of the two objective functions are created using neural networks. After creating the respective response surfaces, genetic algorithm is used to locate the optimum design. The design with the greatest width and minimum height of the microchannel generates the maximum DNA amplification yield (~2.1 % increase in the DNA concentration and ~33.5% volume reduction), while at the same time reducing the total number of PCR cycles required for the case study of 94 bp DNA sample by one. The design with the minimum width and height of the microchannel generates the minimum pressure drop (~95.2 % decrease in the pressure drop and ~37.5 % volume reduction). Multi-objective optimisation is performed with the use of a Pareto front plot, generating designs with the maximum width and varying height.

Deformation-driven rupture of gas cavities within a soft porous medium

Oliver Paulin, Liam Morrow, Matthew Hennessy, Christopher MacMinn

University of Oxford

Gas bubbles can be found in otherwise liquid-saturated granular media as a result of various physical processes, such as the microbial decomposition of organic matter. The gas formed is typically non-wetting and, as such, it is energetically costly for bubbles to invade the narrow pore throats between solid grains. If the solid skeleton is sufficiently soft, gas can instead displace solid grains to form open macroscopic cavities. A transition between gas existing within the pore space and in open cavities can be triggered through actively controlling the confinement on a particular sample. When an external deformation is applied to the solid skeleton, the subsequent increase in elastic stresses forces the closure of open cavities, with gas correspondingly being forced into the pore space. We investigate this phenomenon by using a close packing of squishy hydrogel beads as a model soft porous medium, and observe the behaviour of gas cavities within this system. We complement our experimental observations by deriving a thermodynamically consistent phase-field model, which captures the competing effects of elasticity and capillarity within a rigorous kinematic framework. Here, we focus on gas cavities within a 1D setting, with experiments based in a capillary tube. We identify the parameters which determine the confinement needed to force pore invasion, the effect of varying the rate of applied compression and the reversibility of the bubble rupture process.

Using Dissipative Particle Dynamics for investigating surfactant solutions under shearing

Rachel Hendrikse,Andrew Bayly, Peter Jimack, Xiaojun Lai

CDT Fluid Dynamics, University of Leeds

Surfactants are present in many everyday products such as detergents and shampoos. Because of the amphiphilic nature of surfactant molecules, they self-assemble into lyotropic liquid crystal structures when in solution. There exists a wide range of possible solution phase structures, e.g. micellar, hexagonal, lamellar, etc, depending on the solution composition. The structure of these phases leads to distinct phase dependent rheologies. The rheology can be very difficult to predict numerically, and therefore is often measured experimentally for such systems. The specific surfactants to be studied in this work are alkyl ethoxysulfates (AES). These anionic surfactants are one of the most common components of personal care products. This talk will focus on the use of Dissipative Particle Dynamics (DPD), to understand the effects of phase structure on the rheology of the material. DPD is an off-lattice, mesoscopic simulation technique which involves a set of particles moving in continuous space. While similar, DPD has benefits over Molecular Dynamics (MD) techniques, in particular MD struggles to reach the long time scales involved in the self-assembly process. Most existing DPD research focuses on understanding equilibrium behaviour. However, the complex behaviour of surfactant solutions under shear flow is not well understood. In our research we investigate phase and structural changes that are induced in the fluid, as a result of applied shear. For example, we can show that micelles transform from spheres to worm like micelles under the application of increasing shear. This talk will also present how DPD can be used to calculate the shear viscosity of a fluid, along with the challenges in calculating such properties.

Singularities in surfactant driven cavity flow

Richard Mcnair, Oliver Jensen, Julien Landel

University of Manchester

We model the advective Marangoni spreading of insoluble surfactant at the free surface of a viscous fluid that is confined within a two-dimensional rectangular cavity. This is a fundamental problem similar to the shear or lid-driven cavity flows, with applications for confined surface flows where surfactant can be naturally present, such as for unsteady flow over superhydrophobic surfaces in normal environmental conditions, or artificially added, such as in cleaning problems. Interfacial deflections are assumed small, with triple-phase contact lines pinned to the walls of the cavity; inertia and surface diffusion are neglected. Linearising the surfactant transport equation about the equilibrium state allows a modal decomposition of the dynamics, with eigenvalues corresponding to decay rates of perturbations. Computation of the family of mutually orthogonal two-dimensional eigenfunctions reveals singular flow structures near each contact line, resulting in spatially oscillatory patterns of wall shear stress and a pressure field that diverges logarithmically. These singularities of a stationary contact line are associated with dynamic compression of the surfactant monolayer, but can be regularized by surface diffusion. They highlight the need for careful treatment in computations involving unsteady surfactant transport dominated by advection in confined domains.

Hydrodynamic interactions of sedimenting squirmers

Albane Théry, Eric Lauga

Department of Applied Mathematics and Theoretical Physics, University of Cambridge

Confinement increases contacts between swimmers in dilute suspensions and affects their interactions. In particular, boundaries can experimentally lead to the formation of clusters that would not occur in bulk. How much does hydrodynamics govern these boundary-driven encounters between microswimmers? Here, motivated by recent experiments, we consider theoretically the symmetric boundary-mediated encounters of weak sedimenting squirmers. The classical squirmer model has been used to address the dynamics of a variety of swimmers, including active droplets and biological organisms. We consider the far-field interaction of pair of squirmers, as well as the lubrication interactions occurring after contact between two or more squirmers. In the far field, the orientation of the microswimmer is controlled by the wall and the squirming parameter, which characterises the type of flow created far from the squirmer, distinguishing between pushers and pullers. The presence of a second swimmer affects the orientation of the original squirmer, but for weak squirmers most of the interaction occurs after contact. We therefore analyse the near-field reorientation of circular groups of N squirmers. We show that a large number of swimmers and the presence of gravity can stabilise clusters of pullers, while the opposite is true for pusher swimmers; to be stable, pushers clusters would thus need to be governed by other interactions (e.g. phoretic). This idealised outlook on the phenomenon of active clustering enables us to focus specifically on the hydrodynamic contribution, which can be hard to isolate in large scale suspension simulations.

Influence of thermodynamic model on the shock interaction patterns of CO2 flows

Catarina Garbacz, Fábio Morgado, Marco Fossati

Aerospace Centre of Excellence, University of Strathclyde, Glasgow, G1 1XJ, United Kingdom

Interference between shock waves results in complex flow structures that dictate the location and intensity of aerothermal loads on the surface of a hypersonic vehicle, therefore playing a major role in aerodynamic design. There is considerable interest in CO2 flows given the relevance of this mixture to the atmosphere of Mars, however the literature is still scarce when it comes to shock interactions. In this work, a viscous hypersonic CO2 flow over a double-wedge geometry is simulated. Given the high-temperatures encountered in this type of flows, vibrational relaxation is accounted for since it is expected to have a significant impact on the results. Two different angles of the second wedge are simulated, to identify different flow patterns and understand interactions between the various flow features. To better assess the influence of thermal nonequilibrium due to vibrational excitation of the molecules, different gas models are compared. All the simulations are performed with the open-source CFD code SU2-NEMO that is coupled to the Mutation++ library, which handles all the computations associated with the thermodynamic state of the mixture. Typically, hypersonic flow features are extremely directional and characterized by high-gradients. Therefore, for a better compromise between accuracy and computational cost, AMG library is used for anisotropic mesh adaptation.

The UK's hypersonic multiphysics fluid simulation suite: OP2A

T. J. Greenslade, Minkwan Kim & Nathan Donaldson

University of Southampton

Hypersonic flows are of great concern in a number of engineering fields, including the development of hypersonic cruise vehicles as well as that of atmospheric re-entry vehicles. Understanding of these flows is required to determine such quantities as thermal loads and aerodynamic coefficients. The accurate simulation of such flows is thus critical to the design of hypersonic vehicles, but the UK currently lacks any homegrown hypersonic CFD toolkit. Hence, the University of Southampton is currently developing the Open source multiPhysics Phenomena Analyser, OP2A. OP2A is a modular, object-oriented CFD toolkit capable of modelling flows at velocities ranging from subsonic to hypersonic. OP2A is able to capture the effects of thermo-chemical non-equilibrium on hypersonic flows using multi-temperature models with a finite rate chemistry model. A block-structured Cartesian mesh with adaptive mesh refinement is employed in OP2A with hybrid MPI/OpenMP parallelisation to achieve high performance simulations. Additionally, OP2A's object-oriented design allows for the use of various interchangeable simulation components, and the straightforward creation of new components; allowing OP2A to be forward looking with regard to novel CFD techniques. The OP2A results presented here include a diverse set of validation and verification cases as well as the latest results from the 'Magnetohydro¬dynamic Enhanced Entry System for Space Transportation' (MEESST) project. MEESST is a European collaboration aiming to produce both experimental and numerical fluid data for the development of improved spacecraft re-entry systems. OP2A is currently being utilised as part of MEESST to produce high resolution simulation data mirroring various experimental configurations, using finite volume methods at second order spatial accuracy.

Effect of rarefaction on the evolution of compressible Kelvin-Helmholtz instability

Vishnu Mohan, A Sameen, B Srinivasan, S S Girimaji

Indian Institute of Technology, Madras

The effect of compressibility and rarefaction on Kelvin-Helmholtz instability is studied using a finite volume based solver for the Boltzmann equation called the Unified Gas Kinetic Scheme. The simulations are obtained for a wide range of Knudsen number and Mach number, and ve di erent regimes have been identified. The flow features of these di erent regimes have been studied using the perturbation kinetic energy, thickness of the mixing layer and the vorticity contours. It is seen that compressibility stabilizes the flow by suppressing perturbation kinetic energy, and the growth of mixing layer thickness. Pressure wave seen in compressible Kelvin-Helmholtz instability, roll and unroll the shear layer vortices, stabilizing it. Rarefaction, on the other hand, prevents the amplification in perturbation kinetic energy, and increases the mixing layer growth rate. This is due to the reduction in Reynolds number, which has been shown to increase the stability of flow for all wavelengths of perturbation.

Analysis of grid influence on the simulation of nonequilibrium flow over proximal bodies

Fábio Morgado, Catarina Garbacz, Marco Fossati

Aerospace Centre of Excellence, University of Strathclyde, Glasgow, G1 1XJ, United Kingdom

Accurate simulation of proximal bodies during atmospheric re-entry is a difficult task. The leading objects in the chemically-reactive hypersonic flow generate strong shock-waves which interact with shocks and objects downstream. This interaction may induce high aerothermal loads on the surface, capable of initializing the demise of the object. One key requirement to correctly assess the location and intensity of such loads is the creation of a mesh fully aligned with the flow features. However, during the re-entry process, the grid needs to be recurrently adapted to account for the bodies dynamics. The advancements made in the adaptation of tetrahedral meshes over the years have made them a good candidate for this type of simulations. In this work, steady-state proximal body simulations are conducted with hexahedral and tetrahedral meshes in order to assess the influence of grid alignment on the features and surface loads of both leading and trailing bodies. The AMG mesh adaptation tool is used to produce anisotropic tetrahedral meshes which follow the flow highly-directional features, in order to assess the feasibility of using oriented tetrahedral meshes for multi-body simulations when compared to hexahedral meshes. The open-source CFD code SU2-NEMO is used for all the simulations. The code is coupled with the thermo-chemistry library Mutation++ to provide the CFD solver all the thermodynamic state variables related to the mixture used.

Simulations of rotating detonation waves with adaptive mesh refinement

Han Peng, Ralf Deiterding

Aerodynamics and Flight Mechanics Group, University of Southampton, Boldrewood Campus, SO16 7QF

The highly efficient rotating detonation engine (RDE) has been considered as one of the viable replacements for current propulsion and power generation systems that employ constant pressure combustion. The extreme thermodynamic conditions in RDE make simulations costly and challenging. An effective approach for capturing supersonic combustion and discontinuous flow structures is adaptive mesh refinement (AMR). In this project, we use the open-source mesh adaptation framework Adaptive Mesh Refinement in Object-oriented C++ (AMROC) to solve the rotating detonation problems in a 2-D unwrapped chamber. The Navier-Stokes equations with detailed chemical kinetics are solved within the AMROC finite volume framework. A second-order accurate MUSCL-Hancock scheme with Minmod limiter is used for the reconstruction. The inviscid flux is evaluated by a hybrid Roe/HLL scheme. The viscous terms are solved with a second-order accurate central-difference scheme. Strang splitting is adopted for the reactive source term and a simplified ethylene/oxygen reaction mechanism is employed. The results show that the detonation propagation velocities are in good agreement with the results computed from the theoretical Chapman-Jouguet (C-J) state. Stable multiple-wave modes are captured by the simulations. The number of detonation waves increases with an increasing inlet mass flow rate, which is also observed in the experiments. The discrepancies between premixed injections and partly premixed injections are also studied. The effects of the degree of blending and the non-uniform local equivalence ratio on the rotating detonation propagation are analysed.

Reconstruction of 3D large-scale structures inside a stirred tank from limited velocity measurements

Kirill Mikhaylov (1), Stelios Rigopoulos (2), George Papadakis (1)

(1) Department of Aeronautics, Imperial College London, London, SW7 2AZ, UK
(2) Department of Mechanical Engineering, Imperial College London, London, SW7 2AZ, UK

Stirred tanks are widely used for mixing in many industrial applications. A stirred tank consists of a cylindrical vessel and a rotating impeller at its centre. The impeller blades generate trailing vortices of opposite vorticity that are shed in the wake and the interaction with the vessel walls creates a very complex 3D flow field. The central aim of this study is to apply the subspace identification algorithm, N4SID, to reconstruct the trailing vortices from velocity measurements taken at strategically selected sensor locations (between 1 and 6 probe points were considered). The algorithm reconstructed very accurately the trailing vortices at the design Reynolds number of 600 (even with a single sensor point). Crucially, the requirement of measurements at only a few locations opens the possibility of using experimental data to perform this task. The algorithm also worked well at the off-design Reynolds numbers 500 and 700, demonstrating the robustness of the approach and the potential for constructing the model for a particular Reynolds number and then applying this over a range of Reynolds numbers, giving the algorithm predictive ability.

A minimal data-driven quasilinear approximation based approach for turbulent channel flow statistics

Jacob Holford (1), Yongyun Hwang (1), Myoungkyu Lee (2)

(1) Department of Aeronautics, Imperial College London, South Kensington SW7 2AZ, UK (2) Combustion Research Facility, Sandia National Laboratories, Livermore, CA 94550, USA

Recently, Hwang & Eckhardt (2020) proposed a minimal quasilinear approximation (MQLA) of turbulent channel flow to generalise the theory of Townsend's (1976) attached eddy hypothesis at finite Reynolds numbers. It was found that the MQLA reproduces the qualitative scaling behaviour of turbulence intensities and one-dimensional spanwise spectra for a wide range of finite Reynolds numbers. The MQLA determines a stochastic forcing for a linearised eddy-viscosity based model, with the only `input' required being the turbulent mean profile, readily available through the robust law of the wall. In this talk, we present a `physics-informed data-driven model' which quantitatively improves the MQLA. To this end, the stochastic forcing in the MQLA is determined by incorporating the attached eddy hypothesis in the logarithmic layer, and such the model becomes 'predictive' at any Reynolds number. By considering the statistical structure of the eddies with respect to spanwise length scale for different fixed spanwise wavenumbers, a universal self-similar forcing for the streamwise Fourier modes is determined by formulating an optimisation problem designed to best capture turbulent flow statistics and spectra in the logarithmic region using the results of a high Reynolds number DNS (Lee & Moser 2015). By utilising the universal self-similar forcing based on the attached eddy hypothesis, the proposed quasilinear approximation is then examined as a predictive tool over a range of Reynolds numbers and is shown to reproduce turbulence statistics and velocity spectra with a good accuracy at a drastically reduced computational cost.

CNN-aided flooding prognostic in packed column using Electrical Capacitance Tomography

Yuan Chen, Jiabin Jia, Mathieu Lucquiaud, Yunjie Yang, Chang Liu

School of Engineering, The University of Edinburgh, Edinburgh, EH9 3JL, UK

Flooding is accompanied by a sharp increase in pressure, resulting in inefficient operation and potential damage to equipment. This work aims at investigating the use of an Electrical Capacitance Tomography (ECT) to determine real-time local liquid hold-up, then determine flooding point. The proposed method is validated using both the Multiphysics simulation and experiments. Simulation: A geometric model filled with a simplified crimp shape packing (Mellapak 250.Y) was established in COMSOL Multiphysics together with the presented ECT sensor. The training data were collected at local liquid hold-up between 0% to 17.8% with plastic packing. Both Conventional Neural Network (CNN) and Maxwell equation were shown to be capable of accurate determination of the local liquid hold-up. Simulation results indicate that there is a possibility of obtaining local liquid hold-up from ECT measurements and suggests that it is possible to measure local liquid hold-up with ECT to determine loading point and flooding point. Experiments: Experimental validation is carried out based on the simulation results. An eight-electrode ECT sensor was placed on the outside of a packed column and capacitance data were acquired over gas flow rate from non-flooded to flooded. The results between CNN and Maxwell equation were compared. Additionally, local liquid hold-up began to increase much sooner than global liquid hold-up and global pressure drop. The results from CNN proved ECT could be used non-invasively to monitor the local liquid hold-up, allowing for more accurate measurement of loading point and flooding point.

Ensemble simulation methods for in-silico trials of endovascular medical devices

Michael MacRaild (1,2), Ali Sarrami-Foroushani (2), Toni Lassila (2), Deborah Stocken (3), Alejandro Frangi (2)

(1) Centre for Doctoral Training in Fluid Dynamics, University of Leeds, (2) Center for Computational Imaging and Simulation Technologies in Biomedicine (CISTIB), University of Leeds, (3), Leeds Institute of Clinical Trials Research (LICTR), University of Leeds.

In-silico clinical trials (ISCTs) refer to pre-clinical trials performed entirely, or in part, using individualised computer modelling and simulation to virtually test some aspect of a medical device, drug or clinical procedure across a large range of anatomical morphologies and physiologies. This enables better judgement to be made for the performance and safety of a device across the global population. Our first ISCT focussed on flow diversion in intracranial aneurysms, using computational fluid dynamics to assess treatment success with haemodynamic variables, such as flow reduction in the aneurysm [1]. Our ISCT replicated the results of conventional clinical trials, as well as offering insights that clinical trials cannot provide. In future ISCTs, we would like to explore different device designs, on larger and more various cohorts of anatomies and physiologies, but the high cost of each simulation makes the computational burden prohibitive. This means that performing larger-scale ISCTs in future depends upon the development of inexpensive computational models that can accelerate the simulations without making unacceptable sacrifices in accuracy. Recent developments in data-driven modelling allow us to construct accelerated simulation methods that exploit data previously generated for similar problems. Two possible data-driven modelling approaches for accelerating simulations are Reduced Order Models (ROMs) and Physics-Informed Neural Networks (PINNs). ROMs aim to overcome the “curse of dimensionality” by building low-order representations of high-order models that preserve essential behaviour at the cost of lost accuracy. PINNs are a more recently developed machine learning-based approach that seeks to combine both data-driven and deterministic modelling. Whilst machine learning algorithms typically leverage “big data” in order to train predictive models, PINNs are designed to operate in the “small but rich” data regime that we find in fluid dynamics applications. In this talk, I will discuss data-driven simulation methods for accelerating blood flow modelling in intracranial aneurysms.

Poisson CNN: Convolutional neural networks for the solution of the Poisson equation on a Cartesian mesh

Ali Girayhan Özbay (1), Arash Hamzehloo (1), Sylvain Laizet (1), Panagiotis Tzirakis (2), Georgios Rizos (2), Björn Schuller (2)

(1) Department of Aeronautics, Imperial College London; (2) Department of Computing, Imperial College London

The Poisson equation plays a central role in the computation of the pressure corrections when solving the incompressible Navier-Stokes equations in computational fluid dynamics (CFD). Considering the high computational effort involved in this procedure, machine learning methods offer a unique capability to estimate the result of and accelerate this step. To this end, we propose a novel fully convolutional neural network (CNN) architecture to estimate the solution of the Poisson equation on 2D Cartesian grids, capable of making predictions for problems with differing grid resolutions/sizes and arbitrary Dirichlet or Neumann boundary conditions without re-training. The flexibility in terms of boundary conditions is achieved by an innovative approach whereby the problem is decomposed into one homogeneous Poisson problem plus four inhomogeneous Laplace sub-problems. Training is conducted on a purely synthetic dataset using a novel loss function approximating the continuous L^p norm between the prediction and the target. The resulting model is highly versatile, predicting solutions to Poisson problems with mean percentage errors below 10% and capable of being included in a CFD solver without any further fluid dynamics specific training. Analytical test cases demonstrate that the model is capable of also working well in tandem with traditional iterative Poisson solver algorithms like Multigrid, reducing RMS error after one iteration by almost 95% compared to a zero initial guess.

Experimental investigation of displacement instabilities in straight microchannels with pure viscoelastic fluids

Seng H. Hue, Loïc Chagot, Panagiota Angeli

ThAMeS, Dept of Chemical Engineering, University College London (UCL), UK

Knowledge of the behaviour of a liquid displacement by another immiscible liquid in microfluidic devices is important in both academic and industrial contexts. Examples include the cleaning and decontamination of small tubing and channels in microreactors, tertiary oil recovery from porous rocks and controlled administration of drugs and vaccines in medical devices. The displacement of a liquid by another fluid has been studied extensively during the last decades. As shown by Saffman and Taylor (1958) [1] for porous media and Hele-Shaw cells, when a viscous liquid is pushed by a less viscous one, the interface between them tends to become unstable. This instability often leads to inefficient cleaning as a liquid film remains trapped at the wall. The observations can be extended to microfluidic displacements and many experimental studies have been performed for Newtonian flows ([2], [3]). For non-Newtonian flows, Gan et al. (2007) [4] highlighted that viscoelastic flow instabilities can promote a more effective mixing than in viscous/viscous or viscous/inertial fluid configurations. However, the impact of the viscoelastic properties of complex fluids on the space-time scales of these instabilities is still poorly understood. In the present study, interfacial instabilities were investigated during the displacement of silicone oil by a viscoelastic aqueous solution. The aqueous phase was a Boger fluid comprising of a mixture of polyethylene glycol (PEG), zinc chloride (ZnCl2) and polyethylene oxide (PEO). Experiments were also carried out with a reference Newtonian aqueous solution (PEG + ZnCl2). The density and viscosity ratios of the two immiscible fluids were the same in both cases. Experiments were performed in a circular quartz channel with a diameter of 200 microns, and images were taken with a 12-bit high-speed camera. Results showed that interfacial instabilities, after the initial displacement front, appear earlier for the viscoelastic fluid than the Newtonian one. Moreover, by tracking the displacement interface, two main types of instabilities were found for both the reference and viscoelastic solutions: axisymmetric and asymmetric instabilities. The frequencies of the instabilities were also measured to reveal the characteristic modes. The viscoelastic fluid was shown to exhibit larger instability amplitudes and a more consistent frequency. [1] Saffman PG, Taylor GI (1958) The penetration of a fluid into a porous medium or Hele-Shaw cell containing a more viscous liquid. Proceedings of the Royal Society of London Series A Mathematical and Physical Sciences 245(1242):312/329 [2] Foroughi H, Abbasi A, Das KS, Kawaji M (2012) Immiscible displacement of oil by water in a microchannel: Asymmetric ow behavior and nonlinear stability analysis of core-annular ow. Phy. Rev. E 85(2):026309 [3] Lu Y, Kovalchuk NM, Che Z, Simmons MJ (2020) Interfacial instabilities due to immiscible fluid displacement in circular and non-circular microchannels. Experimental Thermal and Fluid Science 113:110045 [4] Gan HY, Lam YC, Nguyen NT, Tam KC, Yang C (2007) Efficient mixing of viscoelastic fluids in a microchannel at flow Reynolds number. Microfluidics and Nanofluidics 3(1):101/108

05C-2

Reversible trapping of colloids in continuous flows past grooved microchannels by diffusiophoresis

Naval Singh (1), Goran Vladisavljević (1), François Nadal (2), Cécile Cottin-Bizonne (3), Christophe Pirat (3) and Guido Bolognesi (1)

(1) Department of Chemical Engineering, Loughborough University, UK (2) Wolfson School of Mech., Elect. and Manuf. Engineering, Loughborough University, UK (3) Institut Lumière Matière, UMR5306 Université Claude Bernard Lyon 1 - CNRS, France

In recent years, an increasing interest in harnessing chemical energy in the form of solute concentration gradient has led to the exploration of colloidal particle manipulation by diffusiophoresis (DP) in microfluidic devices [1]. Transient salt concentration gradients have been successfully used to achieve enhanced particle transport into dead-end structures by DP and diffusioosmosis (DO) effects [2-3]. In this study, we report a novel mechanism for reversible trapping of particles in dead-end micro-structures via steady-state solute gradients in a continuous flow setting [4]. A microchannel, made of optical adhesive glue, was fabricated by photo-/soft-lithography techniques and fitted with a transverse microgrooved wall. The charged fluorescent colloidal particles were accumulated within the microgrooves by pumping parallel electrolyte solutions into the device junction. The spatial distribution of particles within the channel was characterized via confocal microscopy and a numerical investigation was carried using finite element simulations. As shown by the confocal scans in the figure, we found that particles accumulate within the flow recirculation region beneath the groove entrance due to a combination of DP transport and hydrodynamic effects. The trapping phenomenon is fully reversible, and particles can be cyclically trapped into and released from the grooves by controlling the salt concentration of the parallel streams via a flow switching valve. The proposed method offers great potential for microfluidic bio-analytical testing applications, including bio-particle pre-concentration and signal amplification. [1] S. Shin, Physics of Fluids, 2020, 32, 101302 [2] S. Shin, E. Um, B. Sabass, J.T. Ault, M. Rahimi, P.B. Warren & H.A. Stone, PNAS, 2016, 113,2 [3]M.K. Rasmussen, J.N. Pedersen, & R. Marie, Nature Communications, 2020, 11, 2337 [4] N. Singh, G. Vladisavljević, F. Nadal, C. Cottin-Bizonne, C. Pirat & G. Bolognesi, Physical Review Letters, 2020, 125, 248002.

Coating flow in the presence of an irrotational airflow with circulation

Andrew Mitchell, Brian Duffy, Stephen Wilson.

University of Strathclyde

An analysis of steady coating flow of a thin film of a viscous fluid on the outside of a uniformly rotating horizontal circular cylinder in the presence of an irrotational air flow with circulation shows that the presence of the air flow can result in qualitatively different behaviour of the fluid film from that in classical coating flow. Full-film solutions corresponding to a continuous film of fluid covering the entire cylinder are possible only when the flux and mass of fluid do not exceed critical values, which are determined in terms of the parameters F and K representing the speed of the far-field air flow and the circulation of the air flow, respectively. The qualitative changes in the behaviour of the film thickness as F and K are varied are described. In particular, the film thickness can have as many as four stationary points and, in general, has neither top-to-bottom nor right-to-left symmetry. In addition, when the circulation of the air flow is in the same direction as the rotation of the cylinder the maximum mass of fluid that can be supported on the cylinder is always less than that in classical coating flow, whereas when the circulation is in the opposite direction the mass of fluid can be greater than that in classical coating flow. Recently Newell and Viljoen (Phys. Fluids 31, 034106 (2019)) sought to analyse stability of such flows by extending the analysis of Hinch and Kelmanson (Proc. R. Soc. Lond. 459, 1193{1213 (2003)) to include the presence of the external air flow. We revisit and extend their calculation and find, in disagreement with their original findings, that the flow is always stable for all values of F and K.

Wetting and dewetting dynamics of a thin liquid film spreading on an immiscible liquid surface

Christophe J. Wilk (1), Julien R. Landel (1) and Andrew L. Hazel (1).

(1) Department of Mathematics, University of Manchester

Thin liquid films are present in applications ranging from film coating to biological systems such as tear film. In this study, experiments on the spreading of surfactant-laden immiscible droplets over liquid substrates demonstrated a striking and unusual dewetting instability. As the film spreads over the liquid substrate due to Marangoni forces, it reaches, at least locally, a critical thickness under which intermolecular forces becomes important. Then, van der Waals forces reduce the film thickness and apparent holes in the film start to grow. After the end of the spreading phase, these apparent holes continue to form and grow. The interplay between Marangoni and intermolecular forces enables the system to wet and apparently dewet the substrate at the same time, which seems counter-intuitive. Our experimental data shows that the spreading is driven by a Marangoni flow and that the dewetting velocity (the rate at which the holes grow) is constant across the film interface. The pattern of holes formed in the process was analysed using Minkowski functionals which indicate that holes nucleate following random processes rather than spinodal dewetting, in agreement with previous studies on film dewetting over solid substrates. Our model, based on the lubrication approximation, attempts to explain the key physical mechanisms responsible for the dewetting phenomena observed experimentally. We perform linear stability analysis of a system consisting of a thin immiscible film over a thick liquid substrate, including intermolecular forces with a long-range attractive component and a short-range repulsive one, capillary forces, and insoluble surfactant at the midline. We investigate the most unstable modes to identify the key parameters that can lead to the dewetting phenomena observed experimentally. We discuss our model predictions in comparison with our experimental findings.

Adjoint equation-based estimation of porous media permeability

T. Zhang, K. H. Fraser, H. S. Gill, A. N. Cookson

University of Bath

Contrast enhanced myocardial perfusion imaging uses an injected contrast agent (CA) as a proxy for blood flow in cardiac tissue. This imaging modality has previously been modelled as the advection-diffusion of the CA through a porous medium. Here, an adjoint equation approach is developed for estimating the parameters of this porous medium model, as a route to more accurate and detailed quantitative diagnoses. Several test cases – of varying porous parameters, Peclet number (Pe), source terms, and boundary conditions – were used to generate "observed" data, on which to test the parameter estimation. Forward models were solved using the finite element method, implemented in FEniCS, with adjoints computed via the dolfin-adjoint module. These models were used to estimate the permeability tensor, by minimising an objective function containing two penalty terms: (1) the error between estimated concentration and observed concentration, and (2) the gradient of the permeability tensor. An iterative approach to adjusting the weighting of the penalty terms was developed, which allows the optimiser to converge to a low overall error while resolving local inhomogeneities in the porous medium parameters. For an homogeneous 2D test case with Pe≤0.1, the estimated permeability and concentration matched well with the observed data (with an L2 error norm < 10^(-3) for all cases). A test case with a homogeneous background permeability containing a localised area of lowered permeability (between 0% - 70% of the background value), showed that the method could successfully determine the location and extent of the low permeability region. This method was found to be compatible with features such as noise filtering and spatio-temporal down-sampling. In addition to its potential for analysing perfusion images, this method could prove useful for other applications where experimental determination of porous medium parameters is essential. Future work will test and develop the method on real data.

The effect of large scale incoming turbulence on wind-turbine-blade

ThankGod E. Boye, Zheng-Tong Xie

Aerodynamics and Flight Mechanics (AFM) Group, Faculty of Engineering and Physical Sciences, University of Southampton, SO17 1BJ, United Kingdom

In this study, we investigate the effect of large-scale incoming turbulence on a pitching wind turbine blade, of which the cross section is NACA0012 aerofoil at a moderate Reynolds number Re = 1.35 x 105, based on the chord length and the freestream velocity. An efficient turbulence inflow condition method for large eddy simulation is employed for the study. At the inlet synthetic turbulence with streamwise integral length-scale greater than the chord length are generated for modelling the impact of large-eddies on a pitching wing at reduced frequencies kred = 0.1 and 0.2. The smooth inflow results show a good agreement with the reference data. Large-scale turbulence evidently reduces the lift, drag and moment coefficients at kred = 0.1 compared to smooth inflow, whereas it reduces less at kred = 0.2. This is opposite to the effect of small integral length-scale (i.e. less 0.1c) turbulence in the literature.

An experimental investigation of the flow past a very thick flatback airfoil suitable for wind turbine blades

Antonios Cene1, Francesco Grasso2 Marinos Manolesos1

(1) Swansea University, SA1 8EN, Swansea, UK, (2) Vestas Technology UK Ltd, PO30 5TR Isle of Wight, UK

Very thick flatback airfoil shapes are now commonly used for the inboard part of wind turbine blades. The flow interaction with a flatback airfoil generates a few compelling and challenging problems such as vortex shedding, flow induced noise and vibration, and three-dimensional flow separation. Results from an experimental study of the aerodynamic and aeroacoustic properties of a flatback version of a typical inboard wind turbine airfoil (FFA-w3-360) are presented. Flow control devices such as Vortex Generators (VGs) and Gurney Flaps (GFs) were also used for this study to determine the effects of controlled flow on the aerodynamic and aeroacoustic properties of the airfoil. The aim is to produce understanding into the performance of the different type of flow control devices and the characteristics of the unsteady flow around the flatback airfoil. Experiments were performed at Swansea University Wind Tunnel at a chord Re number of 1.8M. The flatback profile (FB20) had a 20% thick trailing edge (TE) and was fitted with pressure taps, transient pressure sensors and microphones. Microphone measurements are presented with an emphasis on the amplitude and frequency of noise associated with the vortex-shedding tone from the blunt trailing edge wake. The presentation will discuss the effects of the different passive flow control devices on the airfoil performance and noise generation. The GF addition for the free-transition case at $\alpha=0°$ leads to drop in vortex-shedding frequency and drop in the peak amplitude on both sides of the model. Moreover, the addition of the VGs on the pressure side proved to have a more profound effect than placing the VGs on the suction side for both free and fixed transition cases. Finally, for all cases at high angles of attack a very low dominant frequency is observed ($St \approx 0.02$), which is linked with Stall Cell formation.

Harmonic forcing of a laminar bluff body wake with rear pitching flaps

Athanasios E. Giannenas (1), Sylvain Laizet (1), Georgios Rigas (1)

Department of Aeronautics, Imperial College London, London, SW7 2AZ, UK

A numerical study on the response of a 2D bluff-body wake subjected to harmonic forcing imposed by two pitching and thin flaps at the rear of the body is performed. Different aspect ratios (AR=1,2,4) for a 2D rectangle at a low Reynolds number (Re=100) characterised by laminar vortex shedding are considered with two different forcing strategies (snaking and clapping). In the snaking motion, both flaps move in-phase and in the same direction, while for the clapping motion the flaps also move in-phase but in opposite directions. The flaps follow a sinusoidal motion characterised by a monochromatic forcing frequency and an amplitude. A fundamental resonance is observed for the snaking motion when the wake is forced near the vortex shedding frequency of the natural (unforced) flow. This first-order resonance results in a distinct peak on the mean and RMS drag coefficient and a trough on the lift coefficient RMS. No harmonic resonance is present for the clapping motion but a subharmonic resonance can be observed when the forcing is applied at twice the vortex shedding frequency of the unforced flow. This second-order resonance results in a distinct (but smaller than the ones observed for the harmonic resonance) peak of the mean drag and lift RMS coefficients while the drag RMS remains unaffected. Furthermore, it is shown that the snaking motion can result in significant drag reductions of more than 20% through a wake re-organisation mechanism at non-resonant regimes. The clapping motion can achieve even greater drag reductions (even up to 100%) through wake symmetrisation and propulsion mechanisms. Finally, a single scaling parameter is proposed which can be used to predict the mean drag reduction of the forced flow for both snaking and clapping motions over a wide range of forcing frequencies, amplitudes and flap lengths.

Fluid mechanics of sash windows

Gaël Kemp (1), Megan Davies Wykes (1), Rajesh K. Bhagat (2) and Paul F. Linden (2)

(1) Engineering Department, University of Cambridge, Cambridge, CB2 1PZ, UK
(2) Department of Applied Mathematics and Theoretical Physics, University of Cambridge, Cambridge, CB3 0WA, UK

Implementing natural ventilation strategies in buildings is an efficient way of reducing their energy consumption while improving the indoor air quality. In northern Europe, sash windows or counter-balanced vertically-sliding windows are a common feature in domestic buildings. We describe a model for the ventilation flow rate through a sash window for a well-mixed room, when the indoor temperature is warmer than outdoors. Depending on the position of the neutral pressure level relative to the sash window, we identify three flow regimes: either inflow through one opening and outflow through the other opening, or bi-directional flow through the upper opening or through the lower opening. Using an analytic as well as a numeric approach we find the critical height of the lower opening for the flow to transition from one regime to another. Using fresh and salt water, we perform laboratory experiments in a tank to measure the flow rate for different sash window geometries, and compare our experimental results with the model. Using our results, we assess the optimal sash window arrangement for different natural ventilation strategies. The developed model can easily be implemented into control systems and building designs and our results have implications for optimal ventilation rates and control of ventilation in smart buildings. Furthermore, by increasing ventilation rates we can help to limit the risk of transmission of airborne infection.

A comparison of the development of spatially and temporally accelerating flows

Matthew Falcone; Shuisheng He

University of Sheffield

Direct numerical simulations of equivalent temporally and spatially accelerating flows are compared with the aim of understanding the physical and quantitative similarities of the flows. Temporally developing flows have long been used to understand the fundamentals of turbulence in spatially developing flows including in free shear, channel, and more recently boundary layer flows. This is largely due to the reduced computational cost associated with temporally developing simulations. Recent research on idealised spatially accelerating flows, which used an innovative accelerating moving wall to create a relative spatial acceleration, found that on the onset of the acceleration, a new laminar boundary layer forms superimposed on the existing turbulent flow. The spatial development of these flows was found to be characterised by the development and subsequent transition of the new boundary layer. The results were found to closely mirror prior studies of temporally accelerating flows which found that the transient flow development was similarly characterised by the formation, development and subsequent transition of a new laminar boundary layer. Consequently, these similarities raise the prospect of the ability to use temporally accelerating flows to investigate spatial acceleration. In addition, such a comparison can be used to improve the understanding of both flows. To this end, temporally accelerating channel flow has been performed to match spatially accelerating flows (using the moving wall approach described above) maintaining the same acceleration until both flows have undergone transition. A complex picture has emerged from the preliminary results. FIK analysis and the budgets of the momentum equation have indicated that there may be important differences between the effect of spatial and temporal inertia in accelerating flows.

Spectral Relaxation computation of heat transfer in aligned-field MHD Flow of nanofluid over a moving flat plate

Shahina Akter and M. Ferdows

Research Group of Fluid Flow Modeling and Simulation, Department of Applied Mathematics, University of Dhaka, Dhaka-1000, Bangladesh

Nanofluids are typically made up of nanoparticles such as metals, oxides, carbides, or carbon nanotubes, each of which has its own chemical and physical properties. The purpose of this paper is to investigate the steady-state heat transfer phenomena in an aligned MHD flow of an incompressible, viscous and electrically conducting water based nanofluid past a semi-infinite moving flat plate in which the flow velocity and magnetic field vectors far from the plate are parallel. A scaling group of transformation is used to reduce the governing non-linear partial differential equations into a set of non-dimensional ordinary differential equations. The transformed system of differential equations is solved numerically, employing the Spectral relaxation method via MATLAB software. SRM is a simple iteration scheme that does not require any evaluation of derivatives, perturbation, and linearization for solving a non-linear system of equations. The solutions for the flow and heat transfer characteristics are evaluated numerically for various values of the embedded parameters, namely the nanoparticle volume function φ, magnetic parameter M, magnetic Prandtl number P_rm, and Eckert number Ec respectively on velocity, induced magnetic field and heat transfer properties are explored through graphs and tables and discussed in details. The skin-friction coefficient and heat transfer rate have also been studied. We assumed the nanofluid is homogenous and therefore, we have used the single-phase method to solve the governing equations and also for investigation of heat and fluid flow in nanofluids to show this model is still employing and that is accurate enough to capture the thermal and hydrodynamic effects of a nanofluid.

Electrically driven Alfven Wave investigation

S. Lalloz123, A. Potherat2, L. Davoust1 and F. Debray3*

(1) IMAP - Science et Ingénierie des MAtériaux et Procédés, 1130 rue de la Piscine, Saint Martin d'Hères, France (2) Center for Fluids and Complex Systems, University of Coventry, Priory Street, Coventry CV1, UK (3) National Laboratory for High Magnetic Fields, 25 avenue des Martyrs, BP 166, 38042 Grenoble CEDEX 9, France

Magneto-mechanic waves were first theorized by H. Alfven in 1942. these are waves propagating in conductive media subjected to a magnetic field, in the form of transverse oscillations of both magnetic and velocity fields. After 70 years of research, they are still difficult to even reproduce in a laboratory and most of their dynamics remains elusive. In particular, the existence of these waves in a resistive MHD medium is still raised by the literature. During the presentation, a semi-analytical model, based on the linearized full-MHD equations in a bounded geometry, will be firstly discussed. The main results, showing the propagation of Alfven waves in a resistive medium under specific conditions, will be presented. In order to confirm results from the model, Alfven waves will be investigated by using an experimental rig, the Flowcube device, originally designed by Klein. Thus, in the second part of the presentation, the last version of the Flowcube will be presented. In particular, the methods of wave forcing, using an AC current injection through an electrode array, and of wave detection, using the Electric Potential Velocimetry, will be detailed. Finally, by looking at the physical characteristics of the Flowcube device, we will see that the latter can meet the conditions for observing waves established by the model.

Is machine learning robust for the prediction of chaotic flows?

A. Racca (1), L. Magri (1)(2)(3)(4)

(1) University of Cambridge, (2) Imperial College London, (3) The Alan Turing Institute, (4) Institute for Advanced Study, Technical University of Munich (visiting)

The time-accurate prediction of a chaotic system, such as turbulence, is challenging because its evolution becomes unpredictable after the predictability time. This is because infinitesimal errors in a chaotic system increase exponentially, i.e., two nearby time series diverge from each other. Echo State Networks (ESNs), which are a class of Reservoir Computing, can accurately predict the chaotic dynamics well beyond the predictability time by learning temporal patterns from data. Existing studies, however, also showed that small changes in the hyperparameters may markedly affect the network's performance. In this work, we improve the robustness in the selection of hyperparameters (validation) in ESNs for the time-accurate prediction of chaotic solutions. We define robustness as the ability to select hyperparameters that perform consistently between validation and test sets. We propose the Recycle Validation, and the chaotic versions of existing validation strategies, to specifically tackle the forecasting of chaotic systems. We analyze model-free and model-informed (hybrid) ESNs for the prediction of chaotic and quasiperiodic solutions in prototypical nonlinear systems in fluid dynamics, such as the Lorenz-96 and Moehlis-Faisst-Eckhardt (MFE) model. We show that the proposed validation strategies outperform the state-of-the-art validation strategies. Because the strategies are principled --- they are based on chaos theory such as the Lyapunov time --- they can be applied to other Recurrent Neural Networks architectures with little modification. This work opens up new possibilities for the robust design and application of ESNs, and Recurrent Neural Networks, to the time-accurate prediction of turbulent systems.

Koopman with control for constitutive law identification

Emily Southern, Dr. Eric Keaveny

Imperial College London

Constitutive laws relate the stress in a material to its deformation. Determining the constitutive law experimentally typically involves subjecting the material to a prescribed deformation and measuring the force required to achieve it. Such experiments can often assume that the sample experiences homogenous shear and its response is linear. The decision on which constitutive law to fit to the resulting data is based on the rheologist's knowledge about the fluid in relation to the catalogue of standard models appearing in the literature. In this talk, we present an alternative approach based on the application of Koopman operator theory and Dynamic Mode Decomposition (DMD) in the context of control, which extracts material parameters that arise in stress-evolution equations of viscoelastic fluids directly from data. We will present results from various tests of the framework that highlight its accuracy in identifying material parameters, as well as its robustness to measurement noise. We will discuss how data should be chosen, but also demonstrate how data from multiple experiments can be combined to improve resolution. Finally, we will show that our approach provides a natural way to utilise data from the nonlinear regime and extends to higher-dimensional data sets where spatial data within a sample might be available.

An Eulerian-Lagrangian approach to simulate particle-laden flows

Boyang Chen, Bruño Fraga

School of Engineering, University of Birmingham, UK

Particle-laden flows are common in many engineering and environmental scenarios, including the suspension of solid particles in wastewater, sediment transport in rivers and gas-solid mixing in fluidized bed reactors. Such flows are often turbulent implicating a wide range of scales of motion, which is a challenging task for computational fluid dynamics (CFD) models. An accurate treatment of the continuous and dispersed phases as well as the coupling and interaction between the two is critical to provide reliable prediction on the complex mechanism of dispersed multiphase flows. Large-eddy simulation of particle-laden flows is applied within an Eulerian-Lagrangian framework to investigate the dynamics of the dispersed and carrier phases. Four-way coupling was implemented, with the interaction between particles being handled by a soft-sphere collision model. The parallel efficiency of computing these two phases is supported by the MPI implementation. The simulation results were used to explore the dynamics of particle-driven gravity currents. The aim of this work is to present the first simulation and validation on turbidity currents by considering actual dispersed particles. The inertial particles within gravity currents are modelled as point-particle influencing the velocity and the density field of ambient fluid. The Boussinesq approximation is applied to compute the density variance generated by the suspension of solid particles. The accuracy of the solver is validated versus laboratory measurement and shown a well agreement with experimental data. The results obtained are compared with density-driven gravity currents, highlighting the influence of inertial particles on the front speed of the current and the scale of the coherent structures generated at the shear layer. Furthermore, significant insights are gained into the sedimentation of solid particles and its influence on the energy budget of turbidity currents.

Combining the method of characteristics with a reduced order modeling technique for economically solving the population balance equation

Fabian Sewerin

Emmy Noether Group for Dispersed Multiphase Flows, Chair of Mechanical Process Engineering, Otto-von-Guericke-Universität Magdeburg, Universitätsplatz 2, 39106 Magdeburg, Germany

The prediction of size distributions describing fine, polydispersed particles constitutes an important, albeit challenging task in many engineering applications. In the case of metal combustion, for example, the size distribution of the precipitated oxide fines is critical for an assessment of the oxide retrievability. The size distribution of soot particles, on the other hand, is known to control the particle respirability and toxicity. The spatial and temporal changes in the particle size distribution owing to spatial transport, particle-fluid and particle-particle interactions can be described by the population balance equation (PBE). In this presentation, we report on a methodology for solving the population balance equation that is based on the notion of particle size characteristics and involves a constrained Galerkin projection onto solution spaces constructed from frequent shapes of the particle size distribution. These shapes are determined with the aid of a principal orthogonal decomposition (POD) of training datasets assembled from adaptive grid solutions of the PBE in simplified flow configurations. In order to analyze the ability on part of the trained shape functions to generalize beyond the training configurations and to quantify the achievable accuracy, we consider the dispersion and growth of a particle population in a laminar plane jet. Here, the shapes are trained in a perfectly stirred reactor and a steady-state plug flow reactor, respectively. Our findings confirm that the trained shapes are applicable also in unseen flow configurations if the training and testing cases share the same physical processes controlling the particle size distribution. For moderate accuracy requirements, the combined PBE-POD approach leads to a significant reduction in the number of degrees of freedom parameterizing the particle size distribution compared to existing adaptive grid methods. This is particularly advantageous for the incorporation of the PBE into probabilistic methods for turbulent particle-laden flows.

A numerical study on suspended sediment transport in a partially vegetated channel flow

Mingyang Wang, Eldad Avital

Queen Mary University of London

Turbulent structures generated by vegetation patches play a dominant role in the dispersion of suspended sediment, which in turn is of great significance for ecosystem cycling and river geomorphology development. High fidelity Large Eddy Simulations (LES) coupled with the Discrete Phase Method (DPM) were used to explore the particle distribution and its variance (the non-uniformity in temporal and spatial space) in a partially vegetated straight channel. The novel findings and conclusions are outlined here. Firstly, the contour of the vertical vorticity component coincides well with particle preferential gatherings in the outer edge of the mixing layer in the near-bed region. Large-scale turbulent structures grow in mixing layer along the side of a vegetation patch (VP), which deplete particles away from the mixing layer into the neighbouring region. Also, higher vegetation densities (Dn) promote this depletion trend. Secondly, the Probability Density Function (PDF) and its variance were defined to quantify these phenomena, illustrating that the VP continuously interrupts the flow condition and promotes higher non-uniformity of particle distribution among the vegetated and non-vegetated regions. The variance of the PDF in the non-vegetated region is significantly higher than that in the neighbouring vegetated region located in the same streamwise location. The particle parcels are highly unevenly located along the periphery of the large eddies and are exchanged by the mixing flow between the non-vegetated and vegetated regions. Finally, the vertical entrainment of particles occurs in the vegetated region of the present cases. This is because the horseshoe structures provide an upwards velocity for the current Dn conditions (Dn<0.1) and an increase of Dn (Dn<0.1) accelerates the upward suspension. These findings complement our understanding of particles' transportation in both spanwise and vertical directions.

A deep learning-CFD method for Reynolds-Averaged Navier-Stokes simulations of arterial flows

Konstantinos Lyras, Jack Lee

King's College London

Obstructive arterial disease is a leading cause of cardiovascular mortality and remains a heavy burden on public health. In pursuit of an effective non-invasive means for diagnosis, collaboration between medicine and engineering has proved computational fluid dynamics (CFD) to be an effective tool for predicting flow patterns in arteries. A good balance between accuracy and computational speed is required, with CFD simulation times that, depending on the problem, are too long to be clinically useful. This study investigates the accuracy of deep learning models for the inference of CFD solutions obtained from Reynolds-Averaged Navier-Stokes (RANS) simulations for obtaining CFD predictions in a fraction of a second. We focus on a modernised convolutional neural network (CNN) architecture and evaluate a large number of trained neural networks with respect to their accuracy for the calculation of haemodynamics. We present a novel CNN-CFD method that, based on a database from previous CFD simulations allows the instantaneous prediction of pressure and velocity in the whole computational domain of new unseen arteries and without running a new simulation. A core idea of the method is that it enforces constraints dictated by the underpinning physics by naturally including them in the training of the CNN. This approach leads to predictions which ensure physical consistency of the model outputs. Numerical experiments are conducted on a number of internal flows relevant to haemodynamics applications, and different types of arterial stenosis and a wide range of Reynolds numbers are examined. We illustrate how training data size influence the accuracy of the solutions. With a relatively small amount of data our best models can arrive at a mean relative pressure and velocity error of less than 5% across a range of previously unseen lesion shapes. This can be further halved when using more data for training the network.

A family of non-monotonic toral mixing maps

Joe Myers Hill, Rob Sturman, Mark Wilson

University of Leeds

A fundamental model for laminar fluid mixing by chaotic advection is that of orthogonal shears on the torus. Composing such shears gives an area-preserving, discrete time map, modelling the iterative process of stretching and folding. The Arnold Cat Map is the canonical such example and mixing properties can be proved straightforwardly. Here we present a new family of maps, incorporating a non-monotonic shear to more closely resemble realistic fluid velocity profiles, and prove mixing results for a wide parameter range. In 2005 Cerbelli and Giona proved the mixing property for a particular case, focusing the first shear precisely half-way along the domain. This simplified the dynamics significantly, allowing for a direct approach. We build on their geometric arguments and follow a scheme from Katok and Strelcyn to extend the result to this broader family of non-monotonic maps. Importantly, this tells us that non-monotonic shears are not a barrier to analytical methods, only surmountable when the dynamics are simplified by some unique feature of the map. Aside from applications, our parameterised family is a rich source of varied dynamics; including the uniformly hyperbolic Arnold Cat Map, the non-uniformly hyperbolic (yet well behaved) map from Cerbelli and Giona, an entirely periodic map, and everything in between.

Transpiration cooling of hypersonic flow past a flat plate with porous injection

Pushpender K. Sharma, Ralf Deiterding, Neil D. Sandham

University of Southampton, Aerodynamics and Flight Mechanics Research Group

There is a renewed interest in recent times for alternate methods of effectively cooling the surface of high-speed vehicles, such as injection of coolant into the flow stream. These strategies are supposed to reduce the overall weight and cost of thermal protection systems (TPS). It is noted that the direct effusion of coolant. Fluid through large holes or slots is not very effective in providing the film cooling as it leads to separation and enhanced mixing inside the boundary layer. An alternate strategy of transpiration-based cooling is adopted where the coolant is injected into the flow through a porous layer. This leads to a gradual and more uniform distribution of the coolant into the boundary layer and hence higher effectiveness. In the present study, effects of conjugate heat flux wall boundary condition as compared to the isothermal wall are explored. The conjugate heat flux boundary condition at the wall is more realistic and allows the wall temperature to adjust according to the equivalence of the heat flux from solid and fluid mediums at the interface. The coolant is injected through a porous layer composed of a staggered arrangement of cylinders in two dimensions (2D), while a staggered arrangement of spheres is used in 3D. Lower pressure ratios resulting in relatively small blowing ratios are used, which are found to be more e ective in the experiments done by the Oxford Hypersonics group. Also, to mimic similar turbulent conditions in the wind tunnel, wall-bounded disturbances are introduced upstream of the porous layer such that reasonable mixing of coolant is allowed inside the hypersonic boundary layer.

Direct numerical solution of compressible flow over rough surfaces

Raynold Tan, Neil D. Sandham, Ralf Deiterding

University of Southampton, Aerodynamics and Flight Mechanics Research Group

In hypersonic flows, irregular surface roughness can be found on high speed vehicles experiencing high enthalpy flow. These vehicles would typically ablate material as a means of alleviating the high heat flux. As such, the rough surfaces found on these vehicles typically show limited resemblance to the artificially constructed rough surfaces that have been the basis of most fundamental research on turbulent flow over rough walls. In this work, flow passing an irregular rough surface is investigated, based on a scan of grit blasted surface that serves as a typical example of an irregular rough surface found in engineering applications. The scanned map of surface height versus lateral coordinates is filtered in Fourier space to remove features on very small scales and to create a smoothly varying periodic representation of the surface. The surface is then extended with a solid flat base and written into a triangulated surface mesh. This surface mesh is then used as a no-slip isothermal wall boundary in Direct Numerical Simulations (DNS) of Compressible Turbulent Channel Flow (CTCF). A level set Ghost Fluid Method (GFM) is used to represent the irregular rough boundary within the simulation. The flow statistics of a smooth and rough channel simulations are compared.

Design and characterisation of a gust generator for aeroelastic wind tunnel testing

Davide Balatti, Cameron Turner, Marinos Manolesos, Hamed Haddad Khodaparast and Michael I. Friswell

Swansea University, Faculty of Science and Engineering

Researchers have been looking for solutions to reduce structural stresses at the aircraft wing root caused by gust encounters, either by using passive or active methods. The unsteady nature of a gust flow and strong coupling between resulting aerodynamic loads and structural deformations make the modelling dynamic response a challenging task. To validate numerical models in the wind tunnel, a gust generator (GG) is required. The present work discusses the design, installation and commissioning of a GG at the Swansea University wind tunnel. The system comprises two electronically operated airfoil shaped horizontal vanes, each one bound to a shaft and placed upstream of the working section. Two independent servo-motors connect to the shafts through low backlash gearboxes are used to rotate the blades in a synch. The system is able to create single and continuous gusts. Finite Element analysis and hammer tests have been used to structurally characterise the set up. Experimental data, collected from a crosshot wire sensor, confirm that the system is reliably capable of creating single and continuous gusts and match Computational Fluid Dynamics predictions. In the case of single gust, measurements have been taken for airspeed in the range of 10 m/s to 26 m/s, considering as vane maximum amplitude rotation 5°, 10°, 15° and 20° and the vane rotation frequency between 1 Hz and 14 Hz, corresponding to a maximum reduced frequency of 0.9. To characterise the flow-field, measurements have been taken at nine vertical positions and seven horizontal positions. In the case of continuous gust, measurements have been taken for airspeed in the range of 10 m/s - 26 m/s, considering as vane maximum amplitude rotation 5°, 7.5° and 10° and the vane rotation frequency between 1 Hz and 10 Hz, corresponding to a maximum reduced frequency of 0.6. Wind tunnel tests have been planned to validate the aeroelastic model of a wing with a hinged wingtip subjected to single and continuous gusts. Additional information: https://doi.org/10.5281/zenodo.4748519

Travelling waves in the asymptotic suction boundary layer

M. Engel (1) , M. Linkmann (1) , H. Knahl (2) , E. Jelli (2) , B. Eckhardt (2)*

(1) School of Mathematics and Maxwell Institute for Mathematical Sciences, University of Edinburgh, Edinburgh, EH9 3FD, United Kingdom (2) Department of Physics, Philipps-University of Marburg, D-35032 Marburg, Germany (2) deceased 7th of August 2019*

The asymptotic suction boundary layer (ASBL) is a flow that develops over a at bottom plate in the presence of suction through that plate, resulting in a constant boundary layer thickness. As such, it shares certain properties with parallel shear flows and spatially developing boundary layers. Travelling-wave solutions with large-scale low-speed areas that extend into the free stream, reminiscent of large-scale low-momentum zones that influence mixing and extreme events in turbulent boundary layers, have been found in the ASBL. Here, we continue such a solution with respect to Reynolds number and domain size up to domains of size (Lx/delta,H/delta,Lz/delta) = (46, 20, 23). It appears through a saddle-node bifurcation in Reynolds number and generally disappears through saddle-node bifurcations in domain size. In short domains, the structure is known to localise in spanwise direction, we nd that it does not do so in the streamwise direction. We study the spatial structure of its dominant instabilities as a function of domain size, leading to a phenomenological description of breakdown scenarios of travelling-wave type free-stream coherent structures in the ASBL. This work is supported by the Priority Programme SPP 1881 "Turbulent Superstructures" of the Deutsche Forschungsgemeinschaft (DFG) under grant LI3694/1.

Comparison of Lagrangian curvature und torsion statistics in open and wall-bounded turbulent flows

Y. Hengster (1), M. Lellep (2), A. Tully (1), D. Schanz (3), F. Huhn (3), D. Garaboa Paz (4), E. Bodenschatz (5), M. Novara (3), R. Geisler (3), M. Bross (6), A. Schröder (3), M. Linkmann (1)

(1) School of Mathematics and Maxwell Institute for Mathematical Sciences, Univ. Edinburgh, UK. (2) School of Physics and Astronomy, Univ. Edinburgh, UK. (3) German Aerospace Center (DLR), Institute of Aerodynamics and Flow Technology, Department of Experimental Methods, Göttingen, Germany. (4) Group of Non-linear Physics, University of Santiago de Compostela, Spain. (5) Max Planck Institute for Dynamics and Self-Organization, Göttingen, Germany. (6) Inst. of Fluid Mechanics and Aerodynamics, Bundeswehr Univ. Munich, Neubiberg, Germany.

As curves, Lagrangian particle trajectories are fully described by curvature and torsion. Involving different temporal derivatives, the mathematical expressions of curvature and torsion mix large and small time scales, whose statistical signatures may differ for different types of turbulent flows. Here, we analyse Lagrangian particle data from two experiments obtained by means of high-resolution Lagrangian Particle Tracking using the Shake-The-Box framework [1] , turbulent von-Kármán flow and a zero-pressure-gradient boundary layer over a flat plate. For von-Kármán flow, we recover experimental results for curvature probability density functions (PDFs) [2], and provide the first measurement of instantaneous torsion PDFs from experiment, reporting a scaling exponent of −3 for large values of torsion in agreement with that obtained from numerical simulations of homogeneous isotropic turbulence [3] . For the boundary layer, the statistics are conditioned on wall distance. In the buffer and logarithmic layers, curvature and torsion PDFs have the same power-law tails, which agree with those measured for von-Kármán flow and for homogeneous isotropic turbulence. This work was supported by DFG (Grant No. SCHR 1165/5-1, Ka 1808/21-1 and Li 3694/1) as part of the Priority Programme 'Turbulent Superstructures' (DFG SPP 1881). The experimental work was partly funded by the EuHIT [4] project of the European Union. [1] D. Schanz, S. Gesemann, A. Schröder, Exp. Fluids 57, 70 (2016), [2] H. Xu, N. T. Ouellette, E. Bodenschatz, Phys. Rev. Lett 98, 050201 (2007), [3] W. Braun, F. De Lillo, B. Eckhardt, J. Turbul. 7, 1 (2006), A. Scagliarini, J. Turbul. 12, N25 (2011), [4] https://www.euhit.org/

Spatiotemporal dynamics of a vortex induced vibration system in presence of noise

M. S. Aswathy, Sunetra Sarkar

Department of Aerospace Engineering, Indian Institute of Technology Madras, Chennai, India, 600036

In the present study, the influence of parametric noise on the spatiotemporal dynamics of a vortex induced vibration system is examined. The study has been carried out using a Navier Stokes solver at a Reynolds number 100 and highlights the relevance of noise in bringing about substantial qualitative and quantitative dynamical changes in the system even for low Reynolds number flows. Parametric noise advances the instability regime and alter the frequency characteristics of the responses by predominantly exciting additional system frequencies. The study reveals that the time scale of the input noise is a significant factor in deciding the kind of dynamics exhibited by the system. A short time scale invokes aperiodic dynamics in the responses, whereas a long time scale noise invokes switch states/intermittent states. Noise has been seen to affect the separation phenomenon by delaying or aiding it, depending on the regime of operation. For the long time scale noise case, we observe a new qualitative dynamical state in the structural response called as Noise Induced Intermittency. As the bifurcation parameter approaches lock-in, the structure oscillates intermittently between low(corresponding to the outside lock-in regime) and high (corresponding to the lock-in regime) amplitude oscillations. This is manifested as bursts of high amplitude oscillations interspersed with lower amplitude oscillations. In such a scenario, the flow-field is a combination of strong Karman, weak Karman and transition aperiodicities. With short time scale noise, the flow-field dynamics are predominantly characterised by aperiodic phenomena such as vortex pairing, vortex diffusion, and deflected wakes. Examining its complex spatio-temporal dynamics, from a stochastic framework, adds a fresh perspective to the VIV problem.

Numerical investigation of fluid-structure interaction of a surface mounted elastic plate in a cross flow

Cakir, M and Malalasekera, W

Wolfson School of Mechanical, Electrical and Manufacturing Engineering, Loughborough University, Loughborough, LE11 3TU, United Kingdom

Flow around a surface-mounted blunt bodies is extensively studied problems in fluid dynamics. However, a few studies have been performed to observe their characteristics of fluid-structure interaction (FSI). Therefore, the research reported in this paper is an attempt to analyse such problems. In this work, a turbulent flow around a surface-mounted elastic plate has been studied using a single-platform coupling methodology adapted by STAR- CCM+ CFD software. A fully coupled FSI of a three-dimensional airflow with 10 m/s inlet velocity is modelled using the k-ω turbulence model. From the study, displacements and drag forces are obtained. The work undertaken here mainly consists of validation and a parametric study. The model is first validated with numerical results reported in the literature and then applied to perform a comprehensive parametric study. For the parametric study, the selection of materials, geometric configurations have been considered as the key parameters for the modelling of external flow FSI problems. The results show that simulation results and literature data match well. Relative differences between fluid and solid material properties affecting the characteristic of FSI have been analysed. The results demonstrate how the formation of flow-induced oscillations is influenced by the changes in the geometric configurations. Possible reasons for this are discussed. Consequently, it can be stated that analysis and validation of a single-platform FSI modelling strategy as in this study allows the simulation of full-scale systems that exists in engineering applications.

Energy harvesting potential of an Unimorph Flexible Beam in Uniform Flow

Rajanya Chatterjee (1), Chhote Lal Shah (2) Sunetra Sarkar (2),Sayan Gupta (1)

(1) Department of Applied Mechanics, IIT Madras, India (2) Department of Aerospace Engineering, IIT Madras, India

In the current study, the flow induced vibration of a flexible 2D beam laminated with piezo-electric material on one side (Unimorph) has been simulated. The flexible flag like structures deform significantly in the presence of underwater currents or wind. The subsequent deformation in the attached piezoelectric generates strain, which is converted to electrical energy by application of a resistance load. The electric field, in turn, contributes to strain energy in the structure to deforms it further or dampens depending on parametric condition. This voltage-induced deformation in structure now affects the fluid load leading to modified Fluid-Structure interaction behavior. The simulations have been carried out using an in-house three-way coupled fluid-solid-energy solver, comprising of a discrete forcing immersed boundary method (IBM)-based incompressible Navier-Stokes solver coupled with a finite difference method (FDM)-based solid solver with an energy equation being coupled to the solid internally. A parametric investigation has been done to understand the effect of varying flexibility and mass ratio of the beam on the dynamic response flow field, and variation in voltage output. The nonlinear dynamic study of the structural response indicates existence of different periodic and aperiodic regimes. The flow fields at the downstream of the flexible body correspond to the associated dynamics of the beam tip and the mode shapes of the beam. It is necessary to identify the parametric regimes for which the dynamic response is periodic or aperiodic. The regularity in the vibrating motion is directly related to the longevity of the piezoelectric material and the modeshapes decide the high strain regions, dictating the ideal placement of the piezo-patches if necessary. Simulations have been run mainly at lower Re = 200. The detailed study will be discussed in the presentation.

Numerical investigation of pressure spaces in solving fluid-structure interactions using an ALE finite element method

Gregory Walton, Peter Jimack, Mark Walkley, Yongxing Wang

University of Leeds

Solving fully coupled fluid-structure systems can be numerically challenging. In this work we consider a monolithic approach that is based upon an arbitrary Lagrangian-Eulerian (ALE) finite element method. This is a moving mesh approach that fits a single mesh to the evolving fluid-structure interface. We will demonstrate that a key aspect in accurately capturing the solid deformation depends on representing the pressure field jump at this interface. A comparison of the use of different pressure spaces to approximate the discontinuous pressure at the fluid-structure interface will be described for a challenging model problem involving an incompressible fluid. The stable P2-P1 element pair will be considered, as well as the impact of enriching the pressure space using a constant in each element (P1+P0). Alternatively using separate pressure spaces for both the fluid and solid regions is considered, as well as P2-P0 elements. A numerical study exploring both the stability and efficiency of solution of the FSI system is presented. The implications of using different pressure spaces in terms of the accuracy of the resulting simulations will be described first. Second, we will consider the efficiency of the linearised solver in conjunction with both preconditioned iterative and direct solvers.

Computational study of vibration-based leak detection approach in 90-degree pipe elbow

Ahmed. A. Abuhatira1(a.abuhatira@dundee.ac.uk), Salim. M. Salim2

(1) School of Science and Engineering, University of Dundee, (2) College of Engineering, Swansea University

Vibration-based leak detection (VBLD) approach in pipeline systems has been a topic of research interest. Its identified as effective method for early leak detection and popular choice as its non-invasive and more suited for monitoring than inspection. Previous publications investigated VBLD approach experimentally using straight pipe with a small diameter in water loop systems. The current study investigates this approach computationally in oil and gas sector particularly in a large scale 90-degree pipe elbow. Reynolds Stress Model (RSM) is coupled with a finite element structural model to simulate the fluid-structure interaction (FSI) using one-way coupling. The RSM turbulence model and FSI model are validated against published experimental and numerical results [1] [2] [3]. Preliminary results presented in the figure below indicate that vibration signal increases as the leak size increases. The VBLD is found capable of assessing leakages with different damage severities. The FSI model developed in this current study is useful to design experiments within the context of the Fluid–Structure Interaction for pipeline systems including the vibration-based leak detection method. The FSI model can be used to gain the number of sensors required to detect a certain level of leakage, for a given case study, and thus can be useful for improving novel sensors and measurement technique about this topic. The findings from this work, provides some quantitative of different vibration measurements for comparison purpose for similar case studies.

A pseudo-spectral high-order methodology for incompressible smoothed particle hydrodynamics

G. Fourtakas and B.D. Rogers

Department of Mechanical, Aerospace and Civil Engineering, University of Manchester, Manchester, M13 9PL, UK

Herein, we present a high-order pseudo-spectral incompressible smoothed particle hydrodynamics (FFT-ISPH) methodology. Incompressible smoothed particle hydrodynamics (ISPH) has received significant attention in particle methods due to the near second-order convergence characteristics and accurate pressure fields. Typically, with the classical fractional projection step method [1], the linear system arising from the pressure Poisson equation (PPE) is solved iteratively using solvers such as the biconjugate gradient with Jacobi preconditioner and recently more advanced solvers [2]. It is widely accepted that the computational cost of solving the PPE by iterative solvers is high and much larger for ISPH than mesh-based schemes due to the number of neighbouring nodes in the locally compact kernel support. The proposed scheme is an intermediate between a fully spectral and a standard incompressible SPH solver where the solution of the PPE is performed in spectral space whereas the discretisation and time integration are performed in physical space. The spatial discretisation is based on the high-order ISPH variant of Lind and Stansby [3] and the time integration uses a low-storage (fully) third-order Runge-Kutta scheme resulting in higher order convergence rates [4]. To the best of the authors' knowledge this is the first time such an approach has been demonstrated in SPH. Performance analysis shows more than two orders of magnitude gains in computational cost. Most importantly the FFT-ISPH performance is independent of the number of neighbouring nodes which hindered ISPH schemes in comparison with mesh-based solvers. Further, we go beyond the classical periodic cases usually encountered in pseudo-spectral schemes and demonstrate how different boundary conditions can be applied to the FFT-ISPH with periodic test cases , wall bounded no-slip test cases and finally a mixed boundary conditions test case. The accuracy and robustness of the FFT-ISPH scheme exhibits fourth (and higher) order convergence rates. [1]. Cummins, S.J. and M. Rudman, An SPH projection method. Journal of computational physics, 1999. 152(2): p. 584-607. [2]. Guo, X., et al., New massively parallel scheme for Incompressible Smoothed Particle Hydrodynamics (ISPH) for highly nonlinear and distorted flow. Computer Physics Communications, 2018. 233: p. 16-28. [3]. Lind, S.J. and P.K. Stansby, High-order Eulerian incompressible smoothed particle hydrodynamics with transition to Lagrangian free-surface motion. Journal of Computational Physics, 2016. 326: p. 290-311. [4]. Fourtakas, G., Rogers B.D. and A.M.A. Nasar, Towards pseudo-spectral incompressible smoothed particle hydrodynamics (ISPH), Computer Physics Communications, [In Press].

High Weissenberg number simulations with incompressible Smoothed Particle Hydrodynamics and the log-conformation formulation.

Jack King and Steven Lind

Department of Mechanical, Aerospace and Civil Engineering, University of Manchester

Viscoelastic flows occur widely, and numerical simulations of them are important for a range of industrial applications. Simulations of viscoelastic flows are more challenging than their Newtonian counterparts due to the presence of exponential gradients in polymeric stress fields, which can lead to catastrophic instabilities if not carefully handled. A key development to overcome this issue is the log-conformation formulation, which has been applied to a range of numerical methods, but not previously applied to Smoothed Particle Hydrodynamics (SPH). Here we present a 2D incompressible SPH algorithm for viscoelastic flows which, for the first time, incorporates a log-conformation formulation with an elasto-viscous stress splitting (EVSS) technique. The resulting scheme enables simulations of flows at high Weissenberg numbers (accurate up to Wi=85 for Poiseuille flow). The method is robust, and able to handle both internal and free-surface flows with large deformations, and a range of linear and non-linear constitutive models. Several test cases are considered including flow past a periodic array of cylinders and jet buckling. This presents a significant step change in capabilities compared to previous SPH algorithms for viscoelastic flows, and has the potential to simulate a wide range of new and challenging applications

High-order simulations using the Local Anisotropic Basis Function Method

Jack King, Steven Lind & Abouzied Nasar

Department of Mechanical, Aerospace and Civil Engineering, University of Manchester

Mesh-free methods have significant potential for simulations in complex geometries, as the time consuming process of mesh generation is avoided. Smoothed Particle Hydrodynamics (SPH) is a widely used mesh-free method, but suffers from a lack of consistency, making it unsuitable for direct numerical simulations (DNS). High order accurate, consistent, and local (using compact computational stencils) mesh-free methods are particularly desirable. We present a novel framework for generating local high order difference operators for arbitrary node distributions, referred to as the Local Anisotropic Basis Function Method (LABFM). Weights are constructed from linear sums of anisotropic basis functions (ABFs), chosen to eliminate all errors below a given order. LABFM is able to generate high order difference operators with compact computational stencils. Spatially varying resolution and accuracy are included naturally in the formulation. We validate the method against canonical PDEs and canonical flows, demonstrating up to 10th order convergence. We show the capability of the method with examples of high Rayleigh number convection, high-Reynolds number 2D-DNS through porous media, and violent free-surface flows. LABFM is a promising new mesh-free method for the solution of PDEs in complex geometries, with potential for Arbitrary Lagrangian-Eulerian schemes with natural adaptivity of resolution and accuracy. The method is highly scalable, the computational efficiency is competitive, and in the low order limit the method collapses to SPH. As a mesh-free method with the geometric flexibility of SPH and the accuracy of high order finite differences, LABFM has significant potential to transform the future of DNS.

Dealing with density discontinuities in planetary SPH simulations

S. Ruiz-Bonilla (1, 2), V. R. Eke (1), J. A. Kegerreis (1), R. J. Massey (1), L. F. A. Teodoro (3, 4)

(1) Institute for Computational Cosmology, Durham University, South Road, Durham DH1 3LE, UK (2) Institute for Data Science, Durham University, South Road, Durham DH1 3LE, UK (3) BAERI/NASA Ames Research Center, Moffett Field, CA 94035, USA (4) School of Physics and Astronomy, University of Glasgow, G12 8QQ, Scotland, UK

One of the fundamental features of SPH is how the density at any point in space is computed as a weighted sum of the masses of the neighbouring particles. This implies that the density varies smoothly in space, making density discontinuities difficult to represent. However, differentiated planets in hydrostatic equilibrium can contain density discontinuities both where there is a change of material (e.g. the core to mantle boundary) and at the surface of the planet. When trying to represent a planet in an SPH simulation, the smoothing of SPH particle densities across these discontinuities gives rise to well known problems. The incorrect pressures induced by the incorrect densities in these regions effectively create an artificial force that repels different material layers from each other, or in the case of the free surface, particles in the outermost shells of the planet will tend to expand and contract until they reach a new equilibrium state. These problems have further consequences when studying planetary giant impacts, such as the one that formed the Moon. However, it is not well understood exactly how these problems affect the mixing between materials and the thermodynamical evolution of material placed into the post-impact debris disk. In this work we present a novel method to address both problems at once. We define a statistic that measures how afflicted a particle is by being close to a boundary or free surface. The densities of these particles are then corrected in a smooth way using that same statistic. Preliminary results show this method greatly alleviates the two problems mentioned. Further tests are being performed at the moment.

Effect of flow parameters on transonic buffet at moderate Reynolds numbers

Pradeep Moise(1), Markus Zauner(1),(2), and Neil Sandham(1)

(1) Aerodynamics and Flight Mechanics Group, University of Southampton, (2) Department of Aerodynamics, Aeroelasticity, Acoustics, ONERA

Self-sustained, periodic, flow oscillation observed over an aircraft wing in the transonic regime is referred to as transonic buffet and can be detrimental to aircraft performance and safety. Although buffet has been established to arise as a global instability, the physical mechanisms underlying the instability and the influence of flow parameters on it remain unclear. In this study, we examine buffet characteristics under free transition conditions by varying free-stream Mach (M) and Reynolds numbers (Re), angle of attack and sweep angle by performing large eddy simulations of flow over infinite wing sections based on Dassault Aviation's V2C profile. With increasing M, we observed that flow oscillations that initially develop are eventually damped out for M just below an onset value, Mo. Above Mo, large-amplitude shock motion occurs, as is typical of a supercritical Hopf bifurcation. Both buffet frequency and amplitude are affected by M. With increasing angle of attack and Re, the buffet amplitude increases in the range studied, but its frequency remains unaffected. Interestingly, multiple shock-wave structures seen at lower Re reduce in number, tending towards a single shock wave with increasing Re. Further, we used spectral proper orthogonal decomposition to extract dominant coherent features for the various cases simulated. In addition to the buffet mode which occurs at a Strouhal number of St = 0.1 approximately, we also observed coherent structures associated with the shear layer with significant energy content for St ~ 1. Flow-field reconstruction based on the extracted buffet mode shows that shock wave motion is accompanied by a pressure rise in the rear part of the aerofoil's suction side for all cases simulated suggesting that the latter could be an important factor driving the instability. This study highlights the influence of flow parameters on transonic buffet and essential buffet characteristics that are conserved when they're varied.

Effect of boundary layer transition on shock buffet instability

Wei He(1) and Sebastian Timme(1); Pradeep Moise(2) and Neil Sandham(2)

(1) University of Liverpool, Brownlow Hill, Liverpool L69 3GH, UK (2) University of Southampton, Highfield, Southampton SO17 1BJ, UK

Stability of transonic shock buffet, an unsteady flow phenomenon resulting from strong shock-wave/boundary-layer interaction, is systematically investigated on the laminar V2C aerofoil assessing the impact of fixed boundary-layer transition. We consider two levels of the aerodynamic hierarchy for the flow modelling, specifically large eddy simulation (LES) and Reynolds-averaged Navier-Stokes (RANS) simulations. For LES, transition is forced by synthetic jet, whereas laminar flow is ensured upstream of the fixed transition point by deactivating the production term of the RANS turbulence model, specifically the common negative Spalart-Allmaras model with compressibility correction. The chord Reynolds number ranges from 500,000 to 3.2 million with Mach number around 0.73 and angle of attack near the onset of shock buffet. Early results show that there is overall good agreement for low Reynolds number and low (transonic) Mach number flows between the two modelling approaches, and it is found that the fixed transition will stabilise the shock buffet instability. Both LES and RANS show that the frequency of the aerofoil buffet mode increases with Reynolds number and Mach number. At the same time, compared to LES, an earlier boundarylayer separation is observed in RANS, which results in a more unstable flow. As Reynolds number increases, the growth rate of the shock buffet instability is increased and reaches a maximum value at Mach number of 0.73. Interestingly, at lower angle of attack, the frequency of the aerofoil mode is reduced by fixed transition compared to fully turbulent flow. In contrast, the frequency of the unstable mode is increased by the fixed transition as the angle of attack is increased. More results demonstrating the significance of boundary-layer transition will be presented at the meeting. The work is sponsored by EPSRC grant EP/R037027/1.

Numerical and experimental analysis of transonic buffet for different airfoil geometries at moderate Reynolds numbers

Markus Zauner(1)(2), Ananda Renou(1), Julien Dandois(1), Vincent Brion(1), Pradeep Moise(2), and Neil Sandham(2)

(1) Department of Aerodynamics, Aeroelasticity, Acoustics, ONERA (2) Aerodynamics and Flight Mechanics Group, University of Southampton

Laminar-flow wings can significantly reduce aircraft emissions, but still require improved understanding. Of particular interest is a complex aerodynamic instability known as transonic buffet, which leads to periodic detachment or thickening of boundary layers and oscillating shock waves, causing undesirable vibrations. Recent studies showed significant differences between Dassault Aviation's V2C and ONERA's OALT25 laminar-flow profiles in terms of low-frequency buffet oscillations. Furthermore, drastic changes of the buffet mechanism were reported in previous studies for the OALT25 airfoil when comparing cases with natural and tripped transition. In the scope of the present work, we attempt numerical as well as experimental comparison of OALT25 and V2C wing sections at Re=500,000. In a first part, implicit large-eddy simulations (ILES) are performed for the OALT25 airfoil at various Mach numbers to study buffet onset condition. Results are compared with numerical studies of the V2C profile, which were previously published by the authors. Buffet frequencies obtained by simulations are lower for the OALT25 airfoil (St≈0.08) compared to the V2C airfoil (St≈0.1). Buffet onset appears for the OALT25 airfoil at higher Mach numbers, where shock oscillations are particularly pronounced at intermediate frequencies (St≈0.5). Furthermore, an ILES at Re=1,000,000 is carried out to analyse scaling effects. In a second part experimental studies of both airfoils at similar flow conditions are performed to validate and extend numerical investigations. The present study highlights the significance of airfoil geometries in terms of buffet behaviour at onset-conditions. Acknowledgments: The authors want to thank UKTC (EP/L000261/1), EPSRC (EP/R037027/1), ARCHER, and Iridis5 (University of Southampton) for computational resources

Practical resolvent analysis applied to shock buffet on the common research model elastic wing

Jelle Houtman, Wei He, Sebastian Timme

University of Liverpool

Shock buffet on wings is a phenomenon in the transonic regime revealed through self-sustained unsteadiness and caused by strong shock-wave/boundary-layer interaction. It can lead to an unfavorable structural response called buffeting with impact on the fatigue life, efficiency and passenger comfort. Recently, triglobal shock-buffet studies by the authors, both considering the fluid-only and the fluid-structure coupled systems, gave a modal explanation for the phenomenon and this work builds upon those previous studies. The resolvent method has become ubiquitous in the field whereby the optimal response due to harmonic forcing is sought. Specifically, we will present details of an implementation of the resolvent method into an industrial computational fluid dynamics solver (DLR-TAU), extended to deal with the aeroelastic problem in edge-of-the-envelope flow and applicable to practical full aircraft cases. The method relies on an iterative scheme to converge to the optimal forcing and response solutions at defined frequencies. We will first consider the Goland wing for verification purposes by comparing with direct solution methods from Matlab. Subsequently, the capability is demonstrated on the elastic configuration of the NASA Common Research Model. The fluid-structure coupled results are scrutinised with respect to those from the fluid-only system to determine the impact of coupling on the otherwise pure aerodynamic shock-buffet instability. The work is supported by an EPSRC Industrial CASE scholarship in partnership with Airbus

Shock train response to back pressure forcing

Alex Gillespie, Neil Sandham

University of Southampton

Numerical simulations are performed to study the effect of back pressure disturbances on a shock train. The basic problem consists of a shock train within a constant-area channel with a supersonic turbulent inflow and a back pressure applied to the outlet. The shock train is composed of numerous shock waves which form in response to the back pressure condition. There is a high degree of coupling between the mostly-inviscid supersonic region and the highly viscous turbulent mixing layer. A study of the grid sensitivity is performed where we find that the internal structure of the shock train is less sensitive than its equilibrium location. By subjecting the shock train to different fixed back pressures, it is shown that the shock train length varies linearly with back pressure, while the shock spacing is conserved. With sinusoidal back pressure forcing the shock train oscillates at the applied forcing frequency. Higher frequency disturbances are filtered as they travel upstream through the shock train. Lower frequencies induce larger shock oscillations and produce upstream offsets in the shock positions. Further examination of the flow reveals a region of low frequency motion below the leading shock which becomes strongly amplified by the lowest back pressure forcing frequency.

Indirect noise in multi-component nozzle flows with dissipation

Animesh Jain (1), Luca Magri (2,1)

(1) Department of Engineering, University of Cambridge, Cambridge CB2 1PZ, UK;
(2) Imperial College London, Aeronautics Department, London, UK

Aircraft engine manufacturers are striving to make cleaner and less noisy engines. In addition to direct noise (caused due to unsteady heat release of flame), flow inhomogeneities can accelerate in the nozzle vane downstream of the combustor with the mean flow and generate indirect noise. If these sound waves are convected downstream, they add to the noise pollution, however, if they are reflected upstream, they can create thermoacoustic feedback. Most of the studies in the literature assume the flow to be isentropic, however, the presence of losses due to factors such as flow recirculation and wall friction causes the flow through real nozzles to be non-isentropic. In this work, a quasi-one-dimensional model is mathematically developed and analyzed numerically to predict the sound produced by temperature and compositional inhomogeneities accelerated through nozzles with pressure losses. The physics-based model is compared with existing non-isentropic models, which make use of semi-empirical parameters. The results compare favorably with experimental data of the literature. It is found that friction has a significant effect on the gain/phase of the reflected and transmitted waves for compact and non-compact nozzles. Furthermore, the effect of friction on thermoacoustics is studied highlighting the sensitivity of thermoacoustic oscillations to the dissipation in the nozzle. The study opens new possibilities for the accurate prediction of indirect noise from nozzles.

Data assimilation of thermoacoustics

Andrea Nóvoa (1), Luca Magri (1,2,3,4)

(1)Cambridge University Engineering Department, Trumpington St, Cambridge CB2 1PZ, UK (2) (2) Imperial College London, Aeronautics Department, Imperial College London, Exhibition Road, London SW7 2AZ, UK (3) The Alan Turing Institute, 96 Euston Rd, London NW1 2DB, UK (4) Institute for Advanced Study, TU Munich, Lichtenbergstraße 2 a85748 Garching, Germany

The prediction and control of thermoacoustic instabilities are challenging tasks due to their intricate nonlinear and multi—physics nature. Traditionally, these phenomena have been modelled using low-order models that provide qualitatively accurate solutions at low computational cost. We design an ensemble square-root Kalman filter to make these models quantitatively accurate. The aim of the filter is to perform state and parameter estimation to make the algorithm more versatile by enabling model learning on the fly. The algorithm is tested and validated through a multimicrophone experiment. We perform simulations using sampling strategies based on the Nyquist-Shannon criterion in non-chaotic regimes, and positive Lyapunov exponents for the chaotic regions. Bifurcation analysis indicates that the thermoacoustic model is extremely sensitive to small changes in the parameters, which give rise to nonlinear regimes identified with nonlinear timeseries analysis (limit cycles, frequency-locked, quasiperiodic, and chaotic oscillations). This sensitivity is key during the filtering process, as large changes in the state can result in unphysical solutions to the thermoacoustic parameters. We overcome this issue by a combined rejection-inflation covariance strategy. The numerical simulations show that the filter is robust and capable of recovering the true state even for chaotic regions with large uncertainty in the measurements. This work can help design and manufacture stable premixed gas turbines with the introduction of both on-the-fly assimilation of experimental data and parameter calibration into computationally inexpensive low-order models.

Diffusiophoresis: a mechanism to produce small-on-top stratified coatings

Clare R. Rees-Zimmerman & Alex F. Routh

Department of Chemical Engineering and Biotechnology, University of Cambridge

Stratification in drying films – how a mixture of differently-sized particles arranges itself upon drying – is examined. It is seen experimentally that smaller particles preferentially accumulate at the top surface, but it is not fully understood why. Understanding this could allow the design of coating formulations which self-assemble during drying to give a desired structure. Potential applications are across a wide range of industries, from a self-layering paint for cars, to a biocidal coating in which the biocide stratifies to the top surface, where it is required. On the basis of diffusional arguments alone, it would be expected that larger particles stratify to the top surface. However, other physical processes, including diffusiophoresis, may also be important. By deriving transport equations, the magnitude of different contributions can be compared, and numerical solutions for the film profile are produced. Diffusiophoresis is the migration of particles along a concentration gradient of a different solute species. A particular diffusiophoresis mechanism that has been hypothesised to cause small-on-top stratification is an excluded volume effect. This work probes the significance of this type of diffusiophoresis: to the diffusional model, a diffusiophoresis term is added that can be varied in strength. For hard spheres, it is predicted that diffusiophoresis counteracts the effect of diffusion, resulting in approximately uniform films. When the diffusiophoresis strength is increased, the small particles are predicted to stratify to the top surface. This suggests that diffusiophoresis does contribute to experimental observations of small-on-top stratification, but it might not be the only promoting factor.

Optimising anaerobic baffled reactors performance through combined numerical and experimental analysis

Riza Inanda Siregar(1),(2), Bruno Fraga(1), Soroosh Sharifi(1), Mike Jesson(1)

(1) University of Birmingham; (2) Universitas Sumatera Utara

Anaerobic baffled reactor (ABR) is an anaerobic water treatment method that uses a series of baffles to force the wastewater containing pollutants to flow under and over several baffles to promote mixing with the sludge. The most significant advantage of this anaerobic treatment is the ability to separate biochemical reactions in each chamber due to the arrangement of baffles. This condition permits different bacterial groups to grow under the most favorable places and allows methanogenesis. ABRs provide poor effluent quality, which needs post-treatment or additional equipment to improve it. While other anaerobic treatment methods have mechanical mixers to improve the mixing process, ABRs use a mechanical principle based merely on the baffle arrangements. In order to enhance the mixing performance, changes in the structure and arrangement of baffles are tested using three-dimensional computational fluid dynamics (CFD) modelling. The effect of different configurations on the homogeneity of the mix, the residence time, and the dead spaces and flow patterns are analysed using a passive scalar tracer. The more the tracer is distributed and stays in the tank (particularly in contact with the bottom), the more potential contact between the wastewater, which brings nutrients, and the active biomass where bacteria are present. Several simulations with different Reynolds numbers are observed in a periodic domain. Three factors are considered in the baffle arrangements, these include: the effect of the ratio between downflow and upflow part, the angle of the slanted edges, and the distance of the baffle with respect to the bottom of the reactor. The CFD model is validated with experimental work to obtain the flow patterns and the instantaneous and averaged velocities, which are investigated using particle image velocimetry (PIV).

Energy-saving transportation of liquid in a Bend Tube: an endo-anodizing approach

Wei Li (2), Honghao Zhou(1), Kaiqi Zhao(2) Jian Jin(2) , Xue Chen(1) Muhammad Wakil Shahzad(1) , Yinzhu Jiang(2) , Sheng Dai(2), Lidong Sun(2) and Ben Bin Xu (1)

(1) Mechanical and Construction Engineering, Northumbria University, Newcastle upon Tyne, NE1 8ST, UK, (2) State Key Laboratory of Mechanical Transmission, School of Materials Science and Engineering, Chongqing University, Chongqing 400044, PR China

The bending-shaped tubes have been widely used in both industry and experiment applications. Here, we propose an endo-anodization strategy to attain an uniform superhydrophobic coating of TiO2 nanotube arrays throughout the deformed/bend Ti tubes inner surface. Led by a hybrid carbon fibre cathode, conformal electric field can be generated to adaptatively fit the complex geometries in the deformed tube, where the structural design with rigid insulating beads can self-stabilize the hybrid cathode at the coaxial position of the tube with the electrolyte flow. As a result, a superhydrophobic coating with a high water contact angle (157°) and low contact angle hysteresis (less than 10°) can be obtained. For the anodized U-shaped tube, the overall drag reduction can up to 25.8%. Furthermore, we demonstrate on spatially coat tubes with complex geometries, to achieve energy-saving liquid transportation. The CFD analysis is also conducted to understand the drag reduction effect on tube with different angles. This facile coating strategy has great implications in liquid transport processes with the user-friendly approach to engineer surface regardless of the deformation of tube/pipe, thus holding great promises in the applications as bio-medical, micro-fluidics, chemical engineering and oil/off-shore engineering.

Detachment of a tilting plate suspended by a capillary bridge

Matthew Butler (1,2) & Dominic Vella (1)

(1) University of Oxford, (2) University of Birmingham

When a small amount of liquid bridges the gap between two solid surfaces, its surface tension can provide a force that adheres the solids together. We consider a scenario where this capillary adhesion force is used to attach an otherwise free plate to a fixed planar surface. When a given load is applied to the centre of mass of the plate, there is a single equilibrium separation of the surfaces which is unstable. For a plate that is constrained to remain parallel to the surface, this unstable equilibrium acts as a limit to adhesion: if the plate is initially closer than this separation then it is pulled by surface tension towards the substrate, whereas if the separation is larger that this then the plate ultimately detaches. However, adding the effect of plate tilting (as well as possible translation of the liquid bridge) can change the ultimate fate of the plate. We use a simple 2D lubrication model to show that a small initial tilt can grow to eventually dominate the resulting dynamics. On longer timescales, we show that this tilting mechanism can cause anomalous detachment, and investigate how this detachment depends on the initial perturbation and size of the plate.

Capillary-scale solid rebounds: experiments, modelling and simulations

Carlos A. Galeano-Rios (1,2), Radu Cimpeanu (3,4,5), Isabelle A. Bauman (6), Annika MacEwen (6), Paul A. Milewski (2), Daniel M. Harris (6).

(1) School of Mathematics, University of Birmingham, (2) Department of Mathematical Sciences, University of Bath, (3) Mathematics Institute University of Warwick, (4) Mathematics Institute, University of Oxford, (5) Department of Mathematics, Imperial College London, (6) School of Engineering, Brown University.

A millimetre-size superhydrophobic sphere impacting on the free surface of a quiescent bath can be propelled back into the air by capillary effects and dynamic fluid forces, whilst transferring part of its energy to the fluid. We focus on the dependence of the coefficient of restitution, contact time and maximum surface deflection on the different physical parameters of the problem, covering the range from minimum impact velocities required to produce rebounds to impact velocities that cause the sinking of the sphere. Experiments, simulations and asymptotic analysis reveal trends in the rebound metrics, uncover new phenomena at both ends of the Weber number spectrum, and collapse the data. Direct numerical simulations using a pseudo-solid sphere successfully reproduce experimental data whilst also providing insight into flow quantities that are challenging to determine from experiments. A model based on matching the motion of a perfectly hydrophobic impactor to a linearised fluid free surface is validated against direct numerical simulations and used to efficiently explore the low-Weber-number regime. Our combination of experimental and modelling approaches allow us to explore the entirety of our target parameter space within this challenging multi-scale system.

Substrate curvature affects droplet splashing

Thomas C. Sykes (1), Benjamin Fudge (1), Miguel A. Quetzeri-Santiago (2), J. Rafael Castrejon-Pita (3), Alfonso A. Castrejon-Pita (1)

(1) Department of Engineering Science, University of Oxford, UK. (2) Faculty of Science and Technology, University of Twente, NL. (3) School of Engineering and Materials Science, Queen Mary University of London, UK.

Droplets impinging on solid substrates are encountered in a plethora of scenarios, including inkjet printing, bounded sprays and natural phenomena (e.g. raindrop interaction). For low impact velocities, droplets tend to simply spread across substrates, before potentially retracting, whereas for higher impact velocities droplets splash, breaking-up into a number of satellite droplets. The propensity of a droplet to splash is also affected by the droplet fluid properties, other pre-impact dynamics (e.g. direction and droplet shape), the surrounding gas dynamics, and substrate adhesive/cohesive forces (wettability). Understanding the conditions under which splashing occurs is often important (in e.g. printing, coating and contamination), usually to prevent the formation of smaller droplets with indiscriminate trajectories and unwanted aerosolisation of fluid. We assess the effect of geometry on droplet splashing, focusing on the effect of substrate curvature. Using high-speed imaging, we study a uniquely wide range of dry curved substrates, from (convex) spheres of radius order 1 mm, through flat substrates, to concave lenses, in addition to a wide range of impact velocities. The precision of our experiments enables the transition between simple deposition and splashing to be precisely delineated across the range of curvatures considered, with the effect of droplet-sphere alignment determined. We also examine the spreading dynamics post¬-impact using image processing, including contact angles, making progress towards uncovering the physical mechanism underpinning splashing on substrates with curvature.

Bifurcations and control of bubbles in Hele-Shaw cells

Alice Thompson & Joao Fontana

University of Manchester

The propagation of a deformable air finger or bubble into a fluid-filled channel with an imposed pressure gradient was first studied by Saffman and Taylor. Assuming large aspect ratio channels, the flow can be depth-averaged and the free-boundary problem for steady propagation solved by conformal mapping. Famously, at zero surface tension, fingers of any width may exist, but the inclusion of vanishingly small surface tension selects symmetric fingers of discrete finger widths. At finite surface tension, Vanden-Broeck later showed that other families of 'exotic' states exist, but these states are all linearly unstable. Introducing an axially-uniform centred constriction in the channel changes the bifurcation structure and leads to observable symmetric, asymmetric and oscillatory states. These phenomena can be predicted via depth-averaged modelling, and also observed in our experiments, with quantitative agreement between the two in appropriate parameter regimes. The centred constriction is a form of passive control, with relatively limited influence over linear stability achieved by altering the solutions themselves. In this talk, we will discuss the potentially far greater power of active feedback control to stabilise the many different solution branches in this system. Although depth-averaged models will never perfectly predict the solutions and bifurcation structure in the full 3D system, they can still be useful to design control strategies are effective in the full system. Finally, we discuss some of the practical obstacles we will encounter towards our ambition to combine feedback control with control-based continuation for direct experimental observation of stable or unstable propagation modes.

Colloidal particle focusing and sorting in microchannels via solute concentration gradients

Adnan Chakra, Naval Singh, Goran Vladisavljević, and Guido Bolognesi

Department of Chemical Engineering, Loughborough University, UK

In recent years, an increasing interest in harnessing chemical energy in the form of solute concentration gradient has led to the exploration of colloidal particle manipulation by diffusiophoresis (DP) and diffusioosmosis (DO) in microfluidic devices (MFDs) [1]. Different variations of MFDs have been used to filter and sort charged particles with the aid of other electrokinetic phenomena such as DP and DO [2]. In this study, we report a novel phenomenon that enables the pre-concentration and sorting of colloidal particles dispersed in a continuous flow within a straight microchannel. A bespoke Ψ-junction microfluidic device is used to generate a steady-state salt concentration gradient in the direction perpendicular to the flow. As a result, the charged colloidal particles dispersed in the inner solution accumulate into two symmetric regions of the channel by forming two distinct focusing spots (or peaks) at the junction. Downstream the channel, these two peaks converge towards the centre while their intensity increase. A few millimetres from the junction, they almost merge and a particle focusing of ca. 60% is achieved. In the absence of salt contrast or particle surface charge, no focusing effect occurs and the colloids remain homogeneously distributed in the inner solution. Therefore, the proposed particle manipulation strategy can be easily controlled by adjusting the salinity level of the outer solution and it can be exploited to sort particles based on their surface charge. Despite a similar colloid behaviour being reported in previous studies with a similar flow configuration [3], our experiments show that the observed particle dynamics is driven by a novel unreported physical mechanism, that combines DO, DP and hydrodynamic effects. The proposed method offers great potential for microfluidic bio-analytical testing applications, including bio-particle pre-concentration and filtration. [1] S. Shin, Physics of Fluids, 2020, 32, 101302 [2] M.K. Rasmussen, J.N. Pedersen, and R. Marie, Nature Communications, 2020, 11, 2337 [3] Abecassis, Benjamin & Cottin-Bizonne, Cécile & Ybert, Christophe & Ajdari, A & Bocquet, Lyderic, New Journal of Physics, 2009 , 11. 075022

Analysis of wall mass transfer in a turbulent pipe flow using extended proper orthogonal decomposition

Rasmus Korslund Schlander (1), George Papadakis (1), Stelios Rigopoulos (2)

(1) Department of Aeronautics, Imperial College London, London SW7 2AZ, UK (2) Department of Mechanical Engineering, Imperial College London, London SW7 2AZ, UK

A method is presented for investigating the role of coherent structures in turbulent flows on the timeaverage wall mass transfer by using extended proper orthogonal decomposition (EPOD). The data used for the analysis is from a direct numerical simulation (DNS) of a pipe flow with a Reynolds number of 5300 based on bulk velocity. A Fukagata Iwamoto Kasagi (FIK) identity equation is used to relate the fluctuating velocity and scalar fields to the time-average Sherwood number (Sh). The results show that the scalar-velocity correlations account for up to 65.8% of Sh depending on downstream location for a passive scalar. By applying the FIK identity on the POD/EPOD modes, which due to homogeneity are Fourier modes in the azimuthal direction, it is now possible to compute the individual contribution of the coherent structures to the average wall mass transfer. By applying the method on the DNS data set, it can be shown that the first 15 wavenumbers and 10 POD modes can reconstruct 50% of the Sh number for a passive scalar using velocity modes while containing only 30% of the turbulent kinetic energy.

On the effects of in-plane solidity on the different regimes of canopy flows

S. Nicholas (1), M. Omidyeganeh (1), A. Pinelli (1)

(1) Department of Mechanical and Aeronautical Engineering, City, University of London, UK.

We have carried out highly resolved Large Eddy Simulations (LES) of an incompressible, turbulent, open channel flows bound by a canopy of rigid laments that are flush-mounted, slender, normal to an impermeable wall. The Immersed Boundary (IBM) formulation that is adopted allows the computation to tackle the region occupied by the canopy directly by imposing the zero velocity condition on every single stem. In the current investigation, we have considered five canopy configurations that differ only in the solidity (lambda) obtained by varying the in plane solid fraction (obtained by increasing the stems number density) with the same canopy height. The selected solidity nominally falls in the so called 'transitions - dense' canopy configurations (i.e. lambda in [0:1 0:6]). While the above regime was investigated recently by altering the canopy height alone, the variation of the in-plane solid fraction and its effect on the overlying turbulent flow have not been explored. The former discriminates the scales of the outer flow that can penetrate into the canopy and control the overall interactions of the inner and outer flows. All the mean velocity profiles exhibit two inflection points, one located at the top of the canopy and the other closer to the bottom wall. While the origin of the inner flow corresponds to the solid wall, the origin for the outer layer can be identified by a virtual location below the tip of the canopy, similar to the one adopted in describing turbulent flows over rough walls. While the coarsest and the transitional scenarios share some features with the flows sharing the same solidity, obtained by changing the canopy depth, the densest case features a radically different behaviour with an almost complete decoupling of the interaction between the outer and the intra canopy flows. This suggests the existence of a new quantitative measure that characterises the transition beyond the value of lambda.

Linearised predictions of secondary currents in turbulent channels with topographical heterogeneity

Gerardo Zampino, Davide Lasagna, Bharath Ganapathisubramani

University of Southampton

When lateral variations of the surface topography are imposed to turbulent wall-bounded flows, secondary currents in the form of streamwise-aligned vortical structures are observed. In this work, a rapid tool for the prediction of these currents is proposed. The approach uses a small perturbation hypothesis, where the small parameter is proportional to the peak-to-peak amplitude of the topography. Linearized Reynolds-Averaged Navier-Stokes (RANS) equations are then derived, using the Spalart-Allmaras turbulence model to describe the transport of the eddy viscosity and a nonlinear Reynolds stress model to capture the anisotropy of Reynolds stress tensor. A Fourier decomposition method is utilised to obtain the solution over surfaces where the topography has arbitrary spectral characteristics. Since the governing equations are linear, the solution at each spanwise wavenumber can be obtained independently from the others. Surfaces with streamwise-aligned rectangular ridges in symmetric channels are then investigated as a function of the ridge width W and the separation (gap) between ridges G. The flow topology is in agreement with experimental and numerical data available in literature. In particular, the flow structure shows an upwash above the ridges and a downwash within the troughs. The streamwise vortical structures tend to fully occupy the space available within the through (or over the ridge) and the channel half-height when increasing G (or W). Weak tertiary structures occur when the gap (or width) is increased. Tertiary flows modify the local topology at the center of the through (or ridge) and flow reversal can be observed, in agreement with experimental and numerical data. The amplification of secondary structures as a function of W and G is also studied. Two main configurations are found for which the strength of secondary flows is maximum: one for a constant gap equal to about 0.6 and one for a constant width equal to about 0.6. These results suggest that the spanwise period alone is not sufficient to fully characterize the effect of surface modulations. Extensions of the approach to lateral variations of the surface roughness, commonly occurring in applications, are possible and will be discussed in the talk.

Heat transfer in next generation Gyroid Heat Exchangers.

I Qureshi (1), D. Borman (2), A. Burns (3), J. Summers (4), E. Greiciunas (5)

(1) ESPRC Centre for Doctoral Training in Fluid Dynamics, School of Computing, University of Leeds, LS2 9JT (2) School of Civil Engineering, University of Leeds (3) Chemical and Process Engineering, University of Leeds (4) School of Mechanical Engineering, University of Leeds (5) EG Fluids LTD, Leeds, United Kingdom, LS25 1NB

The purpose of this project is to identify and evaluate opportunities to signi cantly improve the heat transfer performance of Heat Exchangers (HE) by exploiting Additive Manufacturing (AM) techniques and Triply Periodic Minimal Surface (TPMS) architectures to create novel heat transfer surfaces which enhance mixing and address flow maldistribution. HE are devices which facilitate the transfer of heat between two or more fluid streams. The optimal design and efficient operation of HE are of particular interest in the aerospace industry. Here, HE are a crucial component in thermal management systems where they condition working fluids for various thermal duties throughout aircraft. Typically, enhancements in heat transfer performance are accomplished by increasing the e ective heat transfer surface area, generating turbulence or disrupting the thermal boundary layer. To that effect, secondary heat transfer surfaces, turbulators and corrugations are employed but require an increase in the required pumping power to overcome their associated frictional losses. This presents a large variety of possible surface configurations which can be utilised to balance heat transfer enhancement verses frictional losses, making the design process of HE challenging. Until recently, innovation of HE design was restricted by available manufacturing possibilities, however, modern AM techniques allow for the fabrication of compact and complex three-dimensional geometries which were previously impossible. This enables us to develop novel and efficient AM-HE geometries with true freedom of design. TPMS are a promising target for the next-generation of AM-HE. TPMS architectures are composed of two interpenetrating volume-domains separated by a single area minimising wall. Their complex geometries are only possible with AM. They demonstrate a number of properties that make them ideal candidates for heat transfer surfaces; Large surfaces area to volume ratios, superior mechanical and transport properties and fouling resistance.

To tread or not to tread?

Swathi Krishna; Alexander Gehrke, Karen Mulleners

Ecole Polytechnique federale de Lausanne, Switzerland

The wing kinematics of hovering insects are limited by their physiology to a conventional stroke (back and forth motion), and a rotation (change in angle-of-attack) predominantly. This rotation at the end of each half-stroke has implications on the large scale flow structures around the wing by pushing the leading edge vortex away, which further breaks down and leads to a loss in lift. Unlike biological fliers, engineering models like flapping wing micro air vehicles have fewer limitations in terms of joints and actuation, and can be designed to avoid this kind of rotation. Motivated by the possibilities in engineering applications, the benefits of a modified hovering motion, termed as the water-treading mode, is discussed here. As opposed to a designated leading and trailing edge in the normal hover, the water-treading mode allows for a dynamic interchange of the leading and trailing edge throughout the stroke. In this mode, the wing is rotated such that the leading edge becomes the trailing edge and vice-versa in every consecutive half-stroke. This eliminates the bluff-body dynamics observed in biological hover modes, which means that the leading edge vortex generated in one half-stroke does not breakdown and decay by the end of that half-stroke. Instead, it is captured by the trailing edge and continues to evolve in the next half-stroke. We hypothesize that this kinematic pattern would be more efficient than the normal flapping kinematics and improves the lift characteristics. Phase-locked particle image velocimetry and force measurements are carried out on a dynamically scaled robotic model in quiescent flow. The differences in the aerodynamic forces are explained with the help of the shear layer velocity and the relevant flow patterns observed.

Redistribution of a passive tracer in a porous substrate under a surface washing flow at high Péclet number

Emily C. Butler (1), Merlin A. Etzold (2), Francesco P. Contò (2), Stuart B. Dalziel (2), Joel Daou (1), Julien R. Landel (1)

(1) Department of Mathematics, University of Manchester, UK (2) Department of Applied Mathematics and Theoretical Physics, University of Cambridge, UK

We study the cleaning of a liquid chemical contaminant from the surface of a flat, wet porous medium. We focus on quantifying the extent to which cleaning the surface of the porous medium can spread the contaminant over the porous material. This modifies the concentration distribution within the porous material - potentially increasing the area that has been contaminated - compared with the case if there were no cleaning flow. To gain an understanding of this redistribution phenomenon, we propose a simple model for a shear washing flow over a small, flat drop releasing a dilute passive tracer of constant concentration, initially situated on the surface of the porous medium. As the flow progresses over the surface, the tracer is transported from the drop into the flow through advective and diffusive processes, while molecular diffusion across the flow-porous interface encourages mass transfer into the porous medium downstream of the drop. We will discuss an important asymptotic regime, based on an intermediate time scale for high-Péclet-number flow, valid at distances far downstream of the drop. In this regime, streamwise diffusion in the porous medium can be neglected, and, at leading order, a time-dependent one-dimensional diffusion equation governs the concentration field of tracer in the porous medium. As the formation of the diffusive boundary layer occurs faster than the intermediate time scale, mass transfer in the flow phase can be described by a quasi-steady two-dimensional advection-diffusion equation. In order to substantiate our analysis, we performed finite element simulations using COMSOL. The results of these simulations will be compared with our experimental data to confirm the mechanism redistribution of contaminant during the decontamination of porous media

Optimising tool channel design to improve coolant distribution and reduce cutting temperatures in machining processes

Eleanor Harvey (1), Harvey Thompson (1) , Adam Johns (2), Gaetano Pittala (2)

University of Leeds (1) Sandvik Coromant (2)

Coolants are frequently used in machining of hard-to-cut materials in order to combat the high thermo-mechanical loads experienced near the cutting edges of the tool. Managing the temperatures at the cutting edge has many benefits, including an extended tool life and improved surface integrity of the workpiece. A hostile environment near to the tool cutting edge, along with the area being visually obstructed by both the tool and workpiece, makes the behaviour of coolant difficult to observe and analyse in practical machining. Computational Fluid Dynamics (CFD) is a powerful tool in these applications. It can both provide valuable insight into the underlying flow behaviours which allow for improved machining results observed experimentally, and be utilised in the design process for developing new tools with optimised features. A multiphase Volume of Fluid model is combined with a conjugate heat transfer approach across the tool, coolant and the workpiece for the first time and applied to a machining case to investigate the milling of a titanium alloy. The model is then used to investigate the tool design by altering the coolant channel parameters. Also, the tool design is optimised for both maximum coolant coverage along the cutting edge and minimisation of tool temperatures in the same region.

Collective dynamics of active filaments on spherical surfaces

Timothy A. Westwood and Eric E. Keaveny

Imperial College London

Cilia are slender, microscopic organelles which are ubiquitous in the natural world. Large collections of cilia are used by a variety of microorganisms and swimming cells for propulsion, as well as by larger animals to facilitate fluid transport in their airways, brains and reproductive tracts. Cilia rely on their ability to spontaneously organise their collective motion to establish large-scale, coordinated patterns such as metachronal waves that break the time-reversal symmetry associated with viscosity-dominated flows. Despite the broad range of environments in which cilia are found, mathematical models that explore their coordination typically involve rectangular grids of model cilia on fixed, planar boundaries. While this provides a natural starting point, the effects of surface motion, as well as surface geometry and topology, on the coordinated state remain largely unexplored. Furthermore, collections of cilia are frequently modelled using simple rotor systems or filaments with prescribed kinematics . As such, the role of the cilia elasticity in their large-scale coordination also requires further study. In this work, we model cilia as elastic filaments whose motion is dynamically driven by a dynein-mimetic compressive force applied to the free end of the filament. As the magnitude of this force is increased the straight filament shape becomes unstable, leading first to three-dimensional whirling, then to planar beating, and finally to a `chaotic' writhing as the force is increased further. Each filament is discretised along its length into hydrodynamically-interacting segments according to the filament model of Schoeller et al. (2021). The filaments are clamped to planar boundaries, as well as spherical surfaces that are either held fixed or free to translate and rotate. The collective behaviour is observed for various filament densities and applied force magnitudes for these different surface conditions. On planar surfaces neighbouring filaments tend to synchronise both the phase and orientation of their motion. On fixed spheres, the filaments will still synchronise their motion, however, the change in topology introduces poles which we find can disturb the motion of filaments that are sufficiently close to them. When the sphere is no longer held fixed, the pole-induced differences in filament motion lead to translation and rotation of the surface, which further alters filament behaviour. This change occurs at both the level of individual filaments, where we find both novel motions and those associated with different force values, and at the collective level, where we observe the emergence of metachronal waves.

Shape, shear and diffusion effects on bacterial orientation distributions near walls

Smitha Maretvadakethope (1), Andrew Hazel (2), Bakhti Vasiev (1) and Rachel Bearon (1)

(1) Department of Mathematical Sciences, University of Liverpool. (2) School of Mathematics, University of Manchester

A common problem when developing healthcare technologies, like catheters, is the formation of bacterial biofilms which can lead to infections in patients. In the case of motile swimming bacteria, the early stages of biofilm colonization are dependent on the properties of the bacteria and the flow environments they are in. Specifically, physical characteristics like bacterial shape, bacterial swimming speeds, and fluid shear rate, affect the rate of surface colonisation and the effective diffusion of different bacteria. In our work we study the initial stages of biofilm development in two-dimensional Poiseuille flows. We study the effects of bacterial geometry and linearly varying shear throughout the bulk-flow on bacterial trajectories, and how the history of bulk-flow dynamics affects bacteria wall-interactions in a dilute suspension. We obtain population level features of the equilibrium problem through continuum modelling and use individual based stochastic models to analyse the observed features, explore cell behaviours throughout the flow, and study the resulting orientation distributions of swimmers in the absence of hydrodynamic wall-interactions. We determine the importance of the deterministic effects of swimming in Poiseuille flow on the probability orientation distribution of swimmers at the wall via analytic solutions, and use individual based models to study the roles of rotational and translational diffusion on the trajectories of bacteria interacting with the channel walls. These insights are used to determine how boundary conditions for continuum models need to be refined to account for shape and shear effects.

Combined effects of wall transpiration and plate movement on self-similar biomagnetic flow

Sadia Anjum Jumana and M. Ferdows

Research Group of Fluid Flow Modeling and Simulation, Department of Applied Mathematics, University of Dhaka, Dhaka-1000, Bangladesh.

The study of biomagnetic fluid with magnetic particles has received considerable attention from researchers recently owing to its wide range of application on biomedical and bioengineering science due to the biocompatibility, biodegradability, facile synthesis of magnetic particles. Magnetic Resonance Imaging (MRI), magnetic drug and gene delivery, cancer treatment, diagnosis and tumor, human eye treatment, hyperthermia purposes etc. are most common example where magnetic particles are used. A steady flow and heat transfer of self-similar biomagnetic (blood) boundary layer with magnetization over a permeable moving flat plate is considered. In this study, transpiration and movement velocity along the wall is imposed. The dimensionless equations (similarity equations) are generated by transforming the governing equations of the biomagnetic fluid due to magnetic dipole using the procedure of similarity technique. The explicit numerical method has been used to demonstrate the dimensionless non-linear, coupled, differential equations. The existence of dual solutions and their stability is reported by considering stability analysis suggested by existing literature. Through our analysis, we have found that two solutions exist for the arbitrary values of transpiration, movement velocity and biomagnetic interaction parameters on flow and physical parameters and also found stable and unstable solution. The achieved results are shown graphically and in tabular form.

Scalar and momentum transport across patches of cylinders

D.D. Wangsawijaya, C. Nicolai, B. Ganapathisubramani

Aeronautical and Astronautical Engineering, University of Southampton, Boldrewood Innovation Campus SO16 7QF, United Kingdom

Canopy flow is a widely encountered phenomenon in both environmental and engineering applications. Typically, a boundary layer develops over a surface and encounters an array of obstacles. Examples of such scenario include: flow past aquatic and terrestrial vegetations, atmospheric boundary layer developing over a cluster of tall buildings and wind farms. These obstacles are known to affect the scalar, momentum, and energy transport process in the flow downstream of the obstacles. In this study, we aim to explore the e ect of various canopy arrangements on the scalar and momentum transport across a simplified canopy model. A parametric study of turbulent boundary layers impinged by a circular patch consisting an array of cylinders, whose density is varied by systematically increasing the number of cylinders in a patch, is conducted in a water flume. Simultaneous PIV (particle image velocimetry) and PLIF (planar laser-induced fluorescence) measurements capture the instantaneous velocity and scalar concentration fields downstream of the circular patch. Preliminary results suggest that increasing the density of cylinders yields shorter wake behind the canopy and weaker horizontal (streamwise) scalar and momentum dispersion past the canopy. However, we also observe a stronger vertical scalar and momentum dispersion within the canopy that extends above the height of the cylinders as the density increases.

Cell-scale haemodynamics and transport in canonical disordered porous media: microfluidic experiments and theoretical models

Kerstin Schirrmann (1,2), Qi Zhou (3), Eleanor Doman (4), Oliver E. Jensen (4), Miguel O. Bernabeu (5), Timm Krüger (3), Igor L. Chernyavsky (1,4,6), Anne Juel (1,2)

(1) Manchester Centre for Nonlinear Dynamics, University of Manchester, UK; (2) Department of Physics and Astronomy, University of Manchester, UK; (3) School of Engineering, Institute for Multiscale Thermofluids, University of Edinburgh, UK; (4) Department of Mathematics, University of Manchester, UK; (5) Centre for Medical Informatics, Usher Institute, University of Edinburgh, UK; (6) Maternal and Fetal Health Research Centre, School of Medical Sciences, University of Manchester, UK

While extensive research has been devoted to fluid flows through porous media, the underlying mechanisms for the flow and transport of blood and nutrients in biological organs/tissues such as the highly porous human placenta are still unclear. As the size of flow channels within these systems becomes comparable with that of a red blood cell (RBC, diameter about 8 μm), the particulate character of blood gives rise to complex nonlinearity by introducing spatiotemporal heterogeneities that require microscopic interrogation. In this work, we aim to characterise the microscopic blood flow within canonical porous media. Porous media models are constructed by introducing different levels of disorder to regular obstacle arrays arranged on a square grid. Microfluidic experiments are performed with monodisperse oil droplets (diameter about 250 μm, modelling RBCs) in a water-glycerol mix. In our numerical model we use the lattice Boltzmann and immersed boundary methods to simulate blood flow as a suspension of deformable RBCs in plasma. Network models are used to rationalise observations from simulations and experiments. Our results show an intricate interplay of structural disorder, rheological properties and time-dependent effects on the localisation of droplets and RBCs within a porous medium, which is highly heterogeneous presenting preferential paths as well as emerging path occlusion. The overall hydrodynamic resistance of the suspension flow in a porous medium is shown to be more sensitive to the level of geometric disorder than in the case of the Newtonian flow. The role of RBCs in the intervillous space within the human placenta is multifold. The RBCs facilitate the transport of oxygen, CO2 and other solutes, whereas RBCs' localisation can significantly affect the flow patterns in the porous media. A multi-disciplinary approach, cross-validating simulations and experiments and extracting generalised constitutive relationships for RBC flow in complex geometries, will help bridge the gap between microscopic characterisation and organ-level modelling.

Pore-scale large-eddy simulations of turbulent flow in a composite porous-fluid system

Jadidi M., Revell A., Mahmoudi Y

Department of Mechanical, Aerospace and Civil Engineering (MACE), The University of Manchester, M13 9PL, UK

In the present study, the flow features for a channel partially filled with two different porous blocks and a solid block are investigated numerically using a detailed pore-scale large eddy simulation (LES) approach. The effect of the porous block on the coherent structures and large reverse flow patterns along the top surface of the blocks are investigated and compared with the solid block. Moreover, characteristics of the wake flow and the flow leakage from the porous region to the clear flow through the porous-fluid interface are examined in detail. Pore-scale LES results show that some portion of the fluid entering the porous blocks is pushed upwards toward the porous-fluid interface due to the negative vertical pressure gradient inside the porous block and leaves the porous structures to the clear region (positive leakage). This tendency of flow leads to a formation of counter-rotating vortex pairs (CRVPs) of the fluid flow within and above the porous block and changes the coherent structures above the porous-fluid interface. It is also argued that the magnitudes of turbulent kinetic energy and turbulent velocity fluctuations are decreased drastically over the porous block as compared with the solid block since the reverse flow on the porous-fluid interface is deteriorated by the positive leakage.

Shock-induced collapse of cavitating surface nanobubbles

Duncan Dockar, Livio Gibelli, Matthew K. Borg

School of Engineering, University of Edinburgh, Edinburgh EH9 3FB, United Kingdom

Cavitation is the phenomenon where a bubble of gas or vapour can spontaneously form within a liquid, in response to a local drop in pressure, and the collapse of these cavitation bubbles often releases a high-speed liquid jet, capable of surface damage. Engineers are currently looking to exploit the concentrated impacts of these jets for nano-/microscale applications, such as in surface cleaning, cancer treatment, drug and gene delivery, and waste-water treatment. In our previous work, we determined the pressure drop required to induce the rapid, unstable expansion of spherical cap-shaped surface nanobubbles resting on a solid substrate. However, despite their important role in heterogeneous cavitation, their collapsing dynamics have not yet been explored. Here, we present our current research on Molecular Dynamics (MD) simulations of shock-induced surface nanobubble collapse, with comparisons to the collapse of spherical bubbles near a substrate, investigating the differences in their jetting dynamics and potential damage capacity. We found that jets from the collapsing surface nanobubbles were weaker, with slower velocities, and wider and flatter leading profiles, due the underlying solid inhibiting their development. Consequently, the weaker jets resulted in lower impulsive impact pressures and reduced pitting damage, where the pit depths were dependent only on the surface nanobubble's contact angle, and not their initial diameter. We evaluated the differences in jet impact in terms of the local surface impulse across the substrate, which is closely related to the change in fluid momentum during jet development, and we show a linear scaling between the peak impulse and surface damage volume, across all tested bubble cases. For novel applications of collapsing surface nanobubbles, the weaker jets could be better suited around more delicate materials, and we also propose that the distinct pitting shape could allow unique identification of surface nanobubble collapse in experiments.

Dynamics of interfacial peeling

Sepideh Khodaparast

Leeds Institute of Fluid Dynamics (LIFD), School of Mechanical Engineering, University of Leeds

Interfacial peeling is a novel technique that employs the motion of a three-phase contact line to peel and detach soft hydrophobic films from a solid substrate. Here, we discuss the role of surface wettability, film thickness, liquid viscosity and interfacial tension on the peeling efficiency of this technique, based on experimental analyses using air-water interfaces on various solid substrates. Additionally, we monitor the effectiveness of this approach as the propagation speed of the air-water interface increases. At a threshold interfacial velocity (around 1 mm/s), the peeling efficiency dramatically decreases, and the liquid phase wets the film instead of creating a crack between the film and the substrate. The failure is caused by the local pinning-depinning of the contact line yielding a jumping behaviour with an amplitude that surpasses the film thickness at higher interfacial velocities. Finally, we report on the application of the interfacial peeling technology for removal of bacterial biofilms from their host surfaces. We show that the principles derived from the experimental measurements on synthetic polymeric films apply to viscoelastic biomaterials and allow removal and transfer of intact biofilms from one surface to another. Using fluid-based peeling for removal of hazardous soft materials offers new opportunities for chemical-free cleaning of surfaces contaminated by various sources of biofouling.

Hydrodynamics of flow over a flat plate with generic roughness elements

Melike Kurt, Jonathan Massey, Rodrigo Vilumbrales-Garcia, Gabriel Weymouth and Bharathram Ganapathisubramani

University of Southampton,

There have been signicant efforts to understand the hydrodynamic function of shark denticles due to their drag reducing and lift enhancing effects on the flow. Often, the researchers investigate these effects by using bio-mimicry, replicating the morphological properties of the shark denticles. However, it is unclear if it is the shape of the denticles that is important or if generic roughness on the surface would have the same effect. To test this hypothesis, we carry out experiments over a flat plate with hemispherical roughness elements that has similar dimensional to denticles. A parameter sweep is considered where different surface roughness distributions (i.e. coverage area) of plates at varying angles of attack and flow speeds. Using flow visualization and force measurements, the results from this parameter sweep will be presented in comparison with the flow over smooth surface case. The data will allow us to better understand the effect of such roughness elements on the flow physics and the resulting hydrodynamic performance.

Eigenmode decomposition as a foundation for exploring high-efficiency swimming

Amanda Smyth (1), Anna Young (2), Luca Di Mare (1)

(1) University of Oxford, (2) University of Bath

Replicating the high-efficiency swimming motion of marine animals has been one of the primary challenges in the field of bio-mimicking and bio-inspired underwater robotics. Significant difficulties are often encountered in attempts to reproduce the swimming motions seen in nature, and motions are often simplified to a combination of heaving and pitching of the tail fin. This work aims to contribute to improved understanding of the complex features of high-efficiency swimming motion seen in nature, through the study of hydrodynamic eigenmodes. In this work the eigenmodes are used as basis functions for the hydrodynamic response to external forcing, such that the modes define the set of possible responses of the hydrodynamic system to flow perturbations or body motion. Given the compactness of using eigenmodes to represent the connection between body motion and hydrodynamic response (circulation), we hypothesise that the eigenmodes correspond to high-efficiency swimming. Given this hypothesis, this work uses an inviscid, 3D vortex lattice model to investigate the properties of the hydrodynamic eigenmodes of different bio-inspired 3D geometries, and explores how these correspond to swimming motions seen in nature. The eigenmodes show remarkable similarity to qualitative observations of swimming in nature, and also correspond well to existing classification of swimming motion into "undulatory" and "oscillatory" swimmers. The ability of the eigenmodes to generate large thrust with low power is explored, and it is found that motion with high hydrodynamic efficiency can be achieved using any eigenmode from any of the body geometries tested. However, the "cut-in" frequency for positive thrust generation varies between modes and geometries. Likewise, the magnitude of thrust produced also varies between modes and geometries; as such, it is likely that the optimal choice of eigenmode to be used for swimming motion is goverened by practical constraints such as thrust requirements and body elasticity, rather than hydrodynamic efficiency. Overall, the ability of hydrodynamic eigenmodes to explain and explore the properties of high-efficiency swimming is promising for use in marine animal research, and in the development of autonomous underwater vehicles.

Recirculation regions in wakes with base bleed

Konstantinos Steiros (1), Nikolaos Bempedelis (1), Liuyang Ding (2)

(1) Imperial College London, Department of Aeronautics (2) Princeton University, Department of Mechanical and Aerospace Engineering

The appearance of detached recirculation regions in wakes with base bleed determines the aerodynamic properties of many natural organisms and technological applications. In this work we introduce an analytical model which captures certain key dimensions of the recirculation region in the wake of porous plates of infinite aspect ratio, along with the porosity range over it exists, when vortex shedding is absent or suppressed. The model is used to interpret why the recirculation region (i) emerges, (ii) migrates away from the body with increasing base bleed, (iii) disappears at a critical bleed and (iv) is partially insensitive to variations in the Reynolds number. The model predictions show considerable agreement with data from laboratory experiments and numerical simulations.

PIV and LES investigation of the flow around a succulent-inspired cylinder

Oleksandr Zhdanov, Richard Green, Angela Busse

James Watt School of Engineering, University of Glasgow

Unlike most other plants, tall arborescent succulents cannot rely on reconfiguration to cope with aerodynamic loadings due to their inflexible structure. Nevertheless, they can withstand high winds without being uprooted. The aerodynamic properties of a succulent-inspired cylinder with four ribs, whose cross-section resembles a square with concave sides and rounded corners, is known to be similar to the conventional square cylinder, e.g., they are insensitive to Reynolds number (Re) variations. In addition, as for the square cylinder, force coefficients and Strouhal number show a strong dependence on the angle of attack reaching their extrema at a critical angle of attack, but the succulent-shaped cylinder exhibits improved aerodynamic behaviour, e.g., a reduction of the drag coefficient at low angles of attack [1]. In the present study, the flow past an infinitely long succulent-inspired cylinder with four ribs was investigated using experimental and numerical approaches at two biologically relevant Reynolds numbers to obtain detailed insight into the flow field with focus on its changes around the critical angle of attack. Particle image velocimetry (PIV) measurements were taken at Reynolds number 50,000, while large eddy simulations (LES) were performed at Re = 20,000 for a range of angular orientations with respect to the freestream. Quantitative flow visualisation revealed that the presence of the critical angle of attack is associated with the reattachment of the separated shear layer to the back rib. In addition, characteristic large-scale flow structures around the cylinder and in its wake were identified and analysed using proper orthogonal decomposition (POD). [1] Zhdanov, O., Green, R. and Busse, A., 2021. Experimental investigation of the angle of attack dependence of the flow past a cactus-shaped cylinder with four ribs. Journal of Wind Engineering and Industrial Aerodynamics, 208, p.104400

Turbulent entrainment in large wind farms

N. Bempedelis (1), G. Deskos (2), S. Laizet (1)

(1) Department of Aeronautics, Imperial College London, London, UK, (2) National Renewable Energy Laboratory, Colorado, USA

In a recent press release, the UK government set out its plans to make the UK the world leader in clean wind energy. This target will be achieved mainly by deployment of large-scale offshore wind plants. In such plants, unlike isolated turbines or small wind farms, the physical mechanism that is primarily responsible for transporting the energy of the wind to the turbines' location is turbulence, as energy enters the system vertically, via entrainment, rather than horizontally. As a result, entrainment is a critical factor for the performance of very large wind farms. The main goal of this study is to characterize and quantify the role of the different mechanisms that govern energy transport and extraction in large wind farms. To this end, we use the well-established wind farm simulator Winc3d, which is part of the Xcompact3d framework of high-order finite-difference solvers dedicated to turbulence, to perform high fidelity turbulence-resolving simulations of the flow in wind farms of different layouts and sizes. The simulations reveal details about the role of entrainment, its relative importance according to the position of a turbine within the farm, and its overall relationship with the plant's power output. Finally, based on the above information, we explore the concept of position-varying farm designs to achieve increases in the power output of large-scale wind farms.

PIV investigation of flow inside a rotating tangent cylinder

Rishav Agrawal, Alban Potherat and Martin Holdsworth

Fluid and Complex Systems Research Centre, Coventry University, Coventry CV1 5FB, UK

The Earth's rotation provides a Taylor-Proudman constraint to its liquid core, which creates an imaginary cylindrical boundary inside the liquid core tangent to the solid core. This imaginary `tangent cylinder' (TC) extends up to the core-mantle boundary and opposes the fluid exchange between the regions inside and outside it. A few studies have been carried out in the past to study the effect of TC on the flow dynamics in the Earth's liquid core, however, there are still many open questions, for example, what are the possible convective states inside TC and how they behave? Answering these questions might provide a better insight into the Earth's core convection. The present study aims to explore the dynamics of flow inside the Earth's TC. A cylindrical geometry is chosen as the domain, inside which the TC will be generated. To create a TC, an electrically conducting fluid is to be simultaneously subjected to rotational, magnetic, and buoyancy forces. This is achieved by heating a cylinder filled with sulphuric acid from the bottom and cooling it from the outside (buoyancy) using water, placing it atop a rotating platform (rotating force), and then placing the entire platform inside the bore of a magnet (magnetic force). Sulphuric acid is used as the working fluid as it has high electrical conductivity and is also transparent which enables us to utilize Particle Image Velocimetry (PIV) technique. Both horizontal and vertical planes inside the TC are visualized using 2D-PIV measurements. As the first step, a stability study of the onset of convection will be carried out, and the resulting critical scaling for the relevant dimensionless parameters (Rayleigh number and Ekman number) will be obtained and analyzed.

Volcanic plumbing systems and the dynamics of magma ascent

Caitlin Chalk, Janine Kavanagh, Kate Williams, David Dennis

University of Liverpool

Volcanoes are arguably one of the world's most exciting and dangerous natural phenomena, yet most activity occurs beneath the surface of the Earth and is hidden from view. Prior to eruption, magma (molten rock, comprising melt, crystals and bubbles) is transported from kilometres deep within the interior toward the surface through a complex network of fractures in the Earth's rock layers. This subsurface movement of magma can be directly linked to measurable quantities such as seismic signals and ground surface deformation – data that are used by volcano observatories to monitor volcanic activity and anticipate if an eruption is imminent. A complete understanding of the dynamics of viscous magma ascending through solid rock – a multiphase fluid dynamics problem – is therefore vital for accurately forecasting volcanic eruptions. I will present ongoing work on experiments of fluid injected into a transparent, solid elastic material that provide an analogue for magma propagating through rock. Tomographic Particle Image Velocimetry is used to capture the three-dimensional dynamics of the ascending fluid in order to decipher the hidden mechanisms of volcanic plumbing systems.

Bubble curtains used as barriers across horizontal density stratifications

Alexis Bacot (1), Daria Frank (2), Paul Linden (2)

(1) École polytechnique, 91128 Palaiseau, France (2) Department of Applied Mathematics and Theoretical Physics, University of Cambridge, Wilberforce Road, Cambridge CB3 0WA, UK

Bubble curtains are multiphase line plumes that are used to reduce buoyancy-driven flows between two water zones at different densities. They are similar to air curtains, plane turbulent jets, that are installed in doorways of buildings between two climatically different environments. In this study, we establish a formal analogy between bubble curtains and air curtains and unify the two frameworks for their description that had previously been used. By means of small-scale laboratory experiments conducted in a channel with freshwater and brine solutions, we study how effectively a bubble curtain acts as a separation barrier for a wide range of density differences as well as different air fluxes and water depths. Qualitatively, two regimes of operation of a bubble curtain are identified and we establish the optimum operating conditions on the basis of quantitative measurements and theoretical considerations. We develop a theoretical model to calculate the infiltration flux of dense water across the bubble curtain that is in very good agreement with experimental measurements and yields a theoretical upper limit on the effectiveness of the bubble curtain. We also discuss the zones of mixed fluid around the bubble curtain, provide a scaling law for their horizontal extent as well as theoretically predict the water density inside these mixed zones.

Quasi-static magneto-hydrodynamic convection in a rotating cylinder

Anthony Rouquier, Abhishek Kumar, Alban Potherat

Fluid and Complex Systems Research Centre, Coventry University, United Kingdom

We present a numerical study on the topic of magnetoconvection in a cylinder under the quasi-static magneto-hydrodynamic approximation. The earth background rotation causes the convection of the Earth core to be divided in two zones due to the Taylor-Proudman constraint; inside and outside of a theoretical cylinder aligned with the Earth's rotation axis and touching the solid inner core, the tangent cylinder (TC). The aim of this work is to obtain a better understanding of the convection process in planetary cores by simulating a flow in similar condition as the one in the Earth TC. We consider the motion of an electrically conducting fluid in a closed cylinder, heated at the bottom, with electrically insulated boundaries. An external homogeneous magnetic field is imposed across the cylinder length. A spectral element approach is used to model the flow. Its specificity of the spectral-element method is that mesh refinement is mainly achieved by using polynomials of much higher order than a finite element method, allowing a higher degree of accuracy. A linear stability analysis of the system is conducted to find steady states and the onset of convection, then branch tracing, using a Newton-Raphson solver, will be used to study the phase space and find potential bifurcation points. The objective was to study how the onset of convection and the convective steady states are affected by the presence of both a magnetic field and rotation. The addition of either rotation and a magnetic field delay the onset of convection. However, for a given rotation rate, a magnetic field, will have the opposite effect. Steady states are also markedly different in the case where there is a magnetic field, the flow concentrate almost exclusively in the boundary regions.

Evaporation of sessile droplets on slippery liquid-like surfaces and slippery liquid-infused porous surfaces (SLIPS)

Steven Armstrong(1), Glen McHale(1), Gary G. Wells(1), Rodrigo Ledesma-Aguilar(1)

(1) Wetting, Interfacial Science & Engineering Lab, Institute of Multiscale Thermofluids, University of Edinburgh, UK

In sessile droplet evaporation, contact line pinning leads to a constant contact area mode and/or a receding contact line displaying stick-slip behaviour. However, this causes undesirable effects such as ring stain formation in applications such as inkjet printing and coating. Recently we have shown a droplet evaporating in a pinning-free mode at a constant hydrophobic contact angle determined by a "liquid Young's law"[1,2]. Here, we report pinning-free evaporation and droplet lifetimes across a far broader range of contact angles from 67°-105° using voltage programmable control of contact angles on two pinning-free surfaces[3]. The first uses a conformal slippery liquid-infused porous surface (SLIPS)[4], and the second used a Slippery Omniphobic Covalently Attached Liquid-like (SOCAL) solid coating[5]. We find the total time of evaporation over the contact angle range studied is only weakly dependent on the value of the contact angle. On these types of slippery surfaces, droplet lifetimes can be predicted and controlled by the droplet's volume and physical properties (density, diffusion coefficient, and vapour concentration difference to the vapour phase), largely independent of the precise value of contact angle. These results are relevant to applications, such as printing, spraying, coating, and other processes, where controlling droplet evaporation and drying is important. [1] G. McHale et al., Langmuir 35, 4197 (2019). [2] S. Armstrong, G. McHale et al., Langmuir 35, 2989 (2019). [3] S. Armstrong, G. McHale et al., Langmuir 36, 11332 (2020). [4] T-S Wong et al., Nature 477, 443 (2011). [5] L. Wang and T.J. McCarthy, Angew. Chemie - Int. Ed. 55, 244 (2016).

Kinetics of breath figure templating on photocurable polymer substrates

Francis Dent, Sepideh Khodaparast

University of Leeds

Novel nano/microscale surface patterning techniques enable enhanced functionality across domains spanning photonics to biomaterials. This application-driven pique has motivated the development of a range of smart fabrication technologies. However, many practical approaches are impaired by a lack of fundamental knowledge of the core mechanical mechanisms, limiting their adaptivity and controllability. One such patterning approach is a fluid-based droplet templating (breath figure) methodology initiated by drop-wise condensation on the surface of a polymer film. The breath figure method relies on interfacial self-assembling forces to order the stabilised water droplets into a hexagonally packed lattice prior to polymer curing. To achieve tunability in the final design, condensation is induced by sub-cooling the films below the dew point with a thermoelectric cooler; the droplet growth period is altered by photocuring the patterned polymer at varying process times. Here, we perform real-time systematic microscope measurements on a variety of photocurable films, investigating the kinetics of droplet nucleation, growth and assembly. We find all samples exhibit a maximum fractional area coverage of around 0.7, with the initial droplet diameter growth scaling with time to the power 1/3, as predicted in previous theoretical analysis. Subsequent curing and analysis of the resultant solid micropatterned surfaces are performed using atomic force and scanning electron microscopy

Dielectrowetting of low dielectric films in high dielectric liquids

Andrew M. J. Edwards (1), Élfego Ruiz-Gutiérrez (2), Michael I. Newton (1), Gary G. Wells (2), Rodrigo Ledesma-Aguilar (2), Glen McHale (2) and Carl V. Brown (1)

(1) SOFT Group, School of Science and Technology, Nottingham Trent University, Clifton Lane, Nottingham NG11 8NS, UK. (2) Institute for Multiscale Thermofluids, School of Engineering, University of Edinburgh, The King's Buildings, Mayfield Road, Edinburgh EH9 3FB, UK.

The static wetting behaviour of a solid surface by a droplet of an electrically insulating liquid in air can be modified using surface localised non-uniform electric fields, created using a voltage applied to embedded interdigitated electrodes. Our previous work has shown how liquid dielectrophoresis can force a droplet in air to spread over the surface, reducing the contact angle in an effect known as dielectrowetting. Conversely, we have further shown that when devices are fully immersed in a dielectric liquid our dielectrowetting techniques can detach a captive air bubble, levitate the bubble at a voltage controlled distance below the surface, and reattach the bubble on demand. Here we demonstrate dielectrowetting control over a low dielectric constant liquid film which fully wets a solid surface in equilibrium, creating a liquid layer between the surface and an immersing immiscible high dielectric constant liquid. When the voltage is applied, we show how negative dielectrowetting can dewet the film and create a liquid bubble with a contact angle that can be controlled over the entire range from 0° to 180°. In the limit that the liquid film is thick (> 100 μm) we have found excellent agreement with the dielectrowetting modified form of Young's law across the full contact angle range. Moreover, when the thickness of the low dielectric permittivity film is significantly reduced, the liquid dielectrophoresis forces induce destabilisation of the film producing various interesting rupture patterns. These observations are well described by our recently developed Lattice-Boltzmann simulations that solve the hydrodynamic equations for dielectric liquids in the presence of electrostatic fields.

Multifaceted design optimization for superomniphobic surfaces

J. R. Panter (1), Y. Gizaw (2) and H. Kusumaatmaja (1)

(1) Department of Physics, Durham University, Durham DH1 3LE, UK. (2) Procter and Gamble Co., Winton Hill Business Center, 6210 Center Hill Avenue, Cincinnati, OH, USA.

Superomniphobic textures are at the frontier of surface design for vast arrays of applications. Despite recent substantial advances in fabrication methods for reentrant and doubly reentrant microstructures, design optimization remains a major challenge. We overcome this in two stages. First, we develop readily generalizable computational methods to systematically survey three key wetting properties: contact angle hysteresis, critical pressure, and minimum energy wetting barrier. For each, we uncover multiple competing mechanisms, leading to the development of quantitative models and correction of inaccurate assumptions in prevailing models. Second, we find these wetting properties are antagonistically coupled. Interestingly, combine these analyses simultaneously leads us to an optimal texture that is very similar to mushroom shaped patterns utilised by the hexapod springtails to repel liquids and breathe through their skin.

Droplet wetting and self-propulsion on liquid surfaces

Glen McHale (1), Gary G. Wells (1), Rodrigo Ledesma-Aguilar (1), Halim Kusumaatmaja (2)

(1) Institute for Multiscale Thermofluids, School of Engineering, University of Edinburgh, The King's Buildings, Mayfield Road, Edinburgh EH9 3FB, UK. (2) Department of Physics, Durham University, Durham DH1 3LE, UK

The equilibrium state of a droplet on a solid surface is often described using the Young's equation for the contact angle. However, chemical and physical heterogeneities mean this equilibrium contact angle is rarely observed on real solid surfaces where contact line pinning is common. Recently, slippery lubricant impregnated surfaces (LIS), inspired by the surface of the Nepenthes pitcher plant, with little or no contact line pinning have been created. Droplets on these surfaces have little or no contact with the underlying solid therefore raising the question of what it means to have a contact angle for a droplet on a liquid or lubricant surface. Here, we interpret LIS as having effective interfacial energies described by Cassie-weighted area fractions of lubricant and solid interfacial tensions. This allows us to describe sessile droplets on thin (conformal) lubricant impregnated surfaces by a general Young's equation. In the limit of a continuously lubricant coated surface provides a liquid form of Young's law and allows droplet-on-liquid wettability to be defined. We show these concepts allow the design of low friction surfaces for uni- and bi-directional droplet self-propulsion. These results have wide applicability to creating topographically simple and complex, slippery lubricated surfaces for applications such as microfluidics.

Effects of porosity on the flow structure in the outer region of turbulent boundary layers

Prateek Jaiswal and Bharath Ganapathisubramani

University of Southampton, Southampton, UK

Two aspects of porous materials that substantially influence the turbulent boundary layer convecting over them are surface roughness and permeability. Although their individual effects on the flow are well documented, their combined action and non-linear interaction are poorly understood. Nevertheless, nature and engineering are replete with examples of such flows. Therefore, detailed experimental measurements of turbulent flows over and past polyethene foams, with varying porosity, have been performed. In particular, three foams with a varying pore size (s), permeability (K) and thickness (h) were used, which resulted in six distinct cases. The principal aim of the present study is to quantify the impact of porous surfaces on the large-scale energetic structures within the outer layer. A single hot-wire probe and PIV with a high dynamic spatial range were used to estimate first, second, third-order moments of velocity statistics and power spectral density of velocity. The results confirm Townsend's outer-layer hypothesis holds when the velocity is scaled with outer-layer variables, in either of the six cases studied.

Vortical structures of stratified shear layers in an inclined duct

Xianyang Jiang, Adrien Lefauve, Stuart B. Dalziel and P. F. Linden

Department of Applied Mathematics and Theoretical Physics, Centre for Mathematical Sciences, University of Cambridge, Wilberforce Road, Cambridge CB3 0WA, UK

We study Eulerian vortical coherent structures to improve our understanding of stratified turbulent mixing in shear flows. We focus on the sustained stratified shear flow in an inclined duct and use three dimensional, time-resolved experimental datasets providing the density and three-component velocity field in various flow regimes [1]. We analyse the velocity gradient tensor from our experimental data to decompose vorticity into a pure shearing part and a rigid rotational part, the latter defining a convenient 'rotex vector' [2]. We report weighted conditional statistical distribution of the vortical structures from the Holmboe wave regime to the fully turbulent regime. The results show that the vortices in the turbulent stratified shear layers tend to be hairpin-like, which appear to develop from confined Holmboe waves [3]. The vortices are stretched by the shear layer at higher Reynolds numbers, with their inclined angle relative to the streamwise axis approximately 20 deg. We also discuss the role of vortical and shearing structures on the buoyancy flux and energy dissipation, to shed light on mixing efficiency. For example, the lift-up behaviour of these vortical structures induces vertical motions that enhances the transport of momentum and buoyancy. [1] Partridge, Lefauve & Dalziel, Meas. Sci. Tech. 30 055203 (2019) [2] Gao & Liu, Phys. Fluids 30, 085107 (2018) [3] Lefauve, Partridge, Zhou, Caulfield, Dalziel & Linden, J. Fluid Mech. 848, 508-544 (2018)

Experimental study of vortex ring impingement: three-dimensional flow field, wall shear stress and heat transfer

Qianhui Li, Zhengzhong Sun, Qiang Zhang, Christoph H. Bruecker

School of Mathematics, Computer Science and Engineering, City, University of London, Northampton square, London, EC1V 0HB, UK

A scanning PIV set-up is integrated to the flow rig to study the time-resolved 3D velocity field. A rotating glass cube is used for scanning the light sheet through the volume at 100 scans/sec while the camera is continuously recording in high-speed mode at 5kHz frame rate. This generates a set of 50 overlapping sheets of illuminated volumes which are later processed into voxel stacks with the method of isotropic voxel-reconstruction using a Gaussian interpolation technique. 3D Cross-Correlation provides the time-resolved velocity field data. In addition, arrays of 500μm long micro-pillar sensors are implemented to map the WSS evolution. Finally, the heat transfer characters are measured by infrared (IR) thermography.Results show that the spatio-temporal evolution of the heat transfer coefficient (HTC) is correlated to the WSS induced by vortex ring impingement. The spatial peak WSS and HTC appear under the core of the primary vortex ring where there is the thinnest boundary layer. The temporal peak values of the WSS and HTC occur when the boundary layer starts rolling up. Thereafter, both values decrease during the development of secondary vortical structures. These results have beneficial implications for the design of the heat transfer enhancement fluidic device. Acknowledgment: Support from The Engineering and Physical Sciences Research Council (EPSRC) [EP/T006315/1] is gratefully acknowledged; The position of Professor Christoph Bruecker is jointly funded by BAE System and the Royal Academy of Engineering [Grant number RCSRF1617\4\11].

Experimental properties of continuously-forced, shear-driven, stratified turbulence

Adrien Lefauve and P. F. Linden

Department of Applied Mathematics and Theoretical Physics, Centre for Mathematical Sciences, University of Cambridge, Wilberforce Road, Cambridge CB3 0WA, UK

We study the experimental properties of exchange flows in a stratified inclined duct (SID), which are simultaneously turbulent, strongly stratified by a mean vertical density gradient, driven by a mean vertical shear, and continuously forced by gravity [1]. We focus on the 'core' shear layer away from the duct walls, where these flows are excellent experimentally-realisable approximations of canonical hyperbolic-tangent stratified shear layers, whose forcing allows mean and turbulent properties to reach quasi-steady states. We analyse state-of-the-art data sets of the time-resolved density and velocity in three-dimensional sub-volumes of the duct in 16 experiments covering a range of flow regimes (Holmboe waves, intermittent turbulence, full turbulence) [2]. In this talk we focus on the turbulent energetics and mixing statistics. We derive the kinetic and scalar energy budgets and explain the specificity and scalings of SID turbulence. Then, we assess the relevance of standard mixing parameterisations models relying on uniform eddy diffusivities, mixing lengths, flux parameters, buoyancy Reynolds numbers or turbulent Froude numbers, and we compare representative values with the stratified mixing literature. The dependence of these measures of mixing on controllable flow parameters is also elucidated, providing asymptotic estimates that may be extrapolated to more strongly turbulent flows, quantified by the product of the tilt angle of the duct and the Reynolds number. These insights may serve as benchmark for the future generation of experimental data with superior spatio-temporal resolution required to probe increasingly vigorous turbulence. [1] Meyer & Linden, J. Fluid Mech. 753, 242-253 (2014) [2] Lefauve, Partridge & Linden, J. Fluid Mech. 875, 657-698 (2019)

Generalised Quasilinear approximation of a two-dimensional Kolmogorov flow exhibiting spatially localised turbulent states

Hannah Kreczak (1), Steve Tobias (2), Brad Marston (3)

(1) Newcastle University, UK. (2) University of Leeds, UK. (3) Brown University, USA

Recently Hiruta and Toh (2020) confirmed a subcritical laminar-turbulent transition occurs in direct numerical simulation of a purely two-dimensional simple Navier-Stokes flow without walls. The flow is a doubly periodic, extended 2D Kolmogorov flow with a linear drag and finite unit flow rate imposed. For large enough Reynolds number, the system exhibits bistability of laminar and turbulent states resulting in the formation of spatially localised turbulent states (SLTs) which can persist indefinitely within the domain. Quasilinear theory (QL) is often utilized to approximate the dynamics of fluids exhibiting significant interactions between mean flows and eddies, and the Generalised Quasilinear (GQL) approximation is an extension of this theory. GQL includes large scale mode interactions via a spectral filter, such that nonlinear interactions involving only small zonal scales are removed. We examine the effectiveness of QL and GQL in capturing the subcritical transition to turbulence and SLTs in the extended 2D Kolmogorov flow. Early results show that the QL approximation is unable to replicate SLTs and that for GQL SLTs may persist although not at the length-scale they exhibit in direct numerical simulation.

Shear driven magneto-buoyancy under the influence of rotation

Craig Duguid, Paul Bushby, Toby Wood

Newcastle University

The primary candidate for the production of many of the surface and atmospheric phenomena we observe on the Sun (such as sunspots, flares and prominences) is the magneto-buoyancy instability. The idea of magneto-buoyancy is that within a magnetic flux tube the gas pressure is reduced as a result of the Lorentz force (which acts as a magnetic pressure). As such the gas within the tube has lower density than the ambient and so becomes buoyantly unstable and rises. One key mechanism in this process is the Solar differential rotation profile which consists of a strong shear region at the base of the convective zone, known as the tachocline. This vertical shear stretches out the poloidal (vertical) field converting it into toroidal (azimuthal/horizontal) field. It is this toroidal field which rises and breaks out the surface of the Sun resulting in the observed phenomena. Although previous works have explored the interaction between an imposed vertical shear and a vertical (poloidal) field, the influence of rotation has not been fully explored. We will present preliminary work on our exploration of this problem.

A new residual distribution solver for galaxy formation simulations

Ben Morton, Sadegh Khochfar

Institute for Astronomy, University of Edinburgh

The residual distribution (RD) family of solvers provide a powerful, truly multi-dimensional method for modelling the behaviour of fluids. Built around an unstructured mesh of triangular elements, this approach performs the flux integral in all dimensions in a single calculation, avoiding all dimensional splitting found in most finite volume approaches. We present our new implementation of a full 3D gravo-hydrodynamic solver, with our focus targeted at possibilities for galaxy formation simulations. RD solvers have not been widely utilised for astrophysical simulations, and so have significant untapped potential for modelling the complex flows present in many systems, across a wide range of mass, and length, scales. We cover extensive testing of the implementation's abilities to resolve important features, such as shocks and explosions, and include direct comparison to state-of-the-art methods currently used in the field. We also include our novel adaptive time-stepping mechanism for desynchronised residual calculation, which significantly improves the computational efficiency of the solver.

Diffusion and dispersion in anisotropic magnetohydrodynamic turbulence

J. Pratt (1), A. Busse (2), W.-C. Mueller (3)

(1) Georgia State University, (2) University of Glasgow, (3) TU Berlin

Magnetohydrodynamic (MHD) turbulence structured by a large-scale magnetic field is an essential aspect of interstellar or interplanetary plasmas. Here we investigate diffusion and dispersion in anisotropic MHD turbulence. We adopt the Lagrangian viewpoint, the natural point of view to study diffusion, and construct statistics based on the trajectories of Lagrangian tracer particles. From the motions of these tracer particles, we produce Lagrangian statistics such as single-particle diffusion, two-particle dispersion, and velocity autocorrelations. We also demonstrate new Lagrangian statistics developed to understand anisotropic turbulent dispersion. Simulation results will be presented that are performed using grid sizes up to 2048^3. Diffusion and transport processes in turbulent plasmas constitute fundamental astrophysical problems; a clear understanding of these processes is needed in order to produce improved theoretical models for the diffusion and transport of energetic particles, including cosmic rays.

Magnetic layering in the solar radiative zone

Fryderyk Wilczynski, David W. Hughes, Evy Kersalé

School of Mathematics, University of Leeds

A horizontal magnetic field, stratified with depth, can become unstable to the instability mechanism known as magnetic buoyancy. This instability is important in an astrophysical context, being the primary candidate for the release of magnetic field from the solar interior. Magnetic buoyancy is a form of double-diffusive instability – one in which the density depends on two components that diffuse at different rates, with the two components in this context being magnetic field and heat. Double-diffusive systems are important both in an astrophysical and geophysical context. The most well-studied such system, significant in an oceanographic context, is that of thermohaline convection, in which the two components are heat and salt, with the diffusivity of heat much greater than that of salt. The most important aspect of thermohaline convection is that the long-term evolution can lead to layered (or staircase) profiles in density, temperature and salinity, which leads to dramatic enhancements in turbulent transport. Under certain restrictive assumptions there exists a non-trivial analogy between equations governing magnetic buoyancy instabilities and those describing thermohaline convection. The restrictions are that the flow is two-dimensional, and that the atmosphere is weakly stratified. Under these restrictions, it necessarily follows that diffusive magnetic buoyancy instabilities will lead to staircases (in magnetic field, entropy and density) and to enhanced turbulent transport. The fascinating question that we set out to investigate is: what is the nature of the layering process in the astrophysically more realistic case of strongly stratified atmospheres?

11B-5

Non-linear simulations of tidal flows in an adiabatic convective shell

Aurélie Astoul, Adrian Barker

Department of Applied Mathematics, School of Mathematics, University of Leeds, Leeds, LS2 9JT, UK

In close two-body astrophysical systems, like binary star or Hot-Jupiter systems, tidal interactions often drive dynamical evolution on secular timescales. Many host stars feature a magnetised and convective envelope, and such layer may also be present in the envelopes of giant gaseous planets. Tidal flows that are generated there, due to the tidal potential of the companion, are dissipated by viscous friction mechanisms which lead to the redistribution of angular momentum in the convective shell. In the tightest systems, non-linear effects are likely to have a significant impact on the tidal dissipation and change the zonal flows, triggering differential rotation, as demonstrated in the hydrodynamical study of Favier+2014. In this context, we investigate how the addition of non-linearities affect the tidal flow properties, and the energy and angular momentum balances, thanks to 3D non-linear simulations of a neutrally-stratified (adiabatic) and incompressible convective shell using the pseudo-spectral code MagIC. In this talk, we will mostly present the hydrodynamical case which is intended as a preliminary study before adding a magnetic field. Unlike the above-mentioned study, we have chosen a body forcing where the equilibrium tide (the quasi-hydrostatic tidal flow component) acts as an effective force to excite tidal inertial waves, while using stress-free boundary conditions. With this more realistic set-up, we show new results for angular momentum evolution, for the amplitude of the energy stored in zonal flows, and its consequences on tidal dissipation.

Controlling the breakup of toroidal liquid films on solid surfaces

Glen McHale (1), Carl V. Brown (2), Andrew M. J. Edwards (2), Élfego Ruiz-Gutiérrez (1), Michael I. Newton (2), Gary G. Wells (1), Rodrigo Ledesma-Aguilar (1)

1) Institute for Multiscale Thermofluids, School of Engineering, University of Edinburgh, The King's Buildings, Mayfield Road, Edinburgh EH9 3FB, UK. (2) SOFT Group, School of Science and Technology, Nottingham Trent University, Clifton Lane, Nottingham NG11 8NS, UK

The ability to create liquid films and control how they break up into droplets is important for processes such as drying, printing, coating and lubrication. The break up of a film often involves the nucleation of holes and the growth of dry spots in a relatively uncontrolled manner. Here we report the use of non-uniform electric fields to create liquid films in carefully design shapes (stripes, disks, squares and rings) on surfaces that are intrinsically non-wetting. By removing or reducing the electric field it is then possible to study the dynamics and stability pathways for dewetting from film to one or more droplet states. This allows the identification of different stages of dewetting from an initial constant speed retreat of the contact line to an exponential approach to an equilibrium shape. In the specific geometry of a ring, the dewetting stage is followed by a Rayleigh-Plateau instability, which causes the ring to break up into one or more droplets. We demonstrate control over these instability pathways and show the selection of breakup modes.

Dynamics of respiratory saliva droplets

Avshalom Offner and Jacques Vanneste

School of Mathematics, The University of Edinburgh

We theoretically study the dynamics of respiratory saliva droplet as they travel through air while vaporising. Our model is governed by a set of ordinary differential equations (ODEs) for the droplet position and velocity, radius and temperature, reflecting the conservation of momentum (Maxey-Riley model), mass and energy, respectively. For small Reynolds number droplets, the ODEs for the radius and temperature can be decoupled from the others, allowing analytic approximations to be derived for the terminal droplet radius as well as for the time until this radius is reached. To numerically solve the full system of ODEs, a model for the air velocity is required. Attempting to provide sensible estimates on the droplet airborne time before touching the ground, we simplify the model by considering a one-dimensional problem (aligned with gravity), in which an indoor vertical air velocity is described using an Ornstein–Uhlenbeck process. The model results suggest that saliva droplets can stay aloft for up to several hours. Droplets typically shrink to their terminal radius in about 1-10 seconds, during which they may fall a considerable distance depending on their initial size. Having reached their terminal radius, droplets are almost fully entrained with the air flow. Air velocity fluctuations result in non-negligible probability of droplets rising above their original height. In practice, this means that people within indoor spaces may inhale infectious droplets by passing through a location previously occupied by a person carrying the virus. These results may assist in shaping and adjusting social-distancing guidelines, as well as affect regulation on the allowed number of people within indoor spaces.

Robust interpolation schemes for dispersed particle flows using the Full Lagrangian Approach

Chris Stafford, Oyuna Rybdylova

Advanced Engineering Centre, School of Computing, Engineering and Mathematics, University of Brighton, Brighton BN2 4GJ

The ubiquitousness of dispersed multiphase flows in industrial and environmental applications has resulted in the development of a range of modelling and simulation approaches for describing the behaviour of these flows. Of central importance in the modelling of such flows is accurate representation of the spatial distribution of the dispersed phase, which can exhibit wide variation in unsteady or turbulent flows. Whilst Eulerian continuum models are capable of capturing this spatial variation, their averaged nature necessarily means that some of the detail in areas of high variation is lost in the ensuing description of the dispersed phase. The Full Lagrangian Approach addresses this problem by tracking individual particles through the flow, and by computing the Eulerian-Lagrangian transformation tensor as the flow evolves, this method is able to provide the evolution of the local particle number density along each trajectory. The advantage of such an approach is that relatively few trajectories need to be tracked in order to produce a faithful representation of the Eulerian number density field, when compared to traditional box-counting methods. Reconstruction of the number density field does however require that a suitable numerical scheme is used to interpolate the Lagrangian data between trajectories. This work demonstrates the application of different meshfree approaches for estimating the particle number density field. In particular, a Delaunay triangulation is used to produce a simple linear interpolation, whilst a kernel smoothing approach is able to provide a higher degree of accuracy, and the efficacy of both methods is compared against a traditional Cloud-In- Cell procedure. A synthetic turbulent flow field is used to assess the performance of the developed interpolation procedures in situations where the dispersed phase has defined clusters and voids, and the methodology is also applied to a simple flow configuration in OpenFOAM.

Nanobubble nucleation from acoustothermal physics

Saikat Datta, Rohit Pillai, and Matthew K Borg

School of Engineering, Institute of Multiscale Thermofluids, The University of Edinburgh, Edinburgh EH9 3FB, UK

Nanobubbles have immense potential in applications as diverse as selective targeting of cancerous tumour cells, ultrasound contrast agents for disease diagnostics, and deep cleaning of precision tools. Ultrasonic vibration of a surface adjacent to the liquid is a well-established technique to produce macroscopic bubbles on a surface. The underpinning physics behind this spontaneous nucleation of bubbles is cavitation, which is driven by local oscillations in fluid pressure. However, recently we have revealed that high-frequency nanoscale vibrations can generate large "acoustothermal" forces, which superheats the liquid within few nanometres of the surface. This can rapidly take the liquid to the vicinity of the liquid-vapor spinodal, causing boiling, and this generates nucleation of bubbles near the surface. Current macroscale experiments suggest that, when compared to hydrophilic surfaces, the onset of nucleate boiling occurs at a lower temperature, and cavitation occurs at a lower vibration threshold, for hydrophobic surfaces. On the contrary, molecular dynamics studies have demonstrated that the opposite occurs at the nanoscale. In this work, high-fidelity molecular dynamics simulations are performed to clarify this longstanding contradiction in the literature by elucidating the underpinning mechanisms behind acoustothermal nucleation of surface nanobubbles, a configuration which allows us to control both vibrations and heating precisely. We show that the type of nucleation event (cavitation vs boiling) and preferred surface type (hydrophilic vs hydrophobic) can be tuned by varying vibration parameters only. We unveil for the first time the counteracting effects of surface wettability on vibration-induced nanobubble nucleation – higher wettability increases the energy requirements for nucleation; however, it also produces higher energy dissipation upon vibration compared to lower wettabilities. This work has broad scope for future research and may find applications in many future generations of clinical and engineering devices.

Efficacy of turbulence modeling techniques in capturing dynamic stall at low Reynolds numbers

Chandan Bose (1), Vincent Terrapon (1), Grigorios Dimitriadis (1)

Department of Aerospace and Mechanical Engineering, University of Liège, Liège 4000, Belgium

Dynamic stall is a subject of continued interest in the aerodynamics and aeroelasticity communities due to its broad range of engineering and biological applications, such as helicopter rotor blades, rapidly maneuvering flight, wind turbines, jet engine compressor blades, or flapping of insect wings. This study focuses on investigating the unsteady flow physics of the dynamic stall phenomenon associated with a harmonically pitching NACA 0012 airfoil for different reduced frequencies (k) at Re = O(104) and comparing the evolution of the unsteady flow structures. To that end, the effect of turbulence modeling on the prediction of Dynamic Stall is investigated using Unsteady Reynolds Averaged Navier Stokes (URANS) and Improved Delayed Detached Eddy Simulation (IDDES) techniques. The k- ω Shear Stress Transport (SST) model is adopted for URANS closure and the Spalart–Allmaras-based IDDES strategy is used in this study. The simulation results are compared to those obtained from low-speed wind tunnel experiments. It is observed that 3D IDDES outperforms 2D URANS in capturing the reattachment phase and the dynamic stall onset due to its ability to resolve finer turbulence scales that play an important role in highly separated flows. The present findings hint that there is a need for 3D scale resolving techniques to capture the 3D nature of dynamic stall. This study will further be extended for three different airfoil sections, i.e., a flat plate, a NACA0012, and a NACA0018 in order to investigate the effect of thickness on the nature of dynamic stall, such as light and deep dynamic stall.

Far-field acoustic investigation surrounding a separated aerofoil using time-resolved PIV

Douglas W. Carter, Bharathram Ganapathisubramani

University of Southampton

We present an analysis using time-resolved particle image velocimetry measurements (PIV) to reconstruct the instantaneous pressure field and extract the far-field acoustics emanating from NACA 0012 and NACA 65410 aerofoils at a chord-based Reynolds number Re=75000. By redirecting a high-speed Nd:YLF laser sheet into the test section simultaneously from opposing sides, the entirety of the velocity fields in the suction, pressure, and trailing regions of the aerofoils was captured using three overlapping high-speed cameras. The fidelity of the planar pressure reconstruction from the data is evaluated using a comparison to 2D RANS simulations under dynamically similar conditions. We find good agreement in the pressure fields except near the leading edge where the mean velocity gradients are highest. Using Curle's acoustic analogy the far-field acoustic pressure is extracted and mechanisms of noise generation are suggested for angles of attack ranging from attached to fully stalled conditions. Difficulties, caveats, and shortcomings of the approach are also discussed

Modelling rough and porous media with an adaptive mesh refinement Lattice Boltzmann Method

Mikaël Grondeau, Ralf Deiterding

University of Southampton, Engineering and Physical Sciences Faculty, Aerodynamics and Fluid Mechanics Research Group, Southampton SO17 1BJ, UK

Flows over porous and rough media can be found in nature and in industrial devices. The combination of permeability and roughness effects leads to an outer flow that is significantly different than flows over impermeable flat or rough walls. However, the phenomena at play are still not fully understood. We employ the lattice Boltzmann method (LBM) with D3Q27 operator and block-structured adaptive mesh refinement (AMR) to enable accurate computation of turbulent structures generated by the porous medium by refining the mesh in areas of interest. The aim of this study is to use the LBM-AMR coupled with large eddy simulation (LES) to capture the flow in and over several porous media configurations. All calculations are made with our AMROC software. The first configuration consists of two layers made of spheres and is described in Stoesser et al. (2006). The Reynolds number based on channel height and average flow velocity is 17,630. The second configuration is described in Kuwata et al. (2016) and the porous media is made of several interlaced cubes layers. The Reynolds number is 2,900. Simulations are realized with a Wall-Adapting Local Eddy-Viscosity model. Results are found to be in good agreement with the reference experiment and Direct Numerical Simulation. The validated AMR-LBM-LES model is then used to study the influence of the surface roughness in the context of a rough and porous medium. This is done by modelling the aforementioned porous medium made of cubes with different surface rugosities.

Effects of fractal-like multiscale roughness on turbulent boundary layers

T. Medjnoun (1), E. Rodriguez-Lopez (1), M. A. Ferreira (1), T. Griffiths (1), J. Meyers (2) and B. Ganapathisubramani (1)

(1) Aerodynamics and Flight Mechanics Research Group, University of Southampton, Hampshire, SO17 1BJ, UK, (2) KU Leuven, Mechanical Engineering, Celestijnenlaan 300, B3001 Leuven, Belgium

In the previous two decades, multiscale surface roughness has earned special attention from both the numerical and experimental communities. These complex surfaces generally exhibit fractal-like characteristics and are widely present in urban areas and natural landscapes commonly studied in Atmospheric Boundary Layers (ABL). One of the challenges when examining flows over such surfaces is the determination of the smallest spatial resolution needed in order to accurately predict the bulk flow quantities. In fact, unless direct numerical simulations (DNS) or well-controlled laboratory measurements are accessible, computational fluid dynamics (CFD) methods such as large-eddy simulations (LES) or Reynolds-averaged Naviers-Stokes (RANS) require modelling of the unresolved roughness features. Therefore, wind tunnel experiments in addition to the modelling techniques can prove beneficial with the different lines of enquiries, resulting in appropriate modelling strategies and better predictions of flows such as in urban environments. To uncover the effects of the multiscale roughness on the turbulent boundary layer, an experimental study based on fractal-like cuboid elements roughness is carried out, and the effects of roughness scale hierarchy on the mean and turbulent flow quantities are investigated. Three iterations have been used with the first iteration of large-scale cuboids onto which subsequent smaller cuboids are uniformly added, with their size decreasing with a power-law as the number increases. The drag is directly measured through a floating-element drag balance, while particle image velocimetry allowed the assessment of the flow field. The effects of scales on the boundary layer characteristics as well as implications on the classical similarity laws will be additionally discussed.

A combustion instability mechanism in rocket engines produced by orifice whistlin

Philipp Brokof (1), Juan Guzmán Iñigo (2), Wolfgang Armbruster 3), Justin Hardi (3)

(1) RWTH Aachen), (2) Imperial College London, (3) German Aerospace Center (DLR)

The high energy density available in the combustion chamber of rocket engines favours the excitation of combustion instabilities which can severely damage them. In hydrogen/liquid-oxygen (H2/LOX) engines, fuels are usually injected by coaxial shear injectors. For these kind of injectors, injection-coupling is a well-known cause of high frequency combustion instabilities. A common explanation for this phenomena is that the injector acoustics are excited by pressure fluctuations produced in the combustion chamber. In a previous study, it was proposed that the LOX injector is self-excited by the coupling of the longitudinal modes of the injector tube with the whistling of the injector's inlet orifice. This orifice, which is used to acoustically decouple the combustion chamber from the propellant feed system, can absorb or generate acoustic energy (the latter is termed whistling). However, due to the lack of sufficient pressure oscillation data from within the injector elements of the rocket combustion chamber, this hypothesis could not be proven experimentally. To study the feasibility of this mechanism, we use an acoustic network approach. This approach requires the knowledge of the acoustic transfer function of the orifice, that we compute numerically. The numerical approach is based on the linearisation of the compressible Navier-Stokes equations (LNSE) around an Reynolds-averaged mean flow.

Particle image velocimetry of a jet impacting a tightly packed tube bundle

T. Charpentier (1)(2), J. Vétel (1), F.P. Gosselin (2).

(1) Laboratoire de dynamique des fluides, Department of Mechanical Engineering, Polytechnique Montréal, Montréal,QC, Canada (2) Laboratory for Multiscale Mechanics, Department of Mechanical Engineering, Polytechnique Montréal, Montréal, QC, Canada

---

Nuclear fuel assemblies are exposed to primarily high-speed axial flow during operation. To mitigate the risks of high pressure gradients in a loss-of-coolant accident (LOCA), strategically located LOCA holes have been located around the core. The side effect of these LOCA holes is that they allow for localised crossflow jets to impinge on the fuel assembly, which can give rise to flow-induced vibrations and possible mechanical degradation of the fuel rods over the long term. Studies of this particular geometry are mostly focused on the vibrations of the rods themselves and not on the flow patterns inducing them. In this study particle image velocimetry (PIV) to obtain velocity profiles and fluctuations in a confined six by six cylinder array for a jet Reynolds number of 17500. The goal is to understand the contribution of each cylinder in the flow pattern and identify those responsible for turbulence effects. We consider various configurations of equally spaced cylinders (pitch to diameter ratio, P/D = 1.32) as well as different eccentricities of the jet impacting the bundle. When the nozzle is aligned on the pitch zone between two cylinders, a biased flow pattern is identified, leading to an asymmetrical distribution in the bundle and a bi-stable situation. In contrast, when the alignment is done on the centre of a cylinder, a rather symmetrical and more stable flow pattern emerges. In both cases, the momentum of the jet is first lost to the streamwise normal Reynolds stress component within the pitch zones of the leading set of cylinders and later to the transverse normal Reynolds stress component in their wake. The velocity in the bundle after three columns of cylinders is only 20% of the initial velocity, resulting in a significant decrease in flow rate downstream. Strouhal numbers after the first columns differ from known experimental results of tandem and side-by-side cylinders for similar spacing ratios. For tightly spaced bundles, it is not possible to infer the behavior from the flow past a pair of cylinders. We acknowledge the funding of Framatome and the Natural Sciences and Engineering Research Council of Canada

Stability, receptivity and sensitivity of mixed baroclinic convection in a cavity

Abhishek Kumar and Alban Pothérat

Centre for Fluid and Complex Systems, Coventry University, Coventry CV1 5FB, United Kingdom

In this study, we are seeking the critical conditions for the oscillatory instability that arises in a nearly semi-cylindrical cavity with an upper free surface where fluid can be fed in, and with porous lower boundaries where fluid can escape [1]. This generic configuration is representative of numerous problems where solid materials are melted, for instance, metallurgical casting processes [1,2]. In this geometry, the flow is driven by an unusual type of mixed convection, which is the combination of the buoyancy indirectly caused by the shape of the boundaries and the through-flow. The cavity is made of a semicircular lower boundary, two adiabatic sidewalls and infinitely extended in the third direction. The non-dimensional equations governing stratified flows under Boussinesq approximation are solved using the spectral-element code NEKTAR++[3]. Our system is characterised by three non-dimensional parameters: the Rayleigh number (Ra), the Prandtl number (Pr), and the Reynolds number (Re) based on the strength of through-flow. The linear stability analysis and direct numerical simulations are performed to identify the observable states. The analysis reveals that the unstable modes are three-dimensional. Furthermore, the receptivity analysis shows that the through-flow in the middle of the inlet is most receptive. The sensitivity analysis reveals that region at the bottom of the domain is most sensitive.
References: [1] Flood and Davidson, Mat. Sci. Tech. 10, 741 (1994). [2] Kumar and Pothérat, J. Fluid Mech. 885, A40 (2020). [3] Cantwell et al., Comput. Phys. Commun. 192, 205 (2015

Mass transfer from small spheroids suspended in a turbulent fluid

J. M. Lawson, B. Ganapathisubramani

University of Southampton

By coupling direct numerical simulation of homogeneous isotropic turbulence with a localised solution of the convection-diffusion equation, we model the rate of transfer of a solute (mass transfer) from the surface of small, neutrally buoyant, axisymmetric, spheroidal particles in dilute suspension within a turbulent fluid at large Péclet number, Pe. We observe that, at Pe = O(10), the average transfer rate for prolate spheroids is larger than that of spheres with equivalent surface area, whereas oblate spheroids experience a lower average transfer rate. However, as the Péclet number is increased, oblate spheroids can experience an enhancement in mass transfer relative to spheres near an optimal aspect ratio $\lambda=1/4$. Furthermore, we observe that for spherical particles, the Sherwood number Sh scales approximately as Pe0.26, which is below the Pe1/3 scaling observed for inertial particles but consistent with available experimental data for tracer-like particles. The discrepancy is attributed to the diffusion-limited temporal response of the concentration boundary layer to turbulent strain fluctuations. A simple model, the quasi-steady flux model, captures both of these phenomena and shows good quantitative agreement with our numerical simulations. The results will be compared against experimental measurements of mass transfer from particles in homogeneous turbulence.

A phase-field model for capillary bulldozing

: Liam Morrow, Oliver Paulin, Matthew Hennessy, and Christopher MacMinn

University of Oxford

The invasion of non-wetting gas into a horizontal, liquid-filled tube or Hele-Shaw cell is a classical problem in fluid mechanics that has been studied extensively from a variety of perspectives. However, the addition of a sedimented granular material to the defending liquid phase can fundamentally change the mechanics of the problem by introducing friction, leading to a class of ``multiphase frictional flows'' that remain relatively poorly understood. For example, recent experiments [Dumazer et al., 2016, PRL] show that, in a capillary tube, the motion of the gas-liquid interface will bulldoze the granular material, accumulating a pile of grains on the liquid side of the interface that will grow until it forms a plug and clogs the tube. Here, we present a phase-field model for capillary bulldozing. The model involves three phases -- gas, liquid, and solid -- and takes the form of a coupled pair of nonlinear conservation laws and a linear elliptic equation for the velocity of solid. We solve our model numerically for a variety of different scenarios to develop insight into the roles of sliding friction, rearrangement, capillarity, viscosity, and plug formation during capillary bulldozing.

Boiling flow dynamics in non-isothermal microchannels with conjugate heat transfer

Federico Municchi, Mirco Magnini

Fluids and Thermal Engineering, Faculty of Engineering, University of Nottingham

Multiphase flows with phase-change represent a viable technological solution for cooling high power density microdevices since, compared to single phase flows, additional energy can be dissipated in the form of latent heat. Furthermore, two-phase flows maintain uniform surface temperatures, vital for the correct operation of components; and respond positively and passively, i.e. without the need to increase the mass flux, to localised "hot-spots", as the heat transfer coefficient increases with heat flux. Despite the large number of experimental studies conducted on boiling heat transfer in microchannels, there is still general disagreement on the underlyng dynamics, because the existing experimental techniques cannot yet access the small spatial and temporal scales of the flow with sufficient resolution. Numerical simulation methods have advanced rapidly in the last decades and the direct numerical simulation of boiling flows with interface-tracking methods may provide valuable information on fluid dynamics and governing heat transfer mechanisms. We present a numerical method for solving the conjugate heat transfer equation coupled with a Volume of Fluid (VoF) scheme for simulating incompressible two-phase flows with phase change in micro-channels, assuming a parallel channel microevaporator model. We detail the numerical and modeling aspects, and we present a validation of the model. We investigate the effects of different geometrical configurations of the channels (channel and fin size) on the heat removal capacity for multiphase flows under different transient regimes. Both the cases of one isolated bubble and train of bubbles are investigated. This work is supported by the UK Engineering & Physical Sciences Research Council (EPSRC), through the BONSAI (EP/T033398/1) grant. We acknowledge the use of Athena at HPC Midlands+, which was funded by the EPSRC grant EP/P020232/1, as part of the HPC Midlands+ consortium.

Towards optimization of a wave-to-wire energy device in a breakwater contraction

Jonathan Bolton, Onno Bokhove, Duncan Borman, Harvey Thompson

University of Leeds

A novel wave-energy device has recently been proposed. The device consists of a channel with a V-shaped contraction, in which a buoy is placed. Attached to the buoy is a magnet (or multiple magnets) which, by constraining the buoy to move along a fixed trajectory, travel through a fixed induction coil when the buoy is displaced by the waves. This electromagnetic induction system converts the energy of the waves into electrical energy (via the buoy's motion). A complete nonlinear mathematical formulation for the device has been derived, coupling the 3D potential-flow hydrodynamics, buoy-dynamics and electromagnetics. In addition, a proof-of-concept was built. These developments satisfied the first three of nine recognized Technological Readiness Levels (TRLs): establishing the basic principles, introducing the modelling concepts and demonstrating a proof-of-concept. Going forward, we aim to address TRLs 4, 5 and 6: laboratory validation, design optimization and near-scale testing. The final three TRLs concern the upscaling of the device, testing within an operational environment and successful delivery. Concerning the design optimization, we aim to show: a) A 2D asymptotic water-wave model. While full 3D nonlinear modeling is desirable, optimization is generally done via surrogate models. The 3D potential-flow model includes the most computationally costly degrees-of-freedom. Instead, a 2D Benney-Luke approximation, eliminating the vertical degree-of-freedom while keeping accurate long-wave dispersion, is derived. b) Optimization of a simplified ODE model, in which the hydrodynamics are replaced with an explicit wave-form, feeding into the buoy and electromagnetic equations. In this, we will investigate optimization of the electromagnet geometry.

A Bayes factor comparison of the Brownian and Langevin models of passive particle transport

Martin Brolly, Jacques Vanneste, Aretha Teckentrup, James Maddison

School of Mathematics and Maxwell Institute for Mathematical Sciences, University of Edinburgh, King's Buildings, Edinburgh EH9 3FD, UK

The dynamics of passive particles in turbulent flows is often represented by stochastic models for applications such as pollutant transport. These models provide an inexpensive way of computing particle trajectories by representing motion at unresolved scales using simple stochastic processes. These processes are typically defined by families of stochastic differential equations whose parameters are estimated from observational or simulation data. Bayesian inference provides a systematic framework to carry out this estimation, incorporating relevant prior information. Stochastic models of varying degrees of sophistication have been proposed. A basic question is then which of these models can better explain observed dynamics given prior uncertainty in their parameters. The question is complicated by the need to balance the accuracy of the models against their complexity and cost. Bayesian model comparison offers an attractive means of assessing quantitatively the relative explanatory power of competing hypotheses in the face of empirical data, through the computation of the so-called Bayes factor. We employ this methodology to compare two elementary stochastic models of passive particle dynamics, namely the Brownian and Langevin models, given trajectory data from simulations of homogeneous two-dimensional turbulence. We show that the Langevin model is preferred only on short timescales and when sampling times of observations are sufficiently short (relative to the decorrelation time of the flow) to constrain the models' parameters. On longer time scales or sampling times, the simpler Brownian model is preferable. We present this work as an initial demonstration of the use of Bayesian model comparison in fluid dynamics and discuss the wider applicability of this methodology to related problems. Berloff, P. S. & McWilliams, M. C., (2002), 'Material Transport in Oceanic Gyres. Part II: Hierarchy of Stochastic Models', Journal of Physical Oceanography, 32(3), pp.797-830. Thomson, D., (1987), 'Criteria for the selection of stochastic models of particle trajectories in turbulent flows', Journal of Fluid Mechanics, 180, 529-556. Ying, Y. K., Maddison, J. R. & Vanneste, J., (2019), 'Bayesian inference of ocean diffusivity from Lagrangian trajectory data', Ocean Modelling, 140. Kass, R. E., & Raftery, A. E., (1995), 'Bayes Factors', Journal of the American Statistical Association, 90, pp. 773-795.

Predicting the statistics of chaotic systems using resolvent based modelling

Thomas Burton, Sean Symon, Davide Lasagna

University of Southampton

Modal analysis techniques decompose complex fluid flows into a set of fundamental spatial modes that can represent the same motion with fewer degrees-of-freedom (reduced-order models). Combining these spatial modes using temporal amplitude coefficients would define the evolution of these structures in a similarly efficient manner. However, these temporal amplitude coefficients are not known, and are generally determined alongside the spatial modes from the experimental/simulation data. In the work presented, a Resolvent based optimisation method is used to find these amplitude coefficients by generating 'quasi-solutions' to the governing equations. The only data required for this method is the mean state of the system, which can be assumed a priori or obtained via cheap methods (e.g. empirical, Reynolds-Averaged-Navier-Stoked solutions). The method uses nonlinear, non-convex optimisation techniques to minimise a residual functional measuring the difference between the modes and the nonlinear (triadic) interaction of the same modes transformed by the resolvent operator. An efficient basis for the optimisation is found via a singular value decomposition of the resolvent operator, yielding a set of modes ranked in terms of the magnitude of response to the nonlinear forcing. The results is a 'quasi-trajectory' that captures the fundamental structure of the 'turbulent attractor' and also its associated statistics, with the potential for less computational effort due to the dimensionality reduction. As a simple proof of concept, the method is applied to the Lorenz '63 system, for which a dimensionality reduction can be exploited because of the linearity of the first equation. The effects of the dimensionality reduction on the convergence rate of the optimiser (as a form of pre-conditioning) is also discussed.

Real-time parameter inference in reduced-order flame models with heteroscedastic Bayesian neural network ensembles

Maximilian L. Croci, Ushnish Sengupta, Matthew P. Juniper

University of Cambridge

The estimation of model parameters with uncertainties from observed data is an ubiquitous inverse problem in science and engineering. In this paper, we suggest an inexpensive and easy to implement parameter estimation technique that uses a heteroscedastic Bayesian Neural Network trained using anchored ensembling. The heteroscedastic aleatoric error of the network models the irreducible uncertainty due to parameter degeneracies in our inverse problem, while the epistemic uncertainty of the Bayesian model captures uncertainties which may arise from an input observation's out-of-distribution nature. We use this tool to perform real-time parameter inference in a 6 parameter G-equation model of a ducted, premixed flame from observations of acoustically excited flames. We train our networks on a library of 2.1 million simulated flame ideos. Results on the test dataset of simulated flames show that the network recovers flame model parameters, with the correlation coefficient between predicted and true parameters ranging from 0.97 to 0.99, and well-calibrated uncertainty estimates. The trained neural networks are then used to infer model parameters from real videos of a premixed Bunsen flame captured using a high-speed camera in our lab. Re-simulation using inferred parameters shows excellent agreement between the real and simulated flames. Compared to Ensemble Kalman Filter-based tools that have been proposed for this problem in the combustion literature, our neural network ensemble achieves better data-efficiency and our sub-millisecond inference times represent a savings on computational costs by several orders of magnitude. This allows us to calibrate our reduced order flame model in real-time and predict the thermoacoustic instability behaviour of the flame more accurately.

Turbulent flow field reconstruction from sparse measurements

Francis De Voogt, Douglas W. Carter, Bharathram Ganapathisubramani

University of Southampton

Recent work has demonstrated the use of sparse sensors in combination with the proper orthogonal decomposition (POD) to produce data-driven reconstructions of the full velocity fields in a variety of flows. In this work, we aim to combine the outcomes from POD based analyses with Machine Learning strategies to improve the prediction capabilities using sparse sensors. We utilise a time-resolved Particle Image Velocimetry dataset obtained in a water channel experiment of a NACA 0012 aerofoil at Re-C = 75000 at an angle of attack $\alpha = 12°$. We use POD-based sensor selection strategies that allows us to determine the location of sensors to predict a reduced-state of the flow field. The reduced state is based on a limited number of POD modes (which is related to the number of sensors that can be used). To further improve the accuracy of reconstruction, non-linear Machine Learning methods based on Shallow Neural Networks (SNN) are used to augment the output from the POD-based technique. Different variants of SNN are applied with different goals in order to reduce the reconstruction error. Preliminary results indicate that a simple 2-layer network can reduce the reconstruction error by nearly 25%. Limitations of using SNN for these types of problems will be discussed.

DNS of incompressible Rayleigh--Taylor instabilities at low and medium Atwood numbers

Arash Hamzehloo, Paul Bartholomew and Sylvain Laizet

Turbulence Simulation Group, Department of Aeronautics, Imperial College London

The Rayleigh-Taylor (RT) instability develops at the perturbed interface of two fluids of different densities subjected to relative acceleration. Here, direct numerical simulations of two-dimensional (2D) and three-dimensional (3D), single-and multi-mode, incompressible immiscible RT instabilities are performed using a newly-developed two-phase solver within Xcompact3d, a high-order finite-difference computational fluid dynamics framework. A phase-field approach based on the Allen-Cahn formulation with a continuum force surface tension model is implemented. Various combinations of Atwood number (a measure of the density ratio of "heavier" and "lighter" fluids in a two-phase system), i.e. $0.2 \leq At \leq 0.75$, Reynolds number (the ratio of inertial forces to viscous forces), i.e. $300 \leq Re \leq 10000$, surface tension and initial perturbation amplitude are investigated. It is found that at high Reynolds numbers, the surface tension, if significant, could prevent the formation of Kelvin-Helmholtz type instabilities within the bubble region. A new relationship is proposed for the vertical distance of the bubble and spike versus the Atwood number. The spike and bubble re-accelerate after reaching a temporary plateau due to the reduction of the friction drag as a result of the formation of the spike vortices and also the formation of a momentum jet travelling upward within the bubble region. The interface for a 3D single-mode instability grows exponentially, however, a higher Reynolds number and/or a lower Atwood number results in a noticeably larger surface area after the initial growth. In the early development stages, a 3D multi-mode RT instability also displays an exponential interface growth rate similar to single-mode RT instabilities with identical Atwood numbers. However, due to the collapse and merging of the individual single-mode instabilities, the interface growth rate of a multi-mode RT reduces eventually and becomes strongly dependent to the mesh resolution at late simulation times. On the other hand, the ratio of kinetic energy over released potential energy exhibits an almost steady-state behaviour after the initial growth, with values around 0.4, independently of the mesh resolution.

Do ambient shear and thermal stratification impact wind turbine wake breakdown?

Amy Hodgkin (1), Sylvain Laizet (1), Georgios Deskos (2)

(1) Department of Aeronautics, Imperial College London, (2) National Wind Technology Center, National Renewable Energy Laboratory, CO, USA

As wind turbines grow in size, they experience increasingly uneven inflow conditions due to the structure of the atmospheric boundary layer. These conditions include shear and thermal stratification which alter the shape and length of the wakes and in turn impact the loading and power generation of turbines that exist the wake of an upstream turbine. This study aims to further understand the effect of shear and thermal stratification on the wake properties and tip-vortex break-up mechanisms using high-fidelity simulations and data-driven techniques such as the proper orthogonal decomposition (POD). A suite of high-resolution simulations is carried out with the LES wind-farm simulator, WInc3D, one of the high-order finite-difference solvers of the XCompact3d framework dedicated to turbulence. WInc3d is based on sixth-order schemes on a uniform Cartesian mesh to solve the incompressible Navier-Stokes equations. The wind turbine rotor is modelled using the actuator line method. The flow is perturbed harmonically around the blade tips to trigger instabilities and enable transition to turbulence. The combined effect of shear and thermal stratification on the wake is investigated in the region of 0 - 3.5R downstream of the turbine. Results show that perturbation frequency affects the breakdown location thanks to the mutual inductance instability. Fourier analysis is used to compute the growth rate of the perturbation. Shear is shown to increase the growth rate along the bottom and decrease it along the top, resulting in a slanted wake which is longer at the top. Thermal stratification acts to suppress the wake, hence, increasing its wake diameter at the sides. Furthermore, we examine the structure, energy content and frequency of the POD modes to infer the wake's breakdown and instability mechanisms as well as provide an estimate for its stable length by gradually synthesizing the effects of the atmospheric ambient turbulence.

Simulating atmospheric boundary layer flow in a recirculating water tunnel

H.D. Lim, Christina Vanderwel

University of Southampton

Atmospheric boundary layer flow is simulated in the University of Southampton's newly constructed Recirculating Water Tunnel. The boundary layer flow develops along a false floor inserted in the tunnel's test section which measures 8.1m long and 1.2m wide. A system of spires at the test section entrance and roughness elements along the false floor have been designed to create a thickened urban boundary layer with an appropriate log-layer for scale model testing of buildings and cities. Particle Image Velocimetry measurements will be presented to validate the flow quality, the flow uniformity, turbulence statistics, boundary layer thickness and Reynolds number. This research is one of the first steps in an ongoing project funded under a Future Leaders Fellowship to simulate urban air pollution in urban landscapes – the end goal will be to study the flow around a three-dimensional scale model of the city of Southampton using fluorescent dye as a proxy for air pollution.

Simulating surface wave dynamics with Convolutional Neural Networks

Mario Lino (1), Chris Cantwell (1), Eduardo Pignatelli (2), Stathi Fotiadis (2), Anil Anthony Bharath (2)

(1) Department of Aeronautics, Imperial College London; (2) Department of Bioengineering, Imperial College London

We investigate the performance of Convolutional Neural Networks (CNNs) to simulate the spatio-temporal dynamics of surface waves described by the shallow water equations, on open and closed fluid domains surrounded by complex wall geometries. The proposed data-driven model acts as a neural solver, inferring the height field at each time-point from previous predictions at past time-points. This approach results into predictions between three and four orders of magnitudes faster than the ground truth numerical simulations, obtaining a root-mean-square error between the predictions and the ground truth of order 10-4 times the characteristic length for at least 20 time-steps. The CNN was trained with a loss function that incorporates the spatial gradients into the loss function, which serves to increase the accuracy of the predictions by reducing oscillatory artifacts. We assess the ability of the proposed approach to capture wave propagation, reflection, interference and diffraction; and to extrapolate to long-term predictions. Importantly, we analyse how well the neural model generalises to geometric configurations not seen during training and demonstrate that it is capable of accurately predicting the height distribution of waves on a liquid surface within curved and multi-faceted open and closed geometries, when only simple box and right-angled corner geometries were seen during training. The speed-up over numerical solvers is partly attributed to the lower spatial and temporal resolutions, so to complement the physics model we additionally propose an independent neural network that time-interpolates on the physical predictions.

Machine learning modelling of transonic aerodynamic loads for aeroelastic analysis of an airfoil

David Massegur, Andrea Da Ronch, Marcello Righi

University of Southampton

A major problem in aerospace and automotive industries is that the computational cost of deploying three-dimensional computational fluid-dynamics (CFD) simulations for full-scale applications is prohibitive. Simplified models exist but these are lacking physical interpretability, leading to expensive redesigns and potentially catastrophic misshapes. Further, this translates into a multitude of problems: more pollutants released into the environment, higher levels of noise for the community, and less profitability for the business. Although this disproportion between the requirement of computational power and the availability of such will persist for the foreseeable future, there is a new hope: artificial intelligence (AI) has recently experienced a significant breakthrough due to advances in machine learning (ML), the science of computer algorithms which learn complex functions from data. Given a set of measured inputs and target outputs, a ML algorithm finds the function that best fits such data. The prediction accuracy of the models improves considerably when sophisticated algorithms called neural networks (NN) are used. These consist of multiple non-linear regressions of different architectures (dense, convolutions, recurrent) stacked in sequence to form deep mathematical structures and extract complex relationships from the data. The resulting surrogate model is used in place of the high-fidelity model to compute new simulations at a fraction of the original cost. In this work, a ML model is developed to predict the aerodynamic loads of an elastically-suspended airfoil in transonic regime under forced pitch and plunge motion. Only a few CFD responses are required to train the model. New simulations are then computed with the surrogate model for the aerodynamics. It was found that the computational saving compared to CFD is up to 99%. Similar approaches can be thought for many other applications in industry. Thus, AI will make a huge impact on the next generation of engineering methods.

Turbulent channel flow over streamwise-aligned ribs

Mattias Nilsson, Bharathram Ganapathisubramani

University of Southampton

We investigate secondary flow structures produced by streamwise-aligned, smooth rectangular ribs of several different widths placed at two different spanwise spacings in a fully developed channel flow. The pressure drop in the channel is measured to determine the increased skin friction (drag) of these surfaces compared to smooth wall channel flow. We also use cross-stream stereoscopic particle image velocimetry (PIV) to visualise the secondary motions and analyse the mean flow behaviour.

Swirling vortex rings

Rigoberto Ortega-Chavez (1), Lian Gan (2), Philip Gaskell (3).

Department of Engineering, Durham University, Durham DH1 3LE, UK

A numerical simulation of a swirling vortex ring were performed using the open-source code OpenFOAM. A Reynolds number of Re= 2500 was implemented in order to get a laminar ring as well as 5 different swirl numbers S= 0, 0.25, 0.5, 0.75 and 1 . In previous numerical works, a swirl velocity profile was added to a well-formed vortex ring. However, for this work and in order to get a ring similar to one generated experimentally. The swirl velocity was added to the fluid (as a solid body rotation) before it had been discharged through an orifice. This allowed us to analyse the formation process of the vortex ring. The kinematics of a swirling vortex ring was found to be in a good agreement with previous works1, 4, where there is a decrease in the self-propagation velocity and an increase in the radial velocity appreciable in a faster growing of the vortex radius in compare with a ring without swirl. Also, a time dependent swirl velocity was found in the ring's core as Naitoh et al. (2014) reported in their experimental work. For early time, the rolling up process has a strong influence in the swirl velocity and then the growing of the ring radius becomes dominant. Besides, it is showed that a Gaussian swirl velocity profile at the core proposed in previous numerical works is not possible taking into account the formation process. The formation number that represents the dimensionless time when the vortex ring has acquired its maximum circulation was also studied3. It was found that this dimensionless time is highly sensitive to the swirl number as He et al. (2019) reported.

Sensitivity analysis of flood prediction model performance

Saba Rabab, Ian Pattison

Heriot-Watt University, Edinburgh, UK

Prediction of a flood using a data-driven model has increased recently and it is growing fast with the advancement of technology and data availability. The model performance usually improves with the size of datasets for predictive modelling. However, this depends on the selection of model and type of datasets. Therefore, the knowledge of model performance for a given dataset is critical. To quantify the relationship between flood prediction model performance and length of the dataset, sensitivity analysis was conducted for a data-rich catchment of Eden, UK. Seven gauging stations were selected as input variables with a response station downstream of the Eden River. The dataset was divided into a variable length of the training dataset (10 to 30 years) and a test dataset of 10 years. A linear regression model was adopted that showed a strong relation (R2 ~ 97%) between the input and output parameters. An increasing trend of model performance was found as the length of the training datasets increased from 10 to 30 years. The result showed that the linear regression model for flood prediction followed the general data-driven model norm, however, the sensitivity analysis was performed for a smaller length of dataset. It is recommended to perform the sensitivity analysis on a larger dataset.

Recurrent Neural Network based surrogate modeling of unsteady forces acting on a plunging airfoil

Rahul Sundar (1), Dipanjan Majumdar (1), Chhote Lal Shah (1), Sunetra Sarkar (1)

(1) Department of Aerospace Engineering, IIT Madras, Chennai - 600036, India

Owing to the inherent nonlinearity in flow, both periodic and aperiodic dynamical states are observed in the flow-field of flapping airfoils[1] at various parametric regimes, even when the kinematical model is periodic. Due to significant computational costs of high fidelity simulations of such complex flow problems, developing accurate yet computationally cheap surrogate models for prediction of the aerodynamic forces are of specific interest. Such surrogate models can be used to carry out control or optimisation studies in a robust manner. Traditional time series forecasting methods such as Auto Regressive Integrated Moving Average(ARIMA) are found to model data with underlying linear or periodic relationships better, and therefore might not be best suited for capturing the aerodynamic load behaviour in the aperiodic regimes. Hence, a surrogate model that captures both periodicity and aperiodicity in the force coefficients is investigated. Recurrent neural networks(RNNs)[2] are a class of deep neural networks prominently used for inherently nonlinear sequence modeling problems such as natural language processing[3], video captioning[4] and time series modeling[5]. Hence, in the present study, a RNN based parametric surrogate model is proposed to predict the aerodynamic load coefficients' time histories for unsteady incompressible flow past a plunging airfoil at Reynolds number, Re = 300. The reduced frequency(k) and non-dimensional plunging amplitude(h) are the parameters considered for the surrogate model. Two variants of the model are developed where, (i) given k, h and time histories of the aerodynamic load coefficients for first few cycles, time histories for the next cycles are predicted, and (ii) given only the k and h, the entire time histories of aerodynamic load coefficients are predicted. The effectiveness of the surrogate model variants is demonstrated by comparing the spectral properties of the true and predicted time series and comparing the predictions with that of an ARIMA model for both periodic and aperiodic dynamical states. [1] Sandeep Badrinath, Chandan Bose, and Sunetra Sarkar. Identifying the route to chaos in the flow past a flapping airfoil. European Journal of Mechanics-B/Fluids, 66:38–59, 2017 [2] Zachary C Lipton, John Berkowitz, and Charles Elkan. A critical review of recurrent neural networks for sequence learning. arXiv preprint arXiv:1506.00019, 2015[3] Kanchan M Tarwani and Swathi Edem. Survey on recurrent neural network in natural language processing. Int. J. Eng. Trends Technol, 48:301–304, 2017.[4]Sheng Li, Zhiqiang Tao, Kang Li, and Yun Fu. Visual to text: Survey of image and video captioning. IEEE Transactions on Emerging Topics in Computational Intelligence, 3(4):297–312, 2019 [5] Benjamin Lindemann, Timo Müller, Hannes Vietz, Nasser Jazdi, and Michael Weyrich. A survey on long short-term memory networks for time series prediction. Procedia CIRP, 99:650–655, 2021.

Unsteady flow development in particulate filter channels due to oblique flow entry

C. Samuels, S. Aleksandrova, T.C. Watling, H. Medina, S. Benjamin

Coventry University

Particulate filters used in automotive applications create an additional pressure loss in the exhaust system, these losses can impact the engine's performance. To optimise the filter design, one needs to understand the properties of the flow in the small filter channels and quantify the associated pressure losses. One important question is the flow regime in the channels. The flow in aftertreatment systems is usually characterised by a low Reynolds number typical of laminar flow. However, existing experimental and numerical studies have confirmed the persistence of upstream turbulence in the channels, and the development of unsteady laminar or even turbulent flow. To capture this effect with CFD a high fidelity model, like large eddy simulation (LES), is required. Moreover, most modelling studies assume a flow normal to the front face of the filter. In practice, the entering velocity profile is rarely straight and varies vastly from channel to channel. Therefore, this study focuses on the effect of a highly oblique entry into the filter channel on flow development and provides an insight into the effects of varying entry angle on the size and shape of the recirculation area just beyond the leading edge of the channel. These changes in the flow structure are closely related to the overall pressure losses and flow structure downstream of the channel. Accurate prediction of the flow regime and structure in such configurations is important for the design of other applications such as heat exchangers and automotive catalysts as it will affect the flow losses and heat transfer both in the channel and further downstream. A better understanding of this effect will also help in the assessment and development of simplified modelling techniques such as the porous medium approach and one-dimensional flow models.

Scale-space energy transfer pathways in inhomogeneous compressible turbulence

S. Arun (1), A. Sameen (2), B. Srinivasan (2), S. S. Girimaji (3)

(1) Dept. of Aeronautics, Imperial College London (2) Indian Institute of Technology Madras, Chennai, India (3) Texas A&M University, College Station, Texas, USA

Scale-space energy density function which is defined as the derivative of the two-point velocity correlation is used to quantify the energy distribution across different scales for inhomogeneous turbulent flows. The function describes the turbulent kinetic energy density of scale at a location and can be considered as the generalization of spectral energy density function concept to inhomogeneous flows. We derive the scale-energy density function transport equation for compressible flows to develop a better understanding of scale-to-scale energy transfer and the degree of non-locality of the energy interactions. Specifically, the effects of variable-density and dilatation on energy cascade are identified which are extra pathways for energy transfer across scales in compressible flows. It is expected that these findings will yield deeper insight into compressibility effects on canonical energy cascade leading to improved models (at all levels of closure) for mass flux, density-variance, pressure-dilatation, pressure-strain correlation and dilatational dissipation processes. Direct numerical simulation (DNS) data of mixing layers at different Mach numbers are used to characterize scale-space behavior of different turbulence processes. The scaling of the energy density function that leads to self-similar evolution at the two Mach numbers is identified. It is established that production is influenced by long-distance (order of vorticity thickness) interactions, whereas the pressure-dilatation effects are more localized (fraction of momentum thickness) in scale-space.

Weakly nonlinear theory for deterministic wave forecasting

Raphael Stuhlmeier (1), Michael Stiassnie (2)

(1) Centre for Mathematical Sciences, University of Plymouth, Plymouth, UK. (2) Faculty of Civil & Environmental Engineering, Technion, Haifa, Israel

Deterministic wave forecasting aims to provide a wave-by-wave prediction of the free surface elevation based on measured data. Such information about upcoming waves can inform marine decision support systems, control strategies for wave energy converters, and other applications. Unlike well-developed stochastic wave forecasts, the temporal and spatial scales involved are modest, on the order of minutes or kilometres. Due to the dispersive nature of surface water waves, such forecasts have a limited space/time horizon, which is further impacted by the effects of nonlinearity. I will discuss the application of the reduced Zakharov equation, and simple frequency corrections derived therefrom, to preparing wave forecasts. Unlike procedures based on solving evolution equations (e.g. high order spectral method), such corrections entail essentially no additional computational effort, yet show marked improvements over linear theory.

Three-dimensional modification of Gurney flap to improve the lift-type vertical axis wind turbine performance

Taurista P. Syawitria,b, Yufeng Yaoa, Jun Yaoa and Budi Chandraa*

(a) Department of Engineering Design and Mathematics, University of The West of England, Bristol BS16 1QY, United Kingdom (b) Department of Mechanical Engineering, Universitas Muhammadiyah Surakarta, Surakarta 57162, Central Java, Indonesia

Gurney flap (GF) has been used to improve the performance of lift-type of Vertical Axis Wind Turbine (LVAWT). While GF increases the lift generation, it can also increase the drag of LVAWT that leads to the reduction of the lift to drag ratio. Therefore, it decreases the performance enhancement of LVAWT. Such an increment of drag generation due to GF has significant effect on the reduction of performance of LVAWT, especially at medium to high tip speed ratios (TSRs) when GF actually has lower impact on the increment of lift generation compared to low TSRs. One possible solution is to modify three-dimensional (3D) GF by introducing gaps or holes on the GF geometry to reduce the drag generation and previous researches for aeroplane wings application were found that the introduction of gaps or holes can significantly reduce the drag generation by up to 12%. Therefore, this study attempts to apply gaps and holes in the GF mounted in the blades of LVAWT to reduce the drag generation caused by GF so that to improve the performance of LVAWT. Computational Fluid Dynamics (CFD) solution is utilized to evaluate the effect of gaps and holes to the performance improvement of LVAWT. In order to reduce the computational cost, a quasi-3D simulation with three gaps/holes and three GFs is considered. A hybrid Reynolds averaged Navier Stokes-large-eddy simulation (RANS-LES) model applying stress-blended eddy simulation together with Transition Shear-Stress Transport turbulence model is adopted for CFD simulations as this model can produce more accurate power coefficient prediction, compared to other unsteady RANS turbulence models at all TSRs operation. All CFD simulations are carried out using ANSYS Fluent v19, and results will be presented in the final full paper and presentation.

Data assimilation for RANS using time-averaged PIV of the flow around a NACA0012 airfoil

Craig Thompson, Bharathram Ganapathisubramani, Sean Symon

University of Southampton

Obtaining accurate 3-dimensional Reynolds-averaged simulations of airfoils at angles of attack which induce stall has long been an issue within the aerodynamic community. A large proportion of the inaccuracy results from the inability to precisely capture complex 3-dimensional flow features, such as leading-edge separation and stall cells. This study investigates the use of data assimilation where PIV data obtained in water channel experiments is combined with OPENFOAM based RANS solutions (using k-omega SST turbulence model). The velocity field is obtained by means of time-resolved PIV measurements carried out over a NACA0012 airfoil. A forcing term is computed from the error between the PIV and RANS results at prescribed control locations, hence driving the local solution to agree with the measured information. The effects of altering the density and uniformity of the control locations on the output of the computation are discussed. When comparing the velocity fields, particular attention has been made to ensure the computation regarding the shear layer caused by leading edge separation was improved upon, resulting from inaccuracies when using the baseline k-omega SST. The subsequent improvement to the computational analysis is discussed, as well as its ability to capture a more complex 3-dimensional flowfield

Simulation of turbulent axisymmetric bluff body wake with the effect of pulsed jet forcing

Taihang Zhu and Jonathan Morrison

Imperial College London

The turbulent axisymmetric bluff body wake is studied with a large eddy simulation (LES). The effect of pulsed jet forcing on turbulent wake is investigated. The pulsed jet actuator is included in the computational domain to keep high fidelity. Spectral proper orthogonal decomposition (SPOD) is applied to analyze the wake and the pulsed jet forcing effect. The numerical result shows good agreement with the experimental result and successfully reproduced the multi-stability of the axisymmetric wake. The SPOD result identified the axisymmetric bubble pumping mode (StD = 0.06), axisymmetric-breaking vortex shedding mode (StD = 0.22) and its subharmonic mode (StD = 0.1). When the high-frequency pulsed jet (StD = 9.82) is applied to the wake, the pressure fluctuation on the base and the azimuthal modes are globally suppressed. The mean flow properties show that the inlet flow is driven toward the wake and deviates the shear-layer, forming a concave separation streamline. The entrainment in the wake is suppressed, and the wake is narrowed, leading to a global pressure rise in the whole wake. The shear-layer deviation is caused by the vectoring effect of the pulsed jet. The high-frequency pulsed jet generates concentric pulsed jet and greatly enhanced the entrainment in the vicinity of the separation point. A very low-pressure region is generated, which drives the inlet flow towards the wake. The low-frequency forcing (StD = 2.62) generates diffusive pulsed jet which enhance the mixing and entrainment in its trajectory. These pulsed jets reduce the wake length and accelerate recirculating flow near the base, leading to a base pressure drop. When the low-frequency pulsed jet is applied, the vortex shedding mode (StD = 0.2) is amplified.

Modelling airway mucus: Rayleigh-Plateau instability of an annular viscoplastic liquid film

James Shemilt, Alice Thompson, Carl Whitfield, Alex Horsley, Oliver Jensen

Department of Mathematics, University of Manchester

The evolution of a thin annular film coating the inside or outside of a cylinder has been well studied in the case that the liquid is Newtonian. When the cylinder is sufficiently long, the film is unstable to a surface-tension-driven Rayleigh-Plateau instability. This flow has not, however, been studied when the liquid has a yield stress. Applications of the viscoplastic problem are potentially numerous, with the motivating example for this work being the flow of mucus in lung airways. Here the case of a Bingham fluid film will be discussed, and a description of its evolution using thin-film theory will be presented. The yield stress suppresses the linear instability of a layer with initially uniform thickness, and the dynamics during the nonlinear growth are modified. At late times, the layer approaches a non-trivial shape determined by a balance between capillary forces and resistance to flow from the yield stress. As in the Newtonian problem, collars of fluid form, but in contrast not all of the fluid drains into the collars, with the rest being trapped at the wall. Analysis of the late time dynamics including asymptotic and numerical solutions of the relevant equilibrium equations will be presented. Application to lung airway flows will be discussed, particularly in the context of diseases such as cystic fibrosis.

Left ventricular remodelling: Integrating flow imaging and machine learning into patient-specific models

Fergus Shone (1,2,3,4), Erica Dall'Armellina (2), Alejandro Frangi (3,4), Zeike Taylor (3,4), Peter Jimack (4)

(1) CDT Fluid Dynamics, University of Leeds, (2) LICAMM, University of Leeds, (3) CISTIB, University of Leeds, (4), School of Computing, University of Leeds

Heart failure (HF) is among the leading causes of death worldwide, with an anticipated increase in prevalence of 46% by 2030. Early-stage risk-stratification is vital in the effective diagnosis and treatment of HF, and there is a growing consensus that fluid dynamics in the heart chambers could provide key clinical insights. This study focusses on flow in the left ventricle (LV), quantifying haemodynamic patterns in both healthy and impaired populations. 4D-Flow MRI – a phase-contrast magnetic resonance imaging technique capable of reconstructing time-resolved, three-dimensional flow fields in the cardiovascular system – will be deployed to obtain patient-specific datasets. However, low spatial resolution (1.5 – 3mm3) and the presence of artifacts such as acquisition noise and velocity aliasing preclude the direct usage of this data, particularly in the computation of sensitive flow derivatives such as wall shear stress. Therefore, recent advances in physics-informed machine learning (ML) will be applied to produce high-fidelity reconstructions of the ventricular flow field. Physics-informed neural networks (PINNs) lie on the intersection between phenomenological and mechanistic modelling, providing seamless integration of flow measurements and domain knowledge. Where they differ from purely data-driven methods is in the inclusion of physical constraints, prescribed in the loss function, which restrict the permissible space of solutions by discarding non-physical predictions. This not only constrains the network to produce physically-consistent results, but also allows highly efficient training with far smaller datasets than required for traditional ML. This study will leverage the recent successes of PINNs in modelling fluid flows to address the aforementioned shortcomings of 4D-Flow MRI and provide a predictive model capable of quantifying complex flow patterns in the LV. This study is of particular clinical relevance, as it is hypothesised that flow features such as abnormal vortex formation could be sub-clinical indicators of HF, which could provide early-stage diagnosis, saving lives and costs.

Simulation of behavioural modification effects in multiphase flows

Jacob Anderson, Michael Fairweather, Lee Mortimer

School of Chemical and Process Eng, Uni of Leeds; CDT Fluid Dynamics Leeds; Sellafield Ltd

Direct numerical simulation (DNS) using spectral element code Nek5000 is used to understand the fundamental mechanisms operating in a mesoscopic multiphase pipe flow, with relevance to industrial applications at Sellafield Ltd, Cumbria. Limited numbers of particles are first investigated in isotropic turbulence boxes wherein the key particle behaviour is observed. Gaining knowledge from these focused set-ups then allows the incorporation of key physics in larger-scale simulations. The larger-scale simulations utilise a Lagrangian particle tracking technique which is coupled to a DNS of pipe flow. With the use of the University of Leeds' high-performance computing cluster ARC4, millions of particles are studied in flows of a moderately high Reynolds number (Re_shear = 720, Re_bulk = 12000). The realism later will be further captured by the inclusion of non-spherical particles in the dispersed phase. The impact of this change on quantities such as the dispersion, deposition and agglomeration of the particles will be highlighted by means of a contrast to spherical simulations. Similarly, the impact of the non-sphericity on the turbulence field can be studied. Physics captured in the simulations such as electrochemical forces influence the agglomeration and aggregation statistics of the dispersed particles. The relevance to industry comes from the fact that agglomeration can be controlled to improve pumping efficiency or to cluster waste material together for ease of extraction.

Cavity collapse near porous plates

Elijah Andrews (1), David Fernández Rivas (2), Ivo Peters (1)

(1) University of Southampton, (2) University of Twente

The collapse of a gas or vapour bubble near a solid boundary produces a jet directed towards the boundary. High surface pressure and shear stress induced by this jet can damage, or clean, the surface. A porous boundary, such as a filter, would act similarly to a solid boundary but with reduced effect. Prior research has measured the cleaning effect of bubbles on filters using ultrasonic cleaning, but how the bubble dynamics are fundamentally affected by the porosity of the surface is not known. We address this question experimentally by investigating how both the porosity and pore size of the boundary affects two collapse strength parameters: bubble displacement and bubble rebound strength. We measure these parameters for various porosity values, pore sizes, hole shapes, plate materials, and bubble positions. These experiments demonstrate a clear dependence of both collapse strength parameters on the porosity of the boundary but indicate limited sensitivity at high and low porosity values. Surprisingly, the position of the bubble, over either a hole or a solid part of the boundary, does not significantly influence the collapse strength.

Two-phase flow in the dynamic Earth: Reactive-infiltration instabilities

Danielle Bullamore (1), Sam Pegler (2), Sandra Piazolo (3), Oliver Harlen (2)

(1) CDT Fluid Dynamics, University of Leeds, (2) School of Mathematics, University of Leeds, (3) School of Earth and Environment, University of Leeds

In the lower parts of the Earth's crust, a mixture of crystalline solid and molten rock may exist simultaneously. This system will develop flow instabilities in response to different forcings, such as reaction, density contrasts and tectonic forcing. Three fundamental situations can be envisaged: melt is either "squeezed out", "fluxed through" or "accumulated in" interstitial pores between crystals which will "occupy" different parameter spaces. The motivation for this work is to understand when each of these situations arises and how this could affect melt transfer in the Earth's crust. On a sufficiently large scale (with respect to the pore size) we can describe this system as a two-phase fluid system. In this framework, both the crystalline and the molten rock are treated as viscous fluid continuum both occupying the whole domain with porosity describing their relative proportions. In this study, we are interested in the reactive-infiltration instability and begin by surveying existing work to understand the progress already made. This instability has the potential to create channels in a porous host rock, transitioning from porous flow (on the grain scale) to channelised flow with larger scale features. We will map parameters of the two-phase system with linear analysis in order to understand the onset of instabilities in the system. We also outline plans for future numerical work to study the system over long time scales.

Generation of bubbles by spilling breaking wave groups

Konstantinos Chasapis (1,2), Eugeny Buldakov (2), Helen Czerski (1)

(1) Department of Mechanical Engineering, University College London (2) Department of Civil Environmental and Geomatic Engineering, University College London

Large ocean bubbles dominate the air-sea transfer of carbon dioxide during storms. In this study, we carried out a series of experiments to track the spatiotemporal evolution of bubbles with radius $r>0.1$ mm for different local breaker geometries. A breaking wave in a wave flume was generated with dispersive focusing of a wave group. Running experiments with different phase shifts of the same pre-defined amplitude spectrum showed that when a peak-focused wave (zero phase shift) breaks, then wave groups with other added phase shifts break as well. To investigate differences in the deformation of those breakers a laser imaging technique was used. An algorithm identified the 2D shape of the breaker in successive images. It also separated the crests from bulges based on geometric criteria. We showed that, despite wave groups having same spectra, the extracted bulges differed locally in volume and velocity for each phase shift at the location of breaking. Therefore, breakers ranging from the traditional spilling type, which has a bulge that collapses on the front face of the wave, to the micro-plunging type, which has a pronounced overturning tip, were observed. The evolution of bubbles for each phase shifted bulge was captured by a high speed camera and measured by a feature extraction algorithm. We generally found that spilling bulges created fewer bubbles in total than micro-plungers. They also created fewer larger bubbles, i.e. with radius $r>1$ mm, at all measured flume areas. In contrast, micro-plungers that trap air within a small cavity as they break had less steep size distributions for $r>1$ mm. It is interesting, finally, that the volume of $r>1$ mm did not shift to smaller radii as time passes. This is an indication that those bubbles rise to the surface and burst instead of splitting into smaller ones, irrespectively of the local breaker properties.

A robust microfluidic device for fabricating deformable microcapsules based on water-oil-water double-emulsion templates

Qi Chen(1)(2), Kerstin Schirrmann(1)(2), Igor Chernyavsky(3)(4), Anne Juel(1)(2)

(1) Manchester Centre for Nonlinear Dynamics, University of Manchester, Oxford Road, Manchester M13 9PL, UK; (2) Department of Physics and Astronomy, University of Manchester, Oxford Road, Manchester M13 9PL, UK; (3) Department of Mathematics, University of Manchester, Oxford Road, Manchester M13 9PL, UK; (4) Maternal and Fetal Health Research Centre, School of Medical Sciences, University of Manchester, Manchester M13

Microcapsules consist of droplets surrounded by an elastic membrane. They can provide a useful biomimetic model for red blood cells (RBCs). However, large numbers of capsules are required for this purpose so that robust capsule production is essential to investigate RBC dynamics. Here, we employ a simple and highly reliable method to fabricate hydrophilic PDMS microchannels with specific hydrophobic sections in order to produce water-in-oil-water (W/O/W) double emulsion droplets. The double-emulsion is formed using three flow-focusing devices in series. The first one must be hydrophobic to generate water-in-oil droplets, while the second and third ones are hydrophilic and used for coating the single emulsion and separating the double emulsion droplets, respectively. In order to create a hydrophobic section, adhesive tape is used to mask the required regions on the PDMS slab and glass slides, respectively, before plasma treatment that renders the rest of the PDMS surface hydrophilic and enables subsequent bonding. We use a UV curable monomer for the oil phase which encapsulates the droplets and expose the double emulsion droplets to UV irradiation in a glass capillary which activates the polymerisation reaction to form elastic shells. Both, the capsule size and shell thickness are tunable by changing the flow rates of the inner and middle phases. The membrane properties are characterised using methods reported in the literature.

Surface-washing of contaminated porous substrates

Francesco P. Contò (1), Merlin A. Etzold (1), Emily Butler (2), Julien R. Landel (2), Stuart B. Dalziel (1)

1) University of Cambridge, Department of Applied Mathematics and Theoretical Physics, Wilberforce Road, Cambridge, CB3 0WA, UK; 2) University of Manchester, Department of Mathematics, Oxford Road, Manchester, M13 9PL, UK

The cleaning of porous surfaces is a challenging problem in everyday life and industrial practice since it can lead to a redistribution of the absorbed contaminant within the porous material instead of a complete removal of the unwanted agent. In this work, we present surface-washing experiments modelling the decontamination of porous substrates. Firstly, we report a protocol to manufacture mechanically stable porous media by sintering glass ballotini (< 1 mm) to form free-standing homogeneous porous plates or composite structures where a porous matrix is sintered onto solid glass backing and surrounds. The ability to incorporate directly a solid glass backing provides a method of preventing any liquid leaks through their rear. These samples are then integrated into an apparatus used in a previous work which studied surface-washing decontamination of impermeable surfaces. A dyed fluid is placed onto the porous substrate to simulate the region of contamination. The surface-washing is simulated by a thin (~1 mm) film of water flowing from a reservoir through a gap over an inclined porous-glass surface. The resulting interaction between the cleansing film flow and the contaminating dye is then tracked using dye attenuation, enabling us to analyse the contaminant field in space and time. Our experiments provide insights on the role of initial conditions (wet/dry substrate), the impact of cleaning strategies on industrial performances (e.g., decontamination time), and the relevant transport mechanisms of the contaminant (gravity or capillary advection and diffusion in both water and porous medium). Importantly, they demonstrate a decontamination-induced redistribution of the contaminant within the porous matrix.

Melt percolation in solid rocks: A numerical study of pattern formation in rock-melt mixtures

Giulia Fedrizzi (1), Sandra Piazolo (2), Sam Pegler (3), Oliver Harlen (3)

(1) CDT in Fluid Dynamics, University of Leeds, (2) School of Earth and Environment, University of Leeds, (3) School of Mathematics, University of Leeds

Significant parts of the lower and middle crust of the Earth are known to be partially molten when at depth. The presence of melt within the matrix of solid minerals has consequences on crust rheology and structure. During partial melting, melt forms in segregated pockets, usually localising along grain boundaries. Once the first melt, i.e. fluid, is formed, density differences will cause the fluid phase to migrate. In this process, melt percolates through the rock forming melt pockets, continuous channels and/or a melt channel network. The phenomena that control this movement are still poorly understood. This study aims to numerically reproduce structures found in partially molten rocks from lower and middle crust and provide an explanation for their formation. Numerical simulations are performed to assess the role of the main processes that influence melt percolation in a solid rock. Our model uses a hybrid approach, both Discrete-Element and continuum, to reproduce the behaviour of the two phases: the brittle matrix is represented by a network of elastic springs and melt percolation is modelled as fluid flow in a porous medium. The solid phase can be fractured by external deformation and/or an increase in fluid pressure caused by the formation of new melt. Fractures increase permeability and favour the mobility of the fluid phase, helping separated pockets merge. Therefore, melt viscosity, heterogeneities and rate in melt production greatly affect the characteristics structure of the aggregate. We also investigate how the physical properties of both phases affect melt percolation and compare resulting structures with natural examples.

Effects of shell thickness on cross-helicity generation in convection-driven spherical dynamos

Luis Silva, Parag Gupta, David MacTaggart and Radostin D. Simitev

School of Mathematics and Statistics, University of Glasgow – Glasgow G12 8QQ, UK

Rotating thermal convection is ubiquitous within the interiors and the atmospheres of celestial bodies. These fluid regions contain plasmas or metallic components so vigorous convection drives large-scale electric currents and produces the self-sustaining magnetic fields characteristic of these celestial objects. Here, we are interested in understanding the relative significance of two key mechanisms for the generation and amplification of magnetic fields, namely the helicity and cross-helicity effects of mean-field dynamo theory. This study aimed to test the hypothesis that the turbulent helicity effects (α-effect) are more significant in the case of geodynamics, while the cross-helicity effect (γ-effect) is more important in the case of the global solar dynamo, due to differences between the shell aspect ratio of the solar convection zone and that of Earth's inner core. To assess α- and γ- electromotive effects, we performed, an extensive suite of over 40 direct numerical simulations of self-sustained dynamo action driven by thermal convection in rotating spherical fluid shells, where the shell thickness aspect ratio η is varied at fixed values of the other parameters. The simulations are based on the Boussinesq approximation of the governing nonlinear magnetohydrodynamic equations with stress-free velocity boundary conditions. Two distinct branches of dynamo solutions are found to coexist in direct numerical simulations for shell aspect ratios between 0.25 and 0.6 – a mean-field dipolar regime and a fluctuating dipolar regime. The properties characterizing the coexisting dynamo attractors are compared and contrasted, including differences in temporal behavior and spatial structures of both the magnetic field and rotating thermal convection. The helicity α-effect and the cross-helicity γ-effect are found to be comparable in intensity within the fluctuating dipolar dynamo regime, where their ratio does not vary significantly with the shell thickness. In contrast, within the mean-field dipolar dynamo regime the helicity α-effect dominates by approximately two orders of magnitude and becomes stronger with decreasing shell thickness.

Lateral stress effects on bubbles in Hele-Shaw channels

Jacob Harris, Andrew Hazel, Alice Thompson

The University of Manchester

Hele-Shaw channels are fluid filled regions between two marginally separated parallel plates that are bounded in one of the lateral directions. Generally Hele-Shaw channels appear at small length scales and are used as a proxy for various low Reynolds number problems such as flow within rock fractures or in blood vessels. A modern field of interest is the propagation of bubbles through Hele-Shaw channels, which has applications in drug transport and general applicability in Lab-On-A-Chip experiments. It can be difficult to solve the Stokes equations numerically for these free-boundary bubble problems and so often flow is approximated using a simple depth averaged model (an application of Darcy's law). Using this Darcy model, multiple stable solution branches can be found for the problem of a steadily propagating bubble in a Hele-Shaw channels. Significantly, the Darcy model neglects all lateral stresses, which may have a strong influence on each of these solution branches. I will derive a novel alternative model that is depth averaged, retaining some of the Darcy models simplicity, but now incorporating these lateral stresses. With this new model I'll reproduce the first three solution branches to this steady bubble propagation problem and assess how the presence of lateral stresses influences the overall shape and bifurcation structure of these solutions. In particular, I'll focus on the effect of channel aspect ratio on the existence of the more exotic, unstable, solution branches predicted by the Darcy model.

Viscoelastic fluid flow in microporous media

Victor Ibezim, Robert J. Poole, J.C Dennis

University of Liverpool

The importance of the flow of viscoelastic fluids through porous media in the chemical, biological and oil industries has led to the topic receiving significant attention from researchers over the past decade. Although some progress has been made in this time, the scarcity of comprehensive rheology data as well as the limitation of measurement techniques has restricted a majority of the study to average macroscopically measurable flow quantities like porosity, permeability and pressure gradient. As a result, the subject remains incompletely understood. Here we describe an experimental investigation of the elastic effects of flexible polymer solutions through a distinctive micro-porous structure with a primary focus on relating the bulk flow properties to measurable rheological parameters. A porous glass structure (with typical pore sizes of 500 microns) is formed by removing a pore-forming agent from slightly sintered glass beads (of size 50 microns). Pressure-drop measurements are conducted and non dimensionalized to scale-out the effects of fluid shear viscosity. To investigate the effects of fluid elasticity, aqueous solutions of a polyacrylamide (PAA), a mixture of polyethylene oxide (PEO), polyethylene glycol (PEG) in the concentration range of 12.5-1000ppm, which were characterized in both shear and extensional flows using a shear and a capillary break-up extensional rheometer (CaBER) respectively, were used as working fluids. To quantify the importance of elasticity, a Weissenberg number (Wi) is calculated as a product of CaBER relaxation time and the nominal shear rate in the flows. Results, suitably normalized with Newtonian pressure-drop data show a critical Wi of roughly 0.01 where all working fluids reveal the onset of elastic dominance over viscous forces as the flowrate increases. Such critical value of Wi is due to the estimate of a nominal shear rate based on pore size which severely underestimates the maximum shear rates within the complex pore structure. Significant deviation from the universal behaviour is observed for high concentrations of the PAA, which is thought to be a result of shear-thinning for these systems. Systematic degradation of the polymer caused an exponential decay in elasticity, which is reflected in the pressure-drop measurements.

Fluid dynamics around freely falling ice-particle crystals: An experimental investigation using three- dimensional particle tracking velocimetry

M. H. Khan 1, C.D. Westbrook 1 , M. W. McCorquodale 1,2

(1) Department of Meteorology, University of Reading, Reading, UK (2) Department of Civil Engineering, University of Nottingham, Nottingham, UK

The falling behaviour of atmospheric ice particles is of interest within meteorological contexts as a result of applications within climate modelling and weather prediction. An understanding of the structure of the wake of natural ice particles is required in order to correctly model ice particle growth processes, and thus to predict the terminal velocity and orientation that ice particles adopt in free-fall. The central role of particles in establishing the fate of cloud provides a motivation to investigate the aerodynamics around freely falling ice-particle crystals. The free fall of ice-particles in the cloud was experimentally investigated by dropping ice-particle crystal analogues in a vertical quiescent fluid tank containing water-glycerine mixture at equivalent Reynolds numbers to the atmospheric case. Measurements were conducted using three-dimensional particle tracking velocimetry (3D PTV) for a freely falling 3D printed ice-particle crystal analogue. When the particle falls freely in the fluid, the dynamics of the wake depends on the particle shape and kinematics. Flutter and rotation were noted for some particles, and connected to unsteady fluid flow in the particle wake at various Reynolds numbers which helped identify the flow regimes. The flow regimes are categorised into (i) steady, (ii) periodic shedding and (iii) chaotic regimes. Coherent vortical structures identified using $\lambda 2$ criterion showed smaller vortical regions attached to the particles in the steady regime. They are identified as potential regions where the water droplets accumulate on the particle due to longer residence time of the fluid in the attached vortical region. The observed flow is highly three dimensional due to the irregular geometry and spiralling trajectories of the falling particles. Large coherent vortical structures are shed in the periodic regimes. Smaller eddies were noted at higher fall Reynolds numbers in the chaotic regime.

Flexible sheets in turbulent flow

Marin Lauber, Gabriel D. Weymouth, Georges Limbert

University of Southampton

Fluid-structure interaction of thin flexible structures, such as shell or membranes, are common both in nature and in engineering. Accurate simulations of these systems could enable advances in many biological science and flexible structure engineering. Standard body-fitted method struggle for those type of systems due to the large structural deformation that require constant remeshing leading to low computational efficiency. On the other hands, immersed boundary methods can deal with those large deformations with ease but need special care in the application of boundary conditions and measurement of fluid forces. We present a numerical approach that combines an accurate immersed boundary finite-volume method specially tailored for thin structures, coupled to shell finite-element method to simulate such systems. We provide numerical results of the flow around a flexible plate at an angle of attack as well as the flow over a flexible wing under harmonic motion and demonstrate the numerical accuracy and stability for large deformation fluid/structure interactions.

Robotic inspection of pre-filled medical syringes

Hamza Liaquet, Nikil Kapur, Greg De Boer and Cédric Beaume

University of Leeds Centre for Doctorial Training

Pre-filled medical syringes (PFMS) are the fastest growing method of injectable delivery devices. The PFMS device is manufactured from various materials including as glass, plastic, rubber, and metals. Robotic inspection of the devices for defects is a vital step in ensuring patient safety and satisfying regulatory requirements before being released onto the market. One part of this process checks for foreign particles, in which detection is facilitated by means of fluidisation, where particles are forced away from the walls, allowing for effective visualisation. This is achieved by impulsively rotating the PFMS about a central axis and abruptly stopping the rotation, which is referred to as the spin cycle by manufacturers. Previous studies have investigated the fluid dynamics on the spin-up and spin-down of a cylinder. The resulting flow exhibits a high degree of complexity and the formation of secondary flow structures. The impact of these flow fields on particle motion during the fluidisation step has yet to be considered and provides the motivation for this study. This work, using a combination of experimental and computational methods, aims to link the dynamics of the spin cycle with the fluid flow behaviour and ultimately the particle motion to better understand the automated inspection process. This will support the optimisation of the spin cycle as a key part of bringing new products to market.

A finite volume coupled level set and volume of fluid method with a mass conservation step for simulating two-phase flows

Konstantinos Lyras, Jack Lee

King's College London

The accurate numerical simulation of two-phase flows of two immiscible fluids faces the problem of resolving moving interfaces that deform under the influence of various forces such as surface tension and gravitational forces. In the present numerical study, a novel methodology for simulating two-phase flows based on a conservative level set method is presented. The modelling approach has been developed and tested for immiscible fluids with different density and viscosity. The novel algorithm uses an accurate method for the calculation of the level set function and mass fluxes and is shown to have a reasonable mass loss error. The level set function is calculated with a two-step correction algorithm that uses the calculated liquid volume fraction as a constraint to make sure that mass is conserved and that the fluxes are properly updated. With this approach, no further interface reconstruction step is required, simplifying the evaluation of the curvature and surface tension force calculation. The methodology is evaluated with classical numerical tests such as the rising bubble test, the 3d rotating vortex, and Zalesak's disc for both unstructured and structured meshes. In all the tests, the methodology provides a sharp interface between the fluids with an accuracy of the same order or better than other VOF and level set methods . The results show that the methodology is very accurate when strong three-dimensional deformations, surface tension or buoyant forces are present, improving the accuracy of the standard OpenFOAM code interFoam by an order of magnitude for the majority of the tests here. Consequently, the method can be used for simulating various liquid-gas and liquid/liquid flow applications such as bubbly flows and sprays, offering an accurate representation of the interface dynamics.

Particle patterning inside glass capillary tubes using thin film surface acoustic wave devices

Sadaf Maramizonouz(1), Changfeng Jia(2), Mohammad Rahmati(1), Tengfei Zheng(2), YongQing Fu(1)

(1) Faculty of Engineering & Environment, Northumbria University, Newcastle upon Tyne, NE1 8ST, UK (2) State Key Laboratory for Manufacturing Systems Engineering, Xian Jiaotong University, 710049, Xian, 710048, P.R. China

Capillary tubes can be integrated in microfluidic platforms to manipulate microparticles and biological cells for different applications such as particle patterning and continuous flow particle aligning and focusing. However, the particle patterns formed due to acoustic fields inside capillary tubes with different cross-sections and positioning regarding to the electrode directions have not been thoroughly investigated. Neither acoustic patterning of microparticles inside capillary tubes with and without a continuous flow have been studied. In this research, we systematically investigated the patterning and alignment of microparticles inside glass capillary tubes using thin film surface acoustic wave devices with and without liquid flow. Effects of different cross-section geometries of the capillary tubes as well as their different positioning directions compared to the direction of the electrodes were studied. We found that for rectangular glass capillary tubes, the particle pattern lines are parallel to the sidewalls of tubes when the rectangular glass tube is positioned at different angles regarding the electrodes, which is caused by the standing wave field formed inside the rectangular glass tube. Whereas for the glass capillary tubes with circular cross-section, alignments of particles have shown different patterns at various heights of the circular tubes due to the complex acoustic field generated inside the tube. For cases with continues flow, under the agitation of acoustic waves, particles are patterned in lines parallel to the flow direction (the same as the tube's direction) regardless of the positioning for both the rectangular and circular glass tubes because the liquid flow removes all the other patterns.

The creation of a photoelastic force balance

Bradley McLaughlin, John Lawson, Bharath Ganapathisubramani

University of Southampton

Photoelasticity is an optical method of stress and strain analysis, taking advantage of a mechanical property called birefringence which is found in many transparent polymers. This allows fringe patterns to be observed when imaged through polarized light where the patterns are related to the applied stress. In this study, we examine the feasibility using photoelastic material as the sensor in a floating element balance. We use polyurethane pads as the photoelastic material and embed pins within the material in order to transfer the applied load. The pins themselves are connected to the floating element. The pads are illuminated with LEDs mounted in in the balance and the fringe patterns are imaged using low-cost USB cameras. A series of known loads are used to create a calibration matrix, which is then used to measure the lift and drag curves of a NACA-0015 wing mounted on to the floating element. Initial tests have shown promising results from this method and further testing will be undertaken to fully validate the use of the balance. This optical approach enables remote measurements of loads, e.g. underwater, whilst the simplicity of the design allows it to be scaled up to distributed arrays of sensors. The balance can also be sized to different loads and can be scaled to fit in difficult places as needed. This will make a photoelastic balance a versatile means of force/moment measurement.

Inactivation effect of human thermal plume and upper-room ultraviolet air disinfection on the COVID-19 transmission

Shuo Mi (1), Clive B. Beggs (2), and Eldad J. Avital (1)

(1) School of Engineering and Materials Science, Queen Mary University of London, London, UK (2) Carnegie School of Sport, Leeds Beckett University, Leeds, UK

The COVID-19 pandemic has prompted attention upon potential methods to reduce the spread of viral aerosols or particles in closed spaces. The present study investigates transmission of microbial pathogen aerosols and the inactivation effect of upper-room ultraviolet (UVC range) air disinfection. An Eulerian–Lagrangian method implemented using URANS has been applied through the OpenFOAM package to study the aerosols movement and the process of viral aerosols inactivation with indoor airflow and upper-room UVC disinfection. The thermal plume generated by a person is shown to drive the emitted aerosols towards the upper-room region that is disinfected by 254 nm UVC light (above the person's head), but a fountain effect has also been generated. Under those effects, half of the overall SARS-Cov-2 viral particles could be inactivated by upper-room UVC in 12 minutes if about 30W lamp input per 200 ft2 is used in a small non-ventilated office equipped with one wall mounted lamp, while having higher inactivation rate at the breathing height of the person. Raising the irradiance to 50W per 200 ft2 will yield almost 85% inactivation in such conditions. Adding a ceiling fan will further much accelerate the inactivation process, as will also be discussed in the conference.

Asymptotic framework for flood models comparison

Piotr Morawiecki, Phillipe Trinh

University of Bath

Estimating the river flow given the rainfall and catchment data is key problem in hydrology, especially in flood risk assessment. Many state-of-the-art approaches for estimating flood risk include physical, conceptual, and statistical modelling. Since physical models based on fundamental laws of hydrodynamics are highly data and computationally-demanding, data-based models are often used instead. Conceptual models describe flow pathways using a simplified mathematical formulation, while statistical model describe relationship between input and output parameters using regression methods. Despite their overall good performance, it is observed that data-based modelling approaches at some situations give inaccurate predictions, especially in conditions underrepresented in the training data (e.g. for extreme rainfalls or flow impacted by climate change). Understanding the limits of these models applicability remains an open challenge. Here we present a unified framework using asymptotic analysis, which highlights differences between these modelling approaches. We formulate a benchmark physical model involving coupled subsurface and overland flow based on Richards and St. Venant equations, which we analyse in order to obtain the characteristic scaling laws describing river flow over a range of scenarios. The comparison of scaling laws between the different classes of models reveals fundamental differences between the models behaviour. In consequence, the proposed approach gives a better understanding of uncertainties in hydrologic models, and may lead to development of more theoretically-justified flood estimation methods.

African Easterly Wave precursors to tropical cyclogenesis in a convection-permitting model

Fran Morris (1), Dr Juliane Schwendike (1), Professor Doug Parker (1), Dr Caroline Bain (2)

(1) University of Leeds, (2) UK Met Office

African Easterly Waves (AEWs) are synoptic-scale baroclinic-barotropic disturbances along the African Easterly Jet which propagate over West Africa. AEWs are strongly associated with organised convection, which can both be triggered by the waves and can also act to trigger and maintain them. AEWs and their embedded convection are known to influence the genesis of tropical cyclones (TCs) in the Atlantic Ocean. This connection has been shown through various case studies using observational and reanalysis datasets, and mechanisms for the development of easterly waves into tropical depressions have been proposed. However, AEWs are not always represented accurately in global climate models and their simulation consistently results in a wide range of projections, especially for southern-track AEWs. Since it has been shown that AEW activity directly impacts TC genesis in such climate models, improving their representation is crucial to predicting cyclone activity. Convection-permitting (CP) models have improved the representation of the West African Monsoon, the diurnal cycle over West Africa, and the characteristics of mesoscale convective systems compared to models with parameterised convection. This may allow us to represent the coupling between convection and AEWs better, and therefore more accurately simulate the development of TC precursors. This research uses data from CP4-Africa, a regional CP climate configuration of the Met Office Unified Model, to examine the development of AEWs into TC precursors. We compare this to R25-Africa, an analogous regional model with parameterised convection, to analyse how CP models might add value to the prediction of Atlantic cyclogenesis. Furthermore, we examine the future-climate configurations of CP4 and R25 to investigate how TC precursor behaviour may change under warming.

Pattern formation in polymer droplets spreading on a smooth substrate

Ahmed Othman (1), Andreas Poulos (2), Ophelie Torres (2), Alex Routh (1)

(1) Colloidal Group, University of Cambridge. United Kingdom. (2) Unilever Port Sunlight. United Kingdom.

Controlling the final film morphology in evaporating droplets containing polymer is key for different applications e.g. skincare, inkjet, and biomaterials. Uniform films are often required, yet certain patterns can optimise the film functionality and are required for applications such as microelectronics. The final film morphology has been investigated intensively for droplets and thin films containing different components, such as colloidal particles, resulting in the coffee ring. Drying of spreading volatile droplets containing polymer or binary mixtures is shown to exhibit a fingers-like pattern. This work aims to understand the occurrence and methods to suppresses this instability. Different parameters influence the formation of this instability including, substrate roughness, mixture viscosity, and evaporation rate. The physical mechanism is that the concentration of polymer, or less volatile liquid, increases at the droplet periphery due to the fast evaporation of the volatile liquid. This then leads to solutal Marangoni flow, creating a ridge at the periphery which leads to the instability. The theory presented is based on a thin-film lubrication model and the governing equations are solved near the droplet contact line to show the capillary ridge using numerical methods. We experimentally show the formation of the capillary ridge and the dependence of the instability on polymer concentration.

Jet tilt and the understanding of North Atlantic jet variability and regimes

Jacob Perez (1) , Amanda Maycock (2), Stephen Griffiths (3), Christine Mckenna (2) and Steven Hardiman (4)

CDT in Fluid Dynamics, University of Leeds (1), School of Earth and Environment, University of Leeds (2), School of Mathematics, University of Leeds (3), Met Office (4).

The Polar Jet Stream (PJS) is the main driver of extreme weather events occurring in the mid-latitudes in the Northern Hemisphere. The 'Beast from the East', that brought high levels of snowfall and flooding to the UK in Feb 2018, is a key example of such an event. It is essential to be able to forecast the position of the PJS over the North Atlantic to accurately predict these weather events. Recent studies have shown that the PJS occupies three preferred states, or regimes, over the North Atlantic during the winter representing a Northern, Central and Southern located jet. Each of the regimes influence the surface weather differently, but it is unclear what drives each of the regimes and how the mechanisms that causes the PJS to transition between them. One limitation of current 1-D analyses of the jet latitude index is that they rely on sectoral averaging over the North Atlantic basin, thereby smoothing out the tilted structure of the jet which is a canonical feature of the circulation in the North Atlantic. This study applies an approach to reanalysis data (ERA5) that attempts to account for both the jet latitude and tilt, thereby extending earlier analyses. We consider the extent to which cases with a Greenland tip jet influence the North Atlantic jet latitude index and whether accounting for the tilt of the jet provides additional information about jet variability. Plans for future work will be outlined, with the intent to derive/develop a dynamical systems model that captures the regimes of the PJS, with the intent to use this model for further studies.

Cusps-filaments at receding viscoelastic contact line

Saksham Sharma and D. Ian Wilson

Department of Chemical Engineering and Biotechnology, University of Cambridge, Philippa Fawcett Drive, Cambridge CB3 0AS, UK

Understanding wetting and dewetting phenomena is crucial in many industrial applications, such as making functional or tuneable thin films. In this work, we investigate the dynamics of the contact line of a viscoelastic thin film draining downwards on a vertical plate under the action of gravity. Depending on the properties of the substrate and liquid, the contact line exhibits four distinct regimes: smooth; wavy; wavy with cusps; cusps with laments. While the formation of cusps indicates the tendency of a receding wavy contact line to form drops, polymeric laments trailing these cusps can prevent droplet formation, thereby offering a way to tune the deposition of polymers on the surface. We present experimental results to delineate the regimes and empirical models to describe the transition between them. The dynamics of thin-film evolution is studied in two stages: early stage, where different shape regimes of contact line are shown; and the intermediate stage, where the motion of the contact line is correlated with the substrate wettability and the concentration of polymer in the viscoelastic fluid.

Index of presenters

Abuhtira, Ahmed 07B-5

Adekanye, Damilola 01B-5

Agrawal, Rishav 10B-1

Akter, Shahina 06B-1

Anderson, Jacob PB01

Andrews, Elijah PB02

Armstrong, Steven 10C-1

Arslan, Ali 01B-4

Astoul, Aurelie 11B-5

Aswathy M S 07B-1

Balatti Davide 07A-3

Belcher, Stephen I2

Bempedelis, Nikolaos 10A-5

Bhagat, Rajesh Kumar 02C-5

Bolton, Jonathan PA01

Bose, Chandan 12A-1

Boye, ThankGod Enatimi 06A-1

Brokof, Philipp 12B-1

Brolly Martin PA03

Bullamore, Danielle PB03

Burton, Thomas PA04

Butler, Emily 09B-1

Butler, Matthew 08C-1

Cakir, Meltem 07B-2

Cargill, Solene 03B-1

Carter, Douglas 12A-2

Castelino, Jennifer 01B-2

Cene, Antonios 06A-2

Chakra, Adnan 08C-5

Chalk, Caitlin 10B-2

Charlwood, Samuel 01A-4

Charpentier, Thomas 12B-2

Chasapis Tassinis, Konstantinos PB04

Chatterjee, Rajanya 07B-3
Chen, Yuan 05B-3
Chen, Boyang 06C-1
Chen, Qi PB05
Cherry, Molly 03B-2
Contò, Francesco P. PB06
Cookson, Andrew 05C-5
Corbett, Callan 02C-1
Cousins, Joseph 01C-5
Croci, Maximilian PA05
Cunliffe, Christopher 03C-1
D'Ambrosio, Hannah 01C-4
Datta, Saikat 11C-4
de Langre, Emmanuel I3
De Voogt, Francis PA06
Dent, Francis 10C-2
Dockar, Duncan 09C-4
Duguid, Craig 11B-1
Ebrahimzade, Nima 04A-2
Edwards, Andrew 10C-3
Engel Matthias 07A-4
Falcone, Matthew A 06A-5
Fedrizzi, Giulia PB07
Finke, Jannis 03C-2
Fourtakas, Georgios 07C-1
Frank, Daria 10B-3
Fung, Lloyd 03B-3
Galeano-Rios, Carlos 08C-2
Gambuzza, Stefano 03A-1
Garbacz, Ana Catarina 05A-1
Giannenas, Athanasios 06A-3
Gilbert, Rhys 04A-1
Gillespie, Alexander 08A-5
Goulden, Olivia 01B-3
Gray, Liam 01C-3
Greenslade, Thomas 05A-2

Grondeau, Mikael 12A-3
Gupta, Akanksha 02B-1
Gupta, Parag PB08
Gururaj, Saranraj 02B-2
Hajirasouliha, Farzaneh 03B-4
Hall, Madeleine 03B-5
Hamzehloo, Arash PA07
Harris, Jacob PB09
Harvey, Eleanor 09B-2
He, Wei 08A-2
Hendrikse, Rachel 04C-2
Hengster Yasmin 07A-5
Hickling, Tom 03A-2
Hodgkin, Amy PA09
Holford, Jacob 05B-2
Houtman, Jelle 08A-4
Hu, Xiao 01A-2
Hue, Seng 05C-1
Ibezim, Victor PB10
Jackiw, Isaac 01C-1
Jackson, Stephen D 02A-5
Jadidi, Mohammad 09C-3
Jain, Animesh 08B-1
Jaiswal, Prateek 11A-1
Jiang, Shujie 04A-3
Jiang, Xianyang 11A-2
Jiao, Yuxin 04A-4
Jumana, Sadia Anjum 09B-5
Kemp, Gael 06A-4
Khan, Majid PB11
Khodaparast, Sepideh 09C-5
King, Jack 07C-2
King, Jack 07C-3
Kontogiannis, Alexandros 04B-3
Kreczak, Hannah 11A-5
Krishna, Swathi 09A-5

Kumar, Abkishek 12B-3
Kurt, Melike 10A-1
Kusumaatmaja, Halim 10C-4
Lalloz, Samy 01C-2
Lalloz, Samy 06B-2
Lauber, Marin PB12
Lawson, John 12C-1
Lefauve, Adrien 11A-4
Li, Qianhui 11A-3
Liaquet, Hamza PB13
Lim, Desmond PA10
Lino Valencia, Mario PA11
Lydon, Karl 04A-5
Lyras, Konstantinos 06C-4
Lyras, Konstantinos PB14
MacRaild, Michael 05B-4
Maramizonouz, Sadaf PB15
Maretvadakethope, Smitha 09B-4
Martínez Carmena, Alfonso 03A-4
Masoudi, Esmaeel 01A-1
Massegur, David PA12
Massey, Jonathan 02A-1
Mays, Michael D 01A-3
McHale, Glen 10C-5
McHale, Glen 11C-1
McLaughlin, Bradley PB16
Mcnair, Richard 04C-3
Medjnoun, Takfarinas 12A-4
Mi, Shuo PB17
Mikhaylov, Kirill 05B-1
Mitchell, Andrew 05C-3
Mohan, Vishnu 05A-3
Moise, Pradeep 08A-1
Morawiecki, Piotr PB18
Morgado, Fabio Pereira 05A-4
Morgans, Aimee I1

Morris, Francesca PB19
Morrow, Liam 12C-2
Morton, Ben 11B-2
Moss, John 02B-5
Municchi, Federico 12C-3
Myers Hill, Joseph 06C-5
Nagy, Peter 03A-3
Najvan, Muhammed 03A-5
Newton, Tom 01A-5
Nilsson, Mattias PA13
Nolan, Craig 02A-4
Nóvoa, Andrea 08B-2
O'Brien, Eoin 02C-2
Offner, Avshalom 11C-2
Ortega-Chavez, Rigobeerto PA14
Othman, Ahmed PB21
Ozbay, Ali Girayhan 05B-5
Parapamulla Yapa Arachchige, Gayani 02A-3
Paulin, Oliver 04C-1
Peng, Han 05A-5
Perez, Jacob PB22
Poole, Rob I4
Pratt, Jane 11B-3
Qin, Zhuofan 03C-5
Qureshi, Imran 09A-4
Rabab, Saba PA15
Racca, Alberto 06B-3
Rahul, Sundar PA16
Rees Jones, David 01B-1
Rees-Zimmerman, Clare 08B-3
Rouquier, Anthony 10B-4
Ruiz-Bonilla, Sergio 07C-4
Samuels, Callum PA17
Schäfer, Stephan 03C-4
Schirrmann, Kerstin 09C-2
Schlander, Rasmus Korslund 09A-1

Scott, Isabelle 04B-2

Sewerin, Fabian 06C-2

Shane, Nicholas 09A-2

Sharma, Pushpender 07A-1

Sharma, Saksham 02C-3

Sharma, Saksham PB24

Shemilt, James PA23

Shone, Fergus PA24

Singh, Naval 05C-2

Siregar, Riza Inanda 08B-4

Smyth, Amanda 10A-2

Soman Pillai, Arun PA18

Southern, Emily 06B-4

Stafford, Chris 11C-3

Steiros, Kostas 10A-3

Stuhlmeier, Raphael PA19

Swannell, Barnum 04B-1

Syawitri, Rista PA20

Sykes, Thomas 08C-3

Tan, Raynold 07A-2

Thery, Albane 04C-4

Thompson, Alice 08C-4

Thompson, Craig PA21

Vilumbrales Garcia, Rodrigo 02A-2

Walton, Gregory 07B-4

Wang, Mingyang 06C-3

Wangsawijaya, Dea 09C-1

Westwood, Timothy 09B-3

Wienkers, Aaron 02B-3

Wilczynski, Fryderyk 11B-4

Wilk, Christophe 05C-4

Wise, Nick 02B-4

Yang, Zhuangjian 03C-3

Zagklavara, Foteini 04B-4

Zampino, Gerardo 09A-3

Zauner, Markus 08A-3

Zhang, James 02C-4

Zhdanov, Oleksandr 10A-4

Zhou, Honghao 08B-5

Zhu, Taihang PA22