UK Fluids Conference 2022

University of Sheffield

The Diamond

6-8th September 2022

UK Fluids Conference $6^{th} - 8^{th}$ September 2022

Table of content

Link to conference	1
Conference venue	1
Conference overview	1–3
Invited speakers	4–7
Short programme	8
Oral presentations	9–16
Poster presentations	17–18
Oral Presentation abstracts	19–176
Poster presentation abstracts	177–206

Link to conference

You can access the conference site at this link: `https://ukfluids2022.sheffield.ac.uk/`. The detailed programme of the conference including the oral and poster presentation abstracts can be downloaded using the above link.

Conference venue

The Conference will take place at the The University of Sheffield, Faculty of Engineering, The Diamond Building. Oral session will be held in LT1, LT3 and LT4, and posters will be displayed in the Hall outside the Lecture Theatres, where lunches and refreshments will be served.

UK Fluids Conference series

The UK Fluids Conference aims to bring together all those involved in fluid mechanics research in the UK. PhD students and post-doctoral researchers are encouraged to attend as well as established researchers. There will be a lively programme of invited and contributed talks, and poster sessions, to foster communication amongst the community.

The UK Fluids Conference took place previously in London (ICL, 2016), Leeds (2017), Manchester (2018), Cambridge (2019) and Southampton (online, 2021). This year, the UK Fluids Conference will be held at The University of Sheffield, Sheffield, during 6-8th September 2022.

This year, keynote speakers are Prof Catherine Noakes OBE FREng, University of Leeds, Dr Julien Hoessler, McLaren, and Dr Nathalie Vriend, BP Institute, Cambridge

University. This year's UKFN Thesis Prize Winner Dr Craig Duguid is also invited to give a talk on his prize-winning research.

Prizes

You can vote for best student presentations and posters (highlighted in blue in the schedule) via https://forms.gle/YHkeito21n2YFtsj9 or the QR code below. Please do this before Wednesday 18:00, to help the prizes committee make their choices.

QR code for voting.

Instructions for session chairs for parallel sessions

Each talk is allocated a 13-minute slot (10 minutes talks, 2 minutes questions, 1 minute changeover). The role of session chairs is to keep the sessions to time. Chairs will provide short introduction of talks, invite and moderate questions and getting the next speaker set up. A synchronized clock will be available in each room to show presenting (10 min), Q&A (2 min) and changeover (1 min) time for each talk. Once the Clock shows Q&A time, ask the presenter go to the last slide and conclude their talk. Chairs should be present from 5-10 minutes before the session to meet the session assistant and speakers. 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. Chair should also complete the Student Prize Form at the end of their sessions.

Instructions for keynote session speakers and chairs

Each keynote talk is allocated 45 minutes and please speak for no longer than 35 minutes and leave 10 minutes for questions. The chair should introduce the speaker, invite and moderate questions.

Instructions for presenters in oral sessions

Each talk is allocated 12 minutes with a further minute for change over. You should finish speaking in 10 minutes leaving 2 minutes for questions. A synchronized clock in each room will show presenting (10 min), Q&A (2 min) and changeover (1 min) time for each talk. You should immediately go to the last slide and conclude your talk once in Q&A time. Presenters should be at the front of the room from 5-10 minutes before the session to meet the session chair and assistant. You should copy your presentation to the computer for the projector or test your laptop during this time. Presenters should remain in the room after the session to facilitate informal discussions.

Instructions for poster sessions

Your poster is allocated to one of the 2 poster sessions. Please mount your poster on the allocated display board during the lunch time on the day of your session. Posters can remain on the boards until the end of the conference. Please be present at your poster throughout your session and be ready with a short tour of your poster (1-2 minutes) when you get visitors. Your poster should be in A0 Portrait format.

Conference Organising Committee

Dr Marco Colombo - University of Sheffield
Dr Melika Gul - University of Sheffield
Professor Shuisheng He - University of Sheffield (Chair)
Professor Kostas Kontis - University of Glasgow
Professor George Papadakis - Imperial College London
Professor Ning Qin - University of Sheffield
Dr Maarten van Reeuwijk - Imperial College London
Dr Andrew Ross - University of Leeds
Professor Neil Sandham - University of Southampton
Dr Ashley Willis - University of Sheffield
Dr Elena Marensi – University of Sheffield

Dinner

The Conference Dinner will take place on Wednesday 7^{th} September at the Firth Hall (`https://goo.gl/maps/TXNJX9GJkxbwhVUa7`) starting from 7:00 pm.

Invited speakers

Prof Catherine Noakes - OBE FREng, University of Leeds

Professor Cath Noakes is a chartered mechanical engineer, with a background in fluid dynamics. She leads research into ventilation, indoor air quality and infection control in the built environment using experimental and modelling based studies. She is Deputy Director of Leeds Institute for Fluid Dynamics and she co-chaired the Environment and Modelling subgroup of the UK Scientific Advisory Group for Emergencies (SAGE) focusing on the science underpinning environmental transmission of COVID-19.

The complexity of airborne transmission – from fluid dynamics to policy: COVID-19 has presented us with the most difficult healthcare and societal challenge we have faced in living memory. As a new disease we have had to rapidly build the knowledge base on every aspect of the virus to understand transmission, develop vaccines and therapeutics, and mitigate risks in our everyday lives.

Over the course of the pandemic we have become acutely aware of the role that the environment plays in transmission of respiratory diseases, and how our interactions in indoor spaces determine the risk of infection. Understanding the routes of transmission is challenging, but modelling of aerosols, droplets and indoor airflows can play an important role in identifying mechanisms and determining mitigations. However, this mechanistic approach is only one part of the picture, and successful management of a respiratory disease is significantly influenced by human behaviour, organisational strategy and policy choices.

This talk outlines some of the approaches that are used to understand mechanisms for transmission from respiratory emissions through to ventilation controls. It will highlight some of the scientific understanding and how that has changed over the course of the pandemic. I will also consider how the scientific advice is used to support policy makers and public messaging and some of the challenges and complexities in this process.

Dr Nathalie Vriend - BP Institute, Cambridge University

Nathalie Vriend obtained her ingenieurs (B.Sc. and M.Sc.) degree in Mechanical Engineering (2004) from the University of Twente, The Netherlands and her M.Sc. (2005, in Aeronautics) and Ph.D. degree (2010, in Mechanical Engineering and Geophysics) from the California Institute of Technology, USA. After a short postdoctoral position (2010 - 2011) and obtaining a competitive personal postdoctoral fellowship (2011 - 2013), she started her own research group in 2014 at the University of Cambridge. Since January 2014, Nathalie has been group leader and Royal Society Research Fellow, first as a Dorothy Hodgkin Research Fellow in DAMTP (Department of Applied Mathematics and Theoretical Physics) and now as a University Research Fellow at the BP Institute, both at the University of Cambridge. Her research program on granular materials involves detailed laboratory experiments and targeted field work, but she also employs numerical simulations and theoretical modelling to complement observations, often in collaboration with scientists from multidisciplinary fields. She has active projects in granular rheology and avalanching, dune structure and migration, and sound propagation. In the past she worked on the dynamics of real snow avalanches, singing sand dunes and seismic wave propagation.

Bedform dynamics: interaction, attraction and repulsion of dunes In desert landscapes, we observe individual sand dunes of different sizes, with a characteristic length scale of up to kilometers, which seamlessly interact with each other and their environment. Can we capture the interaction behaviour of these large objects with simple physical laws? Processes in the field occur over long times and are difficult to investigate in detail, but we are able to scale the physics down to the laboratory. During this presentation, we explore a unique, recirculating laboratory experiment in which we create and trace aqueous dunes over long times. We discover when the interaction between two dunes leads to coalescence or ejection of bedforms and probe the stability of the system under long-time behaviour. In addition, we place objects in the path of our model dunes and explore that both object size and shape matter whether a sand dune is blocked or able to overcome an obstacle and reform on the other side. The central mystery of this research topic is the interplay between small-scale granular physics and large-scale landscape features. The fascinating observation is that certain length-scales change by an order of magnitude or more, but the underlying physics remains the same.

Dr Julien Hoessler - McLaren

After studying Mechanical Engineering in France at Ecole Nationale des Ponts et Chaussees, I went for a MSc in CFD at Imperial College London followed by a PhD with Prof. Spencer Sherwin -funded by McLaren Racing - looking at applying Spectral h/p techniques to complex F1 geometries. My focus was on Stability techniques (PSE and BiGlobal stability) and implicit LES applied to vortex dominated flows. I joined in McLaren straight after starting as a CFD engineer, looking mostly at brake thermal modelling, before joining the CFD team where I held different roles from Methodology engineer to Head of CFD.

I now look after the process and HPC estate used to run internal and external CFD simulations for our F1 and IndyCar challengers, with a particular emphasis on optimising the quality and efficiency under the F1 Sporting regulations.

Aerodynamics development using CFD at McLaren Racing: I will present what we do and some of the specific challenges we face when developing and optimising the aerodynamics of our F1 car, focusing mostly on cfd (different flow regimes, a large operating window and parameter space to explore whilst subjected to regulations limiting compute capacity and wind tunnel fan time to ensure a level-playing field).

Dr Craig Duguid - Newcastle University

I completed my B.Sc. Artificial Intelligence and Robotics followed by M.Sc. Information Systems both at The Robert Gordon University in Aberdeen, Scotland. After this I spent time in industry before deciding I wanted to pursue a career in academia and in particular mathematics. I completed my BA Mathematics part time at The Open University in 2016 before joining the EPSRC Centre for Doctoral Training in Fluid Dynamics where completed my PhD under the supervision of Dr Adrian Barker (principal), Prof. Chris Jones, and Dr Sven van Loo. I am currently a research associate (post doc) at Newcastle University where I am working with Paul Bushy and Toby Wood investigating shear driven magnetic buoyancy in the Sun.

The Influence of convection on tidal flows: Tidal interactions are important in driving spin and orbital evolution in various astrophysical systems such as hot Jupiters, close binary stars, and planetary satellites. However, the fluid dynamical mechanisms responsible for tidal dissipation in giant planets and stars remain poorly understood. One key mechanism is the interaction between tidal flows and turbulent convection which is thought to act as an eddy viscosity dampening the large-scale tidal flow. However, the efficiency of this mechanism has long been debated, particularly in the regime of fast tides, when the tidal frequency exceeds the turnover frequency of the dominant convective eddies.

Using hydrodynamical simulations we investigate the dissipation of the large scale (non-wavelike) equilibrium tide as a result of its interaction with convection. Our approach is to conduct a wide parameter survey in order to study the interaction between an oscillatory background shear flow, which represents a large-scale tidal flow, and the convecting fluid inside a small patch of a star or planet. We simulate Rayleigh-Bénard convection in this Cartesian model and explore how the effective viscosity of the turbulence depends on the tidal (shear) frequency.

We will present the results from our simulations to determine the effective viscosity, and its dependence on the tidal frequency in both laminar and weakly turbulent regimes. The main results are: a new scaling law for the frequency dependence of the effective viscosity which has not previously been observed in simulations or predicted by theory and occurs for shear frequencies smaller than those in the fast tides regime; the possibility of anti-dissipation (which could result in inverse-tides); and a strong agreement with the frequency dependence of Goldreich & Nicholson (1977) (despite disagreement with the fundamental mechanism).

These results have important implications for tidal dissipation in convection zones of stars and planets which we will discuss. The results of this work indicate that the classical tidal theory of the equilibrium tide in stars and giant planets should be revisited.

Tuesday 6 September		
12:00-13:00		Lunch
13:00-13:10	LT1	Welcome
13:10-13:55	LT1	Keynote Talk 1 by Dr Julien Hoessler (Chair: G. Papadakis)
14:00-15:30		Parallel sessions I
	LT1	–Session 1A : Boundary layers
	LT3	–Session 1B : Gas-liquid flows
	LT4	–Session 1C : Computational and data-driven methods I
15:30-16:30		Break - Poster Session 1
16:30-18:00		Parallel sessions 2
	LT1	–Session 2A : Aerodynamics I
	LT3	–Session 2B : Droplets
	LT4	–Session 2C : Nuclear thermal hydraulics, heat transfer
Wednesday 7 September		
9:00-9:45	LT1	Keynote Talk 2 by Prof Catherine Noakes (Chair: K. Kontis)
9:45-9:55	LT1	National Fluids Centre Presentation by Matthew Juniper
10:00-11:20		Parallel sessions 3
	LT1	–Session 3A : Jets, buoyancy-driven flows
	LT3	–Session 3B : Solid particles
	LT4	–Session 3C : Magnetohydrodynamics
11:20-11:35		Break
11:35-13:05		Parallel sessions 4
	LT1	–Session 4A : Turbulent flows
	LT3	–Session 4B : Surface engineering, porous media
	LT4	–Session 4C : Microfulidics
13:05-14:00		Lunch
14:00-14:30	LT1	UKFN Thesis Prize Talk by Dr Craig Duguid (Chair: M. Juniper)
14:35-16:05		Parallel sessions 5
	LT1	–Session 5A : Bio-flows, mixing phenomena
	LT3	–Session 5B : Wind and marine energy
	LT4	–Session 5C : Atmospheric, environmental flows
16:05-16:50		Break - Poster Session 2
16:50-18:10		Parallel sessions 6
	LT1	–Session 6A : Aerodynamics and propulsion
	LT3	–Session 6B : Solid particles II
	LT4	–Session 6C : Atmospheric flows II, viscoelastic flows
19:00		Dinner
Thursday 8 September		
9:00-10:30		Parallel sessions 7
	LT1	–Session 7A : Turbulence and transition
	LT3	–Session 7B : Wettability, engineered surfaces
	LT4	–Session 7C : Airborne dispersion and transmission
10:30-10:45		Break
10:45-11:45		Parallel sessions 8
	LT1	–Session 8A : Aerodynamics II
	LT3	–Session 8B : Capillarity, porous media
	LT4	–Session 8C : Computational and data-driven methods II
11:45-12:30	LT1	Keynote Talk 3 by Dr Nathalie Vriend (Chair: A. Willis)
12:30-13:00	LT1	Closing ceremony & prizes
13:00-14:00		Lunch

Please vote for your best student papers (student paper IDs are shown in blue in the below schedule) via this online form (`https://forms.gle/YHkeito21n2YFtsj9`) or the QR code on page 2.

	Tuesday 6 –Boundary layers (LT1) Session 1A–Chair: Melika Gul	Gas-liquid flows (LT3) Session 1B– Chair: Marco Colombo	Compt., data-driven methods I (LT4) Session 1C–Chair: Jens-Dominik Mueller
14:00	**13** Boundary layer flows over solid and viscous stretching sheets N. Hanevy	**101** Nonlinear oscillations of levitated air bubbles in water G. Hunter-Brown	**77** Drag reduction of a square cylinder using deep reinforcement learning P. Mudiyanselage
14:13	**25** Characterisation of permeable-rough surfaces D.D. Wangsawijaya	**91** Shifting gears in liquid-liquid impact R. Cimpeanu	**99** Machine-assisted subgrid-scale modelling for Large Eddy Simulation-Probability Density Function approaches T.H. Un
14:26	**73** Effects of pressure gradient history on rough-wall turbulent boundary layers T. Preskett	**117** A moving-mesh approach for interface-tracking multiphase flow L. Li	**103** Relative importance of physical quantities for data-driven RANS-based turbulence modelling: Creating a list of ingredients for the experimental fluid dynamicist O. Bidar
14:39	**140** A comparison of temporally and spatially accelerating turbulent flows M. Falcone	**86** Conjugate Heat Transfer Effect of Flow Boiling in Microchannels M. Magnini	**114** A DeepONet for predicting groundwater in both steady state and transient systems M.L. Taccari
14:52	**151** High-fidelity comparison of compressible/incompressible boundary layers G. Vivarelli	**21** Dynamics of long bubbles propagating through micro-pin fin arrays I. El Mellas	**128** An automatic differentiation approach to point vortex dynamics with applications to turbulence A. Cleary
15:05	**111** Turbulent mixing of heat and momentum in a turbulent boundary layer perturbed by an effusion film D. Burnett	**92** Computational fluid dynamics driven mass transfer modelling for the prediction of CO2 corrosion in oil/water mixture pipeline flows U. Thorat	**52** Efficient solution of fluid-structure interaction problems using an ALE finite element method G. Walton
15:18	**154** Symmetry-reduced low-dimensional representation of large-scale dynamics in a turbulent boundary layer M. Engel	**14** Bubble collapse near porous plates E. Andrews	**10** On the application of optimal control techniques to shadowing methods for sensitivity analysis of chaotic systems. R. Gilbert

	Tuesday 6 – Aerodynamics I (LT1) Session 2A– Chair: Konstantinos Kontis	**Droplets (LT3)** Session 2B– Chair: Radu Cimpeanu	**Nuc. thermal hydr., heat trans. (LT4)** Session 2C–Chair: Alex Skillen
16:30	**49** Data assimilation method using planar PIV of the flow around a NACA0012 airfoil C. Thompson	**56** Analysis of liquid break up and droplet distribution in a flat fan spray using image processing Z. Nissar	**53** A CFD Study of the conjugate heat transfer in a T-junction with an upstream elbow Y.H. Wong
16:43	**57** Aerodynamic and structural performance of serrated leading edge outlet guide vanes designed for low noise gas turbines C. Cuciumita	**50** Droplet adhesion revisited M. S. Sadullah	**161** Numerical simulation of thermal mixing of liquid sodium in a Y-junction W. Wang
16:56	**58** Topology-free optimisation of turbine blade tip using the adjoint method G. Ye	**64** Evolution of and deposition from an evaporating sessile annular droplet L.M. Mills, H.M. D'Ambrosio, S.K. Wilson, A.W. Wray	**125** LES of a hot horizontal helium jet issued into an air ambient J. He
17:09	**135** Analysis of small-scale statistics in a turbulent recirculating flow around a wing J.C. Bilbao-Ludena	**164** Fingerlike instability in drying droplets A. Othman	**153** Modelling of mixing of helium with air in nuclear reactor cavities B. Liu
17:22	**126** Morphing of a NACA6409 Aerofoil and the use of Periodic Morphing for Aerodynamic Efficiency Gains in Ground Effect D. Clements	**60** Dynamic Leidenfrost Effects: Computational modelling to predict transitions in drop impact P. Lewin-Jones	**95** An experimental study on the thermal management of electronics with the new design branch-type of bio-mimetic heat sink M.Y. Yazici
17:35	**67** Experiments on Transonic Shock-Vortex Interactions C. Hocking	**145** On the evolution of and the deposition from an evaporating droplet H.M. D'Ambrosio	**157** Heat transfer in next generation 3D-printable Heat Exchangers I. Qureshi
17:48	**69** Effects of surface roughness on the propulsive performance of a flapping foil R. Vilumbrales-Garcia	**82** Application of the Fully Lagrangian Approach to unsteady polydisperse gas-droplet flow simulations with inter-phase coupling C. Stafford	**137** Behaviour of the buoyant and forced convective jet in the upper plenum of a pool-type nuclear facility A. Saxena

	Wed. 7 – Jets, buoyancy-driven flows (LT1) Session 3A–Chair: Maarten van Reeuwijk	**Solid particles I (LT3)** Session 3B–Chair: Wei Wang	**Magnetohydrodynamics (LT4)** Session 3C–Chair: Ning Qin
10:00	**59** Fluidic oscillation generated, hybrid synthetic jet dynamics in a fluid filled cylindrical duct W.B. Zimmerman	**33** Effect of low concentration particles in the fluid behaviour of particulate pipe flows R. Raj	**29** The mean electromotive force arising from shear driven magnetic buoyancy and rotation C. Duguid
10:13	**89** The internal structure of forced fountains J. Huang	**87** Resolvent analysis of low-inertia particles in a pipe flow R.K. Schlander	**76** Alfven waves excitation at low magnetic Reynold number S. Lalloz
10:26	**131** Stability of body force influenced flows S. Jackson	**97** Transport of inertial particles in random flows A. Tully	**109** Lagrangian statistics, Floquet stability, and dynamo properties of time-dependent ABC flows C.S. Skene
10:39	**46** Receptivity and sensitivity of mixed baroclinic convection in a cavity A. Kumar	**66** Nonspherical Particle Interactions in Forced Isotropic Turbulence J. Anderson	**51** Nonlinear reduced-order models for magnetoconvection in 2D A. Roy
10:52	**32** Scaling laws in turbulent convection due to internal heating A. Arslan	**100** Hydrodynamics in a three-phase flotation system – fluid following with a new hydrogel tracer for PEPT D. Mesa, M. van Heerden	**72** Experimental characterisation of the flow inside a rotating tangent cylinder R. Agrawal
11:05	**138** Dynamics of multiphase plumes in a stratified rotating environment D. Frank	**80** Robotic inspection of pre-filled medical syringes H. Liaquet	**70** Direct Statistical Simulation: an alternative approach to turbulence K. Li

	Wednesday 7 – Turbulent flows (LT1) Session 4A—Chair: George Papadakis	**Surface eng., porous media (LT3)** Session 4B—Chair: Marco Colombo	**Microfluidics (LT4)** Session 4C—Chair: Radu Cimpeanu
11:35	**115** An investigation on transient flow behaviour in pulsating channel flows O. Maklad	**112** Effect of surface tension and shear viscosity on the slip-length of aqueous flows over superhydrophobic surfaces L. Zhang	**3** Focusing and sorting polystyrene particles and liposomes via diffusiophoresis in straight flat microchannels A. Chakra
11:48	**132** Linearised predictions of secondary currents in turbulent channels with topological heterogeneity G. Zampino	**5** Surfactant dynamics and fragmentation in a thin liquid film spreading on an immiscible liquid surface C. Wilk	**17** Diffusion-slip boundary conditions in modelling flows in micro- and nano-channels A.M. Tomy
12:01	**96** Eliminating turbulence using body force in pipe flow S. Chu	**85** Systematic modulation of bio-inspired micropatterning using temporally-arrested breath figure F. Dent	**102** Pair formation and stability for particles with softness heterogeneity in inertial microfluidics B. Owen
12:14	**24** Instabilities of unsteady triple-deck problems H. Broadley	**149** Understanding melt flow's signature on geometrical patterns in rock-melt mixtures G. Fedrizzi	**139** Numerical analysis of the formation and stability of pairs of particles with different size in inertial microfluidics K. Thota
12:27	**54** On the effect of filament inclination on the different regimes of canopy flows S. Nicholas	**133** Impact of large periodic deformations on solute transport in soft porous media M. Fiori	**105** Application of MPPICInterFoam solver in modeling multiphase droplet microfluidics M. Fatehifar
12:40	**127** Passive attenuation of unsteady transients and vortex streets in a pipe flow J. Hirst	**116** Formation and collapse of gas cavities in a soft porous medium O. Paulin	**167** Investigation of flow anisotropy and its effect on deterministic lateral displacement devices for particle separation C. Mallorie
12:53	**8** Separation angles and wake features of incident flow around polygonal cylinders E. Masoudi	**148** Two-phase flow: Travelling waves in a reactive porous medium D. Bullamore	**168** Simulation of inertial microfluidics using various cross-sectional shapes F. Mirghaderi

	Wed. 7 – Bio-flows, mixing phen. (LT1) Session 5A–Chair: Shuisheng He	Wind and marine energy (LT3) Session 5B–Chair: Melika Gul	Atmospheric, envir. flows (LT3) Session 5C–Chair: Catherine Noakes
14:35	**19** CFD modelling of an aortic narrowing: towards a multi-fidelity predictive capability J. Wang	**9** Wind turbine wakes under the influence of shear, thermal stratification and veer A. Hodgkin	**36** Assessing transient airborne infection risks in a hospital: a mathematical model Alexander J Edwards
14:48	**45** Numerical investigation of particle-vortex interaction in a cross-slot geometry under moderate inertia K. Kechagidis	**11** Impact of analytical wake models on wind farm wake steering strategies F. Gori	**30** Large-Eddy Simulation of Microscale Urban Surface Energy Balances over a Diurnal Cycle S. Owens
15:01	**55** In-silico flow diverter performance assessment in posterior communicating artery aneurysms M. MacRaild	**12** High fidelity simulations of atmospheric boundary layer over complex terrain with application in wind energy C. Jane	**28** Experimentally Simulating Urban Air Pollution around a Single Tall Building T. Rich
15:14	**104** Effect of cardiac motion on right coronary artery haemodynamics P. Kandangwa	**84** Performance investigation of the vertical Axis wind turbine with slotted aerofoil configuration S. Kose	**110** High fidelity simulation of atmospheric dispersion K.K. Maskey
15:27	**121** Fluid flow in poroelastic tissue engineering bioreactor scaffolds G. Booth	**141** Genetic algorithm based active aerofoil deformation in the application of vertical axis wind turbines J.-P. Marx	**165** Understanding regimes and validity of solutions to the hydrodynamic escape of planetary atmospheres R. Chatterjee
15:40	**2** A novel method for understanding the mixing mechanisms to enable sustainable manufacturing of bioinspired silica Y.D. Baba	**120** An experimental investigation into passive pitch systems for tidal turbines S. Gambuzza	**44** Inertia-gravity wave diffusion by geostrophic turbulence: the impact of flow time dependence M. Cox
15:53	**16** Polynomial mixing rates in non-monotonic alternating shear flows J.M. Hill	**47** Experimental investigation of an integrated oscillating water column (OWC) within fixed and heaving breakwaters Y. Almalki	**71** On the spatial evolution of the largest waves propagating over variable coastal bathymetry V. Bellos

	Wed. 7 – Aero. & propulsion (LT1) Session 6A–Chair: Julien Hoessler	Solid particles II (LT3) Session 6B–Chair: Maarten van Reeuwijk	Atmospheric flows II, visc. flows (LT4) Session 6C–Chair: Ashley Willis
16:50	**78** Bio-inspired flexible pillar flow sensor arrays to visualise flow characteristics over an aerofoil downstream of a spanwise variable gust generator A. Court	**155** Periodic orbits in the sedimentation dynamics of thin, rigid disks C. Vaquero-Stainer	**38** Feature based analysis of the north atlantic eddy driven jet J. Perez
17:03	**15** Estimating turbulent aerofoil stall with limited probes via LSE D.W. Carter	**152** Capturing the mechanisms for gravitational settling of inertial particles in turbulence using the kinetic PDF approach C. Stafford	**144** Dynamical impacts of warm-starting operational weather models over Africa F.A. Morris
17:16	**40** RANS analysis of merging supersonic streamwise vortices for enhanced mixing in scramjet engines D. McCaughey	**7** A mechanism for diffusiophoretic motion: silica concentration gradients C.R. Rees-Zimmerman	**94** Poroviscoelastic dynamics of mushy magmatic systems J. Castelino
17:29	**75** The development of a photoelastic force balance B. McLaughlin	**146** Dynamics of particle aggregation in evaporating and de-wetting films of complex liquids J. Zhang	**113** Micro-PIV of viscoelastic fluid flow in micro-porous media V.C. Ibezim
17:42	**41** Assessment of sponge layers for aeroacoustic modeling implemented in Ansys Fluent G. Huang	**130** Ice particle skimming in aircraft icing E.M. Jolley	**61** Time-dependent 3D dynamics in viscoelastic pressure-driven channel flow M. Lellep
17:55	**169** Lagrangian curvature statistics from Gaussian sub-ensembles in turbulent von-Kármán flow Y. Hengster	- -	**166** Flame-patterns and diwhirls route to elastoinertial turbulence in Taylor Couette flows T. Boulafentis

	Thursday 8 – Turb. and transition (LT1) Session 7A–Chair: Elena Marensi	Wettability, eng. surf. (LT3) Session 7B–Chair: Peter Jimack	Airborne dispersion and trans. (LT4) Session 7C–Chair: Daria Frank
9:00	**34** A framework to quantify the robustness of flows subject to finite-amplitude instabilities C. Beaume	**20** Wetting of auxetic metamaterials G. McHale	**4** Experimental investigation of scalar dispersion of pollutant in a room H.D. Lim
9:13	**108** Sensitivity of rough-wall turbulence to changes in spanwise effective slope T.O. Jelly	**22** Analytical models for the wettability of auxetic metamaterials S. Mandhani	**18** Assessing exposure to infected breath in naturally ventilated classrooms C. Vouriot
9:26	**35** Long-wavelength three-dimensional instability in the vortex formation region behind a body A.I. Aleksyuk	**6** Modelling double emulsion formation in planar flow-focussing microchannels H. Kusumaatmaja	**68** Targeted extraction of exhaled breath to reduce potential virus exposure J. Finneran
9:39	**63** Analysis of the new unstable mode in supercritical fluids B. Bugeat	**23** Imaging dynamic wetting ridges on liquid-infused surfaces A. Naga	**81** Breathing simulator to assess effectiveness of masks and control the spread of Covid-19 In quiescent surroundings D. Pickles
9:52	**48** Subcrtical transition to turbulence in quasi-2D shear flows A. Potherat	**124** Finding the point of no return: A dynamical systems approach to the stability and bifurcation of a moving contact line J. Keeler	**163** Coupled CFD and modified Well-Riley model for the prediction of the spatiotemporal variations of airborne infection probability in a mechanically ventilated room. C.F. Castro-Faccetti
10:05	**106** Wakes of perforated normal flat plates M. Cicolin	**123** The Liquid-Solid Amontons' Laws: Friction coefficients for droplets on solids H. Barrio-Zhang	**129** Intelligent Large-Eddy Simulation model to optimise HVAC system energy consumption in a building B. Chen
10:18	**93** Near-wall flow characteristics of flapping foils at intermediate Reynolds numbers A. Zurman-Nasution	**90** Natural wax crystallisation for biomimetic fabrication of hydrophobic coatings S. Khodaparast	- -

	Thursday 8 – Aerodynamics II (LT1) Session 8A–Chair: Ning Qin	**Capillarity, porous media (LT3)** Session 8B–Chair: Wei Wang	**Comp. & data-driven methods II (LT4)** Session 8C–Chair: Elena Marensi
10:45	**79** High-fidelity-simulation assisted optimisation of U-Bend channel R. Quadros	**65** The long cross-over dynamics of capillary imbibition É. R.-Gutiérrez	**27** Benchmarking uncertainty quantification methods in flood modelling applications M. Hajihassanpour
10:58	**74** Effect of winglet serration wavelength on wingtip vortices K. Sabnis	**88** Rough capillary rise J.R. Panter	**119** Adaptive physics-informed neural network for surrogate modeling and design optimization Y. Sun
11:11	**83** Hydrodynamics of flow over hydrofoils with generic roughness elements M. Kurt	**107** Pressure drop and heat transfer in combined porous media A.F. Abusewal	**162** Predicting unseen scenarios with data-driven reduced-order models using sub-sampling and domain decomposition C.E. Heaney
11:24	**122** Direct Numerical Simulation of post-stall NACA0012 at moderate Reynolds Number M. Lahooti	**147** A continuum model for the bulldozing of an immersed granular material in a confined geometry L. Morrow	**160** Performance benchmarking metrics for comparison of machine learning and traditional methods for problems in fluid-structure interaction A.B. Harish

Tuesday 6 Poster session I (15:30-16:30)	
P4	Managing the increasing heat load in advanced military aircrafts with latent heat thermal management systems J. Frank, D. Borman, A. Khan, J. Summers, E. Greiciunas, S. Samarasekera (Uni. of Leeds)
P15	Liquid-Liquid Interactions from Measurement of Apparent Contact Angles on Slippery Liquid Infused Porous Surfaces N. Afify, S. Armstrong, G.G. Wells, R. Ledesma-Aguilar, G. McHale (Uni. of Edinburgh)
P2	Flow Characteristics of a Hyperloop Vehicle A. Lang (Uni. of Leeds)
P34	Investigation and evaluation of scalable liquid-metal MHD solvers for fusion breeder blanket applications. R.W. Eardley, A.J. Dubas, A. Davis (UKAEA)
P24	The affect of hyperdiffusion in rotating convection B. Davy (Uni. of Leeds)
P33	COVID-19 transmission risks associated with environmental contamination in workplace and public toilets C. Higham, L. Fletcher, C. Noakes, M. López-García (Uni. of Leeds)
P21	An investigation of the fluid structure interaction in articular cartilage across disparate scales E. Butler, G. de Boer, D. Head, M. Walkley, M. Bryant (University of Leeds), A. Hart, H. Hothi (Uni. College London)
P29	Electromagnetic effects on the stability on scrape-off layer plasmas G. McGilvray (Uni. of Leeds)
P16	Influence of multiphase microstructure and rheology on the dynamics of bubble growth and release from viscoelastic soft sediments I. Latimer (Uni.of Leeds)
P19	Fluid Mechanics of Polymer Melt Filtration J. Bennett (Uni. of Leeds)
P31	Internal and External Dynamics of Coalescing Non-Isothermal Droplets Y. Darbar, M. Wilson, H. Thompson, D. Harbottle (University of Leeds), T. Sykes (Uni. of Oxford)
P11	Biomechanics of Articular Cartilage: A Quantitative Investigation on the Influence of Density Gradients to the Poroelastic Response of Cartilage C. Thomas, A. Ghanbarzadeh, M. Bryant, G. de Boer (Uni. of Leeds)
P7	Convection in Salt Lakes M.R. Threadgold, C. Beaume, L. Goehring, S.M. Tobias (Uni. of Leeds, Nottingham Trent Uni.)
P6	Patterned Low Friction Lubricated Surfaces M. Pelizzari, G. McHale, S. Armstrong, G. Wells, R. Ledesma-Aguilar (Uni. of Edinburgh)
P37	New development of a coarse-grid CFD for mixed convection in rod bundles C. Zhang, B. Liu, S. He (Uni. of Sheffield)
P23	Predicting vorticity and velocity variations in flow through the pore-scale porous media based on recurrent neural network-based physics-informed machine learning M. Mesgarpour (KMUTT)), O. Mahian, S. Wongwises (Queen Mary), M.S. Shadloo (Normandie Uni.), N. Karimi (NSTDA)
P18	Effects of thermal slip and chemical reaction on free convective nanofluid from a horizontal plate embedded in a porous media A. Alsenafi, M. Ferdows (Kuwait Uni.)
P14	Phase changes of biomimetic respiratory droplets on ultra-smooth surfaces A. Jenkins, G. Wells, R. Ledesma-Aguilar, D.O. Mantecon (Uni. of Edinburgh)

Wednesday 7 Poster session II (16:05-16:50)	
P35	Perspectives of Women in Fluid Dynamics M. Jabbal (University of Nottingham)
P26	Unsteady Wing Loading from Tufts F. De Voogt, B. Ganapathisubramani (University of Southampton)
P5	Efficient acoustofluidics and active surface cleaning using ZnO/glass thin film acoustic waves H. Ong, H. Pang, P. Agrawal, H. Torun, K. Thummavichai, Q. Wu, Y. Fu (Northumbria University, Xi'an University of Science and Technology)
P3	Oscillatory Motions of Falling Snowflakes J. Stout, C. Westbrook, T. Stein, M. McCorquodale (University of Reading)
P36	An Immersed Boundary Method for the DNS solver CHAPSim K. Chinembiri, S. He, W. Wang (University of Sheffield)
P28	Aerodynamic Performance of Flatback Aerofoils for Vertical Axis Wind Turbine K.M. Alzahrani, J.L. Hee, D. Ingham, L. Ma, M. Pourkashanian (University of Sheffield)
P1	Study Of Different Turbulence Models In Predicting 3D Flow Inside A Single Stage Micro Gas Turbine Q. Yu, R. Howell (University of Sheffield)
P30	A Resolvent-Based Variational Method for Statistical Prediction of Channel Flow T. Burton, S. Symon, D. Lasagna (University of Southampton)
P27	New understanding of heat transfer deterioration for supercritical fluid flows Z. Zou, S. He (University of Sheffield)
P10	Towards Full Transient Adjoint Based Aerodynamic Optimisation of Vertical Axis Wind Turbines Using CFD Q. Ren, J.L. Hee, D.B. Ingham, L. Ma, M. Pourkashanian (University of Sheffield)
P8	Optimising the performance of the computational fluid dynamics (CFD) applications written in domain-specific languages (DSLs) B. Yang (University of Leeds)
P17	Stagnation-point heating in high-enthalpy boundary layers T. Ala, N. Sandham (University of Southampton)
P32	Fusion Technology: Experiments with Digital Twinning D. Buta (Uni. of Leeds)

A novel method for understanding the mixing mechanisms to enable sustainable manufacturing of bioinspired silica

Yahaya D. Baba, Mauro Chiacchia and Siddharth V. Patwardhan

Green Nanomaterials Research Group, Department of Chemical and Biological Engineering, University of Sheffield

Bioinspired silica (BIS) has received an unmatched attention in recent times owing to their green synthesis, which offers a scalable, sustainable and economical method to produce high value silica for wide ranging applications including catalysis, environmental remediation, biomedical and energy storage. In order to scale-up BIS synthesis, it is critically important to understand how mixing affects the reaction at different scales. In particular, successful scale-up can be achieved if mixing time is measured, modelled and kept constant across different production scales. To this end, a new image analysis technique was developed by using pH, as one of the key parameters, to monitor the reaction and the mixing. Specifically, the technique involved image analysis of colour (pH) change by using a custom-written algorithm to produce a detailed pH map. The degree of mixing and mixing time were determined from this analysis for different impeller speeds and feed injection locations. Cross validation of the mean pH of selected frames with measurements using a pH calibration demonstrated the reliability of the image processing technique. The results suggest that the bio-inspired silica formation is controlled by meso- and, to a lesser extent, micro-mixing. Based on the new data from this investigation, a mixing time correlation is developed as function of the Reynolds number. Further, we were able to correlate the effects of mixing conditions on the reaction and the product. These results provide valuable insights on the scale-up to sustainable manufacturing for BIS and other nanomaterials.

Focusing and sorting polystyrene particles and liposomes via diffusiophoresis in straight flat microchannels

Adnan Chakra[1], Naval Singh[1], Goran Vladisavljevic[1], Francois Nadal[2], Cecile Cottin-Bizonne[3], Christophe Pirat[3] and Guido Bolognesi[1]

[1]Department of Chemical Engineering, Loughborough University, UK
[2]Wolfson School of Mechanical, Electrical and Manufacturing Engineering, Loughborough University
[3]Institut Lumiere Matiere, UMR5306 Universite Claude Bernard Lyon 1, France

We report a novel phenomenon that enables the pre-concentration and sorting of charged polystyrene particles and liposomes dispersed in a continuous flow within a straight flat microchannel. A 3-inlet junction microfluidic device is used to generate a steady-state salt concentration gradient in the direction perpendicular to the flow. As a result, particles dispersed in the electrolyte solution accumulate into two symmetric regions of the channel by forming two distinct focusing spots. Despite a similar colloid behaviour being reported in previous studies with a similar flow configuration, our numerical and experimental analysis shows that the observed particle dynamics is driven by a novel unreported physical mechanism, that combines diffusiophoresis, diffusioosmosis and hydrodynamic effects.
In addition, we developed a proof-of-concept microfluidic device that exploits this new mechanism to enable the focusing and filtration of colloidal particles based on their zeta potential under continuous and steady-state flow settings. The proposed method offers great potential for microfluidic bio-analytical testing applications, including bioparticle pre-concentration and filtration.

Experimental investigation of scalar dispersion of pollutant in a room

H.D. Lim and Christina Vanderwel

University of Southampton

The scalar dispersion of pollutant continuously released in an indoor space is experimentally investigated by releasing Rhodamine 6G fluorescent dye in a scaled room model. The flow was driven by immersing the scaled room model in the Recirculating Water Tunnel (RWT) to achieve Reynolds number based on inlet of Re ~ 5000. Planar Laser-Induced Fluorescence (PLIF) dye visualization show three distinct flow regions – (1) the near-source region where scalar structures are controlled by large-scales, (2) the wall jet which introduces fresh inlet flow and (3) the well-mixed region where large-scale scalar structures are no longer visible. Time-averaged Particle-Image Velocimetry (PIV) revealed the mean flow pattern with a primary circulating cell near the centre of the room, boundary layer flow along the walls which turns and separates at the corners of the room and high Reynolds shear stresses along the shear layers of the wall jet which indicates efficient entrainment and mixing of the dye with fresh inlet fluid. Instantaneous velocity fields revealed significant penetration of the wall jet's large-scale flow structures into the centre of the room, and advection of these flow structures around the room to the dye source with integral length scales close to the height of the room. This research is one of the first steps in an ongoing project funded under a RAEng UK IC Postdoctoral Research Fellowship to investigate turbulent diffusivity in indoor spaces. The end objective is to investigate the momentum effects of human activities and their influence on turbulent diffusivity of pollutants in indoor spaces.

Surfactant dynamics and fragmentation in a thin liquid film spreading on an immiscible liquid surface

Christophe Wilk, Andrew Hazel and Julien Landel

University of Manchester

Macroscopic thin liquid films are important in many processes ranging from coating flow technology to biological systems, such as the fluid lining in the pulmonary airways or the formation of a tear film. Surfactants can reduce the surface tension at an interface and lead to interfacial Marangoni stresses if surfactant gradients exist. In this study, we investigate the coupling between surfactant concentration and film dynamics by considering the spreading of surfactant-laden immiscible droplets over liquid substrates where endogenous traces of surfactants are present. As the film spreads over the liquid substrate, due to Marangoni forces, a rim is formed at the liquid-liquid-gas contact line of the film which, ultimately, becomes unstable due to a Rayleigh-Plateau instability and breaks up into a myriad of droplets. We hypothesize that this two-dimensional fragmentation instability is due to the intricate interplay between the Marangoni forces and the wetting properties of the fluid which enables the system to form unstable rims at the contact line whilst spreading. We have studied the problem experimentally, theoretically and numerically. Using lubrication theory, a non-linear diffusion equation is derived to describe the spreading of the surfactant in the system. Two continuity conditions of surfactant concentration and flux between the film phase and the liquid substrate are critical to the surfactant dynamical behaviour in the system. By imposing these conditions at the contact line, we observe the formation of a surfactant concentration jump which could explain the rim formation and destabilization observed experimentally. By varying the rate at which the surfactants leave the film, one can significantly change the surfactant concentration profile and, therefore, the spreading velocity of the film. Finally, we discuss our model predictions in comparison with our experimental findings.

Modelling double emulsion formation in planar flow-focussing microchannels

N. Wang, C. Semprebon, C. Zhang, H. Liu and H. Kusumaatmaja

Durham University

Double emulsion formation in a hierarchical flow-focusing channel is systematically investigated using a free-energy ternary lattice Boltzmann simulation method. A formation regime diagram is constructed based on the capillary numbers of the inner, middle and outer phase fluids. The results show that the formation diagram can be classified into periodic two-step region, periodic one-step region, and non-periodic region. Furthermore, in good correspondence with previous experimental observations, different emulsion morphologies are obtained, including the regular double emulsions, decussate regimes with one or two alternate empty droplets, bidisperse behaviours where the resulting emulsions have two different sizes, and structures with multiple inner droplets contained in the continuous middle phase thread. In the periodic one-step formation region, we also propose scaling laws for the double emulsion size and for the size ratio between the inner droplet and the overall double emulsion. Finally, we demonstrate that the interfacial tension ratio can greatly change the morphologies of the obtained emulsion droplets, and the channel geometry plays an important role in changing the formation regimes and the double emulsion sizes. In particular, narrowing the side inlets or the distance between the two side inlets promotes the conversion from the two-step formation regime to the one-step formation regime.

A mechanism for diffusiophoretic motion: silica concentration gradients

Clare R. Rees-Zimmerman and Alexander F. Routh

Department of Chemical Engineering and Biotechnology, University of Cambridge

Analogous to diffusion, diffusiophoresis is the motion of particles along the concentration gradient of a different solute type. One mechanism that can cause diffusiophoresis is a gradient in electrolyte concentration. This work seeks experimental evidence and characterisation of diffusiophoresis of colloids in multivalent electrolyte. Whilst this has been explored for small anions and cations of similar sizes, this work studies much larger anions of colloidal size, and thus uncovers a novel diffusiophoretic motion. The particular system studied is the diffusiophoresis of latex in a background dispersion of sodium-stabilised silica.

Using a Hele-Shaw cell – parallel plates separated by a narrow gap used for studying laminar flow – a thin layer of a latex dispersion in a sharp concentration gradient of silica nanoparticles is established by filling with three layers: water above a thin layer of latex dispersion above a silica dispersion. If the large latex particles were unaffected by the background concentration gradient, then they would be expected to diffuse symmetrically. Any asymmetry can be measured, and is evidence of diffusiophoresis. The quantification involves turbidity readings from backlit images of the flow cell.

Control and test experiments have been run and analysed with silica and latex particles in different salt concentrations. The experiments show that the large latex particles move down a concentration gradient of small silica particles, which can be explained by diffusiophoresis. Reproducible motion of the latex concentration front with time, approximately following a $t^{1/2}$ trajectory, is observed. The rate of this motion is inversely proportional to salt concentration, confirming an electrolyte-driven motion. The evolution of the latex front position is consistent with theory for silica and its stabilising counterion causing the diffusiophoretic motion: a novel mechanism.

Separation angles and wake features of incident flow around polygonal cylinders

Esmaeel Masoudi, Lian Gan and David Sims-Williams

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

In this study, large eddy simulation (LES) is used to investigate shear layer separation of incident flow around polygonal cylinders of side number $N = 5 - 8$ at Reynolds number $Re = 10^4$. In total, six equally distributed incidence angles (α) are studied on each polygon between the face and the corner orientations, thus covering the entire α spectrum. The position of separation points and the instantaneous behaviour of the separated shear layers are studied in detail. It is found that the separated shear layers are highly dynamic manifesting a flapping motion of various condition depending amplitude, which is a key factor that causes flow reattachment and results in a second separation, in addition to the first separation in some incidence angles. The strength of this flapping motion is found to be associated with shear layer penetration distance and shear layer thickness growth rate. The separation angles, which are fixed on the corners, can be calculated by the proposed empirical/analytical equations which are found to be in good agreement with available experimental results. Furthermore, a wake deflection angle is introduced and shown to be a good scaling factor for lift, drag and Strouhal number. Finally, the critical separation angle, which corresponds to the longest shear layer penetration distance, the minimum drag and maximum lift conditions is determined and estimated with analytical and empirical equations.

Wind turbine wakes under the influence of shear, thermal stratification and veer

Amy Hodgkin, Sylvain Laizet and Georgios Deskos

Imperial College London (AH,SL), NREL (GD)

Large modern wind turbines exist high up within the atmospheric boundary layer (ABL) where their wake is impacted by the properties that make it up. The mean shear profile, thermal stratification and veer are major components of the ABL so an understanding of their individual impact on the wake development is important. On top of this, in wind farms, turbines are often situated in the wake of upstream turbines which leads to significant power losses and fatigue loads. Because of this, control methods which aim to shorten the stable wake length of a turbine are growing in popularity. Of particular interest is dynamic pitch control, where the pitch of the blades is varied sinusoidally which disturbs the tip-vortex development leading to early breakdown. We conduct a suite of high-fidelity large-eddy simulations to study the impact of shear, thermal stratification and veer on the stable wake length of a turbine with dynamic pitch control. Instantaneous flow fields from the simulations are used to study the impact that the inflow conditions have on the wake development. We extract coherent structures using decomposition methods and compute the mean kinetic energy fluxes of these modes to better understand the tip-vortex instability mechanisms and structures leading to wake recovery. Our findings indicate that shear can significantly affect the breakup of the wind turbine tip-vortices as well as the shape and stable length of the wake, whereas thermal stratification and veer seem to only have limited contribution to the spatial development of the near-wake field.

On the application of optimal control techniques to shadowing methods for sensitivity analysis of chaotic systems.

Rhys Gilbert and 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 RANS 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 such as the Least Squares Shadowing (LSS), the Multiple Shooting Shadowing (MSS) and the Non-Intrusive Least Squares Shadowing (NILSS) have shown promise. 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 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 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 pre-conditioners 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.

Impact of analytical wake models on wind farm wake steering strategies

Filippo Gori, Sylvain Laizet and Andrew Wynn

Imperial College London

Wind energy is gaining international momentum around the globe driven by the urgent environmental need to reduce carbon emissions and the significant decrease in the cost of renewables. On average, turbine interaction power losses range from 10% to 30%. Wind farm control plays a major role in reducing the drawbacks of wake effects, with wake steering proven to be a promising strategy for power loss reduction and fatigue mitigation. The purpose of this work is to assess the sensitivity of optimal wake steering strategies to both the choice of analytical wake models and optimization parameters. While current model comparisons focus on the discrepancies in absolute power prediction, the conducted investigation aims to determine, in an optimization setting, the discrepancies in optimal decision variables (turbine yaw angles for wake steering). In addition, the behavior of wake models and their impact on the choice of optimization algorithms are assessed. Using the open-source FLORIS framework, different wake models are employed to optimize several farm layouts in various atmospheric conditions. Results show discrepancies in optimization trends, leading to different or even opposite optimal yaw angle settings. The same conclusions hold when varying turbines downstream spacing and ambient turbulence intensity. Furthermore, the sensitivity to initialization of a gradient-based optimizer is considered. Farm power function for all models is found to be multimodal, questioning the suitability of gradient-based algorithms for wake steering control. Solutions corresponding to local extrema lead to potential power losses of up to 15% compared to the obtained global maximum for power production.

High fidelity simulations of atmospheric boundary layer over complex terrain with application in wind energy

Christian Jane, Nikolaos Bempedelis, Rafael Palacios and Sylvain Laizet

Imperial College London

Wind energy is seeing massive growth in recent decades, and it is now acknowledged that it will be critical for the UK 2050 net-zero target. Many onshore wind farms will likely be located on hilly terrain or near urban centres, influencing the atmospheric boundary layer properties and wind turbine wakes. Accurately predicting the changes in wind speed, shear, direction, and turbulence intensity caused by complex terrain throughout wind farms is essential to assess their power output and for optimal wind turbine control. For this purpose, the wind farm simulator Winc3d, part of the high-order finite-difference framework Xcompact3d, is used to perform Large-eddy simulation (LES) of the flow over complex terrain for high Reynolds numbers. The computational framework offers a highly efficient parallelisation strategy with "spectral-like" accurate numerical schemes on a Cartesian mesh. The terrain features are reproduced with an immersed boundary method, which can be combined with a wall model to avoid the prohibitively expensive resolution of the viscous sub-layer. In this talk, we will present comparisons with published wind tunnel experiments and numerical simulations for the flow over two-dimensional and three-dimensional hills. Simulations will also be presented with a single wind farm located before, on top and after the hills.

Boundary layer flows over solid and viscous stretching sheets

Niall Hanevy

Coventry University

Boundary layer flows induced by the extrusion of a surface have received considerable attention since they were first described by Sakiadis [1]. These so-called 'stretching flows' are of practical importance to chemical and metallurgy industries where extrusion processes are commonplace. The vast majority of previous studies in this area consider the stretching sheet to be solid and free from deformation. Until very recently these types of flows were thought to be linearly stable. However, recent work [2] has shown that these flows are, in fact, linearly unstable to disturbances in the form of Tollmien-Schlichting waves. Our current analysis considers the effect of a large temperature gradient between the surface of the stretching sheet and the stationary flow outside of the boundary layer region. We will report on the linear stability characteristics of these flows and will draw comparisons with previous investigations. In addition to this, we will also consider the dynamics of a viscous sheet that thins as it is stretched. Our work is motivated by Al-Housseiny & Stone [3] who were the first to consider the isothermal problem. The coupling of the dynamics of the sheet and the induced boundary layer flow imposes restrictions on the validity of self-similar base flow solutions. We expect to be able to report on a range of physically relevant base flow solutions that take in to account both the viscous nature of the sheet and the large temperature discrepancy between the sheet and the outer boundary layer flow.

[1]Sakiadis, AIChE J 7, 221 (1961).
[2]Griffiths et al., Phys. Fluids 33, 084106 (2021).
[3]Al-Housseiny and Stone, J. Fluid Mech. 706, 597 (2012).

Bubble collapse near porous plates

Elijah Andrews[1], David Fernández Rivas[2] and Ivo Peters[1]

[1]University of Southampton
[2]University of Twente

A bubble collapsing near a solid boundary displaces towards the boundary and produces a high-speed jet. The high surface pressures induced by this collapse can damage, or clean, the boundary. Prior research has focused on simple boundary geometries, such as a solid surface. However, real-world geometries are far more complex, and these complexities affect the bubble dynamics. One such application is porous boundaries, such as those encountered when ultrasonically cleaning filters. In this work, we experimentally show how increasing porosity causes a decrease in bubble displacement during collapse and present a numerical model capable of predicting this behaviour.

Estimating turbulent aerofoil stall with limited probes via LSE

Douglas W. Carter and Bharathram Ganapathisubramani

University of Southampton

The application of linear stochastic estimation (LSE) in combination with the proper orthogonal decomposition (POD) to estimate the state of a stalled aerofoil is demonstrated using time-resolved planar particle image velocimetry (PIV). The experiment focuses on a symmetric NACA 0012 and cambered NACA 65-410 aerofoil at various incidence corresponding to transient and deep stall at a chord-based Reynolds number $Rec = 7.1 \times 10^4$. Three pseudo-pressure probes at the leading edge, mid-chord, and trailing edge are used to generate LSE coefficients to predict the first three POD mode coefficients. The similarity in the low-order dynamics of both aerofoils in either transient or deep stall is demonstrated through the application of LSE coefficients for one aerofoil to predict the state of the other and vice-versa. Consequences for global similarity in low-order dynamics of stalled aerofoils will be discussed.

Polynomial mixing rates in non-monotonic alternating shear flows

Joe Myers Hill, Rob Sturman and Mark Wilson

University of Leeds

Polynomial mixing rates are typical in laminar mixing systems with slow moving fluid near boundaries. Here we present other mechanisms for a slower than exponential mixing rate, illustrated by a variety of non-monotonic alternating shear flows on the torus. In particular we show that a fundamental piecewise linear model exhibits 'ghost boundaries' around which mixing is poor, which are not a result of physical boundary conditions but arise naturally in the dynamics. For this system we establish asymptotic (strong, measure-theoretic) mixing properties and derive rigorous bounds on mixing rates using results from the dynamical systems (billiards) literature. In the non piecewise linear setting we provide numerical evidence on mixing quality and rates, explain the challenges these systems pose to analysis using current dynamical systems techniques, and suggest mixing protocols which maximise chaotic mixing.

Diffusion-slip boundary conditions in modelling flows in micro- and nano-channels

Alwin M. Tomy and S. Kokou Dadzie

Heriot-Watt University

Classical continuum models and existing velocity-slip formulations are insufficient in accurately modelling flows in micro- and nano-channels. Extending the applicability of continuum models to such length-scales necessitates suitable modifications and ad hoc addition of flow-physics such as wall-slip, surface diffusion, Knudsen diffusion etc. In this work, we suggest diffusion-based velocity-boundary conditions that could justify some unconventional flow phenomena observed in micro- and nano- scales. By using these boundary conditions in conjunction with the classical continuum flow equations we investigate Poiseuille flow in microchannels. We present a unified derivation of expressions for mass flow rates and flow profiles in liquid and gas flows. The resultant analytical expressions provide new insights into these flow profiles and fit experimental data even in the free-molecular regime. The methodology considered in the derivation is consistent with recasting the Navier-Stokes equations. The diffusion type of boundary conditions presented in this work may be more appropriate in simulating flows in micro- and nano-systems and may also be adapted as boundary conditions in other interfacial flow modelling.

Assessing exposure to infected breath in naturally ventilated classrooms

Carolanne Vouriot, Henry Burridge and Maarten van Reeuwijk

Imperial College London

In some settings which are occupied for long periods by the same group of people like school classrooms, CO2 measurements have shown to be a very useful method to infer the risk of far-field airborne infection. However, often only a single point measurement is available which cannot capture spatial variations, which are likely to be encountered in naturally ventilated spaces where the room might not be well mixed. As a cost-effective tool, CO2 sensors are in increasingly widespread use and so it is crucial to understand how exposure might differ depending on the source location and/or sensor position. A typical naturally ventilated UK classroom in wintertime is considered. Ventilation is supplied through high- and low-level openings and driven solely by buoyancy to provide a comfortable environment to pupils and staff. We analyse the ratio between actual exposure arising from a single individual and proxy exposure calculated from point measurements of CO2. In doing so, we show that, at the breathing plane, the proxy CO2 exposure is within a factor 2 of the actual far field exposure for measurements taken within approximately 90 % of the room's cross-sectional area. These factors (of up to 2), whilst significant, are small compared with the typical uncertainties associated with the parameterisation of many airborne diseases.

CFD modelling of an aortic narrowing: towards a multi-fidelity predictive capability

Jie Wang

University of Manchester

Due to the challenges in its accurate prediction, the majority of studies to date have not considered the instantaneous fluctuating component of the WSS, but this was recently demonstrated to be significant. arge-eddy simulation (LES) provides the potential for high fidelity scale-resolving simulations capable of capturing instantaneous WSS fluctuations. LES has been successfully used in this domain, and can provide highly accurate patient specific flow predictions but high computational cost. Multi-fidelity modelling provides a framework to use such methods sparingly during parameter pace exploration studies or uncertainty quantification, while still preserving accuracy. In this approach, sparse LES data (at few points in parameter space) is used in conjunction with many cheaper, lower fidelity averaged methods (e.g. obtained through Reynolds averaging). The result is tool which can explore parametric space at a fraction of the cost of LES, but with close to the same accuracy. These techniques has been demonstrated for mixed convection flow, but have not (to the best of our knowledge) been applied to a CoA. Herein, we describe the data-sets developed for a multi-fidelity analysis of a CoA. In the full paper, we will show the multi-fidelity surrogate model's predictive capability.

Wetting of auxetic metamaterials

Glen Mchale[1], Steven Armstrong[1], Shruti Mandhani[2], Gary G. Wells[1], Rodrigo Ledesma-Aguilar[1], Ciro Semprebon[3], Emma Carter[2] and Andrew Alderson[2]

[1]Wetting, Interfacial Science & Engineering Lab, University of Edinburgh, Edinburgh, UK
[2]Materials & Engineering Research Institute, Sheffield Hallam University, Sheffield, UK
[3]Smart Materials & Surfaces Lab, Northumbria University, UK

It is a common conception that when a material is stretched it becomes thinner. However, when an auxetic metamaterial is stretched it becomes wider. This is because the properties of an auxetic material are determined by its lattice arrangement and not by the materials properties of the individual solid elements. Because the expansion of the solid surface area is due to increased space between the solid elements, this allows new types of superhydrophobic materials, and hemi-wicking and liquid-infused surfaces. To illustrate these ideas, we consider a hydrophobic lattice constructed from hexagonal elements with joints that rotate under strain to create a lattice constructed from bow-tie elements. In the former case, this is a superhydrophobic material with a positive Poisson's ratio. In the latter case, this is an auxetic superhydrophobic material with a negative Poisson's ratio. We illustrate these ideas experimentally using PDMS elastomer membranes designed with micro-scale bow-tie lattice structures. We show how the wetting and superhydrophobicity of these membranes behave under strain for both positive and negative Poisson's ratio. To illustrate the general applicability of these ideas as a new class of material, we also show the creation of superhydrophobic auxetic foams.

Acknowledgement. The financial support of the UK Engineering & Physical Sciences Research Council (EPSRC) is gratefully acknowledged (EP/T025190/1and EP/T025158/1).

Dynamics of long bubbles propagating through micro-pin fin arrays

Ismail El Mellas, Federico Municchi, David Hann, Matteo Icardi and Mirco Magnini

University of Nottingham

Two-phase flow through micro-pin fin arrays is encountered in recent micro-pin fin evaporators for micro-electronics cooling and represents a model for flow in unsaturated porous media. While the dynamics of a long bubble in straight channels has been studied for a broad range of cross-sectional geometries and correlations are available to predict the thickness of the lubricating film, detailed studies of the evolution of thin films in a porous media are still missing. In this work, we investigate the dynamics of long gas bubbles and thin films as bubbles propagate through a matrix of cylindrical pins, for different capillary and Reynolds relevant to heat transfer applications and flow in porous media, and different pitch of the cylinders and bubble lengths. Direct numerical simulations of the two-phase flows are performed using the open-source finite-volume library OpenFOAM 1812, where the Volume of Fluid (VOF) method is adopted to capture the interface dynamics. The numerical solver is first validated by simulating the flow of bubbles in straight channels and by comparison with the available literature. The simulations for pin fin arrays emphasise that the film thickness established between the bubble and the cylinders is significantly smaller than that achieved in circular channels with the same capillary number, because the bubble tends to expand in the gap between the cylinders. The liquid film trapped between the bubble and the cylinders exhibits a non-uniform profile with a local dimple at the cylinders side at low capillary numbers. When the bubble is not present, vortex shedding appears at larger Reynolds numbers than in the case of an isolated cylinder. However, the passage of the bubble has the effects of dampening the vortices in the neighbouring cylinders.

Analytical models for the wettability of auxetic metamaterials

Shruti Mandhani[1], Steven Armstrong[2], Andy Alderson[1], Emma Carter[1], Rodrigo Ledesma-Aguilar[2], Ciro Semprebon[3], Gary G. Wells[2] and Glen Mchale[2]

[1]Materials & Engineering Research Institute, Sheffield Hallam University, Sheffield, UK
[2]Wetting, Interfacial Science & Engineering Lab, University of Edinburgh, Edinburgh, UK
[3]Smart Materials & Surfaces Lab, Northumbria University, UK

Typically, when a material is stretched longitudinally, it contracts laterally. Auxetics behave in the exact opposite manner and expand under tension, i.e. they have a negative Poisson's ratio [1,2]. This leads to enhanced properties like high indentation resistance [3], or double curvature [1] that are unachievable in conventional materials. Porous auxetic materials expand by creating additional space in their lattices whilst keeping the solid area constant [4]. Here we model the unique wetting properties caused by strain-dependent changes in the Cassie-Baxter solid surface fraction and Wenzel roughness in auxetic materials. We provide analytical models of the strain-induced changes in solid surface area for different types of auxetic structures, along with the consequent changes in the wetting properties and apparent contact angles. We discuss the implications of these ideas for superhydrophobic and liquid-infused materials.

Acknowledgement: The financial support of the UK Engineering & Physical Sciences Research Council (EPSRC) is gratefully acknowledged (EP/T025190/1and EP/T025158/1).

[1]Lakes, R. Foam Structures with a Negative Poisson's Ratio. Science. 1987, 235, 1038-1040.
[2]Evans, K.E.; Nkansah, M.; Hutchison, I.J.; Rogers, S.C. Molecular network design. Nature. 1991, 353, 124.
[3]Chan, N.; Evans, K.E. Indentation resilience of conventional and auxetic foams. J. Cell. Plast. 1998, 34, 231–260.
[4]Gibson, L.J.; Ashby, M.F.; Schajer, G.S.; Robertson, C.I. The mechanics of two-dimensional cellular solids. Proc. R. Soc. Lond., 1982, A382, 25–42.

Deep learning fluid flow reconstruction around arbitrary two-dimensional objects from sparse sensors using conformal mappings

Ali Girayhan Özbay and Sylvain Laizet

Turbulence Simulation Group, Department of Aeronautics, Imperial College London

The usage of deep neural networks (DNNs) for flow reconstruction (FR) tasks from a limited number of sensors is attracting strong research interest, owing to DNNs' ability to replicate very high dimensional relationships. Trained on a single flow case for a given Reynolds number or over a reduced range of Reynolds numbers, these models are unfortunately not able to handle fluid flows around different objects without re-training. In this work, we propose a new framework called Spatial Multi-Geometry FR (SMGFR) task, capable of reconstructing fluid flows around different two-dimensional objects without re-training, mapping the computational domain as an annulus. Different DNNs for different sensor setups (where information about the flow is collected) are trained with high-fidelity simulation data for a Reynolds number equal to approximately 300 for 64 objects randomly generated using Bezier curves. The performance of the models and sensor setups are then assessed for the flow around 16 unseen objects. It is shown that our mapping approach improves percentage errors by up to 15% in SMGFR when compared to a more conventional approach where the models are trained on a Cartesian grid. Finally, the SMGFR task is extended to predictions of fluid flow snapshots in the future, introducing the Spatio-temporal MGFR (STMGFR) task. For this spatio-temporal reconstruction task, a novel approach is developed involving splitting DNNs into a spatial and a temporal component. Our results demonstrate that this approach is able to reproduce, in time and in space, the main features of a fluid flow around arbitrary objects.

Imaging dynamic wetting ridges on liquid-infused surfaces

Abhinav Naga

Durham University

Inspired by pitcher plants, liquid-infused surfaces (LIS) can repel most liquids. LIS consist of a solid scaffold that is imbibed with a chemically compatible lubricant. The liquid repellent nature of these surfaces is due to the lubricant that effectively minimises pinning forces between the surface and the liquid to be repelled. Consequently, drops start moving as soon as the surface is tilted very slightly (tilt angle $< 5°$). However, in contrast to the fast motion of water drops on dry superhydrohopbic surfaces that have similar tilt angles, the motion of water drops is relatively slower on LIS. Drops typically experience a higher friction force when moving on LIS due to the presence of a wetting ridge - a feature that is absent on superhydrophobic surfaces. Despite the crucial role of the wetting ridge in determining drop friction, microscopic visualisation of its dynamic shape and of the flow profile inside the drop and lubricant are lacking.

To image the dynamic wetting ridge, we fixed a drop on a LIS, directly above the objective lens of an inverted confocal microscope, and then moved the surface at controlled speeds (10 - 2000 μm/s). This home-built setup allows us to monitor wetting ridge geometry over 4 orders of magnitude in the capillary number ($10^{-6} < Ca < 10^{-2}$). During motion, the front side of the wetting ridge changed significantly, decreasing to 30% of its static size at $Ca \approx 10^{-2}$. In contrast, changes in the rear side of the wetting ridge were less pronounced, resulting in an asymmetry between the front and rear wetting ridges. We combine the experimental results with lattice Boltzmann simulations to investigate the velocity profile and the viscous force inside the drop and wetting ridge.

Instabilities of unsteady triple-deck problems

Henry Broadley and Richard Hewitt

University of Manchester

Triple-deck structures have the been the subject of much investigation due to their extensive applications to high Reynolds number flow in scenarios of practical interest. It has been shown by Tutty & Cowley (1986) that such flows can be susceptible to Rayleigh instability for a variety of interaction conditions, though only results for a zero displacement law were given therein. Here two-dimensional flow over an oscillating roughness element placed on an otherwise flat plate is considered for both incompressible and supersonic flow. At first, small amplitude oscillations to O(1) hump heights are considered, requiring a base flow to be calculated from the steady triple-deck equations. These flow profiles are found using a novel (global) numerical method and are shown to agree with existing results, then using the criterion of Tutty & Cowley (1986) we establish which of the steady flows are unstable, before confirming this by considering an appropriate initial value problem. We then allow nonlinear periodic oscillations of the wall roughness and analyse the effect of frequency on the stability of the solutions. It is shown that, on a fixed computational mesh, larger frequencies of oscillation have a stabilising effect on the flow to Rayleigh instability for a given scaled roughness height. Of course the incompressible flow considered herein is also unstable to Tollmien-Schlichting waves, and this is also shown to be captured in our computations.

Tutty, O. R. & Cowley, S. J. 1986 On the stability and the numerical solution of the unsteady interactive boundary-layer separation. J. Fluid. Mech. 168, 431-456.

Deep learning fluid flow reconstruction around arbitrary two-dimensional objects from sparse sensors using conformal mappings

Ali Girayhan Özbay and Sylvain Laizet

Turbulence Simulation Group, Department of Aeronautics, Imperial College London

The usage of deep neural networks (DNNs) for flow reconstruction (FR) tasks from a limited number of sensors is attracting strong research interest, owing to DNNs' ability to replicate very high dimensional relationships. Trained on a single flow case for a given Reynolds number or over a reduced range of Reynolds numbers, these models are unfortunately not able to handle fluid flows around different objects without re-training. In this work, we propose a new framework called Spatial Multi-Geometry FR (SMGFR) task, capable of reconstructing fluid flows around different two-dimensional objects without re-training, mapping the computational domain as an annulus. Different DNNs for different sensor setups (where information about the flow is collected) are trained with high-fidelity simulation data for a Reynolds number equal to approximately 300 for 64 objects randomly generated using Bezier curves. The performance of the models and sensor setups are then assessed for the flow around 16 unseen objects. It is shown that our mapping approach improves percentage errors by up to 15% in SMGFR when compared to a more conventional approach where the models are trained on a Cartesian grid. Finally, the SMGFR task is extended to predictions of fluid flow snapshots in the future, introducing the Spatio-temporal MGFR (STMGFR) task. For this spatio-temporal reconstruction task, a novel approach is developed involving splitting DNNs into a spatial and a temporal component. Our results demonstrate that this approach is able to reproduce, in time and in space, the main features of a fluid flow around arbitrary objects.

Characterisation of permeable-rough surfaces

D. D. Wangsawijaya[1], P. Jaiswal[1,2], and B. Ganapathisubramani[1]

[1]Aeronautical and Astronautical Engineering, University of Southampton, Boldrewood Innovation Campus SO16 7QF, United Kingdom
[2]Bell Textron Helicopters, 12 800 de l'Avenir Mirabel, QC J7J 1R4, Canada

Permeable and rough walls are known to affect the turbulent boundary layers (TBLs) developing over them differently. The effect of roughness on a developing TBL can be characterised by the log-law shift (the Hama roughness function), which is a function of the geometry of the roughness or simply the equivalent sand grain roughness for a "fully rough" flow. The effect of permeability, on the other hand, depends on the pore size and the thickness of the permeable surface. While the effects of these wall conditions are well established independently, their combined effects are still largely unknown.

In this study, we aim to characterise the combined effects of both roughness and permeability on TBLs, and whether these effects can be decoupled. Hot-wire anemometry and wall-shear stress measurements are conducted inside a boundary layer wind tunnel over a wide range of Reynolds numbers ($6000 \leq Re_\tau \leq 30000$), where the boundary layer develops over the following surface arrangements: permeable wall (45 ppi reticulated polyurethane foam), rough walls (metal meshes with 81% and 73% open area, respectively), and permeable-rough walls (the foam topped with metal meshes). Present results suggest that both permeable and permeable-rough walls lead to an increase in drag compared to smooth walls; the increase in drag for the permeable-rough walls is the highest. Characterisation of the combined permeable-rough effects, and an attempt to decouple these effects will follow.

Benchmarking uncertainty quantification methods in flood modelling applicationsy

Mahya Hajihassanpour, Georges Kesserwani and Per Pettersson

University of Sheffield, NORCE Norwegian Research Centre

Evaluation of uncertainty quantification (UQ) methods in flood models, due to the presence of intrinsic uncertainties, is of prime importance today. The Standard Monte Carlo method (SMC) is the traditional choice to conduct the UQ analysis, however, it has a slow canonical convergence meaning to halve its error magnitude the sample size should be quadrupled. This paper compares the performance of a series of alternatives to SMC that are likely to outperform SMC in terms of needing a smaller sample size to achieve a desirable error threshold. The alternative methods are Latin Hypercube Sampling (LHS), Adaptive Stratified Sampling (ASS), Quasi Monte Carlo (QMC) and Haar Wavelet Expansion (HWE). These alternative methods are selected as they can capture any type of probability distribution including the challenging discontinuous and multimodal ones. Probabilistic numerical tests under multiple uncertain inputs are designed to support comparing the alternative UQ methods against the SMC. Tests reveal that (I) All the alternative UQ methods are more efficient than SMC for problems under two uncertain inputs, (II) increasing the number of uncertain inputs to three leads to a reduction in the efficiency of the alternative UQ methods, (III) The higher nonlinearities in the output responses as a function of the inputs, the lower efficiency of the alternative UQ methods, (IV) The deterministic realisation methods (QMC, LHS) usually outperform the random sampling methods (LHS, ASS), especially if the aim is to achieve small error thresholds such as 5%, (V) the QMC due to its flexibility in working with any sample size and excellent coverage of the probability space can be the best alternative candidate to SMC.

Experimentally simulating urban air pollution around a single tall building

Tomos Rich and Christina Vanderwel

Aeronautical & Astronautical Engineering, University of Southampton

Atmospheric flow and air pollution dispersion around a tall building is simulated in a water tunnel using a simplified scale model. This simplified model is a single square cylinder with an aspect ratio of 1.4 , which is immersed within the oncoming boundary layer flow. The building height to boundary layer height ratio is 4.3. Air pollution is modelled by an upstream ground-level point-source release of Rhodamine 6G fluorescent dye. This represents a passive scalar release as the dye is neutrally buoyant in water and is released into the boundary layer with negligible effect on the flow.

Particle image velocimetry (PIV) and planar laser induced fluorescence (PLIF) techniques are used to acquire velocity and concentration measurements. Two-dimensional maps of the streamwise/wall-normal cross section along the centre-line of the flow are stitched together to obtain a field of view which begins ahead of the tall building and covers the long building wake. Stereo PIV and PLIF measurements are also taken in spanwise slices, starting from just behind the tall building and at several further positions downstream.

Results show that presence of the tall building has a significant dispersing influence. The mass of the plume is transported vertically and horizontally when it approaches the building. The building wake consists of a region of recirculation but contains relatively low concentrations. The scalar reappears in the streamwise plane further downstream of the building. The turbulent scalar fluxes are calculated and compared in both the streamwise and spanwise planes.

The mean electromotive force arising from shear driven magnetic buoyancy and rotation

Craig Duguid, Paul Bushby and Toby Wood

Newcastle University

Since the pioneering work of Parker 1955, the leading theoretical paradigm for the Sun's magnetic cycle has been an alpha-omega dynamo process, in which a combination of differential rotation and turbulent, helical flows produce a large-scale magnetic field that reverses every 11 years. The most problematic part of this model is the production of large-scale poloidal field - the alpha effect - which is usually attributed to small-scale convective motions under the influence of rotation.

We revisit the original Parker model, in which the differential rotation in the solar tachocline generates a strong toroidal magnetic field, which then becomes buoyantly unstable. We study this magnetic buoyancy instability to determine whether, in the presence of rotation, it can by itself provide the necessary regenerative effect.

We present results of simulations of a local, rotating (with aligned and tilted rotation vectors), fully compressible model in which an imposed vertical shear winds up an initially vertical background magnetic field which ultimately becomes buoyantly unstable. In particular, we measure the resulting turbulent electromotive force (EMF) where we are primarily interested in the component in the direction of the mean magnetic field, as required for a Parker-like dynamo to operate.

For sufficiently rapid rotation, we find that the mean EMF has a significant component in the direction of the mean magnetic field, as required for a Parker-like dynamo to operate. Our results suggest that magnetic buoyancy contributes directly to the generation of large-scale poloidal field in the Sun.

Large-eddy simulation of microscale urban surface energy balances over a diurnal cycle

Sam Owens

Imperial College London

A methodology is presented for studying the diurnal cycle of the surface energy balance of individual urban facets. An important consideration is the accurate representation of the urban boundary layer (UBL) due to the influence on turbulent surface-atmosphere exchanges. To this end, the Met Office Unified Model (MetUM) was run over London for a 36-hour interval within a sequence of days with clear skies and low wind. During this period, it can be seen from MetUM output and observations that the UBL follows a characteristic, approximately periodic evolution. This data was one-way coupled with uDALES, a building-resolving urban atmospheric large-eddy simulation (LES) framework equipped with a three-dimensional surface energy balance model.

When using LES, it is necessary to generate a turbulent inflow boundary condition. A precursor simulation was used for this purpose, and time-height profile assimilation was used to force the flow towards area-averaged MetUM data. The profiles were adjusted to be fully periodic in time, which makes it possible to simulate multiple days using a single day of MetUM data. The target simulation can then be used to study the surface energy balance and turbulent exchanges whilst not affecting the evolution of the UBL as defined by the inflow boundary condition.

The forcing method is found to reproduce the mean flow well. In order to check the turbulence is also reasonable, the shear stress and heat flux profiles for the uDALES precursor simulation are compared against that of MetUM. The models are found to agree fairly well during the day, with greater discrepancy at night. The models are also compared against observations, with mixed results. Finally, to demonstrate the analysis that this methodology enables, surface energy balances are presented for the uDALES target simulation and compared to that of MetUM in light of the different approaches taken by the two models.

Scaling laws in turbulent convection due to internal heating

Ali Arslan

Imperial College London

Understanding convection constitutes an endeavour to understand the force of buoyancy, which on length scales larger than that of humans is most influential on our lives. Buoyancy due to chemical concentration differences is crucial to the dynamics in the ocean and in the Earths core, while buoyancy due to thermal variations drive atmospheric and mantle physics. The scales involved in these environments lead naturally to turbulent and chaotic motion, such that finding a solution to the governing equations or simulating the flow numerically remain insufficient to obtain a holistic view of the dynamics involved. Instead, one can look to emergent properties of the flow, like dissipation or convective heat transport and care for their value over a long time and in an averaged sense. Thermal convection driven by internal heating is one such simple scenario to investigate buoyancy, where a fluid is heated due to internal energy sources. We look to bound the mean convective heat transport, directly related to the viscous dissipation, in internally heated convection in the asymptotic limit of infinite Prandtl numbers, under different thermal and velocity boundary conditions. Rigorous scaling laws are obtained on the heat transport as a Rayleigh, quantifying the destabilising effect of internal heating, is increased. Finally, to better match convection observed in nature, the techniques are extended to investigate non-uniform heating profiles.

Effect of low concentration particles in the fluid behaviour of particulate pipe flows

Rishav Raj and Alban Potherat

Centre for Fluids and Complex Systems, Coventry University

Particulate flows can be found everywhere in nature, from rivers carrying sediments to the gust of winds carrying dust particles. We can even find them in the oil and gas industry or the biomedical industry. The particles interacting with the fluid affect the fluid advection dynamics and change the stability of the flow.

In the current work, two-phase flow experiments were carried out in a pipe flow facility to study the effect of solid, neutrally buoyant, spherical particles in low concentration particle-laden pipe flow. The pipe facility was connected to a piston-cylinder arrangement to facilitate the flow, and perturbations were introduced orthogonally in the flow through a single impulsive jet injection system. Two-dimensional particle image velocimetry (PIV) technique was used to analyse the flow behaviour. Simultaneously, the particle tracking velocimetry (PTV) technique was used to study the particle behaviour in the flow. The study was carried out with particles of sizes 150μm and 425μm. The effect of particle size, concentration and the jet injection volume on flow stability was studied for various Reynolds numbers.

A framework to quantify the robustness of flows subject to finite-amplitude instabilities

C. Beaume[1], A. Pershin[2], T. S. Eaves[3] and S. M. Tobias[1]

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

Transition to turbulence is an instability in which a laminar flow becomes chaotic. In most canonical shear flows, the laminar state is linearly stable and transition occurs due a finite-amplitude instability. Assessing the robustness of the laminar flow and, thus, the likelihood of observing transition to turbulence is not straightforward: perturbations of different amplitude, but also of different shapes, behave differently. In this talk, we will discuss a new framework to quantify the robustness of the laminar flow based on the computation of the laminarization probability of random perturbations as a function of their energy. This framework is adaptive: it can be tailored to any finite-amplitude instability problem and can be refined by using specific knowledge such as the shape of the perturbations or their energy distribution. We will discuss the application of this framework to plane Couette flow in the presence of a well-known control strategy aimed at suppressing transition and will analyze the performance of the control strategy using simple scalar indicators of the flow robustness and control strategy cost efficiency.

Long-wavelength three-dimensional instability in the vortex formation region behind a body

Andrey I. Aleksyuk and Matthias Heil

Department of Mathematics, University of Manchester, Manchester, United Kingdom

We discuss the physical mechanisms for the long-wavelength (mode A) 3D instability of the 2D time-periodic viscous fluid flow around a circular cylinder. Even though several hypotheses explaining 3D instability in the vortex formation region exist in the literature (e.g., see [Leweke, Williamson, Eur. J. Mech. B Fluids, 1998; Thompson et al., J. Fluids Struct., 2001]), the problem still requires clarification because the actual flow is substantially more complex than is usually assumed in the simplified models. In particular, the base flow in the formation region significantly depends on time and 3D perturbations grow in several subregions (braid shear layers and vortex cores), which can interact with each other.

We show that the pattern of the most rapidly growing perturbations is preserved (and corresponds to a mode A instability) when one continuously increases the perturbation wavelength λ. In the limit as $\lambda \to \infty$, the perturbed solution can be obtained analytically and corresponds to the shift of the base flow solution in time. We use this remarkable similarity of the perturbation pattern to reveal the mechanism for its formation by simplifying governing equations in a large λ limit. We then explain the selection of critical parameters for the instability by analysing the action of the basic physical mechanism for the 3D vorticity change using the approach suggested in [Aleksyuk, Shkadov, Eur. J. Mech. B Fluids, 2018].

The research was supported by the Royal Society Newton International Fellowship (NIF/R1/201343). The authors would like to acknowledge the assistance given by Research IT, and the use of the HPC Pool funded by the Research Lifecycle Programme at the University of Manchester.

Assessing transient airborne infection risks in a hospital: a mathematical model

Alexander J Edwards, Catherine J Noakes, Martín López-García, Marco-Felipe King and Daniel Peckham

University of Leeds

A useful way to understand airborne transmission of disease is by quantifying and evaluating the risk of infection. Quantitative microbial risk assessment (QMRA) can provide an insight into the factors that govern airborne transmission, and those that mitigate against it. Understanding this in hospital settings is important since patients are vulnerable, thus the consequences of infection are severe. Up to now, research has assumed well-mixed air and ignored transient behaviours and effects, which in reality, are ubiquitous in indoor environments. We propose a multi-zone model with inter-zonal flow rates, allowing us to evaluate the consideration of transient effects versus a steady-state model. The adapted Wells-Riley model captures transient concentration of pathogen in the air, and various occupancy activities including a transient infector. To illustrate the methodology, we focus on particular occupancy, ventilation, and disease scenarios with various flow rates within a healthcare setting. Findings suggest that the steady-state model leads to an overestimation of concentration of pathogen in the air and consequently, infection risk for short exposure times. However, for longer exposure times often experienced in indoor environments, a steady-state model could be sufficient depending on the scenario parameters. Our model highlights the need for careful consideration of transient factors and behaviours when assessing airborne transmission and associated infection risks, to ensure the most effective mitigation strategies are implemented.

Feature based analysis of the North Atlantic Eddy Driven Jet

Jacob Perez, Amanda Maycock, Stephen Griffiths, Christine McKenna and Steven Hardiman

University of Leeds - CDT in Fluid Dynamics, University of Leeds - School of Earth and Environment
University of Leeds - School of Mathematics
University of Leeds - School of Earth and Environment, Met Office

The Eddy Driven Jet (EDJ) is a major driver of extreme weather events occurring in the mid-latitudes in the Northern Hemisphere. To quantify the variability of the EDJ, a latitudinal position is defined as the position of the maximum daily zonal mean zonal wind taken over the North Atlantic. Using this measure, three latitudinal states or regimes appear. It has been suggested that the three regimes of the North Atlantic EDJ represent a Southern, Northern and Central jet position during the winter.

This 1-dimensional jet latitude index has become ubiquitous. However, the process of zonal averaging leads to smoothing out of distinct features. The tilted structure of the jet, that is seen in the winter climatology, is lost to this process. There are also difficulties in interpreting days with split jets, broad jets, and Greenland tip jets.

In this work we propose a new feature-based method to diagnose the North Atlantic EDJ and its variability, based on 2-D maps of the low-level zonal wind field. Using this method, we identify jet 'objects' based on contours of a minimum zonal wind threshold. A moment-based analysis is then used to compute the jet latitude, tilt, length, and width. We show cases where this method characterizes the North Atlantic circulation more accurately and more robustly than jet latitude index. Unlike the jet latitude index, we allow for days without a distinct EDJ.

We discuss insights into the morphology of the jet, brought to light by this new method. Most significantly, the distribution of jet latitudes produced by this method results in a unimodal distribution rather than a trimodal structure, casting doubt on earlier interpretations of regime behavior in the North Atlantic EDJ.

RANS analysis of merging supersonic streamwise vortices for enhanced mixing in scramjet engines

Dónal McCaughey, Dr Marco Geron and Dr Rob Watson

Queen's University Belfast School of Mechanical and Aerospace Engineering

Scramjet powered vehicles may have the potential to greatly reduce the weight and improve reusability of some of the currently rocket based stages in space launch vehicles. Among the challenges facing their wider adoption, is that of achieving sufficient mixing of air and fuel in a supersonic flow. The flow remains supersonic within a scramjet engine, leaving a relatively short residence time for fuel to be mixed and spread across the cross section of the combustor. Additionally, the growth of shear mixing layers is suppressed in supersonic flow. Streamwise vortices are studied as a mixing enhancer for scramjet combustors due to their relative insensitivity to compressibility. A specific merging interaction between supersonic co-rotating vortices has been shown in experiments to sustain turbulence production against decay. In this presentation this interaction is investigated further numerically to determine how well it can be replicated with a RANS approach, the effect of increasing the Mach number ($M = 2.5$, 3.0, 3.5, 4.0, 4.5), and the influence of shockwaves on this interaction. The shape and rotation of the merging vortices appears similar to experimental results. The decay of vorticity follows a similar trend to experimental results, however the vorticity was higher in the RANS results. At higher Mach numbers the merging process appears to be pushed further down the domain. Some degree of merging still occurs up to Mach 4.5, indicating this approach may be applicable for mixing enhancement for the Mach number range and length of domain considered.

Assessment of sponge layers for aeroacoustic modeling implemented in Ansys Fluent

Guangyuan Huang, Sidharath Sharma, Stephen Ambrose and Richard Jefferson-Loveday

Gas Turbine and Transmissions Research Centre, University of Nottingham, Nottingham, NG7 2TU

Robust and non-reflective boundary conditions are generally required by transient simulations for aeroacoustic modeling and compressible flow. Ansys Fluent in the latest release of 2022 R1 provides a new acoustic model of sponge layers to remove non-physical pressure waves reflecting from boundary zones. As an alternative to the general non-reflective boundary conditions (NRBC), sponge layers have advantages in suppressing pressure waves that propagate in directions not normal to the boundary surface. The present study aims to assess the capability of this new acoustic model. The aeroacoustic field of a two-bladed Mejzlik propeller installed on a NACA0018 airfoil is modeled by compressible Large-Eddy Simulation (LES) using the general NRBC and sponge layers. Results have shown that odd pressure fluctuation is generated at the junction of the airfoil and the outer boundary with only NRBC applied. On the other hand, the pressure fluctuation is well suppressed near the outer boundary covered by a sponge layer. In addition, the effect of the sponge-layer thickness is investigated. Sponge layers with the thickness of 0, $\lambda/8$, $\lambda/4$, and $\lambda/2$ are implemented, where λ denotes the wavelength of the pressure wave at the first blade passing frequency (BPF). Results have shown that no significant pressure fluctuation is observed near the outer boundary covered by the sponge layer of $\lambda/8$ width. Although the flow close to the inner boundary of the sponge layer is influenced, that being $\lambda/8$ distant from the inner boundary is not influenced. Based on these lines, a recommended setup of sponge layers is purposed.

Inertia-gravity wave diffusion by geostrophic turbulence: the impact of flow time dependence

Michael Cox, Hossein A. Kafiabad and Jacques Vanneste

University of Edinburgh

The scattering of short inertia-gravity waves by large-scale geostrophic turbulence in the atmosphere and ocean can be described as a diffusion of wave action in wavenumber space. When the time dependence of the turbulent flow is neglected, waves conserve their frequency, which restricts the diffusion of energy to the constant-frequency cone. We relax the assumption of time independence and consider scattering by a flow that evolves slowly compared with the wave periods, consistent with a small Rossby number. The weak diffusion across the constant-frequency cone introduced by time dependence leads to a stationary energy spectrum that remains localised around the cone (specifically decaying as $1/\sigma^5$ with σ the angular deviation from the cone) corresponding to a small frequency broadening. We contrast our results with unbounded frequency broadening that arises for surface- or shallow-water waves.

Numerical investigation of particle-vortex interaction in a cross-slot geometry under moderate inertia

Konstantinos Kechagidis, Benjamin Owen, Timm Krüger

School of Engineering, Institute for Multiscale Thermofluids, University of Edinburgh

The characterisation of blood samples in inertial microfluidic flows (IMF) increases the throughput of point of care applications. The cross-slot geometry has been successfully employed to assess the mechanical properties of leukocytes to detect sepsis. Under IMF conditions, a spiral vortex forms at the centre of the cross slot (junction). Although the vortex dynamics has been extensively investigated, the particle-vortex interaction is not yet well understood. Here, we perform lattice-Boltzmann simulations to study the effects of the initial position, confinement ratio and density ratio on the trajectories and dynamics of a single rigid particle in the cross slot. We find that a tracer or a point particle is not sufficient to approximate the behaviour of a finite-sized particle in the cross slot. Results show that the particle motion strongly depends on the initial position. The residence time in the junction depends on the vortex direction and decays rapidly with the initial lateral distance from the inlet's centreline. More considerable variations occur in the profile of the stresses acting on the particle when the vortex forms compared to a case with a stagnation point instead of a vortex. Due to limited space, the particles with sufficiently large confinement ratios stay longer in the junction but do not oscillate. The residence time scales linearly with the particle-to-liquid density ratio, which does not significantly alter the particle trajectory for the investigated density ratios. Finally, we show that the shape of the particle trajectory in the cross slot allows the identification of particle properties. Our research results contribute to the understanding of particle dynamics in IMF cross-slot geometries and open up novel applications in cell cytometry.

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, UK

The aim of this research is to study receptivity and sensitivity in a nearly semi-cylindrical cavity with an upper free surface, where fluid can be fed in, and porous lower boundaries, where fluid can escape. This cavity comprises a semicircular lower boundary, two adiabatic sidewalls and is infinitely extended in the third direction. This configuration describes numerous problems involving solid materials melted, for example, in metallurgical casting processes. Our earlier analysis using the linear stability analysis and direct numerical simulations shows that the unstable modes are three-dimensional [1]. Next, we put forward a mechanism to suppress these unstable modes.

To suppress the instability, we use the receptivity and sensitivity analysis (RSA), which is based on a linear theory. The RSA requires both the direct and adjoint modes, for which we solved the non-dimensional version of the direct and adjoint equations governing buoyancy-driven flows under Boussinesq approximation using the spectral-element code NEKTAR++ [2]. The receptivity analysis reveals that the through-flow in the middle of the inlet is most receptive. The sensitivity analysis indicates that the region at the bottom of the domain is most sensitive. Further, we exhibit suppression of instability using receptivity as a time-dependent body force.

[1]Kumar and Pothérat, J. Fluid Mech. 885, A40 (2020).
[2]Moxey et al., Comput. Phys. Commun. 249, 107110 (2020).

Experimental investigation of an integrated oscillating water column (OWC) within fixed and heaving breakwaters

Yusuf Almalki, Chris Swan and Ioannis Karmpadakis

Imperial College London

The integration of wave energy converters within breakwaters is a promising application towards coastal protection and energy generation. Housing a pneumatic chamber for wave energy extraction in breakwaters can enhance their shoreline protection function without significant additional construction costs. In this study, the efficiency of both fixed and heaving breakwaters with and without OWC integration is investigated experimentally. Their potential to harvest incoming wave energy and protect the coastal zone is quantified by estimating relevant wave reflection and transmission coefficients. This is achieved by considering both monochromatic and random incident wave conditions. The self-rectifying impulse turbine of OWCs has been experimentally simulated using a small opening on top of its pneumatic chamber. The pressure inside the pneumatic chamber and the air velocity across the orifice were recorded to estimate its wave energy extraction potential. The effects of relative breadth and draft of the model on the reflection and transmission coefficients, pressure fluctuation inside its chamber, and air velocity across the topside orifice were parametrically investigated. This information will be used alongside the response of heaving models to provide guidance regarding the viability, optimal design and operation of such devices.

Subcritical transition to turbulence in qausi-2D shear flows

Christopher Camobreco[2], Alban Potherat[1] and Gregory She[2]

[1]Coventry University
[2]Monash University (CC and GS)

The transition to turbulence in conduits is among the longest-standing problems in fluid mechanics. Challenges in producing energy or reducing its consumption often hinge on understanding how to either promote or hinder turbulence. The transition occurs at low flow rates in response to perturbations exceeding a critical amplitude (such transitions are subcritical) through an intrinsically three-dimensional (3D) mechanism. While a global picture is emerging for 3D turbulence, a subcritical transition to turbulence is yet to be observed when flows approach two dimensions,for example, because of intense rotation or magnetic fields. We first show numerically that the optimal perturbation that maximises transient growth in a rectangular duct changes as dimensionality progressively reduced: the optimal streamwise vortices progressively give way to large structures attached to the walls and aligned with the magnetic fields: as such the 3D 'lift-up' mechanism to amplify perturbations in three dimensions is suppressed in favour of a much less efficient 2D 'Stokes mechanism'. Then, in the quasi-2D limit, we use stability analysis and direct numerical simulations to provide the first evidence of turbulence in subcritical quasi-2D shear flows. The scenario leading to turbulence mostly relies on the nonlinear dynamics of so-called Tollmien-Schlichting waves, rather than on perturbations experiencing fast, transient growth. Because these waves are severely damped, the transition cannot take place at such low flow rates as its 3D cousin. This alternative scenario not only calls for a new line of thought on the transition to turbulence but should also inspire the design of more efficient rotating devices, nuclear fusion reactors, and novel strategies to promote or hinder turbulence.

Data assimilation method using planar PIV of the flow around a NACA0012 airfoil

C.Thompson, S.Symon and B.Ganapathisubramani

Southampton University

CFD using a RANS approach is a time efficient method to simulate aerodynamic problems, however from previous investigations RANS simulations have difficulty predicting the transition, separation and reattachment locations on the suction side of stalled airfoils.

With the introduction of PIV, velocity flow fields around airfoils can be obtained experimentally and used to drive CFD simulations, thereby computing more accurate predictions. The resultant predictions can be used to generate missing flow fields, tune simulation parameters, or investigate the far field away from the measurement domain.

In this investigation a state observer-based data assimilation approach is used to improve an incompressible RANS simulation of a NACA0012 airfoil in stall conditions. The investigation shows how an additional spatially varying forcing term within the RANS equation can be optimized to generate a more accurate prediction of the flow field, the physical understanding and significance of this forcing term is explored.

In a two-dimensional domain, deviations of this forcing term with respect to varying the Reynolds number are investigated, particular attention is given to exploring the stagnation of this forcing term at high Reynolds number and the physical reasons for this stagnation. The effect of stall angle on the forcing term is also studied, investigating the accuracy of interpolating the forcing term between the upper and lower limit of stall angles.

Droplet adhesion revisited

Muhammad Subkhi Sadullah

King Abdullah University of Science and Technology

On a solid surface, liquid droplets can be removed by either sliding on or detaching from the surface. While the sliding motion and its retention force have been studied extensively, the detaching motion has only received less attention. In this contribution, we study the force needed to detach a liquid droplet from a solid surface in the direction normal to the solid surface. We employed the centrifugal acceleration balance (CAB) technique to measure the detachment force of a liquid droplet at a given contact angle. We then compared the experimental results with the numerical results obtained from lattice-Boltzmann (LB) simulations. We observed that experimental and numerical results are in good agreement, where the curve for the detachment force decreases monotonically with the liquid contact angle. We then discuss the relation of the detachment force with the work of adhesion. We analyze droplet shapes at equilibrium for different magnitudes of the gravitational force to quantify the possible forces involved in the system e.g., the gravitational, surface tension, Laplace pressure, and adhesion forces. Surprisingly, we found that the gravitational force can be balanced by the difference between the surface tension and Laplace pressure forces alone, indicating that the force due to the work of adhesion does not play a role. The droplet detaches from the solid when the droplet shape can no longer hold a stable shape in response to gravity. This finding is contrary to the previously proposed theory that the detachment force depends on the work of adhesion.

Nonlinear reduced-order models for magnetoconvection in 2D

Alasdair Roy[1], Steven Tobias[1], Christopher Jones[1], Philip Livermore[1], Wayne Arter[2] and Stanislas Pamela[2]

[1]University of Leeds (School of Computing, School of Mathematics) [2]Culham Centre for Fusion Energy

Direct numerical simulation of full state equations can be computationally intensive and result in high-dimensional datasets. While the numerical data is high-dimensional, the flows themselves can exhibit much lower dimensional behaviour. This has led to an interest in finding reduced order models for such systems.

Recently, data-driven model discovery techniques have been shown to be able to find reduced-order models purely from measurements of the system, automating this process. Data- driven methods have the advantages that an optimal set of modes can be used to describe the data through proper orthogonal decomposition. The time series of these modes projected on the measurements can then be used to generate a reduced order model via the sparse identification of nonlinear dynamics (Brunton et al, 2016).

We apply this method to 2D magnetoconvection, where traditional convection can be inhibited by the presence of a magnetic field. These convective processes are important, for instance, at the surface of stars. It was shown that the governing equations for magnetoconvection admit weakly nonlinear solutions given by a system of 5 ODEs (Knobloch et al, 1981) valid close to onset. These are instructive as they are able to predict features of magnetoconvection such as whether convection first sets in as a direct instability or overstability.

We first apply these methods to understand if recovery of the analytic solutions near fixed points is possible, before looking for reduced order models further from onset in regimes with aperiodic motion. By comparison with the solutions of the full PDEs, we assess the utility of the reduced models both for prediction and for reproducing the statistics of the full system; we further test how well a model derived for a certain set of parameters reproduces the dynamics for parameter sets for which no direct data is available.

Efficient solution of fluid-structure interaction problems using an ALE finite element method

Gregory Walton, Peter Jimack and Mark Walkley

University of Leeds

Solving fully coupled fluid-structure systems can be numerically challenging. The mechanics of these fluid-structure interaction problem are highly nonlinear, not only are the governing equations of the constitutive parts nonlinear but the coupling at the interface also contributes to the nonlinear behaviour. In this work we consider a monolithic approach that is based upon an arbitrary Lagrangian-Eulerian (ALE) finite element method. This conformal meshing approach permits the deformation of a single mesh throughout the domain, evolving with the fluid-structure interface.

The importance of an appropriate pressure space approximation when using finite elements with fluid-structure interactions will be demonstrated. Specifically, the benefits of accurately capturing the discontinuous pressure at the fluid-structure interface will be described for a challenging model problem involving an incompressible fluid in three dimensions. The stable P2/P1 element pair will be compared against the lower order but discontinuous pressure pairing, P2/P0. Further discussion will consider the impact of enriching the P1 pressure space using a constant over each element, P2/(P1+P0).

Finally, a numerical investigation will be presented examining the performance and stability of different solving strategies of the linearised fluid-structured system when using a mixed finite element approximation. This will specifically focus on developing an efficient iterative solver, based upon different block preconditioning approaches.

A CFD study of the conjugate heat transfer in a T-junction with an upstream elbow

Y. H. Wong, Y. Duan, L. Lampunio, M.D. Eaton and M. J. Bluck

Imperial College London

The analysis of thermal-mechanical fatigue (TMF) phenomena is important for the assessment of the reliability of nuclear components within the primary and secondary circuits of nuclear power plants (NPPs). TMF can be characterised as either low cycle or high cycle. Low cycle TMF can be categorized as a series of "thermal shocks" whereas high cycle (high frequency) TMF is sometimes known as "thermal striping" and is the focus of this paper. Thermal striping is characterised by the turbulent mixing of hot and cold flow streams that results in temperature fluctuations of the coolant near the wall. The coolant temperature fluctuations may cause cyclical thermal stresses and subsequent cracking and failure of the pipe wall, resulting in unplanned NPP shutdowns. The T-junction of the hot leg pipe, in the primary circuit of a pressurized water reactor, connects the pressurizer with the steam generator and the reactor pressure vessel and is a typical component which is subject to significant thermo-mechanical stresses due to the incomplete mixing of fluids at different temperatures.

The research, presented in this paper, aims to analyse the fluid dynamics within a T-junction with an upstream elbow and the consequent temperature fluctuations near, and within, the pipe wall. Compared to pipes with straight inlets the presence of an upstream elbow may generates low frequency, near wall, temperature fluctuations downstream of the T-junction. The thermal mixing within the fluid domain and the conjugate heat transfer within the pipe wall are modelled and simulated using the CFD solver Star-CCM+. The mesh sensitivity of the CFD model, along with the effect of varying the branch pipe diameter, are studied both qualitatively and quantitatively. This will support further analysis and assessment on the effects of the branch pipe diameter before the T-junction on the flow dynamics and temperature fluctuations near, and within, the pipe wall.

On the effect of filament inclination on the different regimes of canopy flows

Shane Nicholas, Mohammed Omidyeganeh and Alfredo Pinelli

City University of London

We have performed high-fidelity simulations of turbulent open-channel flows over submerged rigid canopies made of cylindrical filaments of fixed length h/H = 0.1 and mean spacing of S/H = 0.066 (with 'H' being the open channel height), which is flush mounted perpendicular to an impermeable wall. The overall shape of the canopy elements was modulated such that the mean effect of a flexible filament was mimicked onto the surrounding flow. The angle of inclination being the free parameter, governs the shape of the filament, which in turn dictates the solidity of the canopy, yielding in different flow regimes (sparse or dense). Although static in nature, the modulation of the inclination angle can also be translated to the mean degree of reconfiguration of the canopy filaments via the Cauchy number to identify the alternative flow regimes. We consider a total of six configurations, in which the filaments are inclined along the flow direction, with the angle of inclination in the range [0,90], with 0 and 90, representing the negligible and very strong reconfiguration regimes and the intermediate values of representing the transitional reconfiguration regimes. Indeed, when the canopy is inclined, the behaviour of the flow (inner and outer) can drastically change based on the direction of the inclination, therefore undermining the notions based on the solidity.

In-silico flow diverter performance assessment in posterior communicating artery aneurysms

Michael MacRaild, Shuang Song, Ali Sarrami-Foroushani, Nishant Ravikumar, Christopher Kelly, Toni Lassila, Zeike A. Taylor and Alejandro F. Frangi

Centre for Doctoral Training in Fluid Dynamics, University of Leeds.
Centre for Computational Imaging and Simulation Technologies in Biomedicine (CISTIB), University of Leeds.

Posterior communicating artery (PComA) aneurysms account for 25% of all aneurysms, but despite PComA aneurysm prevalence, flow diversion (FD) with Pipeline Embolization Devices is not FDA-approved for this subgroup. We performed an in-silico trial (IST) of PComA aneurysm FD, investigating: (i) What is the relationship between FD success and PComA patency? (ii) Which aspect of FD is inhibited in the presence of fetal posterior circulation (FPC)?

We used an automated IST pipeline, consisting of image segmentation, surface mesh pre-processing, device deployment, volumetric meshing, simulation set up and execution, post-processing and analysis. An initial cohort of 17 patients was used. For each patient we simulated transient haemodynamics with two physiological states, rest and exercise. Flow diversion was assessed through post-treatment reduction in space-and-time-averaged velocity (STAV) in the aneurysm. PComA patency was assessed through post-treatment change in STAV at the PComA outlet and change in PComA pressure drop.

We use PComA size as a marker for FPC, as in fetal patients the PComA is enlarged. Preliminary results show a trend towards greater aneurysm flow reduction and increase in PComA outlet flow post-treatment in smaller PComAs. They also show some correlation between treatment success and PComA patency measures, with greater flow reduction in patients with increase in PComA outlet flow and decrease in PComA pressure drop post-treatment.

Larger PComAs demand more flow distally and maintain flow following treatment by PED [3], which is thought to reduce treatment success due to persistent flow through the PED into the aneurysm. We do find worse aneurysm flow reduction in larger PComAs, but conversely, we find larger PComAs are more likely to experience post-treatment flow reduction. We see improved aneurysm flow reduction in cases where PComA outlet flow increased post-treatment and where PComA pressure drop decreased, which conflicts at present.

Analysis of liquid break up and droplet distribution in a flat fan spray using image processing

Zuhaib Nissar, Oyuna Rybdylova and Steven Begg

Advanced Engineering Centre, School of Architecture, Technology and Engineering, University of Brighton, Brighton BN2 4GJ, UK

The study is concerned with the experimental validation of mathematical models for sprays based on the recently proposed Fully Lagrangian Approach (FLA). The FLA can track individual droplets and simultaneously compute continuous number density values along droplet trajectories. This has recently been generalised for the case of poly-disperse evaporating droplets (gFLA). The current work focuses on the development of an image processing algorithm applied to microscopic, high-speed image sequences, in order to characterise the liquid break-up process and the formation of droplets and their distribution in sprays. The study takes into consideration the stochastic nature of these sprays, the data is collected in a systematic grid-wise manner, throughout the spray field and for all timeframes. The proposed method is designed to reconstruct spatial droplet size distribution required for the validation of the FLA. The experimental setup consists of a slot type nozzle, that generates a flat flan spray, and a common rail injection system capable of generating steady-state as well as pulsating sprays. The spray is visualised using a high-speed camera at 5000 Hz and a long-distance microscope fitted with an objective lens; a pulsating LED light source for backlit illumination is synchronised with the camera. The focal plane of the camera is oriented parallel to the spray sheet to ensure fluctuations in the spray are within its depth of field. In the near-nozzle area, an analysis is performed to identify and differentiate between structures in the spray using an in-house image processing algorithm. We include analysis of the mechanisms leading to the break-up and atomisation. In the downstream, dilute atomised area, the position and size of individual particles, are identified in consecutive image sequences and then extracted across all time frames. This statistical data is then used to obtain the particle size distribution in various parts of the spray.

Aerodynamic and structural performance of serrated leading edge outlet guide vanes designed for low noise gas turbines

Cleopatra Cuciumita and Ning Qin

The University of Sheffield, Department of Mechanical Engineering

A parametric study has been conducted to gain novel insight into the aerodynamic and structural behaviour of serrated leading edge guide vanes that were designed to reduce the broadband noise. The study evaluates the total pressure losses and the maximum stress for the vanes with different serrations amplitudes and wavelengths. The aerodynamic computations are first validated against experimental data for the datum geometry for three different regimes: take-off, approach, and cruise. The aerodynamic computations are further coupled with the structural analysis to transfer the pressure distribution on the blade and account for the aerodynamic load. The structural analysis computes the stress for a fan blade off event, which is the worst-case scenario under an ultimate load condition for outlet guide vanes. The rotating unbalance force caused by the shift in the centre of gravity location due to the missing blade is estimated and applied as load. Since this is not a periodic load, the whole vane ring is used for both aerodynamic and structural computations. A total number of 37 different geometries have been computed and response surfaces for the total pressure loss coefficient and for the minimum safety factor have been constructed. The results show a deterioration of the aerodynamic performance with the introduction of leading-edge serrations, which is more pronounced for higher serrations amplitudes and lower wavelengths values. The maximum recorded increase in pressure losses is 0.13% for an amplitude of 7% of chord. The flow mechanism responsible for the increase in total pressure losses is discussed. The maximum stress variation with the amplitude and wavelength of serrations is less linear, but, overall, follows a similar increase pattern, with a maximum increase of 8% as compared to the datum value.

Topology-free optimisation of turbine blade tip using the adjoint method

Guoxi Ye and Ning Qin

Department of Mechanical Engineering, the University of Sheffield.

This paper presents a topology optimisation of turbine tip treatment using the gradient-based adjoint method to alleviate the negative effects of leakage flow. A three-dimensional (3D) numerical calculation was performed for an un-cooled unshrouded turbine blade using Reynolds-averaged Navier-Stokes (RANS) model. All the surface mesh nodes on the blade tip can move in the radial direction to explore the topology-free tip design beyond tip cavities or squealers investigated before. In the adjoint based shape optimisation method, Weighted Inverse Distance morphing method is used to deform the surface mesh and interior volume mesh. To guarantee the mesh quality and minimize the unfavourable effects of decreased mesh quality on the accuracy of flow solution, mesh was smoothened by the Laplacian smoothing method. This gradient-based adjoint method creates a highly flexible design space without prescribed geometry. Before the optimisation started, an initial study of the flat-tip design with a tip gap of 1.5% span was carried out. A carved-in surface design was obtained by the optimisation process. The optimum design indicated a 0.2% improvement in efficiency and 7% reduction of tip leakage mass flow compared to datum design. Vortical flows in the tip gap were studied and analysed among the baseline, optimum and conventional squealer designs. Compared to the conventional squealer, it was shown that the optimum suppresses the harmful vortices while preserving the helpful scraping vortex in the tip gap. The smoothness factor is numerically tested so that the effects of smoothing on the optimal tip shape can be demonstrated.

Fluidic oscillation generated, hybrid synthetic jet dynamics in a fluid filled cylindrical duct

William B. Zimmerman[1], Pratik D. Desai[2], Julia M. Rees[3]

[1]Department of Chemical and Biological Engineering, University of Sheffield
[2]Perlemax Ltd., Sheffield UK
[3]School of Mathematics and Statistics, University of Sheffield

Hybrid synthetic jets, sometimes generated by fluidic oscillators, have many varied uses that stem from their dual features of periodic oscillations and net positive displacement. A new fluidic oscillator of this type, the Desai-Zimmerman fluidic oscillator (DZFO), created for microbubble generation. It features much less internal dissipation due to relatively larger internal features and stronger pulses, while easier to mass manufacture than earlier classes of jet diversion fluidic oscillators. An experiments showing pressure traces within and downstream of the DZFO, from high temporal resolution pressure sensors, are analysed by Fourier power spectrum, among other the techniques. All signals are dominated by the fundamental mode, with the power in the first harmonic practically negligible, but an appreciable power observed in the second harmonic. Surprisingly, the power spectrum and time series data in the downstream duct from one outlet of the DZFO conserves the energy integral of the pressure signal, at distances 100, 200, and 400 diameters down the cylindrical duct relative to the pressure signal at the outlet. Turbulent and laminar boundary layer theories suggest much higher dissipation rates due to the expected formation of fully developed flow certainly before 100 diameters downstream. It is hypothesized that the nearly "on-off" pulse switching of the hybrid synthetic jet is faster than the time scale for either laminar or turbulent boundary layers to develop, hence maintaining practically a slip boundary conditions at the duct walls. A simple functional analysis theory is developed for this downstream regime that leads to a non-trivial solution only when a nonlinear eigenvalue / boundary value problem is satisfied. A Rayleigh-Ritz method is proposed showing successive approximations in closed form, elucidating the modal structure of the candidate excited axisymmetric mode associated with the hybrid synthetic jet motion.

Dynamic Leidenfrost effects: Computational modelling to predict transitions in drop impact

Peter Lewin-Jones, James Sprittles and Duncan Lockerby

University of Warwick

When a liquid drop is placed gently on a sufficiently hot surface it is able to levitate on its own evaporative vapour cushion. This 'Leidenfrost effect' is well understood and has been the focus of much research, motivated by its importance for numerous industrial applications where the dramatic decrease in thermal conductivity caused by it is often detrimental. Notably, however, many technologies, such as spray cooling, actually involve the impact of droplets in Leidenfrost conditions. Recently, experimental studies have probed the dynamic Leidenfrost effect, where droplets can be forced into contact when impact speeds are large enough, uncovering several interesting modes of contact and discovering new unexpected effects, such as an oscillating film height in certain regimes. To provide new insight into the physical mechanisms involved in such phenomena and as an important predictive tool, we have developed a novel computational model for the dynamic Leidenfrost process. This uses lubrication theory to model the evaporated vapour, and the finite element method to solve the Navier-Stokes equations in the drop, implemented in the open source library oomph-lib and has passed a number of benchmarking tests. Our model enables us to explore the parameter space of impacting velocity and solid surface temperature, and probe different regimes of contact and vapour film behaviour, with the aim of predicting the critical impact speed at which contact will occur.

Time-dependent 3D dynamics in viscoelastic pressure-driven channel flow

M. Lellep[1], M. Linkmann[2] and A. Morozov[1]

[1]SUPA, School of Physics and Astronomy, University of Edinburgh, UK
[2]School of Mathematics and Maxwell Institute for Mathematical Sciences, University of Edinburgh, UK

Dilute polymer solutions do not flow like Newtonian fluids. Their flows exhibit instabilities at very low Reynolds numbers that are driven not by inertia, but rather by anisotropic elastic stresses. Further increase of the flow rate results in a chaotic flow, often referred to as purely elastic turbulence. The mechanism of this new type of chaotic motion is poorly understood. In this talk we present the first coherent state in purely elastic parallel shear flows. We consider a model shear-thinning viscoelastic fluid driven by an applied pressure gradient through two- and three-dimensional channels. By starting from a linearly unstable mode recently discovered by Khalid et al. (Khalid et al., arXiv:2103.06794 (2021)) at very large flow rates and very low polymer concentrations, we demonstrate the existence of 2D travelling-wave solutions in such flows. We show that this state sub-critically connects to significantly higher values of polymer concentration and lower flow rates (Morozov, arXiv:2201.01274 (2022)), rendering travelling-wave solutions experimentally relevant. Upon embedding the 2D coherent states in a 3D domain, we observe the emergence of a time-dependent, turbulent-like state, that becomes more complex with increasing Weissenberg number. We perform extensive characterisation of the ensuing dynamics and demonstrate its strong connection to purely elastic turbulence and exact coherent structures known to organise Newtonian turbulence.

Analysis of the new unstable mode in supercritical fluids

Benjamin Bugeat, Pietro Carlo Boldini and Rene Pecnik

TU Delft

A new unstable mode was recently discovered by Ren et al. (JFM, 2019) in the boundary layer flow over a flat plate for supercritical fluids. Its origin, and consequently its link to supercritical fluids, are however unclear. The aim of this presentation is to shed light on these matters.

In the presence of temperature gradients, supercritical fluids can exhibit strong variations of thermodynamic properties, in particular of density and viscosity. This can be accounted for by considering the Navier-Stokes equations in the limit of zero Mach number, ignoring acoustic effects while allowing density to vary with temperature. We carry out an inviscid linear stability analysis that indicates the inflexional nature of the instability observed by Ren et al. for finite Reynolds numbers. We then theoretically show that it is associated with the existence of a generalised inflexion point (GIP), produced by a minimum of kinematic viscosity in the base flow profile.

To test the hypothesis that the profile of kinematic viscosity - rather than other varying properties in supercritical fluids - is the essential feature responsible for this instability, different simple models are studied, allowing the density and dynamic viscosity to be constant or variable depending on the cases. Plane Couette flow and a boundary layer flow over a flat plate are considered. Results support the initial hypothesis and furthermore allow us to characterise the instability, showing the pertinent scales at play in the growth rate, cut-off frequency and phase speed. Furthermore, this analysis reveals why the plane Couette flow of a supercritical fluid can be stable despite the presence of a GIP.

The universality of this instability is eventually discussed beyond supercritical fluids, opening perspectives for shear flows of miscible or non-Newtonian fluids.

Evolution of and deposition from an evaporating sessile annular droplet

L. M. Mills, H.-M. D'Ambrosio, S. K. Wilson, A. W. Wray, B. R. Duffy, K. Sefiane

Department of Mathematics and Statistics, University of Strathclyde, Glasgow
Institute for Multiscale Thermofluids, School of Engineering, University of Edinburgh, Edinburgh.

The vast majority of previous work on evaporating sessile droplets has focused on the case of an axisymmetric droplet with a single, circular contact line. In the present work, we formulate and analyse a mathematical model for the evaporation of an axisymmetric annular droplet with two circular contact lines. Not only is the evaporation of annular droplets of intrinsic scientific interest in its own right, but it also arises in several practical and industrial contexts, such as the evaporation of a droplet in a well, which occurs in the manufacturing of organic light-emitting diode (OLED) displays (see, for example, D'Ambrosio et al. [1]), and in the context of a droplet evaporating on a patterned substrate (see, for example, Schafle et al. [2]). Numerical and asymptotic solutions for the concentration of vapour in the atmosphere are described and used to determine the local, and hence the global, evaporative flux from the droplet in the diffusion-limited regime. The evolution, and therefore the lifetime, of the droplet in various modes of evaporation, as well as the nature of the deposit left behind on the substrate after the droplet has entirely evaporated, are described both in the diffusion-limited regime and in the special case of a spatially-uniform evaporative flux.

The first two authors acknowledge funding from the Engineering and Physical Sciences Research Council (EPSRC) via Research Studentships.

[1] D'Ambrosio, H.-M., Colosimo, T., Duffy, B.R., Wilson, S.K., Yang, L., Bain, C.D., Walker, D.E., "Evaporation of a thin droplet in a shallow well: theory and experiment", J. Fluid Mech. 927, A43 (2021).
[2] Schafle, C., Brinkmann, M., Bechinger, C., Leiderer, P., and Lipowsky, R. "Morphological wetting transitions at ring-shaped surface domains". Langmuir, 26, 14, 11878–11885, (2010).

The Long Cross-Over Dynamics of Capillary Imbibition

Élfego Ruiz-Gutiérrez[1,2], Steven Armstrong[1], Simon Lévêque[3], Célestin Michel[3], Ignacio Pagonabarraga[4,5,6], Gary G. Wells[1], Aurora Hernández-Machado[5,7] and Rodrigo Ledesma-Aguilar[1]

[1]School of Engineering, University of Edinburgh, The King's Buildings, Mayfield Road, Edinburgh EH9 3JL, UK
[2]School of Engineering, Newcastle University, Stephenson Building, Newcastle upon Tyne NE1 7RU, UK
[3]Institut Universitaire de Technologie de Lannion, Rue Édouard Branly, 22300 Lannion, France
[4]Centre Européen de Calcul Atomique et Moléculaire (CECAM), École Polytechnique Fédérale de Lausanne (EPFL), Batochimie, Avenue Forel 2, 1015 Lausanne, Switzerland
[5]Departament de Física de la Matéria Condensada, Universitat de Barcelona, Martí Franqués 1, E08028 Barcelona, Spain
[6]UBICS University of Barcelona Institute of Complex Systems, Martí i Franqués 1, E08028 Barcelona, Spain
[7]Institute of Nanoscience and Nanotechnology (IN2UB), 08028 Barcelona, Spain

Spontaneous capillary imbibition is a classical problem in interfacial fluid dynamics with a broad range of applications, from microfluidics to agriculture. We present new experimental and theoretical results of the spontaneous capillary invasion of dry capillary tubes by viscous liquids. Here we study the duration of the cross-over between an initial linear growth of the imbibition front to the diffusive-like growth limit of Washburn's law. We show that local-resistance sources, such as the inertial resistance and the friction caused by the advancing meniscus, always limit the motion of an imbibing front. Both effects give rise to a cross-over of the growth exponent between the linear and the diffusive-like regimes. We show how this cross-over is much longer than previously thought – even longer than the time it takes the liquid to fill the porous medium. Such slowly slowing-down dynamics is likely to cause similar long cross-over phenomena in processes governed by wetting.

Nonspherical Particle Interactions in Forced Isotropic Turbulence

Jacob Anderson

University of Leeds

The behaviour of particles suspended in a fluid affects the efficiency and safety of many industrial flow systems. The ability to control this behaviour is therefore desirable-particularly in the nuclear industry where particle-laden flows are frequently encountered. Understanding which key parameters encourage or discourage agglomeration can offer avenues for industry to improve their current processing methods. The present work investigates the agglomeration of particles in periodic boxes of forced homogeneous and isotropic turbulence. The Taylor-Reynolds number, particle shape, Hamacker constant and coefficient of restitution are among the parameters that are investigated. The particulate-phase is modelled using an immersed boundary technique which is modified to allow for agglomeration due to electrochemical forces. The fluid-phase is modelled using highly accurate direct numerical simulation through the spectral element solver Nek5000. The turbulence is sustained by a Fourier-space stochastic forcing function, which allows control of the turbulence kinetic energy and integral length scale through a prescribed energy spectrum. Particle nonsphericity is included directly through the particle mesh. Interactions between nonspherical particles and their agglomeration behaviour will be discussed.

Experiments on Transonic Shock-Vortex Interactions

Conor Hocking[1], Kshitij Sabnis[1], Holger Babinsky[1] and Clyde Warsop[2]

[1]University of Cambridge
[2]BAE Systems

Vortices in high-speed flow applications, such as those generated by canards on supersonic aircraft, often interact with shock waves over wings or in engine intakes. When a vortex encounters a strong shock, it may break down. Understanding the conditions under which vortex breakdown occurs is important for accurately modelling the flow on high speed aircraft. This experimental research explores the interactions of wing tip vortices with normal shock waves at Mach 1.3 and 1.5, using high-speed schlieren imaging. Two wind tunnel setups are used, which generate different local streamwise pressure gradients downstream of the normal shock, to compare how this affects the nature of the shock-vortex interaction and the likelihood of vortex breakdown.

Schlieren images of the interaction at Mach 1.3 show that the shock wave bulges upstream in the region where the vortex meets the shock. The strongest vortices, with a high maximum tangential velocity, cause the largest shock-bulging, which corresponds to vortex breakdown. At Mach 1.5, vortex breakdown is not observed (likely because the tangential velocity in the vortex is too low), but schlieren images demonstrate the presence of two different types of shock-wave bulging. Changing the pressure gradient downstream of the shock is found to affect the intermittency of vortex breakdown and the dominant type of shock wave bulging. The flow physics which may have caused these differences will be discussed and suggestions for further experiments on shock-vortex interactions will be presented.

Targeted extraction of exhaled breath to reduce potential virus exposure

Joshua Finneran and Paul Linden

Department of Applied Mathematics and Theoretical Physics, University of Cambridge

Airborne transmission of COVID-19, and other respiratory diseases, occurs though inhalation of virus laden particles that are small enough to be suspended and carried with the exhaled breath of an infected individual. The key mitigation strategy for airborne transmission is ventilation, with the goal being to minimise the concentration of exhaled breath in the occupant's breathing zone to minimise potential exposure. This can be achieved by diluting or displacing exhaled breath. Hence, an understanding of the relationship between the ventilation flows and flows due to buoyancy sources (e.g. body heat, breath, radiators) is essential to predict the room's distribution of exhaled breath, and so transmission risk.

The relative source strength of a rising body plume compared to the buoyant jet of breath (approx. 19:1 ratio) leads to a tendency for exhaled breath to settle at a height lower than the ceiling, forming a so-called lock-up layer. The lower height of this layer means it is more likely to be in the breathing zone of other occupants, or the layer can be more easily mixed into the breathing zone.

We present an experimental investigation of the benefits of targeted extraction of the lock-up layer, rather than placing ventilation outlets at the ceiling, for example. The experiments take place in scaled rooms in a water bath. We observe that, for a given ventilation rate, extraction from the lock-up layer leads to (1) more exhaled breath being extracted, (2) lower average concentration of exhaled breath in the room, and (3) reduced vertical height of the lock-up layer. We find that as the size of the lock-up layer reduces, the dominant mechanism for determining its size is the entrainment from the fountain as the rising breath enters the upper layer in the room.

Effects of surface roughness on the propulsive performance of a flapping foil

R. Vilumbrales-Garcia, M. Kurt, G. D. Weymouth and B. Ganapathisubramani

University of Southampton

Flapping foils propulsive capabilities has been widely documented in past studies, but other aspects that could lead to performance augmentation, such as the effects of surface roughness, are not well understood.

In this study, three NACA 0012 foils with different surface roughness have been built using hemispherical bumps: smooth foil and two different roughness topographies. The first is sparsely rough with 36% area coverage while the second is densely rough with 75% area coverage. Experiments have been conducted in a water flume, under sinusoidal pure pitching 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 have been recorded in terms of forces and moments. PIV acquisition has been carried out to link together any potential benefit to the flow characteristics. Further analysis will include details on how the rough elements can alter the development of the flow over the foil, and how can that impact on its propulsive performance, both in terms of thrust production and efficiency.

Direct Statistical Simulation: an alternative approach to turbulence

Kuan Li

Applied Math, University of Leeds

Direct Statistical Simulation (DSS) is a new theoretical and numerical framework for describing and solving turbulent fluid and MHD problems in the paradigm of statistical physics and fluid dynamics. DSS seeks to approximate the spatio-temporal evolution of the probability distribution of these systems by a set of low-order statistical equations, namely cumulant equations. The low-order cumulants are smooth in space and therefore require many fewer degrees of freedom for numerical computations than those required for direct numerical simulation (DNS). DSS is hence expected to access turbulent dynamical systems in the extreme dynamical regimes beyond the reach of DNS. However, conventional approaches in DSS also suffer a significant challenge, known as the curse of dimensionality, which usually strongly limits the application of this method up to the 2nd-order cumulant only.

We present a series of recent theoretical and numerical developments in DSS, that successfully bypass the curse of dimensionality of the conventional approaches and enable us not only to describe these problems with an extreme efficiency but also to yield the numerical solutions of the low-order cumulant equations with a high statistical accuracy up to the 3rd order. Furthermore, we have simplified the statistical closure, CE2.5, suited for computing the third-order cumulants. In this presentation, we demonstrate the effectiveness and advantages of this new method by solving two representative turbulent problems i) a 2D turbulent fluid dynamical system driven by a Kolmogorov force and ii) a turbulent plasma model, modified Hasegawa-Wakatani in 2-dimensions. We believe that our approach in DSS is now directly applicable for solving 3D turbulent fluid dynamical and MHD problems in extreme dynamical regimes.

On the spatial evolution of the largest waves propagating over variable coastal bathymetry

Vasileios Bellos, Ioannis Karmpadakis and Chris Swan

Imperial College London

Coastal processes have historically attracted engineering and scientific attention due to their influence on the natural and built environment. Despite the many advancements in the description of wave evolution in the coastal zone, there remain significant challenges to be addressed. One of these relates to the physical processes that lead to the formation of the largest waves and their propagation over coastal bathymetry. The present study aims to address this question by providing physical insights into the key physical mechanisms involved. These are primarily manifested as wave amplifications due to nonlinear interactions and dissipation due to wave breaking. Importantly, the competing nature of these mechanisms and their dependence on a variety of environmental parameters increase the complexity of the problem. To address this, a thorough experimental campaign was designed to isolate the contributions of individual parameters, such as the steepness of incident waves, effective water depth and seabed slope. The variation of these parameters was conducted using realistic random wave conditions which capture the stochastic nature of water waves. Importantly, a sufficiently large number of repetitions was realised for each test condition with the aim to minimise statistical uncertainty and investigate extreme events. The experimentally generated waves were allowed to propagate over the coastal bathymetry while being sampled using a densely-packed array of sensors. The fine resolution in spatio-temporal measurements, an efficient wave tracking algorithm and a novel sea-state correlation technique have allowed in-depth insights into the evolution of large wave events. In coupling these experimental measurements with numerical simulations, this study provides a better understanding regarding the origin of the largest waves in storms of varying severity, water depth and bathymetric configuration.

Experimental characterisation of the 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 results in a Taylor-Proudman constraint to its liquid core, which creates an imaginary cylindrical boundary (known as tangent cylinder, TC) inside the liquid core tangent to the solid core. 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 experimentally. 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 sulfuric acid from the bottom and cooling it from the outside (buoyancy), placing it atop a rotating platform (rotating force) and then placing the entire platform inside the bore of a magnet (magnetic force). Sulfuric acid is used as the working fluid as it has a high electrical conductivity and is also transparent which enables us to utilize particle image velocimetry technique. We show that the TC indeed separates two regions of the flow: while the inner region displays Rayleigh-Benard plumes subject to thermally-driven azimuthal wind, the outer zone is driven by the homogeneous cold temperature on the outer boundary of the fluid vessel. This new setup enables us to highlight the detailed topology of both flows and their evolution from the well-known onset states towards a chaotic behaviour in more supercritical regimes.

Effects of pressure gradient history on rough-wall turbulent boundary layers

Thomas Preskett and Bharathram Ganapathisubramani

University of Southampton

Many studies have been under taking the effects of each in isolation however few have looked at the combined effects of these. In this experimental study we present results on flow features of a turbulent boundary layer over a rough wall. These measurements taken in Southampton University's 12m Boundary Layer tunnel taking pressure measurements of the wall pressure history and hot wire measurements of the boundary layer profiles. A series of non-equilibrium pressure gradients are imposed on a rough wall boundary layer using a NACA0012 aerofoil of chord 1.25m mounted in the freestream. Therefore, by varying the angle of attack and the height above the tunnel floor the pressure distribution imposed onto the boundary layer will change. Hot wire data will allow comparison of the effect of different pressure gradient histories on a given rough surface to be compared. Furthermore, the wall shear stress at the measurement location will be measured using a drag balance. Using this data both boundary layer profiles and turbulence spectra will be presented comparing the different boundary layers, thus allowing conclusions to be drawn on the effect of the flow history that a boundary layer experiences and how this impacts the boundary layer characteristics.

Effect of winglet serration wavelength on wingtip vortices

Kshitij Sabnis, Pascal Gehlert and Holger Babinsky

University of Cambridge

Vortices generated at the tips of aeroplane wings are advected downstream, where they can pose significant risk to other nearby aircraft. One area of active research to mitigate the effect of these wingtip vortices involves geometric modifications to the top edge of the winglet, from which the vortex is shed. In particular, serrating this edge has been shown to substantially reduce the peak tangential velocities of the wingtip vortex even 150 chord lengths downstream of the wing. Recent experiments have indicated that a key parameter governing the effectiveness of these modifications might be the wavelength of the serrations. It is believed that this wavelength determines whether a single vortex is shed from the winglet, or whether there are two separate coherent vortices which merge further downstream.

In order to better understand the underlying flow physics, several wing models are constructed which are otherwise-identical but have different wavelength and amplitude serrations on the shedding edge of the winglet. Each wing model is towed through a water tank at a chord-based Reynolds number of 100,000 and the generated tip vortex is studied. Dye-flow visualisation highlights differences in wingtip vortex topology while stereoscopic particle image velocimetry provides quantitative characterisation of the vortex development. These experiments establish a critical wavelength beyond which the tip vortex is substantially weakened and provide a deeper fundamental understanding of the vortex generation and merging processes.

The Development of a Photoelastic Force Balance

B. McLaughlin, J. Lawson and B. Ganapathisubramani

University of Southampton

Photoelasticity is a method of stress analysis that takes advantage of a material property of many transparent polymers, birefringence. In birefringent materials, light rays get resolved into component waves that vibrate parallel to the direction of the principal stresses. The velocity of each wave will be dependent upon the magnitude of the principal stresses. Using a polariscope to bring these components into a common plane allows optical interference. This interference creates fringe patterns, which are proportional to the stress in the polymer. By combining this phenomenon with a floating element balance, lift from a NACA 0015 aerofoil was measured optically. Polyurethane pads were used as the photoelastic material with embedded pins to transfer the applied load. The pads are illuminated by LEDs mounted in the balance, and two low-cost USB cameras to capture the fringe patterns. A series of known forces were applied to create a calibration matrix, which measured the lift curve of the wing at two speeds. The measurements of the photoelastic balance are compared with measurements taken from an ATI Mini 40 sensor. The photoelastic balance has shown to have good agreement with the ATI sensor.

Alfven waves excitation at low magnetic Reynold number

S. Lalloz, A. Potherat, L. Davoust and F. Debray
SIMaP, Coventry University, LNCMI (CNRS)

Alfven waves are magneto-mechanic waves which consists of an oscillation of both magnetic and momentum disturbances propagating along magnetic field lines. They were first theorised by Alfvén in 1942 and discovered seven years later by Lundquist. After 70 years of research, due to the difficulty of reproducing these waves in laboratory, the knowledge of their dynamics remains elusive. In particular, their relevance at the low magnetic Reynolds numbers typical of most liquid metal experiments is controversial.

Here, a revisited version of the Flowcube device, initially developed by Klein, is used to investigate these waves in liquid metal under a magnetic field (up to 10 Tesla). With this device, the wave is forced by injecting an AC current through electrodes located at the top and bottom plates of its rectangular vessel (also called Hartmann plates). This current interacts with the magnetic field to induce azimuthal Lorentz forces and thus generate a vorticity oscillation centred on top of each electrode in use. To observe the waves, these plates are equipped with electric potential probes symmetrically placed on both plates.

The study focuses on three key questions. The first one is whether electrically driven Alfven waves can be sustained at low magnetic Reynolds number, ie when the magnetic diffusion term may potentially prevail over the advective one. Second, how, and in which regimes do they compete with the diffusive nature of the Lorentz force, which, under the current paradigm, prevails at low Rm. Last, can Alfven waves reach a nonlinear regime, relevant to nonlinear processes occurring in plasmas and astrophysical flows? We will show preliminary results indicating that the Lorentz force may indeed underpin wave propagation at low Rm, whilst retaining its diffusive nature and that a nonlinear behaviour may ensue provided the waves are sufficiently excited.

Drag Reduction of a Square Cylinder Using Deep Reinforcement Learning

P. Mudiyanselage, Z. Khatir and F. Gueniat

Birmingham City University

Active flow control (AFC) coupled with deep reinforcement learning (DRL) techniques is rapidly growing in the field of fluid dynamics. From an engineering stand, the ability to control the flow field to improve its efficiency is very crucial. Over the last decades, the field of AFC has been a major area of research in fluid dynamics, however compared to passive flow control it remains very challenging. Challenges arise as more robust design-efficient actuators and sensors are needed and often AFC techniques depend on unrealistic scenarios where a complete knowledge of the environment is provided beforehand. However, recent applications of DRL have demonstrated its effectiveness in performing AFC with scarce data, so that a detailed knowledge of the dynamic is not required. Hence, this paper presents the investigation of a DRL framework to a closed loop active control for drag reduction in a 2D square cylinder wake flow at low to high Reynolds numbers. AFC is performed by an actuator placed at the trailing edge of the cylinder and the DRL agent interacts with the environment by manipulating the actuations. The work presented is fully data-driven and depends on scarce data from sensors to find an efficient control law for drag reduction. For medium Reynolds the results show that the agent can learn a control law that gives a reduction of 32% in average drag. Analysing the flow field, it is shown that the recirculation bubble length is increased compared to no actuation, hence delaying the generation of vortices. Since no prior knowledge of the environment is needed, this approach is suitable for experimental configurations. These results shows that AFC combined with DRL pave the way towards developing new robust control laws for real industrial applications. Results for high Reynolds number will be presented.

Bio-inspired flexible pillar flow sensor arrays to visualise flow characteristics over an aerofoil downstream of a spanwise variable gust generator

Alecsandra Court, Omar Selim and Christoph Bruecker

City, University of London

Using a 2D array of bio-inspired flexible pillar sensors embedded within the suction side of a NACA0012, the effect of upstream flow disturbances can be visualised and analysed on-line for gust control purposes. The flow over the aerofoil in disturbance conditions is investigated and the relevant flow characteristics are recorded with an on-board camera. The sensors are flexible pillar structures with fluorescent tips which are optically tracked in real-time acting as "digital tufts", the motion of the tips can be directly correlated with flow velocities giving a real-time picture of local flow characteristics. The setup is combined with a pneumatically actuated gust generator with a spanwise distribution of spoiler-like flaplets able to deploy independently to generate a variation of disturbance patterns for investigation. This is a combination of two research projects at City, University of London aiming to develop nature-inspired solutions for flow monitoring and control in real-life practical complex flows situations.

The aim of this work is to be able to identify specific flow phenomena which occur in controlled gust conditions. By setting the aerofoil at a critical pre-stall angle of attack, such as Clmax, and initiating a discontinuous gust, the aerofoil is sent into temporary stall. Thus, allowing the identification of flow phenomena such as reversed flow and stall cells. This is an intermediate step towards implementing this system on a wing, that could be used to identify these phenomena in-situ for real-time flow monitoring. Applications include performance enhancement of an aerodynamic system, such as, stall aversion on aircraft or power optimisation for wind energy applications.

High-fidelity-simulation assisted optimisation of a two-dimensional airfoil

Russell Quadros[1], Lionel Cheng[2], Gabriel Staffelbach[2], Mladen Banović[3], Arthur Stück[3] and Jens-Dominik Müller[1]

[1]Queen Mary University of London
[2]Centre Europeen de Recherche et de Formation Avancee en Calcul Scientifique
[3]German Aerospace Center-Dresden

Aerodynamic shape optimisation minimises an objective function of interest (such as lift, drag, etc) by modifying the typically large number of parameters that control the shape. The adjoint approach is the most effective method to compute the sensitivity of the objective function w.r.t. a large number of parameters. RANS adjoint solvers are now widely available, but inherit significant limitations in predicting challenging flow features. LES has been demonstrated to be effective in predicting flow separation in high- lift configuration [1]. However, adjoint solutions of LES blow up if the LES resolves the chaotic nature of the flow [2].
A possible way to involve LES in shape optimization is by tuning the eddy-viscosity field in the RANS model to match the time-averaged LES solution, and then use the frozen eddy-viscosity approach to optimize the geometry. This often results in poor gradient accuracy for shape optimization, and the number of LES runs required towards obtaining the optimal shape are prohibitively expensive [3]. As an alternative, the coefficients of the turbulence model could be adjusted [4] such that the RANS flowfield reproduces the time-averaged LES solution. This allows to include the variation of the turbulence model in the surface sensitivity. This approach been employed as a basis for machine-learning RANS models with better prediction of flow separation for aerofoils [4]. However, to our knowledge, it has not yet been used in shape optimization.
As a first result we will show for the U-Bend case a clear advantage of adaptation of of the model coefficients for the production and the destruction terms to match the LES field [5] over modifying only the production term [4]. We then present the adjoint-based optimisation of the U-Bend aided by the LES-informed RANS model. Optimisation results will be presented comparing shapes, flow structures and pressure drop for the baseline RANS model, a one-time coefficient matching for the baseline geometry, and a more frequent updating of the LES and S-A coefficients to assess the generality of the RANS coefficient set.

[1]Z.J. Wang and Salman Rahmani, Implicit large Eddy simulation of the NASA CRM high-lift configuration near stall, Computers and Fluids, Vol. 220, 2021.
[2]J. Larsson and Q. Wang, The prospect of using large eddy and detached eddy simu-lations in engineering design, and the research required to get there, Phil. Trans. R. Soc. A, Vol. 372, 2014.
[3]G. Alessi and T. Verstraete and L. Koloszar and B. Blocken and J.P.A.J. van Beeck, Adjoint shape optimization coupled with LES-adapted RANS of a U-bend duct for pressure loss reduction, Computers and Fluids, Vol. 228, 2021.
[4]A. P. Singh and S. Medida and K. Duraisamy, Machine-Learning-Augmented Predictive Modeling of Turbulent Separated Flows over Airfoils, AIAA Journal, Vol. 55, 2017.
[5]R. Quadros, L. Cheng, G. Staffelbach, M. Banović A. Stück and J.-D. Müller, LES-Aided Shape Optimisation of U-Bend Channel, ECCOMAS 2022.

Robotic Inspection of Pre-filled Medical Syringes

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

University of Leeds Centre for Doctoral 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.

Breathing simulator to assess effectiveness of masks and control the spread of Covid-19 In quiescent surroundings

D. Pickles, H. Zare-Behtash, A. Cammarano and R. B. Green

University of Glasgow

The emergence of a highly transmissible strain of the severe respiratory syndrome coronavirus 2 (SARS-CoV-2) in 2019 (otherwise known as COVID-19) threatened the health and public safety of humans around the world. To control the spread of the virus, social distancing and face coverings became a requirement globally. This created a need for testing that looks into indoor environments and face mask use. The present work aimed to characterise the air flow patterns of exhaled air for a range of daily breathing conditions produced by an innovative linear actuator-driven breathing test rig characterised using Laser Doppler Anemometry (LDA), Particle Image Velocimetry (PIV) and smoke flow visualisation. The realistic mouth and head profile of the test rig allowed the effectiveness of a wide range of masks (FFP1, FFP2, FFP3, Multi-layer cloth (MLC) masks, cloth masks, face-shields) to be tested over a range of breathing conditions. The results showed that the test rig designed could reliably recreate the flow field when breathing. LDA results, in phase with the movement of the piston, showed that, if worn correctly, the majority of masks prevent the exhaled air from forming a jet that extends away from the subject. At high tidal volumes and frequencies, the exhaled air forms low velocity jets at the areas where masks fit poorly (i.e around the sides of the mask and vertically upwards near the nose). The breathable balaclava, whilst significantly reducing the velocity of the exhaled air, did not prevent it forming a jet which could act as a transmission pathway.

Application of the Fully Lagrangian Approach to unsteady polydisperse gas-droplet flow simulations with inter-phase coupling

Chris Stafford and Oyuna Rybdylova

Advanced Engineering Centre, School of Architecture, Technology and Engineering, University of Brighton, Brighton BN2 4GJ, UK

The need to understand droplet behaviour within industrial spray systems has prompted the development of a range of numerical approaches for the computational simulation of such processes. However, in order to obtain a sufficient level of accuracy these simulations typically require a large number of droplets, leading to long computational runtimes and the need to store large amounts of data. The Fully Lagrangian Approach (FLA) offers an attractive way to address this problem by modelling the droplet phase as a continuum whilst retaining the intrinsic detail of individual droplet behaviour, meaning that only a subset of representative trajectories are required to reconstruct the droplet phase mean-field variables to the necessary level of accuracy, and the computational demand is therefore substantially reduced. Recent work has combined the FLA methodology with the statistical learning technique of kernel regression to obtain a physically consistent representation of the droplet phase, and the present work extends the applicability of this procedure to the treatment of a wider class of flows. In particular, inclusion of a droplet size parameter within the FLA formulation enables the modelling of polydisperse droplets that undergo evaporation, thereby supplying a wealth of information about the droplet distribution in space, velocity, and size. Furthermore, it is demonstrated that the kernel regression procedure is able to numerically generate the initial conditions required by the FLA within arbitrarily complex flows. The efficacy of this approach is illustrated for the case of transient flow around a cylinder using OpenFOAM, and results for the statistics of the droplet size distribution that is naturally produced by the generalised FLA are presented. The ability of the FLA to provide accurate source terms for inter-phase coupling is also analysed, and serves to highlight the suitability of the approach for the simulation of industrially relevant spray systems.

Hydrodynamics of flow over hydrofoils with generic roughness elements

Melike Kurt, Rodrigo Vilumbrales Garcia, Gabriel Weymouth, Bharathram Ganapathisubramani

University of Southampton

The hydrodynamic function of shark denticles has been a subject of interest in recent years predominantly due to their drag reducing and lift enhancing effects on the flow. These effects are often examined by using bio-mimicry and replicating the morphological properties of the shark denticles. However, it is unclear whether the bio-mimicked shape of the denticles has the main role or it is the generic roughness on the surface that causes a re-arrangement in the flow and enhances the hydrodynamic performance. To test this hypothesis, we carry out experiments over a NACA0012 hydrofoil with hemispherical roughness elements that has similar dimensions to that of previously examined denticles. A parameter sweep is performed where different surface roughness distributions (i.e. coverage area) of hydrofoil at varying angles of attack and flow speeds are considered. Using flow visualization and force measurements, the results of this parameter sweep are presented in comparison with the flow over a smooth surface foil. The data allows us to better understand the effect of such roughness elements on the flow physics and the resulting hydrodynamic performance.

Performance Investigation of the Vertical Axis Wind Turbine with Slotted Aerofoil Configuration

Suleyman Kose, Derek Ingham, Lin Ma and M Pourkashanian

Energy 2050, Department of Mechanical Engineering, Faculty of Engineering, University of Sheffield, Sheffield, UK

Obtaining the energy needs from renewable and clean sources is an extremely important issue and wind turbines are renewable energy conversion systems that are widely used in this context. Although vertical axis wind turbines (VAWTs) have many advantages over their horizontal axis wind turbines (HAWTs), their energy efficiency is slightly lower than that of HAWTs. In this study, a detailed aerodynamic investigation of the VAWTs with slotted aerofoil configuration is performed to enhance the power generation at different tip speed ratios. The impact of the slotted aerofoil with different slot gap, slot angle and slot position on the overall performance of the VAWT under different operating conditions is investigated in detail. To illustrate the advantages of slotted aerofoils, a two-bladed VAWT is analysed using Unsteady Reynolds-Averaged Navier Stokes (URANS) simulations with SST k-w turbulence model with the intermittency function. The results show that at low tip speed ratios, noticeable improvement in the torque coefficient is produced in the upstream and downstream regions of the turbine. However, the aerodynamic performance is enhanced only slightly in the upstream region of the turbine at high tip speed ratios while a high disruptive effect is achieved in the downstream region of the turbine with the slotted aerofoil. In addition, it is found that, depending on the blade pitch angle, the enhancement achieved by the slot configuration can increase even more. Finally, the best slot aerofoil configuration for the best aerodynamic performance enhancement of VAWT turbines is observed by investigating the fluid flow contours in detail.

Systematic modulation of bio-inspired micropatterning using temporally-arrested breath figure

Francis Dent and Sepideh Khodaparast

University of Leeds

Tailored microstructured surfaces engender enhanced properties relating to non-wetting, self-cleaning and antibacterial effects. Inspiration for such highly functionalised patterns can be drawn from examples prevalent in nature and effectively implemented through biomimetic engineering design and fabrication methods. Herein, we describe and characterise a microfabrication approach based on an adapted breath figure (BF) technique to produce patterned surfaces similar to the functional epicuticular nanostructured wings of cicada insects. The BF method is a fluid-based templating approach mediated by self-organised condensation droplets that has been used for scalable and energy-efficient surface patterning. However, predictable design and systematic control of BF patterns are limited due to strict temperature and humidity regulation and the complex dynamics of solvent use for passive cooling and polymerisation. In this work, we decouple condensation growth and polymerisation and negate solvent use altogether through the dual application of initiative cooling and deterministic UV curing of photosensitive polymers. Using in-situ optical analysis, we view the condensation kinetics in real-time, demonstrating scale-invariant patterns on substrates where high condensation nucleation density is achieved. By constraining pattern growth to the self-similar diametric power law of $D \propto t^1$, we fabricate homogenous films with feature sizes from hundreds of nanometres to tens of microns at a high packing coverage of 40%. Finally, we modulate the size of the surface structures attained via quasi-instantaneous curing of BF polymer films to create selectively patterned substrates; through spatial UV masking and discrete temporal arresting of BF patterns, we fabricate samples with gradient and multimodal sizing features.

Conjugate Heat Transfer Effect of Flow Boiling in Microchannels

M. Magnini and F. Municchi

University of Nottingham

We present a computational study of saturated flow boiling in non-circular microchannels. The unit channel of a multi-microchannel evaporator, consisting of the fluidic channel and surrounding evaporator walls, is emulated and the conjugate heat transfer problem is solved. Simulations are performed using OpenFOAM v2106 and the built-in geometric Volume Of Fluid method, augmented with self-developed libraries to include liquid-vapour phase-change and improve the surface tension force calculation. A systematic study is conducted by employing water at atmospheric pressure, a channel hydraulic diameter of Dh=229 um, a uniform base heat flux of q=100kW/m2, and by varying the channel width-to-height aspect-ratio and channel fin thickness in the range AR=0.25-4 and Wf=Dh/8-Dh, respectively. The effects of conjugate heat transfer and channel aspect-ratio on the bubble and evaporative film dynamics, heat transfer, and evaporator temperature are investigated in detail. This study reveals that, when the flow is single-phase, higher Nusselt numbers and lower evaporator temperatures are achieved for AR¡1. In the two-phase flow regime, the trends of the Nusselt number versus the aspect-ratio are mixed, although for smaller channel fins an ascending trend of Nu for increasing aspect-ratios is apparent. Nonetheless, due to conjugate heat transfer, Nusselt numbers and evaporator base temperatures follow different trends when varying the aspect-ratio, and channels with AR¡1 seem to promote lower evaporator temperatures than higher aspect-ratio conduits, despite exhibiting slightly worse two-phase convective heat transfer performances.

Resolvent analysis of low-inertia particles in a pipe flow

Rasmus Korslund Schlander, Stelios Rigopoulos and George Papadakis

Department of Aeronautics, Imperial College London

We extend the resolvent framework to particles with low inertia in turbulent flows. The particle velocities are modelled using the equilibrium Eulerian model, which is assumed to be valid for Stokes numbers up to 1. We analyse a vertical turbulent pipe flow with a Reynolds number of 5300 based on diameter and bulk velocity, for Froude numbers $Fr = -4, -0.4, 0.4, 4$ and Stokes numbers $St+ = 0 - 1$. A direct numerical simulation (DNS) for a pipe with a length of 7.5 diameters (D) is performed with the particles released uniformly at the pipe inlet. The particles are re-inserted from the outlet to the inlet in order to artificially increase the domain length and it is shown that the mean concentration profiles become self-similar around 12D downstream of the inlet, and these profiles are then used in the resolvent framework. The resolvent formulation can predict some of the physical phenomena observed in inertial particle flows such as localized high concentration due to the vortical centrifuge effect, turbophoresis and gravitational effects. It also reveals that the upward flow increases particle concentration in the log layer of the pipe. The downward flow increases concentration near the centre of the pipe: both features have been observed in previous Lagrangian simulations as well as experiments. The main effect of Stokes number on the modes seems to be amplifying smaller streamwise wavelengths, therefore, increasing local scale clustering of particles. Increasing the Stokes number increases turbophoresis. The effect of the direction of gravity was also observed using a simplified resolvent model and did not require any mean concentration profiles as input, which simplifies the analysis since no prior simulation or experiment is required for the model to work.

Rough capillary rise

Jack R. Panter and Halim Kusumaatmaja

Durham University

Capillary rise between rough surfaces plays a central role in an immense range of technologies and natural systems. For example, in biology, rough capillaries are highly adapted to enable efficient liquid transport and hydration mechanisms. In engineering and geoscience, invasion by liquids is vital to understand as a significant mechanism of deterioration in fractured materials. In physics and surface science, rough capillary rise is increasingly functionalised for innovations in microfluidics, power generation, and novel carbon capture technologies based on liquid infused surfaces.

Remarkably, there exists no fundamental theory to either predict or explain even the fundamental property of the equilibrium rise heights in rough capillaries. This is surprising given the maturity of the individual fields of capillary rise between smooth surfaces, and hemiwicking within a rough texture. This has led to a severe and erroneous assumption - that the coupling between these modes of liquid imbibition is trivial. Current apparent models are therefore much less understood than assumed, inaccurate, and ultimately of limited utility.

Here, we show how explicitly coupling capillary rise and hemiwicking challenges our conventional understanding and intuitions of wetting and roughness. Firstly, the critical contact angle for hemiwicking becomes separation-dependent so that hemiwicking can vanish for even highly wetting liquids. Secondly, the rise heights for perfectly wetting liquids can be different in smooth and rough systems, even with the same zero-degree contact angle. Finally, the raised liquid volumes are substantially increased in rough compared to smooth systems. Overall, to explain equilibrium rough capillary rise, we develop the Dual-Rise model. This model is analytic, predictive, valid for general roughness, and quantitatively accurate when compared to both experiments and simulations.

The internal structure of forced fountains

Jingzi Huang, Henry C.Burridge and Maarten van Reeuwijk

Imperial College London

Fountains arise when steady ejections of fluid are forced vertically from a localised source against an opposing buoyancy force — fluid travels outwards under the action of the source momentum for some distance before the buoyancy force causes the flow to return backwards towards the source. When the source and environmental fluids are miscible, and of similar density, significant mixing (or entrainment) of environmental fluid by the fountain flow has been evidenced to occur. Herein, the mixing process between the outward and return flows, i.e. the internal structure of the fountain, is studied using direct numerical simulations (DNS). The dynamics of the fountain internal structure is key in determining the fountain height and the total mass of the fountain supported by the forcing at the source. However, there is insufficient studies on the internal structure of fountain and therefore, less understanding on transfer of turbulent momentum and buoyancy. Consideration of the data from our simulations of a quasi-steady, so-called, forced fountain identifies that there at least two insightful definitions to determine the internal boundary between the outward and return flows; namely, these two flows being separated by either the loci of points of zero mean vertical velocity, or those over which zero mass transport occurs (i.e. the separatrix). By analysing the budgets of Reynolds averaged equations, we measure the mean-flow and turbulent transfers of volume, momentum and buoyancy between internal structure of fountain including consideration of a fountain cap region in which the flow reverses. We obtain entrainment coefficients appropriate for the mass transport across relevant boundaries, and further determine coefficients of transport for the turbulent fluxes, comparing these with other relevant flows. In so doing, we present new insights as to the complex internal dynamics of these canonical turbulent buoyant flows.

Natural wax crystallisation for biomimetic fabrication of hydrophobic coatings

Sepideh Khodaparast

University of Leeds

Water-repellent materials have found growing applications in diverse emerging technologies, ranging from self-cleaning fabrics to anti-bacterial coatings for medical devices. Significant non-wetting properties are often achieved by generating high density of micro/nanostructures on the surface to reduce liquid contact with the solid interface. Despite their growing applications, engineering approaches for manufacturing such materials still largely rely on top-down micromachining and/or nanofabrication techniques that are costly and lack scalability, material compatibility and pattern diversity. Efficient natural hydrophobic and superhydrophobic surfaces, on the other hand, exhibit diverse intricate hierarchical micro and nano features extended over large areas, thus providing generous source of model functional coatings. Guided by the inspiring patterns on leaves of plants and wings of insects, we use hydrophobic natural waxes to design and fabricate hierarchically rough surfaces with enhanced non-wetting properties. We use organic, non-polar solvents to achieve homogeneous liquid solutions of waxes as our working material. Upon evaporation of the solvent and subsequent wax crystallisation, micro and nanoscale structures spontaneously emerge in the thin layers of wax solutions coated on various substrates. Using scanning Electron Microscopy (SEM) and contact angle measurements we characterise the crystal shape/dimensions and surface wettability. We demonstrate that the final crystalline structure and wetting properties can be tuned by varying the choice of solvent, wax and substrate, as well as the rate of evaporation. Crystallisation of hydrophobic organic materials through solvent evaporation offers a scalable yet simple and inexpensive approach for fabrication of complex hierarchically patterned surfaces.

Shifting gears in liquid-liquid impact

Radu Cimpeanu

University of Warwick

In this talk I will describe recent experiments, analytical work and high performance computing efforts in tackling the canonical interfacial flow problem of a liquid drop impacting onto a liquid pool. Depending on the impact velocity, the drop may bounce, coalesce or splash, each scenario providing a beautiful multi-scale framework for the development of tools to aid our understanding of highly nonlinear flows. I will be focusing on bouncing and splashing scenarios in particular.

To begin with, consider millimetric drops impacting a deep bath of the same fluid that are generated using a custom syringe pump connected to a vertically-oriented needle. Measurements of the droplet trajectory are compared directly to the predictions of a quasi-potential model, as well as fully resolved unsteady Navier-Stokes direct numerical simulations. Both techniques resolve the time-dependent bath interface shape, droplet trajectory, and droplet deformation. In the quasi-potential model, the droplet and bath shape are decomposed using orthogonal function decompositions leading to a set of coupled damped linear oscillator equations solved using an implicit numerical method. The underdamped dynamics of the drop are directly coupled to the response of the bath through a single-point kinematic match condition, which we demonstrate to be an effective and efficient model in certain parameter regimes.

We then move to the opposite end of the impact velocity spectrum and discuss approaches suitable for high-speed flows in this context. The early time asymptotic treatment based on Wagner theory is tested against direct numerical simulations, with a focus on the characterisation of meaningful morphological features such as the location of the root of the ejected sheet, the shape of the latter, as well as velocities therein. The later stages of the impact (including formation of secondary droplets) are then interrogated numerically and a hybrid modelling framework is proposed.

Computational fluid dynamics driven mass transfer modelling for the prediction of CO2 corrosion in oil/water mixture pipeline flows

Udayraj Thorat, Michael Jones, Richard Woollam, Richard Barker, Harvey Thompson and Gregory de Boer

University of Leeds

Pipeline flows are the most economical method of transporting oil/water mixtures from crude wells to stations for separation and processing. Such pipelines are comprised of steel for practical engineering implementation and cost-related reasons. However, this results in an interaction between the chemical surface properties of steel and dilute electrochemical species carried in the oil/water mixture. This interaction results in the mass transfer of electrochemical species within the fluid boundary layer formed on the pipeline surface. This is well established as CO2 corrosion, and these mechanisms have been extensively characterised experimentally under static conditions. The long-term operation of pipelines is significantly impacted by corrosion which causes degradation of the bounding steel surface, performance breakdown of the pipeline system, and corresponding losses in revenue. To this end, it is necessary to develop models with the capacity to predict accurate corrosion rates to determine pipeline longevity, reduce extensive degradation, and improve effective operation. Underpinning the current state-of-the-art in modelling CO2 corrosion in pipeline flows are empirical measurements of reaction rate kinetics, boundary layer thickness, and turbulent diffusivity all of which restrict predictions to simple geometries, steady-state and single-phase flows. The aim of this investigation is to further the current capabilities of CO2 corrosion prediction in pipeline flows by deriving and implementing a novel method for incorporating computational fluid dynamics with mass transfer modelling. This new model is demonstrated to accurately predict corrosion rates in pipeline flows through validation with experimental evidence. The investigation is then extended to explore more complex situations that conventional tools are unable to describe, including flows in constrictions/expansions, elbows/bends, end caps, transient flow conditions and multiphase fluid flows.

Near-wall flow characteristics of flapping foils at intermediate Reynolds numbers

Zurman-Nasution, Andhini N. and Weymouth, Gabriel D.

University of Southampton

Propulsive flapping foils imitating animals have a potential application for large underwater vehicles but require high Reynolds numbers studies using expensive fully-resolved simulations or complex set-ups experiments. We study flapping foils at Reynolds number (Re) at 1 million using an immersed-boundary solver and implicit LES for two- and three-dimensional simulations. Unlike the flow over a flat plate at Re 1 Million, where the near-wall flow is dominated by the thin turbulent boundary layer, flapping foils at the same freestream Re and an optimum Strouhal number (St) show distinct near-wall flow behaviours. Flapping foils are more affected by the kinematics and adverse pressure gradients. Similar to our previous findings for flapping foil at Re=5300 (Zurman-Nasution et al., JFM 2020), the kinematics at optimum St help to reduce spanwise perturbation, causing more two-dimensional flow and reducing turbulence. The near-wall flow of Re 1 million shows enlarged laminar zones in the inner layer and a diminishing log layer during the peak phase of the local angle of attack. The enlarged laminar zone also increases from the leading to the trailing edges of the foil. Next, we use these findings to compare the wall-resolved flow results with coarse-grid simulations equipped with a wall model suitable for the immersed-boundary solver to study comprehensively flapping foils at Re above 1 million.

Poroviscoelastic Dynamics of Mushy Magmatic Systems

Jennifer Castelino, Susanna Ebmeier, Oliver Harlen and Sam Pegler

University of Leeds

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 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.

While many previous studies have modelled ground deformation due to magma mobilisation using a simple point source or dislocation model embedded in an elastic half space, only a few studies have accounted for the existence of mush as a poroelastic of viscoelastic material. Recent evidence suggests that surface deformation can be caused by both poroelastic and viscoelastic deformation of the mush. With our model we aim to account for this behaviour on a poroviscoelastic spectrum demonstrating the significance of this rheology for observations of deformation.

Here 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 one-dimensional generalised model that describes the behaviour of a poroviscoelastic material. We then adapt this model to a relevant three-dimensional geometries to provide tools for analysing the impact of magma intrusion or eruption for a given system and to provide insight into resulting deformation signals.

An experimental study on the thermal management of electronics with the new design branch-type of bio-mimetic heat sink

Mustafa Yusuf Yazici and Mark Jabbal

University of Nottingham-UK
Samsun University-TUR

Development in the electronics industry has come a long way from nascent low performing devices to advanced devices with high computational speed and power. The advancement in the electronic industry led to an exponential increase in power densities, which in turn simultaneously drove the innovation of smarter and smaller products. The rapid development of electronics, coupled with need of miniaturization in the design and manufacture, make the heat dissipation on the components extremely challenging. If the heat dissipation cannot be effectively dealt with, the resulting overheating problem will cause failures into electrical (slow clock rate, electrical overstress etc.), mechanical (tensile, fatigue etc) and corrosion, and so consequently operation can be interrupted. Therefore, thermal management is becoming critical for performance, life expectancy, and reliability of electronics components/devices.

In this experimental study, biomimetic heat sink design, which focuses on leaves hairiness is introduced based on forced-air cooling for thermal management of electronics. Leaf hairiness regulates the temperature of leaf. So, new branch-type heat sink is designed. Mini pin fins (branches), which mimic leave hairness are arranged on the main pin fin structure (number of 25). The effect of mini pin fins (only number of 12 mini pins fins on each main pin fin) on the thermal performance of heat sink is tested transiently compared to without mini pins (referance case) under two different air velocity of 1 m/s (Re=3225) and 2 m/s (Re=6450) for heat load of 16 W. Temperature and pressure values are measured to evaluate the thermal performance and thermo-economic analyse of the heat sink. Experiments are terminated that the base temperature of heat sink reaches the critical temperature value of 70°C. Thermal performance is considered through thermal resistance and Nusselt number, while thermo-economic analysis is presented through pressure drop.

Eliminating turbulence using body force in pipe flow

Shijun Chu and Ashley P.Willis

School of Mathematics and Statistics, University of Sheffield

Turbulent friction drag is an important factor limiting the performance of many fluids engineering systems, which has led to an extensive search for both active and passive methods for drag reduction. Most research has focused on suppressing turbulence, but progress has recently been made in identifying a class of forces that can laminarize turbulence in pipe flow [1]. These forces accelerate flow near the wall and damp it in the center of the pipe, flattening the base profile. Marensi et al., [2] sought an optimal force that induces the laminarization, extending a popular nonlinear optimization method, which has been successfully used to find the minimal seed that triggers turbulence [3]. Contrary to the previous results, they found it more effective to damp near the wall than in the center. The optimizations, however, were over purely passive (damping) forces. Here we seek the optimal active force (including accelerations) that induces laminarization. We consider purely streamwise forces, three-dimensional forces, and streamwise-independent forces, each of which targets different aspects of the self-sustaining process.

[1]Kühnen, Jakob, et al. "Destabilizing turbulence in pipe flow." Nature Physics 14.4 (2018): 386-390.
[2]Marensi, Elena, et al. "Designing a minimal baffle to destabilize turbulence in pipe flows." Journal of Fluid Mechanics 900 (2020).
31Pringle, Chris CT, and Rich R. Kerswell. "Using nonlinear transient growth to construct the minimal seed for shear flow turbulence." Physical review letters 105.15 (2010): 154502.

Transport of inertial particles in random flows

Aidan Tully, Jacques Vanneste and Mortiz Linkmann

Univeristy of Edinburgh

We examine the transport of inertial particles in smooth random flows with the aim to characterise the clustering of particles in these flows. We focus on the limit of small Stokes number, when the phase-space dynamics can be reduced to a slow manifold, equivalent to the passive advection of non-inertial particles by an effective, weakly compressible flow. We relate the Eulerian statistics of the density of particles to Lagrangian statistics, using large-deviation theory to characterise the latter. We assess the theoretical predictions through a series of numerical simulations of the full inertial particle dynamics and of the effective dynamics. A cloning-pruning importance sampling technique is used to obtain a reliable estimate of the large-deviation rate function for the density distribution within a reasonable number of realisations, overcoming the exponentially low probability of the tails of this distribution.

Machine-assisted subgrid-scale modelling for Large Eddy Simulation-Probability Density Function approaches

Tin-Hang Un and Salvador Navarro-Martinez

Department of Mechanical Engineering, Imperial College London

The application of Large Eddy Simulations (LES) to multi-physics problems, such as reactive, multi-phase flows and magneto-hydrodynamics (MHD), requires models for the non-linear sub-grid phenomena, including chemical reaction, phase change, non-ideal equation of state, etc. These models become less and less general and are very case-specific (e.g. finite-rate chemistry, dilute sprays). Some of these phenomena violate the assumption of isotropy of small scales (e.g. MHD).

The sub-grid probability density function (PDF) approach provides a unified model for complex multi-physics flows as non-linear terms are closed directly by a sub-grid PDF. It has proven successful in modelling reactive flows with LES. The only model needed is the Langevin model which transports the PDF and accounts for the sub-grid pressure gradient and molecular diffusion.

The most commonly used Langevin model is the Simplified Langevin Model (SLM). However, this assumes isotropy of the small scales and uses 40-year-old RANS-based closures and parameters. The Generalised Langevin Model (GLM) is theoretically more complete, but no general closures have been proposed in the literature.

This paper proposes the use of deep neural network (DNN) to model the closures of the GLM. DNN has been used to model sub-grid stresses in LES, with good degree of generality. The present work extends this to predicting the Langevin tensor and dissipation rate. The final model is a parameter-free sub-grid model that can be implemented in multi-physics LES-PDF simulations.

Hydrodynamics in a three-phase flotation system – fluid following with a new hydrogel tracer for PEPT

Diego Mesa, Michael van Heerden, Katie Cole and Pablo Brito-Parada

Imperial College London

Understanding the hydrodynamics of three-phase stirred tanks, such as froth flotation cells, is paramount for the characterisation of turbulence, stability and performance. Although positron emission particle tracking (PEPT) is known for its effectiveness in measuring the hydrodynamics of particles in opaque, high solid content systems, it has not been widely used for characterising the liquid phase. This work presents a new, neutrally buoyant, alginate hydrogel tracer, designed to emulate the density of the liquid phase, which is suitable for high-speed tracking with PEPT.

PEPT experiments were conducted in a bench-scale flotation cell, comparing the new tracer to ion-exchange resin tracers previously used in this system. Results showed statistically significant differences in pathlines, residence time and velocity distribution among the tracers. Moreover, the hydrodynamics of the new tracer agree with existing CFD predictions for the liquid phase. This methodology enables the comprehensive study of relative flow behaviour in complex multiphase systems.

Nonlinear oscillations of levitated air bubbles in water

G. Hunter-Brown , N. Sampara, M. M. Scase and R. J. A. Hill

University of Nottingham

The oscillations of gas bubbles in liquid play a crucial role in several important physical processes, such as the collapse of cavitation bubbles and the radiation of light via sonoluminescence. We will present a novel levitation technique used to observe the nonlinear oscillations of an air bubble in water. Two bubbles are levitated magnetically, as if in zero gravity, in stable mechanical equilibrium away from walls and interfaces, and allowed to coalesce starting from rest. We observed the resulting large-amplitude axisymmetric oscillations of the coalescing bubble using a high-speed camera. Numerical simulations were also carried out using the freely available multiphase Navier-Stokes volume of fluid solver Basilisk. We will describe the technique used to levitate the air bubbles and present results from experiment and numerical simulation and compare these with the analytical model of Tsamopoulos and Brown (1983) for nonlinear shape oscillations of bubbles. In addition to non-linear oscillations, we found that bubble coalescence produced other interesting phenomena, including the ejection of satellite bubbles. We will show observations of the ejection of multiple satellite bubbles which had previously been predicted, here observed experimentally for the first time.

Tsamopoulos, J., & Brown, R. (1983). Nonlinear oscillations of inviscid drops and bubbles. Journal of Fluid Mechanics, 127, 519-537. doi:10.1017/S0022112083002864

Pair formation and stability for particles with softness heterogeneity in inertial microfluidics

Benjamin Owen, Krishnaveni Thota and Timm Krueger

University of Edinburgh

Diseases such as cancer and sepsis cause changes in blood cell structural properties [1]. Inertial microfluidics can be used to diagnosis and treat these diseases by axially ordering cells/particles into trains that migrate to lateral equilibrium positions based on their structural properties. Particles can then be assessed via apparatus such as flow cytometers, or separated based on their structural properties such as softness [2]. The first stage of train formation is pair formation. Previously we have investigated how softness affects pair formation in particles with homogeneous softness, characterising softness using the Laplace number. We demonstrated that softer pairs have lateral equilibrium positions closer to the channel centreline and stable pairs only form when equilibrium positions are off centre [3]. However, the effect of intra-pair softness heterogeneity is unclear. Here, we use a 3D lattice-Boltzmann-immersed-boundary-finite-element solver to investigate a pair of initially spherical, deformable particles with different softness in a rectangular duct under finite inertia. We show that heterogeneous soft particles form stable pairs at lateral equilibrium positions between the bounds of homogeneous pairs. We show that if the leading particle is softer or if lagging particle is softer, has little effect on the pair lateral equilibrium position but has a larger effect on the equilibrium axial distance between the particles. We show that the equilibrium axial distance between particles for heterogeneous pairs is approximately equal to homogeneous pairs of the same softness as the lagging particle.

[1] Gossett et al. (2012) P. Natl. Acad. Sci. 109 (20), 7630–7635.
[2] Martel and Toner (2014) Annu. Rev. Biomed. Eng. 11 (16), 371–396.
[3] Owen and Krueger (2022) J. Fluid. Mech. 937, A4.

Relative importance of physical quantities for data-driven RANS-based turbulence modelling: Creating a list of ingredients for the experimental fluid dynamicist

Omid Bidar, Sean Anderson, Ning Qin, and Ping He

University of Sheffield

In recent years many data assimilation techniques have been introduced for improved Reynolds-averaged Navier-Stokes (RANS) turbulence modelling. While some rely on huge datasets of high-fidelity data, such as those available from direct numerical simulations (DNS), other frameworks have been shown to achieve considerable improvements using limited, experimentally measurable, data. One such framework is field inversion: an approach that involves perturbations of the turbulence model transport equation through a spatial field and the iterative optimisation of this field such that the error between model prediction and data is minimised.

Results reported in literature, and ours, have been primarily dictated by the availability of data for particular test cases, usually generated for benchmarking and validation (i.e. not specifically tailored for data-driven turbulence models). It has been variously observed that the impact of field inversion in terms of error reduction in all physical quantities of interest are highly sensitive to the type of data used. However, there has been no quantitative study of the relative importance of physical quantities for data-driven turbulence modelling. In this work we will attempt to tackle this.

We will use limited data from rich DNS datasets to emulate experimental scenarios (e.g. pressure taps, hot-wire anemometers, or PIV measurements), and perform field inversion. We will then compare the data-driven turbulence model predictions against the rich DNS data in terms of physical quantities of interest (e.g. surface pressure, skin friction, velocity field), and potentially, turbulent quantities (e.g. turbulent kinetic energy, Reynolds shear stress etc.). This will allow us to use error metrics to analyse the relative importance of different quantities, and will be the first step towards creating a list of ingredients for the experimental fluid dynamicist, and bridge the gap between experimental and computational approaches.

Effect of cardiac motion on right coronary artery haemodynamics

Pratik Kandangwa[1,2], Ryo Torii[3], Peter Gatehouse[4], Spencer J. Sherwin[2] and Peter D. Weinberg[1]

[1]Departments of 1Bioengineering and 2Aeronautics, Imperial College London, London, UK
[2]Departments of 1Bioengineering and 2Aeronautics, Imperial College, London, UK
[3]Department of Mechanical Engineering, University College London, UK
[4]National Heart and Lung Institute, Imperial College London, UK

Wall shear stress (WSS) acting perpendicular ("transverse") to the direction of the time-average vector may be atherogenic. Age-related patterns of transverse WSS (transWSS) around rabbit aortic branch ostia correlate well with established age-related patterns of lesions seen in rabbit and human aortas. Coronary arteries are of more clinical interest than the aorta and have a different mechanical environment. Past studies of coronary arteries have either not taken cardiac motion into account or have failed to compute the transWSS metric. We hypothesised that transWSS is sensitive to geometrical changes during the cardiac cycle due to the presence of Dean vortices.

The dynamic geometry and flow waveform of a human right coronary artery (RCA) were obtained from a previous MRI study. Time-dependent CFD simulations were carried out using Star CCM+ and surface deformation of the RCA was accomplished using Blender. Simulations were undertaken to isolate the separate effects of flow pulsatility and wall motion on WSS metrics. Results were compared by converting the 3D geometry into rectangular maps and computing Spearman's rank correlation between them. Effects of non-Newtonian rheology, based on the Carreau-Yasuda model, was also investigated.

Results showed that cross flow index (CFI) and transWSS were more sensitive than the oscillatory shear index (OSI) and time averaged wall shear stress (TAWSS) to wall motion. Spearman's rank correlation coefficient between maps for cases with and without motion was only 0.19 for transWSS but as high as 0.93 for TAWSS. Fluctuating Dean vortices generated by geometrical changes (curvature and torsion) during the cardiac cycle influenced the multidirectional nature of WSS vectors captured by both CFI and transWSS. Including non-Newtonian rheology in the computations changed the magnitude but not the pattern of the maps of WSS metrics.

Application of MPPICInterFoam solver in modeling multiphase droplet microfluidics

Maryam Fatehifar, Alistair Revell and Masoud Jabbari

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

Microfluidic approaches to producing droplets allow precise control over the droplet formation including droplet size and frequency. One intriguing application of droplet microfluidics is particle encapsulation, where single particles/cells are trapped inside droplets. This is attractive for various applications, including nanoparticle synthesis and single-cell analysis. Despite a number of experimental studies of this topic, numerical studies are scarce. Here, we used the open-source fluid dynamics software to simulate the multiphase encapsulation of particles. The solver is MPPICInterFoam and is implemented in OpenFOAM-v2012, originally for modelling fluidized bed flows. The solver couples Eulerian equations, solving fluid-fluid interactions using Volume-of-Fluid method, with Lagrangian equations for the particle-fluid interactions. Coupling Eulerian and Lagrangian approaches allows the motion particles to be predicted as forces such as drag, surface tension and pressure gradient effects are considered. We begin by presenting a thorough assessment of the MPPICInterFoam library, explaining the approximations and limitations therein. We then validate the solver against experimental data for a 2D cross-junction channel flow, while also assessing suitable grid and solver settings. We extend this analysis to explore the droplet volume, droplet frequency and number of particles per droplet and demonstrate the solver is capable of accurately simulating particle movement in a multiphase flow microfluidics. The numerical setup is then applied to investigate the effect of particle concentration on particle encapsulation. Encapsulation efficiency is maximized when the frequency of droplet generation and particle entrance is synchronized, resulting in exactly 1 particle per drop. In our tests for Capillary number, Ca=0.005 this achieved at concentration, C= 2650 particle/mL. Higher C resulted in encapsulation of 2 particles per drop while lower C created empty drops.

Wakes of perforated normal flat plates

M. Cicolin, S. Chellini, B. Ganapathisubramani and I. Castro

Aeronautics and Astronautics, University of Southampton

It is well known that interfering with the wake of a two-dimensional bluff body, e.g. a cylinder or a flat plate, can alter the classical Karman vortex shedding process. In particular, adding a downstream splitter plate or making the body porous can delay the onset of shedding or even remove it completely. Some early work using the relatively simple instrumentation available at the time (hot wires and pulsed wires for measurement of single point data) demonstrated that the eventual removal of the vortex shedding as the porosity of a flat plate increased was accompanied by a relatively rapid fall in drag. Full field data could not easily be obtained at that time and detailed exploration of the wake dynamics was not attempted. Some direct numerical simulations have very recently been reported, albeit at low Reynolds number, and models for wakes containing recirculation bubbles has been developed over recent years, albeit for steady flows. But a number of questions remain open. For example, just how rapid is the 'switch-off' of the vortex shedding as the porosity increases; can vortex shedding occur even in the absence of a recirculating bubble (which moves downstream with increasing porosity); how does the mean structure and geometry of the wake change with porosity; what influence do the details of the porosity geometry itself have on the larger scale wake flow; do the wakes – with or without Karman shedding – display other kinds of instabilities? This paper reports some results of new experimental measurements obtained using a force balance and full-field PIV, undertaken in an attempt to answer these and other questions. The experiments were undertaken using a thin flat plate mounted in a water flume. Plate Reynolds numbers varied between 10,000 and 40,000, and plates with porosities below 50% were used.

Pressure drop and heat transfer in combined porous media

Ahmed F Abusewal, Abdulaziz M Gheit, Mohamed SM Nasr and Robert Woolley

Department of Mechanical Engineering, University of Sheffield

Porous media offer high heat transfer due to their high internal surface areas but at the penalty of pressure drop. When manufacturing a porous media there are a number of design variables: the base material, the porosity, the pore size and the shape of the space. In the experiments reported here the flow and thermal performance of two dissimilar types of porous media have been investigated in isolation and in combination. The media tested were aluminium open cellular matrices and steel wire mesh screens. The samples were tested as regenerators where the fluid, in this case air, was passed through the porous media. Two test rigs were developed to measure the hydraulic and thermal performance of the materials. For the hydraulic performance, the pressure drop with air velocity was measured under steady-state conditions. The Darcy and Forchheimer laminar flow regimes and their transition were identified. Hydraulic parameters such as permeability and the inertia and form drag coefficients in different flow regimes were determined. Thermal performance measurements were conducted using an unsteady-state single-blow technique. The hydrothermal performance of regenerator matrices was characterized in terms of volumetric Nusselt number and Reynolds numbers based on the permeability. The results indicate that the hydrothermal performance of porous media strongly depend on their physical features of media, in terms of the porosity and pore size. A decrease in either lead to enhancement of heat transfer but with higher flow resistance. Comparison between the samples revealed that stacks of steel woven screens had the best performance. The hydraulic and thermal performance of the heterogeneous structures varied based on the proportions of materials employed in the structure.

Sensitivity of rough-wall turbulence to changes in spanwise effective slope

T. O. Jelly, A. Ramani, B. Nugroho, N. Hutchins and A. Busse

University of Leicester

Surface roughness is the rule rather than the exception in real-world flows. It is well known that hydrodynamically rough surfaces experience increased levels of drag compared to smooth surfaces at matched flow conditions, and that surfaces found in practical applications exhibit irregular roughness across multiple length-scales. Yet, predicting the drag penalty of irregular, multi-scale roughness remains extremely challenging and continues to be a focal point of research (Chung et al., Ann. Rev. Fluid Mech, 2021).

The end goal for research in this area is to produce a universal model that correlates key topographical parameters to an equivalent value of Nikuradse's sandgrain roughness, ks. Whilst many models based on roughness height (k), skewness (Ssk) and streamwise effective slope (ESx) have shown promise in predicting the roughness drag penalty, a truly universal correlation will likely depend on additional topographical parameters related to spanwise variations in roughness height, such as spanwise effective slope (ESy).

In this study, direct numerical simulations of turbulent channel flow over irregular, three-dimensional rough walls with systematically varied values of ESy are performed at a friction Reynolds number of 395. A surface generation algorithm is used to synthesise near-Gaussian surfaces with specified ESy for three different values of ESx. Based on an analysis of mean velocity statistics, turbulence co-spectra and the fractional contribution of pressure and viscous drag, the present study shows that ESy can strongly affect the roughness drag penalty — particularly for low-ESx surfaces.

Lagrangian statistics, Floquet stability, and dynamo properties of time-dependent ABC flows

Calum S. Skene and Steven M. Tobias

University of Leeds

We perform 3D numerical simulations of T-periodic, time-dependent ABC flows in a periodic box, where the ABC flow is initially sustained by adding a forcing term to the momentum equations. In the kinematic dynamo regime these ABC flows provide a velocity field through which dynamo action can occur at high enough magnetic Reynolds number, as they are unstable to magnetic perturbations. The links between the dynamo properties of these flows and the Lagrangian statistics are investigated by calculating the finite-time Lyapunov exponents (FTLEs), both in the kinematic regime, where the magnetic field is small, and the non-linear regime, where it is large enough such that the Lorentz force can act to modify the velocity field. The Lagrangian statistics and dynamo properties of a second hydrodynamic state, reached through hydrodynamic instability of the original ABC flow, are also considered.

We find that the FTLEs show significant changes between these three regimes, with the results being highly dependent on the period of the underlying ABC flow. In most cases the FTLEs show a decrease when the ABC flow is altered, which is reflected in the subsequent dynamo behaviour. In all cases though the saturated flow remains chaotic. We will discuss this behaviour in terms of the kinematic growth of the dynamo, its subsequent non-linear saturation mechanism, and possible inability to sustain itself in the non-linear regime. Emphasis will be placed on the role of the period of the ABC flow, the importance of the initial hydrodynamic state, as well as the magnetic Reynolds number.

High Fidelity Simulation of Atmospheric Dispersion

Keertan Kumar Maskey and Ralf Deiterding

University of Southampton

Chemical and Biological hazard predictions have a substantial benefit to public safety. However, the current technology employed in many areas lacks the ability to resolve complex cases. Traditional CFD codes which solve Reynolds Averaged Navier-Stokes (RANs) equations with the Large Eddy Simulation (LES) method have large run-times, making them unsuitable for the aim of conducting relatively quick hazard assessments. The Lattice Boltzmann Method (LBM) is used in this study to produce realistic atmospheric dispersion cases. The success of LBM in the past few years comes from its inbuilt suitability to parallel processing and its inexpensive numerical step, compared to traditional CFD methods. LBM uses a mesoscopic approach to CFD, where it models a sequence of particle distributions colliding and propagating to produce macroscopic quantities (such as density and velocity). The LBM code is currently being developed on the AMROC framework which uses Adaptive Mesh Refinement. A combination of multi-distribution functions alongside scalar transport assumptions is used for modelling the scalar operator. A similar method is used to model the temperature field of the primary fluid on the base weakly-compressible isothermal LBM. Cases have been conducted on single cubes and cube arrays to simulate flow around buildings and compared against experimental findings such as those of the DIPLOS (Dispersion of Localised Releases in a Street Network) project.

Turbulent mixing of heat and momentum in a turbulent boundary layer perturbed by an effusion film

Daniel Burnett and Jonathan Morrison

Imperial College London

To raise the maximum cycle temperature in gas turbines for increased thermal efficiency, more effective turbine blade cooling strategies than conventional film cooling are required, such as effusion cooling - a denser array of smaller holes on the blade surface, leading to attached, full film coverage and higher cooling effectiveness.

To tackle the lack of fundamental understanding of the mixing processes and heat transfer mechanisms which govern effusion cooling, an experimental study is being conducted. Lower temperature air is effused through a porous panel into a large canonical turbulent boundary layer. The obtained high-resolution measurements for this idealised case will serve as a baseline dataset to improve CFD turbulence modelling. The temperature difference between the cold film and the warmer freestream is small (approximately 4°C), so that temperature may be treated as a passive scalar. This enables comparison with Particle Image Velocimetry measurements made for an isothermal film, which showed good agreement in the mean profiles.

Cold-wire measurements of the instantaneous temperature field were performed which revealed that beyond a critical velocity ratio (of the effused gas to the freestream) of 17%, a lower temperature region was observed away the surface, due to the higher momentum films lifting off the surface. Temperature fluctuations were shown to exhibit two peaks above the critical velocity ratio and one peak below it. The common peak was at the mixing layer of film-freestream interface and the second peak was due to local lift-off of the effused gas from immediate upstream holes.

In June, simultaneous cold-wire and Laser Doppler Anemometry measurements will be conducted to measure the turbulent momentum and heat fluxes. The turbulent Prandtl number will then be calculated - a value assumed constant in heat transfer modelling, despite being shown to vary widely in perturbed boundary layer cases such as this.

Effect of surface tension and shear viscosity on the slip-length of aqueous flows over superhydrophobic surfaces

Linsheng Zhang, Yasmin A. Mehanna, Colin R. Crick and Robert J. Poole

University of Liverpool
Queen Mary University of London

So-called ‘Slip-length’ on superhydrophobic surfaces is a crucial indicator for their drag reduction performance in fluid dynamics. Herein, the slip-length of a superhydrophobic surface with various aqueous flows was experimentally investigated. The effects of two fundamental liquid properties (surface tension and shear viscosity) were individually studied by controlling variates (to isolate effects from each other). Superhydrophobic materials were manufactured by using polydimethylsiloxanes and functionalized silica particles. Slip-length measurement was conducted on a rheometer with a cone-and-plate setup and three prepared liquids (sodium dodecyl sulcate, ethanol, and polyethylene glycol solutions) were adopted to generate Couette flow. Experimental results showed that slip-length decreased with decreasing surface tension when the shear viscosity is constant and increased with increasing shear viscosity when surface tension is matched for a fair comparison. Both surface tension and shear viscosity were confirmed to have a linear and direct influence on the slip-length results of the current surfaces. Therefore, it is important to consider the characteristics of the liquid for the application of superhydrophobic materials regarding drag reduction.

Micro-PIV of viscoelastic fluid flow in micro-porous media

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

School of Engineering, University of Liverpool, UK

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 in the understanding of such flows has been made in this time, the scarcity of comprehensive rheology data as well as the limitation of measurement techniques has restricted the majority of these studies to average macroscopically measurable gross flow quantities such as porosity, permeability, and pressure gradient. As a result, the subject remains incompletely understood. The present experimental investigation extends previous work, which focused on relating the bulk flow properties of the polymer solutions to the measurable rheological parameters as it flows through a distinctive micro-porous structure, by using micro-PIV (micro-particle image velocimetry) to measure the velocity distribution and velocity fluctuations within individual pores. The 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). To investigate the effects of fluid elasticity at pore scale, aqueous solutions of a polyacrylamide (PAA) & polyethylene glycol (PEG) in the concentration range of 50-200ppm, which were characterised in both shear and extensional flows using shear and capillary break-up extensional rheometers (CaBER) respectively, were used as working fluids. In order 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. The work to be presented focuses on the velocity field at pore scale, measured using micro-PIV. The velocity field measurement includes the velocity magnitude and fluctuation intensity in several different pores within the porous material across a Wi range of approximately 0.01 to 1 for each of the test fluids.

A DeepONet for predicting groundwater in both steady state and transient systems

Maria Luisa Taccari, Jonathan Nuttall, Xiaohui Chen, He Wang and Peter K. Jimack

University of Leeds

Computational modelling of subsurface flow is able to analyse and forecast the response of an aquifer system to a change of its state. Numerical methods calculate the hydraulic head by iteratively solving an implicit system of equations at each time step in the discretized time and flow domains. Despite great progress in these techniques, running the groundwater model is often prohibitively expensive, especially when the scale of the system is large. In this study, we propose neural network to quickly calculate the response of the system. We leverage a deep operator network (DeepONet) to capture the fundamental input-output relations of the groundwater system and generate solutions for different pumping configurations given the permeability of the ground surface. Results show that the model can easily and accurately predict both the steady state and the transient response of the groundwater system.

An investigation on transient flow behaviour in pulsating channel flows

O. Maklad, P. Taylor, C. Moulinec and M. Seddighi

School of Engineering, Liverpool John Moores University, Liverpool L3 3AF, UK
Scientific Computing Department, STFC Daresbury Laboratory, Warrington WA4 4AD, UK

DNS has been used to investigate the transient behaviour of turbulence for the pulsating flows. An in-house DNS/LES code, CHAPSim, has been adopted and used for the study. Simulations have been performed for channel flow at Reynolds number of Rem=2800, where Reynolds number is based on the time-averaged bulk velocity and channel half-height. A wide range of pulsating frequencies and amplitudes has been simulated. Turbulence statistics and detailed flow behaviour are examined. Where possible, the simulations are validated against the physical experiments.

The preliminarily results show strong similarity between the pulsating and accelerating/decelerating flow for the behaviour of turbulence in the transient period. The transient development of the flow is characterised by a two-stage process resembling pre-transition and transition stages of boundary layer bypass transition (laminar-turbulent) and corresponding stages reported, in the literature, for the individual accelerating/decelerating (turbulent-turbulent) flows. The elongated low- and high-speed streaks are exhibited during the early stages of the transition. This is reflected into immediate but gradual response of the streamwise fluctuating velocities in the near-wall region while it remains almost unchanged in the core region. The wall-normal and spanwise components remain also approximately unchanged during the pre-transition stage and until the onset of transition when the fluctuating velocities and the Reynolds stress exhibit rapid changes.

Formation and collapse of gas cavities in a soft porous medium

Oliver Paulin, Liam Morrow, Matthew Hennessy and Christopher MacMinn

University of Oxford

Gas bubbles can form and grow in otherwise liquid-saturated granular media due to various physical processes, such as corrosion or the microbial decomposition of organic matter. The gas bubbles are typically non-wetting to the solid grains; as such, it is energetically costly for the gas to invade the narrow pore throats between grains. If the solid skeleton is sufficiently soft, the gas can instead displace the solid grains to open macroscopic cavities. An increase in external confining stress can suppress the formation of these cavities and even trigger the collapse of existing cavities, forcing the gas into the pore space. A quantitative understanding of the total volume of gas cavities, their size distribution, and their formation and collapse are important for the macroscopic mechanics of this three-phase system and also for predicting the rate of gas venting to the surrounding environment. Here, we investigate this problem experimentally using a packing of hydrogel beads as a model soft porous medium. We vary the external confining stress to study the formation and collapse of gas cavities within this system. We complement our experimental observations with a phase-field model informed by large-deformation poromechanics, in which two immiscible fluids interact with a poroelasto-plastic solid skeleton and where a phase-field damage model captures the mechanics of the granular solid. Our model captures the competing effects of elasticity and gas-liquid-solid interactions (capillarity). We study the formation and collapse of gas cavities within a 1D setting, identifying the confining stress at which cavities collapse, with gas invading the pore space. We also study the impact of the rate of compression as well as the reversibility of cavity formation and collapse under fluctuating confining stress.

A moving-mesh approach for interface-tracking multiphase flow

Linfeng Li, Jiansheng Xiang, Christopher Pain

Applied Modelling and Computation Group, Department of Earth Science Engineering, Imperial College London, Prince Consort Road, London SW7 2BP

Interface tracking is a key ingredient to the simulation of interfacial flows. Many numerical schemes have been proposed for the advection and reconstruction of the interface to keep the interface sharp and accurate. In this study, we use a novel moving mesh step in the interface-tracking simulation to reduce the error in the advection term. This moving mesh method was inspired by the arbitrary Lagrangian-Eulerian kinematic description widely used in fluid-structure interaction simulation, where a Lagrangian description is usually kept in the solid region. In this study, for multiphase flow, the mesh is advected to minimize a functional of the discretisation error in advection terms. The proposed method is implemented in our in-house code IC-FERST, which uses discontinuous Galerkin discretisation for velocity and control volume discretisation for continuity equation. Interface tracking uses a VOF method with a high-order accurate compressive advection scheme enhanced with the proposed moving-mesh approach. The proposed method is used in the simulation of the single vortex problem [Rider and Kothe, J. Comp. Phy., 141, 112-152] to test its performance. In addition, bubble rising cases are simulated with the proposed method. Both sets of simulations show the accuracy of the proposed method and its promising application in various interfacial flows.

New development of a coarse-grid CFD for mixed convection in rod bundles

Chenxin Zhang, Bo Liu and Shuisheng He

The University of Sheffield

Nuclear power is a low emission and green energy source. There are numerous nuclear power plants currently in operation and more are under construction. In the design of new reactor systems, thermal hydraulic analysis plays a vital role to ensure the safety. The SubChCFD is a novel coarse grid CFD approach that has recently been developed in Sheffield seeking to perform multi-scale thermal hydraulic analysis. This article presents the SubChCFD methodology and new development. Benchmarking against experimental data from NPIC(Nuclear power Institution of China) is also presented.The geometry of the experimental rig is a 2x2 rod bundle within a square channel with round corners. Two supercritical-pressure flow conditions are considered, which are simulated using both conventional CFD and SubChCFD approaches. The strong physical property variations around pseudo critical temperature point are taken into account. In addition, the buoyancy effect is considered in the condition which could be defined as mixed convection. The results from conventional CFD are firstly used to investigate the flow physics and secondly to validate the reliability of SubChCFD methodology. Finally, the velocity profiles and temperature distributions along various lines from conventional CFD and SubChCFD are compared.

Adaptive physics-informed neural network for surrogate modeling and design optimization

Yubiao Sun, Ushnish Sengupta and Matthew Juniper

University of Cambridge

Shape optimization with partial differential equation constraints is challenging as the geometry has to be modified and adapted at every optimization iteration. It is necessary to conduct a complete remeshing or deform existing meshes to accommodate the changing geometry, causing the problem of prohibitive computational costs and lack of robustness. Here we provide a proposition to overcome this challenge, i.e., using surrogate models as robust and accurate approximators to the underlying PDE solution. Given that surrogate models generate computationally cheap evaluations of PDE solutions, we integrate surrogate models into a quasi-Newton optimization method to identify the optimal parameters for a given objective function, and consequently avoid the expensive computational cost. Physics Informed Neural Network (PINN) has been used to build surrogate model for modeling, evaluation and optimization of aerodynamic problems. We not merely explore how to incorporate the Navier–Stokes equations into deep neural networks and constructed PINN model, and but also extend PINN to parameter optimization. It has also been demonstrated that honoring the physics equations contributes to enhanced robustness: while trained only on a certain number of design parameters, PINN model still produces reliable predictions over a wide range of parameter sets completely new to the network. We also present an adaptive surrogate model optimization for efficient numerical approximation of multivariable optimization problems. Our approach integrates the strengths of deep neural networks for rich representation learning, physics embedding and automatic differentiation, remarkably facilitating optimization process. This approach simultaneously determines the flow field and the optimal airfoil shape, and its efficacy and robustness has been demonstrated by obtaining the optimal airfoil shape using PARSEC parametrization.

An experimental investigation into passive pitch systems for tidal turbines

Stefano Gambuzza, Gabriele Pisetta and Ignazio Maria Viola

University of Edinburgh

Tidal currents are a renewable and predictable source of energy that could prove fundamental for energy security and to decrease dependency from fossil fuels. Tidal turbines operate in channels characterised by high turbulence, which causes unsteady loads on the turbine structures, and thus limits the fatigue life of blades and drivetrains. The largest turbines tends to adopt active pitch control systems to reduce the amplitude of the unsteady loads. In this study we present a passive pitch control system, which enables high unsteady loads mitigation and reduces the complexity of the system, with the potential to increase reliability and reduce costs. We consider a blade that rigidly pitches around a spanwise axis, with its motion constrained by a torsional spring. Analytical models show that this can cancel loads fluctuations without affecting the mean loads. We verify these analytical results by measurements of torque, thrust and root-bending moment of a model-scale tidal turbine (diameter of 1.2m) operating in a sheared and turbulent inflow. Measurements of torque and thrust are obtained with a force transducer, while the root-bending moment is measured with strain gauges installed on the blades. Results confirm that mean loads are conserved whilst the standard deviations of thrust, root-bending moment and torque decrease. The unsteady load mitigation varies with the tip-speed ratio. We also show that the passive pitching mechanism can account for changes in the current speed over a tidal period, and could thus potentially entirely replace active pitch control.

Fluid flow in poroelastic tissue engineering bioreactor scaffolds

George Booth, Mohit Dalwadi, Hua (Cathy) Ye, Pierre-Alexis Mouthuy and Sarah Waters

University of Oxford

To engineer functional tissue, we need to provide appropriate biomechanical and biochemical cues to promote cell proliferation and differentiation. Mathematical modelling aims to provide a mechanistic understanding of cell growth, fluid flow and nutrient and waste transport in these systems to inform experimental design. Inspired by human vasculature, our work focuses on modelling electrospun hollow fibre membranes for use in tissue engineering bioreactors.

Experimental collaborators at the University of Oxford have developed a hollow fibre mem- brane scaffold that mimics the deformability and distensibility of human vasculature. The ends of these fibres are fixed in a bioreactor chamber, and cells are seeded to the inner or outer surface. Fluid flow is used to enhance nutrient and growth factor delivery through the fibre lumen, and waste products can also be removed through an additional flow outside of the fibres.

We develop an axisymmetric fluid-structure interaction model of steady Stokes flow in and around a linear poroelastic membrane fibre clamped at its axial ends. Exploiting the small aspect ratio and the large stiffness of the membrane fibre relative to typical fluid pressures, we use asymptotics to derive a coupled system of linear second-order PDEs and ODEs describing membrane displacement and fluid pressure respectively. With this, we solve for fluid flow through the lumen, membrane and extra-capillary space. At leading order, the fluid flow problem reduces to Darcy flow through a rigid porous material, and membrane displacement is driven by the membrane fluid pore pressure. We extend this model to consider how spatially-dependent membrane permeability distributions can be utilised to optimise fluid flow to cells seeded along the length of the fibre surface. In addition to providing mechanistic insights into the mechanical environment of the cells, these models can be used to inform future scaffold design.

Direct Numerical Simulation of post-stall NACA0012 at moderate Reynolds Number

Mohsen Lahooti, Guglielmo Vivarelli, Francesco Montomoli and Spencer J. Sherwin

Department of Aeronautics, Imperial College London

A Direct Numerical Simulation (DNS) of the NACA0012 airfoil at a Reynolds number of 150000 is conducted in the near to post-stall regime. Three angles of attack, $\alpha = 9$, 10 and 12 degrees are studied to reveal the key fluid dynamics of laminar to turbulent transition. Further, we seek insight into the underpinning physics of separation that leads to stall. The DNS simulations are conducted using the high-order spectral/hp method implementation of the incompressible Navier-Stokes equations in the Nektar++ framework. The quasi-3D assumption is used in simulations where the flow is assumed to be spanwise homogenous. Effects of various simulation parameters such as the mesh size, polynomial expansions and the spanwise length on the flow dynamics and aerodynamic coefficients are studied. Moreover, detailed analysis of the flow dynamics such as separation bubble, wall shear stress and vortex dynamics are presented. Finally, in-depth statistical and spectral analysis of turbulent characteristics is performed. Our early results indicate the existence of low-frequency three-dimensional turbulent structures in the spanwise direction near the leading edge. Finally, these results suggest that the accurate prediction of the aerodynamic coefficient requires higher resolution in xy domain compared to the spanwise direction. Such requirement is relaxed as the airfoil passed the maximum lift and was titled deeper in stall.

The Liquid-Solid Amontons' Laws: Friction coefficients for droplets on solids

Hernan Barrio-Zhang, Glen Mchale, Nan Gao, Gary G. Wells and Rodrigo Ledesma-Aguilar

Wetting, Interfacial Science & Engineering Lab, University of Edinburgh

Understanding of the friction between two solids was pioneered by the work of Amontons in 1699, and preceded by the work of Leonardo da Vinci. Since then, the pinning and sliding between solids has been understood by empirical laws tying the normal force between the solids with static and kinetic coefficients of friction. Liquids on solid surfaces also experience resistive forces and have been studied from the perspective of moving contact lines. Recently, there have been works that have identified different resistive regimes when the droplet is static and when it is in motion. This has allowed to view liquid-solid friction from the perspective of static and kinetic regimes of friction. In this work, we show that by considering the surface energy interactions of the liquid-solid-air interfaces, that the frictional force is directly proportional to the normal component of the surface tension force. This allows us to obtain "liquid-on-solid" coefficients of friction, analogous to Amontons' first and second laws, where the normal surface tension force of a droplet replaces the normal load force. For droplets, we show that to overcome the static friction, the force needed is proportional to the contact angle hysteresis. However, in the kinetic regime, the friction experienced by the liquid is proportional to the difference in advancing and receding dynamic contact angles. Comparisons with Amontons' Laws and Furmidge's equation on resistance to droplet motion allows the derivation of the liquid-solid Amontons' Laws, which describe friction coefficients for several liquid-solid systems in the literature. This work provides insight into liquid-solid friction and provides a framework for the design of engineered surfaces to control droplet motion.

Finding the point of no return: A dynamical systems approach to the stability and bifurcation of a moving contact line

Jack Keeler, Satish Kumar, Duncan Lockerby and James Sprittles

University of Warwick

The moving-contact line between a fluid, liquid and a solid is a ubiquitous phenomenon, and determining the maximum speed at which a liquid can wet/dewet a solid is a practically important problem. Using continuum models the maximum speed of wetting/dewetting can be found by calculating steady solutions of the governing equations and locating the critical capillary number, Ca_{crit} , above which no steady-state solution can be found. Below Ca_{crit}, both stable and unstable steady-state solutions exist and if some appropriate measure of these solutions is plotted against Ca, a fold bifurcation appears where the stable and unstable branches meet. In this talk we develop a computational framework to describe this phenomenon and, by applying ideas from dynamical systems theory to the highly-dimensional complex system, show that, rather than just being a consequence of the fold bifurcation, the unstable solutions have profound importance on the transient behaviour of the system. Significantly, the system can become unstable when $Ca < Ca_{crit}$ due to finite amplitude interfacial 'corrugations' are more dangerous than 'stretch' perturbations.

LES of a hot horizontal helium jet issued into an air ambient

Jundi He, Bo Liu and Shuisheng He

The University of Sheffield

The present study is motivated by a postulated loss-of-coolant accident (LOCA) of the high-temperature gas-cooled reactor (HTGR), which is one of the proposed designs of Generation IV nuclear reactors. We are particularly interested in a scenario in which a pipe-break happens in the coolant flow loop followed by the discharge of high-temperature/pressure coolant (helium) and mixing of coolant and ambient gas (air) in the reactor cavity. Of particular relevance is the behaviour of a horizontal jet of one fluid issued horizontally into the ambient of another fluid under the influence of buoyancy.

The study is aimed at gaining a fundamental understanding of the flow physics of a buoyant jet with gas-mixing and heat transfer. High-fidelity large eddy simulations (LES) of buoyant jets at Reynolds number 2892 are carried out using the spectral element code Nek5000, which solves the low-Mach number Navier-Stokes equations, energy and species transport equations. Temperature and species dependent thermophysical properties are considered. The analysis is based on turbulent statistics collected after the jet reaches a stationary state. Three simulations with different jet Richardson numbers (Ri=0.012, 0.064 & 0.137) are carried out to study the effect of buoyancy on the jet spreading rate, laminar-turbulent transition and decaying rate of the maximum velocity and Reynolds stresses. Preliminary results show that the instability triggered by buoyancy causes an earlier laminar-turbulent transition in buoyant jets. Buoyancy production delays the decaying process of the turbulent kinetic energy. Further investigations of the detailed mechanisms will be carried out, which will be reported at the conference.

Morphing of a NACA6409 aerofoil and the use of periodic morphing for aerodynamic efficiency gains in ground effect

Dominic Clements and Kamal Djidjeli

University of Southampton

This research presents an investigation into trailing edge morphing in ground effect of a NACA6409 aerofoil using the FishBAC morphing method. Static morphing was carried out using the two-dimensional Reynolds-Averaged Navier-Stokes (RANS) with morphing beginning at 75%, 85% and 90% chord distance from the leading edge, angles of 0, 2, 3, 4 and 12 degrees and ground clearances of 10%, 20%, 40% and 100% chord. By applying morphing wings to ground effect it is found that the aerodynamic efficiency of aerofoils can be improved. The most aerodynamically efficient trailing edge distance was the 90% morphing at an angle of attack of 3 degrees, with an efficiency of 87.4 compared to 85 with zero morphing. This was then compared to morphing over time with URANS which showed similar lift values but 8% higher drag at 6-degree AoA and 3% higher at 0 degrees. Vortex sheading was observed at 14 degrees AoA in freestream but in ground effect the proximity of the ground eliminated this sheading however stall occurred at 10 degrees AoA in ground effect compared to 12 degrees in freestream. Periodic morphing was carried out using DES starting at 25% along the chord in 10% ground clearance and 4 degrees angle of attack. It was found periodically morphing at a Strouhal number of 0.9 and trailing edge displacement of 0.125% increased the aerodynamic efficiency by 5.4%. The lift reduced but the drag reduced at a much greater rate causing an increase in aerodynamic efficiency. Kelvin-Helmholtz instability of the aerofoil wake was observed and oscillated periodically with a Strouhal number of 0.12 frequency. Applying this technology to a UAV wing in ground effect craft, the flight time and range increased.

Passive attenuation of unsteady transients and vortex streets in a pipe flow

Jacob Hirst, Alistair Revell, Andrea Cioncolini and Mostafa Nabawy

University of Manchester

Instabilities present in an internal fluid flow such as a pipe can lead to unfavourable conditions, such as flow induced vibrations and damage to components such as valves. Moreover, acoustic waves travelling as Fluid-Borne Noise (FBN) can exacerbate noise pollution, leading to undesirable and possibly unsafe working conditions. The modification of pipe wall geometry can alleviate these effects, but current technology, such as Helmholtz resonators and lined silencers are not widely applicable on account of complexity and size restrictions. This work considers a vortex system in the wake of a circular cylinder and the downstream interaction of wall modifications. The present study aims to numerically investigate the impact of modification of the wall. We begin with low Reynolds number flow to develop our understanding for single frequency instabilities, and use a range of flow metrics to measure the effectiveness of the modifications, including flow visualisation and spectral analysis.

An automatic differentiation approach to point vortex dynamics with applications to turbulence

Andrew Cleary and Jacob Page

Maxwell Institute, University of Edinburgh

Our mechanistic understanding of fluid turbulence has substantially improved in recent decades due to the discovery of large numbers of unstable simple invariant solutions to the Navier-Stokes equations. However, the classical approach (recurrent flow analysis coupled with a Newton-Raphson solver) is known to struggle to both identify and converge solutions at high Reynolds numbers. Here, we introduce automatic differentiation (AD) as a technique for finding solutions with specific properties via suitable loss functions. We use a fully differentiable point vortex solver [jax-pv] as a playground to test this loss-based approach, first revisiting the classical problem of finding energy-minimising relative equilibrium states in a rotating superfluid (Campbell & Ziff, Phys. Rev. B 20, 1979) for very large numbers of vortices. We then show how the approach can be extended to search for more complex simple invariant sets that are not detectable by standard methods, focusing on homoclinic and heteroclinic orbits. We extend our analysis to a weakly turbulent channel flow using a fully differentiable Navier-Stokes solver (Kochkov et al, Proc. Nat. Acad. Sci. 118, 2021) to search for "bursting" connections between low dissipation relative equilibria. Time permitting, we may also discuss the utility of jax-pv to generate low order models of scalar mixing.

Intelligent large-eddy simulation model to optimise HVAC system energy consumption in a building

Boyang Chen, Christopher Pain, Pablo Salinas and Claire Heaney

Department of Earth Science and Engineering, Imperial College London, UK

Heating, ventilation and air condition (HVAC) systems are a building's main energy consumer and can mitigate air pollution and airborne virus transmission. However, the most of buildings are challenging to achieve energy conservation by operating these systems more efficiently. The design of such systems often relies on empirical assumptions (of occupancy levels etc.) that significantly lead to excessive energy usage. A well-designed HVAC system requires a fundamental understanding on the fluid mechanisms driven by ventilation and heating system. This work will manipulate the air fluids involving different scales of motion in indoor environments and its contents to minimise the energy consumption of HVAC systems while retaining safe and comfortable environment for people to live and work. Large-eddy simulation is applied to investigate the performance of ventilation systems in primary school by using the open-source CFD code Fluidity and Indoor Geometry Generator. Individual classrooms in the school are modelled and the flow patterns induced by HVAC system is reproduced for airborne transport. The model solves the Navier-Stokes equation for unsteady, incompressible and viscous flow with an advanced adaptive mesh capability. The simulation involves the virus infection between the individuals in a room and monitor the components in the air, such as carbon oxide (CO2) and particle matters (PMs). The results are used to explore the mechanisms of airborne virus transmission and its mitigation by ventilation in thermally stratified environment. Based on the CFD dataset obtained, AI modelling is performed to establish a data-driven model that is adopted to predict the air quality in the school by Generative Adversarial Networks (GANs) and qualify the uncertainties of system designs. Furthermore, this work will contribute to a predictive tool to aid designer or practitioners aiming to optimise ventilation, save energy and improve air quality in buildings.

Ice particle skimming in aircraft icing

Ellen M Jolley and Frank T Smith
UCL

As an aircraft flies through cloud at temperatures below freezing, it encounters ice particles and supercooled droplets which results in the accretion of ice onto its surfaces and hence deformation of its aerodynamic shape. This can, in worst cases, cause series accidents. The study of the dynamics of such particles is thus essential to developing effective icing protection systems. The problem is complex because of the wide range of sizes and shapes of possible particles.

Here, we focus on tackling the situation where there is a thin layer of water on the aircraft surface and the particles are similarly thin such as to be able to interact with the water layer. The technique which has allowed us to make significant headway with this problem has been to study surface fluid layers of arbitrary density (hence including fluids with density similar to that of the air), which can then be used to study the water case by taking the layer fluid density to be one thousand times that of the air. Under suitable assumptions (including that the Reynolds and Froude numbers are large, and that the body is much denser than the air) the model allows the shape of the layer interface and pressure profile beneath the body to be calculated for a given body position. This in turn allows the forces on the body to be calculated and hence for the motion of the particle to be computed in full. The result is a wide range of possible motions of the particle, including both 'sink' cases (the particle enters the water and becomes submerged) and 'skim' cases (where the particle is launched back off the surface of the water following contact). The latter case has analogy with traditional 'stone skimming/skipping' games found around the world.

Stability of body force influenced flows

Stephen Jackson and Shuisheng He

University of Sheffield

In many engineering fluid flows, including examples from magnetohydrodynamics, mixed convection and certain flow control methods, the dominant flow behaviour can be explained by a simpler model problem, namely, where an aiding/opposing body force is imposed on the flow. In this case the conventional understanding is that turbulence will increase/decrease respectively when compared to an equal flow rate (EFR) reference flow. Recent studies have introduced a new framework to study these phenomena whereby body force influenced flows act similarly to their Equal Pressure Gradient (EPG) flows. In particular, streaks simply strengthen or weaken depending on the direction of the body force, seemingly without any changes in streak length and spacing, but the vorticity is largely unchanged. The reasons why this is the case are so far unknown.

By using a combination of direct numerical simulations to determine base profiles, and linear methods to calculate the stability of each base flow, we aim to investigate the relationship between EPG and body force influenced flows. This includes how the nature of the stability changes with varying body force profile shapes, as well as the use of stability methods designed for non-normal systems to capture transient growth. Several different types of base flows are considered in order to characterise the similarities / differences between the general flow instabilities and streak instabilities. These include, laminar, time averaged, and streak imposed base flows.

Linearised predictions of secondary currents in turbulent channels with topological heterogeneity

Gerardo Zampino, Davide Lasagna and Bharath Ganapathisubramani

University of Southampton

Wall-bounded flows subjected to a lateral roughness heterogeneity develop secondary currents which strength and flow organization depend on the surface characteristics. The secondary flows in pressure-driven turbulent channels with topological heterogeneity in the spanwise direction are investigated using linearised Reynolds-Averaged Navier-Stokes (RANS) equations. Following the small perturbation assumption, we assume that the strength of secondary flows is proportional to the equivalent sandgrain roughness height (Nikuradse 1933), here used to characterise the surface roughness. The effect induced by the roughness is a downwards shift of the logarithmic layer and thus, the Spalart-Allmaras turbulence model is modified in order to account for this aspect. Finally, the nonlinear Reynolds stress model is implemented to capture the anisotropy of the of the Reynolds stress tensor that generate the secondary currents. Symmetric channels with strip-type roughness aligned to the streamwise direction are investigated as a function of the high roughness and low roughness widths. The flow organization agrees with the experimental and numerical data available in literature and we predict a downwash and an upwash over the high and low roughness regions, respectively. For increasing strip width, the secondary currents tend to entirely occupy the half-height channel. Tertiary flows are also predicted for wide strips that modify the flow organization. The amplification of such structures is studied as a function of the strip width. Some similarities and differences between strip- and ridge-type roughness are also observed and discussed for some cases. These results suggest that a RANS-based model is enough accurate to predict the flow organization and the size of the secondary flows for both roughness types. Since the equations are linear, a combination of the two effects is possible and will be discussed in the talk.

Impact of large periodic deformations on solute transport in soft porous media

Matilde Fiori, Christopher W. MacMinn and Satyajit Pramanik

University of Oxford

From soils to soft biological tissues, there are many examples of materials that can be modelled as highly deformable porous media, characterised by a strong coupling between mechanical stimulation and fluid flow due to complex rearrangements of the pore space. In both contexts – subsurface geomechanics and living-tissue biomechanics or tissue engineering – the effects of large periodic deformations on solute transport and mixing can be of great interest for predicting and/or controlling the motion of contaminants or nutrients. Here, we propose a 1D continuum model based on large-deformation poroelasticiy that links an applied periodic deformation to the resulting solute transport and mixing. Transport occurs through advection, molecular diffusion, and hydrodynamic dispersion, all of which are affected by the deformation in specific ways. We explore the effects of several dimensionless parameters on the problem, focusing on the ones regulating the applied periodic load. We find that the amplitude and period of deformation influence the mechanical response of the material, which can belong to either a Quasi-Static or a Fast-Transient regime. These mechanical regimes directly characterise the resultant movements of solute.

An Immersed Boundary Method for the DNS solver CHAPSim

Kenneth Chinembiri, Shuisheng He and Wei Wang

University of Sheffield

CHAPSim 2.0 is a Direct Numerical Simulation code developed by the Collaborative Computational Project – Nuclear Thermal Hydraulics (CCP-NTH) as an open-source UK nuclear community code. The solver is fast, efficient, and capable of simulating turbulent thermal flows with strong physical property variation. This paper discusses the methodology and validation of an Immersed Boundary Method (IBM) for complex geometries in the solver CHAPSim 2.0. When adopting this method, the effect of the solid body to flow field is mimicked by introducing a forcing term to the governing momentum equations of the CFD solver [1]. The forcing term allows the user to impose a desired target velocity at the grid nodes of the complex solid boundary and is computed courtesy of the direct forcing approach. There are two methods adopted for the reconstruction of the fluid-solid interface. In the first method, we enforce zero velocity at the grid nodes closest to the immersed surface (no -velocity interpolation). In the second method, we use trilinear interpolation to determine the velocity at the ghost node (the node adjacent to the IB surface on the solid side) that will satisfy the no-slip condition at the fluid-solid interface. Validation is performed by comparing mean velocity profiles against the idealised roughness data by Schultz and Flack [2]. These tests consider pyramid roughness at differing heights and slope angles.

[1]E.A. Fadlun, R. Verzicco, P. Orlandi, J. Mohd-Yusof, Combined immersed-boundary finite-difference methods for three- dimensional complex flow simulations, J. Comput. Phys. 161 (2000) 30–60.
[2]Schultz, M. P. & Flack, K. A. 2009 Turbulent boundary layers on a systematically varied rough wall. Phys. Fluids 21, 015104.

Analysis of small-scale statistics in a turbulent recirculating flow around a wing

Juan Carlos Bilbao-Ludena

Imperial College London

We investigate the deviation from local isotropy of small-scale statistics inside and around a recirculating bubble. More specifically, we consider the flow around a NACA 0018 wing with a square wingtip profile at Rec =5000 and at 10 degrees angle of attack and perform Direct Numerical Simulations. We investigate the balance of the turbulent kinetic energy, enstrophy, and total strain rate transport equations along the cross-stream direction at different streamwise locations. Particular attention is given to enstrophy production and strain self-amplification mechanisms. We find that contributions from mean-velocity gradients (i.e. large scale straining) are important in certain regions of the flow. Geometrical statistics will be also presented in terms of alignments of the fluctuating vorticity with the eigenvector of the fluctuating and mean strain rate tensors.

Behaviour of the buoyant and forced convective jet In the upper plenum of a pool-type nuclear facility

A. Saxena, M. Falcone, X. Huang and S. He

University of Sheffield

Large eddy simulations have been conducted of the upper plenum of E-SCAPE, a 1:6 scale test facility of a pool-type lead bismuth eutectic-cooled fast nuclear reactor (MYRRHA). In the upper plenum of E-SCAPE, there is an above core structure (ACS) region above the core outlet, which is surrounded by a perforated tube, which has five rows of holes. As the flow passes from the core outlet through these holes, jets are formed which are crucial for the overall flow behavior in the plenum. The nature of the jets is varied with some impinging on the components of E-SCAPE (viz Silicon-doping devices, In-vessel fuel handling machines) and some are free. The study is aimed to understand the behavior of the free and impinging jets under the mixed and forced convection conditions. The mean temperature and velocity profiles and turbulence statistics such as the turbulence kinetic energy temperature variance, and Reynolds shear stress are analyzed to understand the turbulence characteristics of the jets and the effect of buoyancy in the two flow conditions investigated.

Dynamics of multiphase plumes in a stratified rotating environment

Daria Frank (presenting), Julien R. Landel, Stuart B. Dalziel and Paul F. Linden

University of Cambridge

The explosion at the Deepwater Horizon (DwH) oil rig in 2010 led to a formation of a long-lived turbulent multiphase plume consisting of a mixture of saltwater, oil droplets and gas bubbles. The DwH plume was discharged into a stratified rotating environment and its duration of several months implies that the Earth's rotation likely had a considerable impact on the plume dynamics. Additionally, the multiphase components of the plume possessed the so-called slip velocity by means of which oil droplets and gas bubbles could separate from the entrained seawater plume. Thus, modelling the dynamics of a multiphase plume in a stratified rotating environment is important for understanding and predicting the transport and the ultimate fate of oil released into the ocean. In this talk, we present results from small-scale laboratory experiments on bubble plumes in a stratified rotating environment. We use sugar instead of salt to create background stratification, which allows us to control independently the three parameters of the problem: the stratification frequency, the Rossby number and the slip velocity of the bubbles. In our data analysis, we focus in particular on the vertical distribution of the multiphase effluent within the water column and discuss the changes in the structure of lateral intrusions of the multiphase plume compared to the non-rotating case.

Numerical analysis of the formation and stability of pairs of particles with different size in inertial microfluidics

Krishnaveni Thota, Benjamin Owen and Timm Krüger

University of Edinburgh

Inertial microfluidics is an emerging technology for the passive manipulation, focusing and sorting of particles. Due to hydrodynamic interaction of particles, pairs often form in inertial microfluidics. It is unclear how and under what conditions particles of different sizes form pairs. In this work, we numerically study the formation and stability of a pair of particles of different sizes in a pressure-driven flow through a rectangular duct under mild inertia. Two types of particle arrangements are considered in our study: staggered pairs (each particle located on opposite sides of the channel centre) and linear pairs (both particles located on the same side of the channel). We employ a combination of the lattice Boltzmann, finite element, and immersed boundary methods to simulate the fully coupled system. We observe that both particles first migrate close to their single-particle equilibrium positions and then form a pair under certain conditions. In particular, we study the effect of particle size ratio on the stability of particle pairs. We find that there exists a critical size ratio beyond which there is no pair formation, and this critical ratio is larger for linear pairs than for staggered pairs.

A comparison of temporally and spatially accelerating turbulent flows

Matthew Falcone and Shuisheng He

University of Sheffield

Spatially accelerating flows occur in a range of engineering applications, including turbo-machinery and aerofoils, particularly those in modern high-lift aircraft. These flows have interesting and not well-understood phenomena such as the reverse transition process, which may result in a relative flow laminarisation. Recently, significant progress has been made in the understanding of temporal acceleration, which has characterised the transient flow development as being the result of the formation of a new boundary layer on the onset of the acceleration. The development of this boundary layer, which initially does not significantly alter the flow behaviour, eventually leads to the transition of the flow to a state commensurate to the increased flow rate. Recent work has extended this theory to an idealised spatially accelerating flow, which has used accelerating moving walls to create a relative spatial acceleration without the wall-normal contraction present in more conventional spatial accelerations.

In this study, we compare the development of spatially and temporally accelerating flows using Direct Numerical Simulation to understand the extent of the similarities and differences between the two flows. The initial results have shown that the overall characteristics of the two flows are similar, with downstream growth of the streamwise Reynolds stresses from the onset of the acceleration with the other turbulence components remaining largely constant. Further analysis will be conducted in the coming months, including an analysis of the streak instability process at the onset of transition and an analysis of the inter-component energy transfer.

Genetic algorithm based active aerofoil deformation in the application of vertical axis wind turbines

Juan-Philip Marx, Derek B Ingham, Lin Ma and Mohammed Pourkashanian

University of Sheffield

The performance of Vertical Axis Wind Turbines' (VAWTs) is significantly impacted by the dynamic stall and static stall phenomena, resulting in poor aerodynamic efficiency and reduced power generation. This work utilises a Genetic Algorithm in order to find the optimal profile of camber magnitudes for active deformation of the turbine blade, at user controlled rotational positions, in order to improve the turbine power generation. This characterised deformation profile is applied in a commercial CFD solver, ANSYS Fluent, in order to evaluate the performance improvement in comparison to a symmetric NACA0015 aerofoil. A high tip speed ratio VAWT is modelled using the URANS approach with a user defined function to change the blade camber magnitude with time, according to the output of the genetic algorithm. The modified turbine blade results in an increase in the torque coefficient, because of the reduced blade aerodynamic separation, and an increase in the profile lift coefficient. Furthermore, the deformation profile of the turbine's upwind rotation significantly changes the wake of the turbine blade experienced in the downwind rotation. This work shows the potential of applying active blade deformation in VAWTs to substantially improve the power efficiency as well as the advantages of genetic algorithms to efficiently evaluate optimal turbine deformation profiles for vertical axis wind turbines.

In this study, we compare the development of spatially and temporally accelerating flows using Direct Numerical Simulation to understand the extent of the similarities and differences between the two flows. The initial results have shown that the overall characteristics of the two flows are similar, with downstream growth of the streamwise Reynolds stresses from the onset of the acceleration with the other turbulence components remaining largely constant. Further analysis will be conducted in the coming months, including an analysis of the streak instability process at the onset of transition and an analysis of the inter-component energy transfer.

Capillary imbibition in SLIPS channels

JRodrigo Ledesma-Aguilar, Sergi Granados Leyva, Hernan Barrio-Zhang, Andreu Benavent, Gary G. Wells, Glen McHale, Ignacio Pagonabarraga and Aurora Hernandez-Machado

University of Edinburgh
University of Barcelona

Spontaneous capillary imbibition is an important process in printing, agriculture and micro-heating. Typically, a fluid invades a solid porous medium leading to "slowing-down" dynamics. But what if the solid medium was replaced by a fluid interface? In this talk we present new results on the spontaneous imbibition of fluids in the presence of a slippery lubricant layer. Using theory, simulation and experiment, we show that the slippery layer provides a qualitative change in the dynamical regimes of an advancing fluid front.

Quantification of nonlinear multiphase flow in porous media

Yihuai Zhang, Branko Bijeljic and Martin J. Blunt

Imperial College London

Multiphase flow in porous materials is conventionally described by an empirical extension to Darcy's law, which assumes that the pressure gradient is proportional to the flow rate. Through a series of two-phase flow experiments, we demonstrate that even when capillary forces are dominant at the pore scale, there is a nonlinear intermittent flow regime with a power-law dependence between pressure gradient and flow rate. Energy balance is used to predict accurately the start of the intermittent regime in both hydrophilic and hydrophobic porous media. The pore-scale explanation of the behavior based on the periodic filling of critical flow pathways is confirmed through 3D micron-resolution X-ray imaging.

Dynamical Impacts of Warm-Starting Operational Weather Models over Africa

Fran A. Morris, Juliane Schwendike, Douglas J. Parker and Caroline L. Bain

Institute of Climate and Atmospheric Sciences, University of Leeds

Weather models which allow for some representation of explicit convection have been shown to add value to weather forecasting by improving the intensity and timing of precipitation, and thus improving the dynamics of precipitating systems. This is particularly valuable in the tropics, where diurnal convection dominates the weather and organised convective systems contribute up to 90% of the rainfall in the boreal summer months. However, implementing high-resolution convection-permitting models operationally can be challenging. One major problem is the initialisation of the models in regions such as Africa where there are sparse measurements to use for data assimilation. Currently, operational convection-permitting models are initialised using global low-resolution models, but these have no fine dynamical structure and as a result there is a relatively long spin-up time of around 12-18 hours before the model begins to accurately reflect precipitation after initialisation. To mitigate this issue, a method called "warm-starting" has been trialled. Warm-starting the operational model takes the large-scale features from the global model used for standard initialisation but retains the fine-scale fields below a certain scale length from a previous run of the model. This approach shows demonstrable improvements in representation of precipitation, and our work investigates why this is effective by examining the atmospheric fluid dynamics at initialisation and evolving afterwards, comparing the warm-start method to simulations where the model has simply been initialised with the global model (a "cold-start"). We produce statistics of vortex objects and examine the energetics of the initialisation, and provide measures of how these features link to precipitation in convection-permitting models, aiming to provide a justification for the scale length at which we include structure from previous model runs using this technique.

On the evolution of and the deposition from an evaporating droplet

Hannah-May D'Ambrosio (presenter), Stephen K Wilson, Brian R Duffy and Alexander W Wray

University of Strathclyde

The evaporation of sessile droplets occurs in numerous physical contexts, with applications in nature, industry, and biology, including coating, chemical decontamination, and inkjet printing. As a consequence of the wide variety of everyday applications, the evolution of, and the deposition from, an evaporating droplet has been the subject of extensive investigation in recent years, with particular interest in droplet lifetimes and in 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. First, we consider the effect of spatial variation in the local evaporative flux on the deposition from a pinned sessile droplet. We determine the flow velocity, the concentration of particles within the droplet, and the evolution of the deposit for a one-parameter family of local evaporative fluxes which captures a wide range of different behaviours and includes diffusion-limited and spatially-uniform evaporation as special cases. We identify three qualitatively different deposit types depending on the spatial variation in the local evaporative flux, and discuss numerical solutions for the particle paths within the droplet. Second, we explore the effect of particle-substrate adsorption on the deposition from an evaporating droplet for both diffusion-limited and spatially-uniform evaporation. We determine the evolution of the mass of particles within the droplet, on the substrate, and in the deposit ring at the contact line, depending on the rate of particle-substrate adsorption. We also discuss the potential the mathematical model has to study the effect of contact line motion and intermittent masking on the deposition of an evaporating droplet. If time permits, we will also discuss work investigating the effect of substrate geometry, droplet size, and droplet orientation on the evolution, and hence the lifetime, of an evaporating droplet.

Dynamics of particle aggregation in evaporating and de-wetting films of complex liquids

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

Loughborough University

We consider the dynamic wetting and de-wetting processes of films and droplets of complex liquids, focusing on the case of colloidal suspensions, where the particle interactions cause agglomeration. This leads to complex dynamics within the liquid and of the liquid-air surface. Incorporating concepts from thermodynamics, we construct a model consisting of a pair of coupled partial differential equations that represent the evolution of the liquid film and the effective colloidal height profiles using the thin-film approximation. The model extends to also include mass transfer effects due to solvent to evaporation and condensation. 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 simulations. 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.

A continuum model for the bulldozing of an immersed granular material in a confined geometry

Liam Morrow

University of Oxford

The flow of immersed granular materials in confined geometries is difficult to characterise due to the complex interactions among the grains, between the grains and the ambient liquid, and between both materials and the walls. Here, we present a reduced-order continuum model for the bulldozing of an immersed, sedimented granular material by a rigid piston in a fluid-filled gap between two parallel plates. This scenario has been studied previously using ad-hoc models and discrete-element simulations. In our continuum approach, the granular pile and the overlying fluid layer evolve as coupled thin films. We model the solid phase as a dense, viscous porous material that experiences Coulomb-like friction along the walls. Conservation of mass and momentum lead to a linear elliptic equation for the local velocity of the grains that is coupled with a nonlinear conservation law for the height of the granular pile. We solve our model numerically for a variety of different scenarios in order to develop insight into the interactions between wall friction, internal viscous-like stresses, and fluid flow both above and through the pile.

Two-phase flow: Travelling waves in a reactive porous medium

Danielle Bullamore, Sam Pegler and Sandra Piazolo

University of Leeds

There are many situations where a fluid flows through a porous medium and reacts with it to alter the porosity and permeability of the medium. Important examples include flows in the Earth's mantle and crust, and formation of karst topography. An everyday example is the flow of water poured through a bed of dissolving sugar. Instabilities in these systems can create remarkable fingering patterns, a reaction-infiltration instability arising from the localisation of flow in high permeability pathways. However, there remains a question of how to model such systems within a simplified continuum framework that allows for a smooth transition between fully reacted and non-reacted components, and to explore the fundamental solutions of such models. We develop a continuum model describing such systems with a two-phase fluid approach. In this framework, both the porous medium and fluid occupy the whole domain, with the relative proportions of porous medium and fluid described by a spatially variable porosity. We show that in the configuration of a 1D transition from reacted to non-reacted regions of the porous solid, there exist travelling wave solutions, and we use the model to predict the spatial scales on which a transition between two such regions occurs. We show that the spatial extent of the transition depends on the initial material porosity and on the relative importance of reaction and diffusion in the system. This work provides basic solutions to lay the footings for future numerical work on the non-linear development of the system (over long time scales) in a computationally efficient manner.

Understanding Melt Flow's Signature on Geometrical Patterns in Rock-Melt Mixtures

Giulia Fedrizzi[1], Sandra Piazolo[1], Daniel Koehn[2] and Sam Pegler[1]

[1]University of Leeds
[2]Friedrich-Alexander University of Erlangen-Nuremberg

Large regions of the Earth's lower and middle crust are known to be affected by the production and migration of melt. Once the conditions for partial melting are met, the molten fraction can remain within the same volume of rock or flow towards shallower levels. Rocks, i.e. polycrystalline mineral aggregates, that are now observed on the Earth's surface can show "frozen-in" patterns characterised by the spatial distribution of melt. Currently, it is unclear how material properties and the relative rate of processes involved (e.g., melt production versus deformation) govern the resultant mineral-melt pattern. The aim of this study is to quantify and classify the impact of melt production and melt flow on geometrical patterns with the aid of numerical simulations.

Our numerical code uses a 2-grid, hybrid approach for the simulation of the two phases. For the fluid, a continuum model with porous flow simulates pervasive percolation of melt. For the solid, a discrete-element model reproduces the elastic behaviour of the host rock. The solid particles are connected by a network of springs that can break if the local stress reaches the failure criterion. This represents the formation of a fracture and results in an increase of local permeability enhancing melt flow.

Numerical experiments show the importance of fluid (melt) pressure on the formation of melt-filled fracture networks. These structures are essential for the onset of preferred melt pathways which eventually evolve into larger melt channels. We show how their spatial distribution and orientation is highly affected by the relative rate of processes such as external deformation, local melt production, fluid pressure diffusion and external influx of melt. By varying fundamental parameters, we can define different regimes that can help the interpretation of patterns observed in natural rocks.

Multi-scale pore network modeling of fluid dynamics in porous rocks

Sajjad foroughi, Branko Bijeljic and Martin J blunt

Imperial College London

Different scales of heterogeneity determine fluid dynamics properties in porous materials. A reservoir rock, for example, has pores with a wide range of size covering many orders of magnitude. As X-ray tomography is a powerful tool to digitise the pore structure. In the digital rock analysis, it is often not possible to visualize pores of all scales due to the resolution/sample size trade-off. For some porous material this sub-resolution pore space is a significant fraction of the total porosity. We consider this sub-resolution as micro-porosity. Differential imaging makes it possible to identify micro-porosity in a micro-CT experiment and to quantify its porosity map of a rock sample. When the volume fraction of unresolved porosity becomes significant, ignoring it has a great impact on modeling the pore scale of porous materials based on the digital images. Because it affects the volume of pores and the connectivity of pore space. Ignoring unresolved porosity leads to lower estimates of pore volume as well as poor connectivity. Including micro-porosity in the pore network model leads to correct saturation and pore volume, this elaborated version of pore network model can be called multiscale pore network model. it is essential to improve our prediction of connectivity and permeability. Furthermore, it has a significant effect on the accessibility to network elements. In this study microporosity is modelled using a multiscale pore network model. Sensitivity analysis on the effect of micro-porosity in the modeling demonstrates the importance of microporosity for both single-phase flow and multiphase flow.

High-Fidelity Comparison of Compressible/Incompressible Boundary Layers

G. Vivarelli, J. A. Isler, F. Montomoli, S. J. Sherwin and P. Adami

Imperial College London/Rolls-Royce

In this work, the effects of compressibility will be analysed by considering different aerofoils with identical pressure loading run at incompressible and compressible conditions. The first step in the overall computation will be the determination of the boundary layer height. This will be done in the incompressible solution by means of an automated methodology employing total pressure and the vertical axial velocity distribution convergence towards an inviscid flow assumption [1]. Through appropriate scaling, this will then be used to determine the compressible boundary layer integral quantities and validated against the boundary layer velocity profile. Similarity and differences between the two solutions due to density effects will be discussed. A second, more involved, comparison will also be carried out on a flat plate at high-Reynolds number with a strong adverse pressure gradient and Mach number just shy of the 0.3 threshold. In this case, the flow separates just upstream of the 10% plate length mark with turbulent transition occurring shortly thereafter. The very same conditions will be used for the incompressible and the compressible flow solver. Similarities and differences in the separation region and turbulent transition will be shown by means of wall shear-stress and boundary layer integral quantities. Similarly, to the first case, these will be determined employing a boundary layer estimate in the incompressible solution.

[1] "General method for determining the boundary layer thickness in nonequilibrium flows" K. P. Griffin, L. Fu & P. Moin, Physical Review of Fluids 6, 024608 (2021)

Capturing the mechanisms for gravitational settling of inertial particles in turbulence using the kinetic PDF approach

Chris Stafford[1], David Swailes [2], Michael Reeks[3]

[1]Advanced Engineering Centre, School of Architecture, Technology and Engineering, University of Brighton, Brighton BN2 4GJ, UK
[2]School of Mathematics, Statistics and Physics, Newcastle University, Newcastle NE1 7RU, UK
[3]School of Engineering, Newcastle University, Newcastle NE1 7RU, UK

The modification of average settling velocity for inertial particles in turbulence is a striking feature of dispersed particle flows. Whilst the preferential sweeping of particles in downward flowing regions occurs for all particle sizes, the manifestation of this effect through the centrifuge mechanism has been shown to be dominant only for small particles, with path history effects playing an increasingly important role for more inertial particles. From a modelling perspective, it has been demonstrated that the kinetic probability density function (PDF) approach provides an exact representation for the increase in settling velocity of particles subject to a Stokes drag force within a Gaussian flow field, and is able to account for the distinct physical mechanisms that occur across the entire range of particle sizes. In this work an asymptotic analysis of the kinetic PDF approach is contrasted with previous work, and it is shown using kinematic simulation that a quantitatively greater proportion of the increase in settling velocity is accounted for as the particle inertia is increased. In particular, even with the reduced asymptotic description, the kinetic PDF approach is able to accurately capture the peak increase in settling velocity that occurs at non-negligible particle inertia. Crucially the appearance of the path history is not only retained, but is in fact the dominant contribution to the settling velocity increase for small particles. These observations suggest that the instantaneous centrifuging of particles from vortical structures within the flow field that occurs for small particles is the limiting case of a more general mechanism underlying the increase in settling velocity. This emphasises the multiscale nature of the description provided by the kinetic PDF approach, and illustrates the implications that extending this framework to the consideration of full dynamical turbulence could have for the modelling and simulation of dispersed particle flows.

Modelling of mixing of helium with air in nuclear reactor cavities

Bo Liu, Shuisheng He, Jundi He, Charles Moulinec and Juan Uribe

The University of Sheffield (Current affiliation: Science and Technology Facilities Council Daresbury Laboratory)

As one of the six proposed designs of Generation IV nuclear reactors, the High Temperature Gas Reactor (HGTR) is being designed to have various passive safety features. The system safety performance has been studied experimentally or numerically to understand some relevant phenomena. For example, in a postulated accident of a pipe break in the main loop, high-temperature/pressure helium can be discharged into the reactor/steam generator cavities, giving rise to very complex gas mixing and double diffusive natural circulations. To study the behavior of the system during and after the blowdown, experimental investigations are being carried out at City College of New York (CCNY).

In parallel, numerical simulations are performed for both the experimental and more extreme HTGR conditions using computational fluid dynamics (CFD). For the relatively moderate experimental conditions where the Mach number of the nozzle is much smaller than 1, simulations are carried out using an open source low Mach number solver Code_Saturne, and the predictions are validated against the experimental data collected at CCNY. The CFD simulations agree well with the experiment in terms of time evolutions of local gas temperature and oxygen concentration at various locations inside the cavity. For the real HTGT condition, additional challenges are posed to the numerical simulation due to large variation of flow regimes in the cavity where highly compressible supersonic jet is formed near the exit of the nozzle, while low speed natural convection dominates in the rest of the domain. To capture the long transient with reasonable computing cost, a pseudo-transient approach is currently being developed using OpenFOAM, in which an overset mesh approach is adopted with partially overlapped subdomains for the jet and the cavity. Some preliminary results show good agreement with a single-domain calculations.

Symmetry-reduced low-dimensional representation of large-scale dynamics in a turbulent boundary layer

Matthias Engel, Omid Ashtari, Tobias Schneider and Moritz Linkmann

School of Mathematics and Maxwell Institute for Mathematical Sciences, University of Edinburgh

An important but poorly understood feature of turbulent boundary layers are persistent large-scale coherent structures in the flow, which influence mixing and extreme events. Here, we use Dynamic Mode Decomposition (DMD), a data-driven technique designed to detect spatio-temporal coherence, to construct optimal low-dimensional representations of such large-scale dynamics in the asymptotic suction boundary layer (ASBL). In the ASBL, fluid is removed by suction through the bottom wall, resulting in a constant boundary layer thickness in streamwise direction. This allows a description of spatio-temporal coherent structures without having to account for variations in the Reynolds number. That is, the streamwise advection of coherent structures by the mean flow ceases to be of dynamical importance and can be interpreted as a continuous shift symmetry in streamwise direction. However, this results in technical difficulties, as DMD is known to perform poorly in presence of continuous symmetries, which can lead to spurious results, for instance. Here, we compare different techniques for symmetry reduction such as optimal Galilean shifts and the method of slices [1,2] in conjunction with DMD, to construct optimal low-dimensional models for the large-scale dynamics of the ASBL.

[1]N. B. Budanur, P. Cvitanovic, R. L. Davidchak, and E. Siminos, Reduction of the SO(2) symmetry for spatially extended dynamical systems, Phys. Rev. Lett. 114 (2015), 084102.
[2]E. Marensi, G. Yalniz, B. Hof, and N. B. Budanur, Symmetry-reduced dynamic mode decomposition of near-wall turbulence, 2021, arxiv:2101.07516.

Periodic orbits in the sedimentation dynamics of thin, rigid disks

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

University of Manchester

We present a numerical investigation of the behaviour of 2D, deformed circular disks sedimenting under gravity in a quiescent 3D viscous fluid at low Reynolds number. At high Reynolds number, it is common to observe sedimenting particles undergoing steady or unsteady rotations in their terminal state, for example steadily rotating helicopter seeds or the unsteady fluttering motion of falling leaves. However, such terminal rotation modes are driven primarily by inertia. We show, through direct numerical simulation, that cylindrically-deformed circular disks also undergo such motions but at zero Reynolds number, where the translation-rotation coupling is a consequence of geometry alone. We find that such particles do not possess a stable spin-free terminal state; instead their orientation is unsteady in general, following periodic orbits which are uniquely determined by the initial orientation. The centre of these orbits is a steady rotation mode about an axis parallel to gravity.

The sharp edge of such an object moving in a viscous fluid generates singularities in the fluid pressure and velocity gradient, which presents a serious challenge to numerical simulation. At low Reynolds number, the singularities mean a significant contribution of the total drag comes from the vicinity of the disk edge; this implies that any under-resolution of the pressure and the velocity gradient leads to critical errors in the sedimentation velocity and rotation, but such singularities have a severe impact on the convergence rate of standard finite-element (FE) discretisations. To address this, we have developed a novel augmented FE method which allows analytic (singular) functions of unknown amplitude to be subtracted from the full solution in a sub-domain around the disk edge, rendering the FE remainder part of the solution regular and thus restoring the standard FE convergence rate.

Heat transfer in next generation 3D-printable Heat Exchangers

L. Cope, S. M. Tobias and J. B. Marston

University of Leeds

Heat Exchangers (HE) are devices which facilitate the transfer of heat between two or more fluid streams. These devices are ubiquitous in industry and operating in a myriad of flow settings. The optimal design of HE is 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 an aircraft.

Typically, enhancements in heat transfer performance are accomplished by increasing the effective heat transfer surface area and/or promoting mixing within the fluid streams. To that effect, secondary heat transfer surfaces and turbulators are employed but require additional pumping power to overcome the associated frictional losses. This presents a large variety of possible surface configurations which can be utilised in order to balance heat transfer enhancement verses frictional losses, making the design process of HE challenging.

Recently, the emergence of 3D-printing techniques such as Additive Manufacturing (AM) have facilitated the fabrication of complex three-dimensional geometries which are unmanufacturable by traditional techniques. This enables the development of novel and efficient AM-HE with true freedom of design.

Triply periodic minimal surfaces (TPMS) are a promising target for AM-HE. TPMS are a family of mathematically described architectures whose complex geometries are only possible with AM techniques. They boast large surfaces area to volume ratios, superior mechanical and transport properties and fouling resistance making them ideal candidates for next-generation heat transfer surfaces.

In this talk I will exploit the freedom of design enabled by AM techniques to create next-generation heat transfer surfaces using TPMS architectures, investigate their thermal performance using CFD for a range of Reynolds numbers and showcase opportunities for further enhancement by geometrically variation.

Performance benchmarking metrics for comparison of machine learning and traditional methods for problems in fluid-structure interaction

Ajay Bangalore Harish

University of Manchester

In the recent years, there has been an explosion in the application of machine learning (ML) for problems involving fluid flows and fluid-structure interaction (FSI). Most of the works have focused on the "idea of prediction" and thus a "speed-up" in simulation times. Thus, today, our community stands divided on this issue of ML. While some believe that ML will solve all the problems, the other school consider ML as a complete waste of resources. However, these differences make it even more pertinent today to realize and accept that the reality stands somewhere in between these two extremes. While ML might not replace the traditional methods, it has its advantages in certain applications. It is pertinent to realize that ML is a tool just like many of our numerical methods and needs to be fairly benchmarked. In this work, we will be comparing various ML techniques that are used today, their potential future and drawbacks for application towards FSI problems. The benchmark metrics will further compare the performances and limitations of ML vs. traditional numerical methods when applied to FSI problems. The work will conclude with outlining potential areas for future applications of ML in FSI.

Numerical simulation of thermal mixing of liquid sodium in a Y-junction

Wei Wang, Charles Moulinec, Shuisheng He, Juan Uribe AND David R. Emerson

UKRI STFC Daresbury Labratory

Because of their high power-density and efficiency, liquid metals have been used as a heat transfer medium in low carbon energy-related industries including solar and nuclear. However, previous investigations of heat transfer and thermal mixing of conventional fluids, such as liquid water, water vapour or ideal gas, cannot be directly applied to liquid metals. In order to gain a better understanding of the heat transfer characteristics of liquid metals, it is important to address some physical issues, such as thermal mixing and striping, which could arise during thermal mixing in pipes, such as three-way pipe junctions, which are widely used in industry to facilitate fluid distribution and mixing. In this study, numerical simulations of flow of liquid sodium in a Y-junction were carried out. Three important thermal hydraulic parameters, i.e. the fluid momentum ratio, the temperature differences of two branches and the inlet Reynolds number, were investigated to study their influence on flow development and the mixing characteristics of the flow in the main tube. It is observed that the momentum ratio plays a key role in the mixing flow development and temperature distribution, while the inlet temperature difference has very limited influence on both flow pattern and wall temperature distribution. The inlet Reynolds number has also little influence on the flow pattern but it dramatically changes the wall temperature distribution. This study of thermal mixing of liquid metal in one of the most widely used pipe junctions reveals different thermal-mixing patterns and thermal stress development under different thermal hydraulic conditions which could benefit the design of robust pipe junctions and valves, and overall safer piping systems.

Predicting unseen scenarios with data-driven reduced-order models using sub-sampling and domain decomposition

Claire E. Heaney, Hanna Go, Xiangqi Liu and Christopher C. Pain

Imperial College London

Given current advances in computational power, high-resolution computational fluid dynamics (CFD) simulations are possible for a wide range of flows, however, can take days or weeks on a cluster or high-end workstation. Reduced-order models (ROMs) are one way in which to address this issue and have been used to generate high-resolution solutions for flows in real-time. Data-driven or non-intrusive ROMs are increasingly exploiting neural networks for both dimensionality reduction and predicting the evolution of the system in time. How well these ROMs can predict for unseen scenarios remains an open question.

A recent approach based on data-driven reduced-order modelling, a sub-sampling technique and domain decomposition, has been shown to make reasonable predictions of flows for unseen scenarios. CFD results for flow around several hundreds of buildings (in 2D) were used as training data for the neural networks. This approach was able to make predictions for domains larger than the one used to generate the training data for the ROM with about four times the number of buildings. This paper investigates how successfully this approach can solve for unseen scenarios produced by varying the wind direction. To establish the accuracy of the method, statistical properties from the ROM are compared with those from the CFD results.

Predicting unseen scenarios with data-driven reduced-order models using sub-sampling and domain decomposition

C. F. Castro-Faccetti, C. Noakes, M. F. King and Khan, A.

School of Civil Engineering, University of Leeds

A healthcare-associated infection (HCAI) is an infection of a patient during the process of care by a health-care facility, which was not present or incubating at the time of admission. WHO (2011) reported that 1 in 10 patients get an infection whilst receiving care, with respiratory infections being one of the most common types of HCAIs. Airflow distribution is a defining factor on the probability of acquiring airborne respiratory infections in health-care facilities. In this study, time-dependent CFD simulations of an aerobiological chamber located in the University of Leeds were conducted to understand airflow and pathogen distribution under different scenarios. An unsteady RANS methodology was implemented. Turbulence closure was achieved by solving the k-omega SST model. The solution-adaptive mesh refinement feature of the commercial package ANSYS Fluent 19.2 was used. Carbon dioxide (CO2) was used as an indicator of airborne particles, and its transport was solved using the advection-diffusion equation. A CO2 mass flow rate of 80 litres per second was released for 15 and 30 seconds, either from the air supply inlet or from a cylindrical shaped mannequin that represented a patient or a health-care worker. The ventilation rate scenarios were 1.5, 3, 6 and 8 ACH. Numerical values and experimental measurements of the time-varying CO2 concentrations were compared for different scenarios and locations in the room. A modification of the Wells-Riley model was proposed and implemented to determine the probability of airborne infection at different locations. Significant spatial variations were found for the probability of infection. Short release durations do not imply zero probability of infection and even in a well-ventilated room (8 ACH), infection probability after a 20-minute exposure to a 15-second CO2 release can be as high as 1.4% for a quanta rate of 100 quanta per hour, with higher risks found for lower ventilation rates.

Fingerlike instability in drying droplets

Ahmed Othman, Andreas Poulos, Ophelie Torres and Alexander Routh
University of Cambridge

Understanding the drying and spreading of polymer thin films is key for product stability and the final film solid state. The spreading dynamics of a liquid containing polymers is important for different applications e.g. skincare, inkjet, biomaterials, and photovoltaics as it controls the final film morphology. In this work, the spreading and evaporation of droplets containing Poly(vinylpyrrolidone) and Ethanol on a smooth solid substrate exhibit a fingerlike instability observed at the edge of the film leading to inadequate coverage of the coated surface. A uniform liquid thin film is experimentally observed to spread ahead of the meniscus of the droplet due to the occurrence of concentration and surface tension gradients which enhances the droplet spreading. An experimental study indicates that the fast-spreading film develops a capillary ridge that becomes unstable, revealing the relationship between the size of the fingers to the mass fraction of the polymer. In addition, two distinct regimes are observed during altering the viscosity of the polymer solution; identifying the onset between both based on the critical concentration of the polymer in the solution. We derived the long-wave evolution equation for thin films for the free interface near the contact line based on a constant solutal Marangoni shear derived from the concentration equation. The velocity of the contact line is quantified, and the scaling agrees with the experimental values. Linear stability analysis was performed to show the most unstable wavelength which shows an excellent agreement with the experimental observation.

Understanding regimes and validity of solutions to the hydrodynamic escape of planetary atmospheres

Richard Chatterjee and Ray Pierrehumbert

AOPP, University of Oxford

A decade after the first detections of exoplanets, atmosphere lost from a gas giant –HD209458b160 light-years from Earth – to the surrounding space was robustly observed (Vidal-Madjar et al. 2003, 2004). Carbon and oxygen were also detected amidst the tenuous gas of hydrogen and helium beyond the Roche lobe; requiring a hydrodynamic mechanism of bulk outflow to drag those heavier molecules out of the gravitational well. This hydrodynamic escape, primarily driven by EUV and X-ray stellar irradiation, can irreversibly change a planet's atmospheric and even bulk composition (Owen 2019). Consequently, this escape mechanism is key to understanding long-term evolution and planetary habitability, and is even effective enough to carve out a valley in the mass-radius distribution of planets called the sub-Neptune Desert. The mathematics of the mechanism are rich: the escaping flow is transonic (analogous to the Parker solar-wind model), while solutions exhibit sharp thresholds and cutoffs due to the non-linear nature, and the validity of solutions requires an understanding of kinetic theory for when the flow becomes collisionless.

Here we bring together previous results from the literature and add to them, focusing on analytic solutions to simplified problems: how we can use them to understand flows with more complex forcing and when those flows can be sustained. Implications for the interpretation of advanced numerical models that solve the spherically symmetric Euler Equations governing the outflows will be discussed. Time permitting, insights into time-stepping using the CIP/-CLS2 scheme (Kuramoto et al. 2013), both as way of integrating to a steady state and in response to physical forcing, will be presented.

Flame-patterns and diwhirls route to elastoinertial turbulence in Taylor Couette flows

T. Boulafentis, T. Lacassagne, N. Cagney and S. Balabani

Mechanical Engineering Department, University College London, Torrington Place, WC1E 7JE London, United Kingdom

Since the seminal work of G.I.Taylor, almost a 100 years ago, the Taylor-Couette (TC) flow, the flow between two concentric cylinders with one or both of them rotating, has served as a paradigm for studying flow instabilities. Despite the simplicity of the geometry, in the case of viscoelastic TC flows, a variety of transitions, leading to the onset of Elasto-Inertial Turbulence (EIT), have been reported. For shear-thinning or moderately elastic fluids, a vortex merging and splitting transition (MST) has been reported [1] whereas for fluids of higher elasticity a chaotic state of strong radial inflow jets, often termed flame-pattern (FP), appears [2]. Stationary structures of strong inflow jets during ramp-down, called solitary vortex pairs or diwhirls (DW), have also been reported as a hysteretic transition [3]. Place, WC1E 7JE London, United Kingdom

In our experiments we investigate the effect of different elasticities (El=0.057-0.341) on the instabilities induced by Boger fluids in a TC flow with a rotating inner cylinder, both for ramp-up and ramp-down conditions in the range of Re=0-200. A combination of flow visualization experiments and PIV measurements have been used. Spatiotemporal maps were produced using mica flakes in order to identify the transitions. In low elasticity, the previously reported MST route to EIT is observed. In higher elasticity, the MST gives rise to FP and hysteretic DWs are observed during ramp-down. FPs become progressively more chaotic with increasing inertia, leading to EIT. By using PIV measurements, the nature of the FP/DW instability and the route to EIT are elucidated in selected Reynolds numbers.

[1] T. Lacassagne, N. Cagney, J. J. J. Gillissen, and S. Balabani, "Vortex merging and splitting: A route to elastoinertial turbulence in Taylor-Couette flow".
[2] N. Latrache and I. Mutabazi, "Transition to turbulence via flame patterns in viscoelastic Taylor–Couette flow".
[3] A. Groisman and V. Steinberg, "Solitary vortex pairs in viscoelastic couette flow".

Investigation of flow anisotropy and its effect on deterministic lateral displacement devices for particle separation

Calum Mallorie, David W. Inglis, Rohan Vernekar and Timm Krueger

The University of Edinburgh

Deterministic lateral displacement (DLD) is a microfluidic particle separation method that accurately separates suspended particles based on size or deformability. This method has shown promise in diagnostic applications such as isolation of tumour cells, viruses and more. Recently there has been growing interest in using DLD at high flowrates, in order to decrease sample processing time. However, a significant decrease in critical size of separation has been reported when operating at moderate Reynolds number, due to inertial effects in the suspending fluid. Previous studies have shown that anisotropic flow behaviour, where flow deviates from the direction it is forced in, can occur at moderate Reynolds numbers in all common device geometries, and that this anisotropy has a deleterious effect on the device performance.

We have used 2D lattice Boltzmann simulations to study the effects of inertia on the flow field in DLD devices in the absence of particles. Our simulations show that increasing fluid inertia decreases the critical separation size based on the flow field for almost any non-zero Reynolds number, but that these changes are insufficient to explain the much greater decrease seen in experiments. Understanding the behaviour and effect of inertia-induced anisotropy may help explain the difference in critical separation size observed in simulations and experiments. We find that anisotropy in DLD devices becomes significant at Reynolds numbers as low as 10, and that anisotropy exhibits a complex non-linear behaviour with increasing fluid inertia. We also show that macroscopic flow field observables, such as recirculation zones, do not readily correlate with trends in anisotropy.

Our work will will help to predict how these hydrodynamic interactions affect device behaviour at high flowrates, allowing manufacturers to anticipate and account for performance changes at different operation conditions with minimal design iterations.

Simulation of inertial microfluidics using various cross-sectional shapes

Fatemehsadat Mirghaderi and Timm Krüger

The University of Edinburgh

Inertial particle microfluidics (IPMF) has attracted considerable attention due to the advantages it offers, such as simplicity, high throughput and precise control of particles for particle separation and particle manipulation for downstream processing including diagnostics and therapeutics, chemical and biological analysis, and cell biology. IPMF strongly depends on the balance of two major inertial lift forces: the wall-induced lift force and shear-gradient lift force. It is known that the number and location of focusing positions of particles in the channel cross-section strongly depend on a number of factors, including the cross-sectional shape, particle size and Reynolds number. A variety of channel cross-sectional shapes has been used experimentally to study the effects of the channel shape on the flow field and particle behaviour. Inertial microfluidic devices are mostly made of soft materials, such as PDMS, which deform under operation, i.e, high operating pressures. The effect of operating conditions on the device's deformation and performance has not been fully explored. Deformation will change the channel's cross-sectional shape from its originally designed geometry. This deformation will affect the inertial lift forces and thus the expected flow field and particle behaviour. In our work, we study and compare the effect of various cross-sectional shapes, such as varying triangles, rectangular and semi-circular cross-sections on the flow field and particle behaviour using a bespoke high-performance lattice-Boltzmann code. Mimicking the deformation of channel cross-section by varying channel cross-sectional shapes through simulation will lead to a better understanding of inertial lift forces, flow field and particle behaviour in real-world devices.

Lagrangian curvature statistics from Gaussian sub-ensembles in turbulent von-Kármán flow

Y. Hengster[1], D. Schanz[2], F. Huhn[2], D. Garaboa Paz[3], E. Bodenschatz[4], M.Wilczek[5] and M.Linkmann[1]

[1]School of Mathematics, University of Edinburgh, UK
[2]German Aerospace Center, Institute of Aerodynamics and Flow Technology, Department of Experimental Methods, Göttingen, Germany
[3]Group of Nonlinear Physics, University of Santiago de Compostela, Spain
[4]Max Planck Inst. for Dynamics and Self-Organization, Göttingen, Germany,
[5]Department of Physics, University of Bayreuth, Germany

A salient feature of fully turbulent flows far from onset is the intermittent occurrence of extreme fluctuations at small spatial and temporal scales. Here, we derive an expression for the curvature probability density function (pdf) for the ensemble of tracer particle trajectories in isotropic turbulence that includes effects of spatio-temporal intermittency. We obtain a master curve for the pdf for near-Gaussian sub-ensembles, generated by conditioning on the squared acceleration coarse-grained over a few viscous time units, where an analytic form of the pdf is known. Using this expression, we calculate the pdf for the full ensemble resulting in a re-scaled version the master curve. The scaling factor is related to moments of the coarse-grained acceleration, and thus includes the effect of spatio-temporal intermittency. The derived pdf agrees qualitatively and quantitatively with the curvature pdf sampled from tracer particle data in von-Kármán flow obtained by Shake-The-Box processing.

Study of different turbulence models in predicting 3D flow inside a single stage micro gas turbine

Qiming Yu and Dr Robert Howell

Mechanical Engineering Department, The University of Sheffield

The objective of this study was to investigate the flow behaviour in a Wren100 micro gas turbine (MGT) stator-rotor system supplied by Turbine Solutions Ltd. The said turbine consists of one nozzle guid vane (NGV) and one rotor disk with blade heights less than 10mm. It is known that the flow in MGTs generally has relatively low Reynolds numbers due to the extremely small blade sizes, meaning large portion of transitional flow could exist within the blade passages. Also, because of the tiny blade sizes, surface roughness could become an important factor that affect the overall engine performance. Thus, to study the aerodynamic performance of the MGT system, it is important to select an appropriate turbulence model.

In this paper, the performance of three different turbulence models (k-epsilon, k-omega SST and transitional SST) in predicting the 3D flow in the Wren100 stator-rotor system were compared with the experimental data. Due to the lack of the original 3D CAD files of the Wren100 turbine parts, simulations were carried out based on the validated discrete reverse-engineered models. It was found that both transitional SST and k-omega SST could predict turbine mass flow rate and thrust with quite high accuracy, which the transitional SST model has slightly better performance. The mass flow rate and thrust predicted by the k-epsilon model have the same trend as the experimental data, but with much higher values. The inadequate performance of the k-epsilon model could be caused by that the boundary layer separations near the wall was not correctly modelled. Overall, the transitional SST model would generate the closest results compared to the experimental data, meaning it is the best model that can be used to generate the initial design of MGTs for further optimisations.

Flow characteristics of a hyperloop vehicle

Alex Lang

University of Leeds

Hyperloop is a proposed mode of next generation, high-speed ground transport, promoted as a faster, cheaper and more efficient alternative to high-speed rail and short-haul aviation. The theoretical Hyperloop system consists of passenger-carrying 'pods' which are transported through a network of tubes, held at a partial vacuum. This low pressure environment reduces the aerodynamic drag on the pods, which can account for over 75% of the total resistance encountered by a conventional high-speed train. The flow regime and aerodynamic characteristics of a Hyperloop will be in a class of their own, combining transonic speeds at relatively low Reynolds number with strong compressibility and blockage effects. In this study, computational fluid dynamics is used for the preliminary aerodynamic design and optimisation of the Hyperloop system. The first component of the work has involved generating a validated methodology for the simulation of the low Reynolds number flow generated by a Hyperloop, using the commercial package Ansys Fluent. This includes consideration and comparison of various turbulence models, as conventional RANS models are known to perform poorly for low Reynolds number aerodynamics. The second stage of the work will involve multi-objective optimisation for the performance of the Hyperloop system (e.g. drag reduction), using design variables including the pod shapes, cruising speed and tube pressure.

Oscillatory motions of falling snowflakes

Jennifer Stout, Chris Westbrook, Thorwald Stein and Mark McCorquodale

University of Reading

The aerodynamics of falling snowflakes are not well understood, due to their complex and irregular shapes. We want to understand the physical parameters which control whether the particle falls steadily or in fluttering, spiralling, or chaotic trajectories. This is important information needed to relate snowflake size distributions to observed radar Doppler velocity spectra, with unsteadiness in falling snowflake trajectories and orientations broadening the width of measured Doppler spectra and influencing dual-polarisation parameters.

3D-printed analogues falling in a tank of water-glycerine mixture and micro laser machined analogues falling in air are used to simulate the behaviours of real ice particles in the atmosphere. Multi-view cameras are used to observe the fall motion of a range of snowflake analogues, and digital reconstructions of the trajectories and orientations of the analogues are analysed to develop fundamental understanding of how ice particles fall, and what properties control their motion.

In this presentation we contrast planar ice crystals with the well-understood trajectories of circular discs. The falling behaviour is found to be a function of Reynolds number, dimensionless moment of inertia, and particle shape , such as how branched the crystal is. The inclination angle planar snowflakes make with the horizontal plane is found to have a sinusoidal oscillation. The Strouhal number corresponding to the frequency of this oscillation is used to quantify and characterise the unsteadiness of oscillating cases. Strouhal number is found to vary weakly with Reynolds number for a given disk aspect ratio, but the Strouhal number decreases with dimensionless moment of inertia, such that thinner particles oscillate more frequently, corresponding well with literature for circular disks.

Managing the increasing heat load in advanced military aircrafts with latent heat thermal management systems

Julie Frank, Duncan Borman, Amirul Khan, Jon Summers, Evaldas Greiciunas and Saman Samarasekera

University of Leeds, CDT Fluid Dynamics

In recent years, the heat load in small high performance military aircraft has steadily been increasing, causing a strain on thermal management systems (TMS). Current TMS are designed to absorb the maximum heat load, but these typically only happen for a small fraction of the flight, leading to over sized heat exchangers for the rest of the flight.

The use of phase change materials (PCM) in latent heat thermal energy solutions (LHTES) has been identified as a possible solution that can be used to store excess energy during peak times, providing increased design freedom for the TMS. As the PCM melts, some of the heat is absorbed from the working fluid, and stored in the material while the PCM remains in the liquid phase. Once the peak heat load period has passed, the temperature drops below the phase change temperature, solidifying the material. The heat is then released back into the fluid or can be extracted and used in another process. A shortcoming of PCM can be the low thermal conductivity leading to long melting times. This creates design challenges in how best to incorporate them within the overall TMS such that they can be responsive. The computational time required to faithfully model the phase change also leads to challenges in modelling the phase change numerically.

This work will be outlining a preliminary numerical model of a shell and tube heat exchanger incorporating PCM. Subsequently, the results from the analysis will be used to assess how the model can be simplified so that it can be used for assessing the performance of novel heat exchanger designs incorporating PCM. Having a reliable numerical modelling framework is important for evaluating and optimising proposed new designs.

Efficient acoustofluidics and active surface cleaning using ZnO/glass thin film acoustic waves

[1]Huiling Ong, Huafeng Pang[1,2], Prashant Agrawal[1], Hamdi Torun[1], Kunyapat Thummavichai[1], Qiang Wu[1], YongQing Fu[1]

[1]Faculty of Engineering and Environment, Northumbria University, Newcastle upon Tyne, NE1 8ST, UK
[2]Department of Applied Physics, college of Science, Xi'an University of Science and Technology, Xi'an, 710054, P. R. China

Glass surfaces used in various applications such as photovoltaics, aerospace, lens, and windows have frequently had soiling issues resulted from dirt, dust, snow, and ice, which can lead to a degraded properties and damages the surface layer on the glass. Various manual and mechanical means are employed to eliminate these contaminants from attaching to the glass surfaces. There are existing solutions applying passive methods to combat soiling issues, e.g., using surface coatings that range from hydrophilic, hydrophobic, and sol-gel coatings. However, this method is not always feasible due to the potential of losing its intended functionalities in extreme conditions where subsequent soiling can occur. This research addresses this soiling issue using ZnO thin film/glass surface acoustic waves (SAWs) based microfluidic device, with incorporation of surface coating for hydrophobicity and icephobicity. Active surface cleaning which is based on acoustofluidics performance involving transportation, jetting and nebulization were evaluated and explored where the acoustic characteristics of both Rayleigh and Sezawa wave modes of the ZnO thin film were studied. Ash particles were used as model contaminants and active surface cleaning of the contaminants was illustrated based on the transportation of water droplets. The optimized SAW powers were identified to cause strong interactions between the ash particles and the water droplet which effectively drive away the particles for efficient surface cleaning, along with surface heating effects which was induced by SAWs.

Patterned low friction lubricated surfaces

Michele Pelizzari, Glen McHale, Steven Armstrong, Gary Wells and Rodrigo Ledesma-Aguilar

Wetting, Interfacial Science & Engineering Lab, University of Edinburgh, Edinburgh, UK

Slippery liquid infused porous surfaces (SLIPS) allow pinning free and low friction motion of droplets across their surfaces. This is because the surface has a thin continuous layer of the liquid (lubricant oil) on which the droplet rests. For a small droplet, its shape is a spherical cap with a contact angle at the three-phase contact line described by a liquid analogue of Young's law. Here we report the development of surfaces with patterns of two different types of lubricating oils. Our surfaces are made by lithographically patterning Teflon AF1600 into a set of square and circular arrays and then coating with GLACO, a commercial liquid solution of hydrophobic silica nanoparticles suspended in IPA, which dewets from the Teflon, but leaves a superhydrophobic coating on the non-Teflon coated regions. This patterned solid surface provides a base which has regions with preferential wetting by Krytox (a perfluoro lubricant) and other regions preferentially wet by silicone and other oils. We report the mobility and contact angles of droplets on these surfaces. We argue that these contact angles obey a liquid analogue of Cassie's law using area weighted averages of the liquid Young's law contact angle for each of the oil regions.

Convection in salt lakes

M. R. Threadgold, C. Beaume, L. Goehring and S. M. Tobias

University of Leeds, Nottingham Trent University

Salt lakes can occur in arid environments where the only outflow of fluid is due to evaporation. Fluid can enter these lakes via rainfall over the neighbouring terrain, which transports salt via groundwater flow. If the evaporation rate exceeds the rate at which fluid enter, lakes can dry up, leading to the formation of a salt crust at its surface. Under the right conditions, the crust forms a connected network of ridges, resulting in a distinctive polygonal pattern. We model salt lakes using a 3D porous medium subject to a uniform through-flow. Our setup leads to a base state that is unstable for large enough Rayleigh numbers where buoyancy-driven convection characterised by the emergence of plumes is observed. We simulate the dynamics numerically from the base state and analyse the sequential stages of the resulting instability by means of characteristic properties of the system (e.g. average surface salinity flux, average pattern wavenumber). As the instability develops, the average wavenumber decreases from the value predicted by the linear theory and settles at a value consistent with field observations and independent of the Rayleigh number. The resulting patterns are similar to those observed in situ.

Optimising the performance of the computational fluid dynamics (CFD) applications written in domain-specific languages (DSLs)

Beite Yang

University of Leeds

Programming a parallel computational fluid dynamics (CFD) application is hardware-dependent if one wants to fully exploit the power of parallelism. Since parallel systems are evolving fast, writing CFD codes on these platforms are usually time-consuming and error-prone. This has been relieved after the emergence of CFD-related domain specific languages (DSLs) such as OPS and OP2. By restricting to a limited scope of domain, these languages have managed to make coding parallel applications on various hardware platforms easier for non-expert programmers. Consequently, programmers' productivity and code portability have been both improved. However, the reported performance of the programs written in DSLs, measured in runtime (seconds), computational throughput (GFlops/s) and achieved bandwidth (GB/s), suggests that there might be room for optimisation.

This project is focusing on defining a set of parameters which has an impact on the code performance and then searching for the parameter configurations with a satisfactory performance, which is a process called parameter tuning. Because the configuration space could be enormous and sparse, hand-tuning can be tedious and prohibitively time-consuming. Therefore, we shall adopt the approach of autotuning. This poster presents the candidate parameters contributing to the code performance and other discoveries so far.

Towards Full Transient Adjoint Based Aerodynamic Optimisation of Vertical Axis Wind Turbines Using CFD

Qiulin Ren, Jee L Hee, Derek. B Ingham, Lin Ma and Mohammed Pourkashanian

ENERGY 2050, Department of Mechanical Engineering, University of Sheffield

Vertical axis wind turbines (VAWTs) have many unique advantages over horizontal axis wind turbines, but the huge gap between the theoretical maximum efficiency and the actual efficiency of VAWTs is of great concern and thus has great research potential. The adjoint optimized blades has shown the potential to significantly improve the power coefficient of vertical axis wind turbines using so called semi-transient adjoint optimization. The full-transient adjoint optimization techniques have not yet been applied to the fluid dynamics of VAWTs. However, it has been applied to the fluid flow past an aerofoil of an aeroplane in motion, and this has resulted in a good improvement in the aerodynamic performance. The difficulties that VAWT aerofoils need to overcome are mainly the complexity of the rotation and pitch motion states and the flow interactions due to the motion of the multiple blades of the turbine. The unsteady adjoint optimization algorithm based on the open source CFD software SU2 can be used to achieve this optimization and is able to define these motion states in transient simulations. Therefore, the present study aims to study unsteady freestream flows over an oscillating aerofoil that simulates the VAWT motion in order to demonstrate the power coefficient improvement of the fully transient adjoint optimization. In addition, the new simulation results will be validated against the previous experimental and computational results. The fluid dynamics and the force generation of the aerofoil will be discussed. It is anticipated that the use of the adjoint optimisation algorithms will produce results that substantially improve the efficiency of the VAWTs.

Biomechanics of articular cartilage: A quantitative investigation on the influence of density gradients to the poroelastic response of cartilage

Craig Thomas, Ali Ghanbarzadeh, Michael Bryant and Gregory de Boer

School of Mechanical Engineering, University of Leeds, Leeds, LS2 9JT, UK

An important and long-standing field of research in orthopaedic biomechanics is the understanding of the stress-strain behaviour of soft, porous solid bodies interstitially filled with fluid such as articular cartilage (AC). To study AC under load will lead to a better understanding of the operation of heathy tissues in natural joints and provide pathways to designing replacement tissues exhibiting the same mechanical response and load carrying capacity with improved biocompatibility. The underlying physical mechanisms that impart transient stress-strain behaviour to the mechanical response of articular AC under compressive loading conditions, were originally established as part of the well-known biphasic theory. Subsequent constitutive models have attempted to capture one or more of AC's bio-mechanical responses e.g., the intrinsic viscoelasticity or permeability of AC under load. This has led to conflated modelling approaches culminating in limited characterisation of the poroelastic theory on which they are based.

The work presented here focuses on the development of a novel nonlinear Finite Volume Method (FVM) numerical solver for the elucidation and computational modelling of the poroelastic response of articular cartilage at a macroscopic scale. The primary objective is to quantify the significance of the underlying physical interactions, which characterise the bio-mechanical behaviour of cartilaginous tissues based on the Biot theory of linear poroelasticity. The model is formulated wherein the case of large deformation (finite strain) and a compressible solid material are considered and combined with porous flow theories. The solver developed is validated on a 3D idealised geometry of a cartilage disc sample under compressive loading conditions e.g., confined, and unconfined compression. Results will show the short period transient mechanical response of AC and will be validated against existing datasets found within the literature.

Phase changes of biomimetic respiratory droplets on ultra-smooth surfaces

Alex Jenkins, Gary Wells, Rodrigo Ledesma-Aguilar and Daniel Orejon Mantecon

University of Edinburgh

The evaporation dynamics of respiratory fluid droplets is a key factor in determining the ability of airborne pathogens to spread. In this study, we evaluate the evaporation of such droplets on pinning-free surfaces that cannot be assumed to have no surface contact. The role of temperature and humidity on the lifetime of respiratory fluids is one of intense study, however unlike previous works our study doesn't rely on lotus effects. Typically when achieving constant contact angle evaporation of fluids on surfaces, the contact of the droplet is assumed to be negligible as the contact angle of the droplet approaches $\sim 180°$. Our study uses slippery omniphobic covalently attached liquid like surfaces (SOCAL), capable of CCA evaporation whilst maintaining a contact angle of $\sim 105°$ allowing for the effect of contact line pinning on the droplet evaporation. Our results show that the evaporation of respiratory droplets on SOCAL is much different to on untreated surfaces and superhydrophobic surfaces. We also observed previously unseen responsiveness to humidity, even when away from the deliquescence limit of the salts within the fluid. The removal of contact line pinning on evaporation presents an alternative approach to evaluating the lifetime of respiratory droplets and new understanding of non-pure liquid evaporations on SOCAL surfaces.

Liquid-liquid interactions from measurement of apparent contact angles on slippery liquid infused porous surfaces

Nasser Afify, Steven Armstrong, Gary G. Wells, Rodrigo Ledesma-Aguilar, and Glen McHale

Wetting, Interfacial Science & Engineering Laboratory, The University of Edinburgh, UK

One approach to reducing liquid friction at a solid surface is to create a slippery liquid infused porous surface (SLIPS). To do so, a completely wetting liquid can be infused into a superhydrophobic surface structure to create a surface composed of a thin continuous layer of the infusing liquid. Recently, we have shown a liquid analogue of Young's law can be deduced for the apparent contact angle of a droplet resting on such a surface despite their never being contact by the droplet with the underlying solid. Here we show that measurements of apparent contact angles of droplets on different SLIP surfaces can be used to determine liquid-liquid interfacial tensions. We show estimates from this new approach are in excellent agreement with the values of liquid-liquid interfacial tensions deduced from the traditional pendant drop technique. Furthermore, we consider the extent to which the Zisman method for characterising the wettability of a solid surface can be applied to the liquid surfaces created using SLIPS. We report measurements of droplet contact angles for both a homologous series of alkanes and water-IPA mixtures on SLIPS surfaces. These measurements show a linear relationship in the liquid form of the Zisman plot provided an effective droplet-vapour interfacial tension is used to allow for whether or not cloaking by the infusing liquid of the droplet-vapour interface occurs. We discuss the interpretation of these results for liquid surfaces using the concept of the Critical Surface Tension (CST) for wetting of a solid surface introduced by Zisman.

Influence of multiphase microstructure and rheology on the dynamics of bubble growth and release from viscoelastic soft sediments

Isabel Latimer

University of Leeds

The growth, retention and release of trapped gas within soft sediments is important in many natural and engineered systems; from methane formation within the muddy aquatic beds and wastewater sludges, to safety concerns from hydrogen gas release in corroded legacy nuclear wastes. Until recently, most research surrounding this topic has centred on laboratory experiments, using X-ray computed tomography techniques to capture images of bubble behaviour within sediments. Even now, the affects of sediment microstructure and fluid rheology on the dynamics of initial single bubble growth, secondary formation of interconnected bubble networks and the resulting influence on continuous gaseous release are not well understood. An increase in computing power has allowed more detailed simulation to be possible especially in regards to the modelling of particulate flows. The aim of this work is to enhance our knowledge of these systems by examining the mechanisms of bubble growth and release in viscoelastic soft sediments from a computational approach. This approach uses a simulation of a packed sediment bed in which fully resolved fluid-structure interaction will be obtained by coupling a lattice Boltzmann and rigid body dynamics solver using the software framework waLBerla (Erlangen, Germany). This poster will detail the current progress of the packed sediment bed simulation alongside initial validation studies for the model and planned extensions for including bubble growth and migration.

Stagnation-point heating in high-enthalpy boundary layers

Teja Ala and Neil Sandham

Aerodynamics and Flight Mechanics Group, University of Southampton

Heat transfer at the front stagnation-point is of a particular interest in an impinging hypersonic flow as it provides the maximum heat flux on the body. The resulting high-enthalpy hypersonic boundary layer can include complex thermochemical states in which the chemical reactions along with molecular vibration can be in non-equilibrium conditions. The classical Fay-Riddell self-similar equations are still widely cited, even though their derivation makes significant approximations in terms of chemical and thermal non-equilibrium in relation to modern formulations, which may reduce the validity of the results. In this contribution we have coded two independent numerical methods to compare with the classical solutions and have been able to identify some missing information in the original paper in terms of model constants. The current study aims to quantify in more detail the inaccuracies of the Fay-Riddell equations by comparing model results for equilibrium and frozen chemistry configurations with numerical solutions of the compressible Navier-Stokes equations. The Navier-Stokes formulation includes a 5-species model for dissociating air, as well as vibrational non-equilibrium of diatomic molecules and multi-species models for viscosity and thermal conductivity. Grid studies are used to assess numerical uncertainty in the computed results for the front stagnation region of a circular cylinder.

Effects of thermal slip and chemical reaction on free convective nanofluid from a horizontal plate embedded in a porous media

Abdulaziz Alsenafi and Mohammad Ferdows

Kuwait University

We consider a two-dimensional, uniform, incompressible and free convection flow of a nano-fluid along a plane. The plate is located facing upward about the porous medium. Throughout the investigation, thermal slip, chemical reaction, heat emission/absorption is considered. In the modeling of nano-fluid we have considered the dynamic effect along with the Brownian and thermophoresis. In obtaining the governing equations, including the boundary conditions, an appropriate scaling is applied. The governing momentum equations, including thermal energy and nanoparticles equations are translated into a group of nonlinear ODEs by using Lie symmetry group transformation. The transformed equations are then solved numerically using the Runge-Kutta-Fehlberg fourth-fifth order. The numerical results of velocity, temperature, and nanoparticle volume fraction profiles for varied physical parameters will be discussed and analyzed at the end. The discussion also includes the local Nusselt and the local Sherwood numbers against several of the systems' physical parameters. It is found that the velocity and temperature decrease with thermal slip and heat absorption whilst it increases by increasing heat generation and chemical reaction order. Our present results will be compared with similar existing literature results.

Fluid mechanics of polymer melt filtration

Joseph Bennett

University of Leeds

The filtering of polymer melts is an essential component of industrial processes such as sequentially stretched film casting as filtration ensures that defects do not appear in the finished film. Filtration is important for maximising the use of recycled material within the production. However, filter performance deteriorates with time due to the capture of contaminants, leading to the need to change filters frequently, which is both costly and disruptive to production, especially if the need for change is only identified through a failure. The work involves the development of computational fluid dynamics (CFD) analysis of complex models of practical filter packs based around two different fundamental designs (known as candle and disc filters). CFD analysis and results of the flow of a clean medium through candle and disc filter packs are presented, to explore and understand the distribution of the highly viscous, high-pressure flow through and around the different filter pack designs. Plans are also outlined for the inclusion of filtration mechanisms in the CFD analysis, and for supporting benchtop validation experiments.

An investigation of the fluid structure interaction in articular cartilage across disparate scales

Emily Butler[1], Greg de Boer[2], David Head[3], Mark Walkley[3], Michel Bryant[2], Alister Hart[4], Harry Hothi[4]

[1]CDT Fluid Dynamics, University of Leeds
[2]School of Mechanical Engineering, University of Leeds
[3] School of Computing, University of Leeds
[4]Royal National Orthopaedic Hospital, University College London

Articular Cartilage (AC) is a highly specialised connective tissue found at opposing surfaces in mammalian joints. Its principal function is to provide a smooth bearing surface, which promotes low friction articulation thus facilitating continuous operation under relative motion. Although AC undergoes high contact pressure with minimal friction and wear, its low capacity for intrinsic healing or repair leaves it prone to degeneration. This results in a high demand for cartilage repair, with approximately 10,000 injuries per year warranting surgery in the UK [1]. Despite this clinical importance, a lack of understanding of the tissue renders successful repair techniques elusive. Currently these are invasive, require extensive revision and fail to produce repair tissue exhibiting the same mechanical and functional properties of native AC. The complex material composition of AC makes it notoriously difficult to model accurately. Intrinsic inhomogeneity, attributed to a stratified composition, contributes to inaccuracies with single scale modelling approaches. This project aims to couple an immersed fibrous network (micro-scale) model with a continuum (macro-scale) model to create an innovative multi-scale poroelastic model of the fluid structure interaction arising within AC. Heterogeneous multi-scale methods are applied, using homogenized stress, strain and velocity from the micro-scale to populate the macro-scale elasticity and permeability tensors. Currently, a continuum-continuum model is in development for self-validation of the multi-scale model. Once established, a fibrous network model of the micro-scale behaviour will facilitate representation of the inherent anisotropy and depth dependent properties of AC. The resulting multi-scale model will be calibrated using experimental data and clinical collaboration.

[1]D. Spitaels et al.,“Epidemiology of knee osteoarthritis in general practice:a registry-based study”, BMJ Open(2020), DOI:10.1136/bmjopen-2019-031734

Predicting vorticity and velocity variations in flow through the pore-scale porous media based on recurrent neural network-based physics-informed machine learning

Mehrdad Mesgarpour, Omid Mahian, Mostafa Safdari Shadloo, Nader Karimi and Somchai Wongwises

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi (KMUTT), Bangmod, Bangkok 10140, Thailand
School of Engineering and Materials Science, Queen Mary University of London, London E1 4NS, United Kingdom
CORIA-UMR 6614, CNRS-University & INSA, Normandie University, 76000, Rouen, France
National Science and Technology Development Agency (NSTDA), Pathum Thani 12120, Thailand
School of Engineering and Materials Science, Queen Mary University of London, London E1 4NS, United Kingdom
James Watt School of Engineering, University of Glasgow, Glasgow G12 8QQ, United Kingdom

Severe issues such as excessive number of grid-points, complex geometry and very small dimensions make the flow through pore-scale porous media (PSPM) one of the most challenging problems in computational fluid dynamics (CFD). As a remedy, here we present a novel method for the computational analysis of flow in PSPM (Re=4,000) which combines conventional CFD with deep learning techniques. In this method, the entire computational domain is resolved in only 10 iterations. In each location, the temporal patterns of velocity and vorticity fields as well as pressure are extracted (Δt=0.001 s). Each parameter's value, location, and time are saved in a separate tensor. After training these tensors with a neural network, an RNN-based model is fitted to the data. Since the RNN model considers time series, the behavior of each parameter's variation is defined as a time series. Due to the existence of a large number of parameters that change over time (over 540 million data-points), developing tensor-based training data is a critical part of this method. The output parameters are then used to generate PDEs based on transfer of momentum in Navier-Stokes equations via automatic differentiation. The boundary condition is a significant bottleneck in the developed Physics Informed Machine Learning (PIML) method. Geometrical boundary conditions are defined in this novel method based on the geometry of PSPM, and as a result, PDE equations can be solved using a matrix of boundary conditions in 12 series of loops. Further, this work includes development of a special type of LES model for use in the PIML method, with a time-dependent loss function. The presented method enables fast prediction of the transient characteristics of the flows passing through PSPM. Most notably, it allows predicting the transient vorticity field in a highly complex geometry like PSPM.

The effect of hyperdiffusion in rotating convection

Bradley Davy

University of Leeds

The Earth's magnetic field is thought to be sustained by dynamo action occurring in the outer core. The flow that characterises core dynamics is complex and difficult to model due to the extreme parameters at which the Earth operates. It is thought that flow in the Earth's outer core is driven by convection but is also affected by both rotation and the resulting magnetic field. As more realistic parameters are attempted to be simulated the range of spatial and temporal scales broaden, requiring increased resolution and prohibitively long run times. Without increased resolution energy builds up in the smallest scales resulting in numerical instability. Hyperdiffusion is a numerical method which can be used to diffuse this energy and thus stabilise the scheme. However, the effect of hyperdiffusion is poorly understood. This work uses theoretical arguments for defining a hyperdiffusive scheme that both maintains gross features of the flow and stabilises numerical schemes. This study will use a holistic approach in order to assess the affect that this numerical treatment has on the flow. A simplified model is first considered which still encapsulates both rapid rotation and vigorous convection. A plane layer is considered rotating around a central axis, features of the flow like the thermal and viscous boundary layers will be monitored to ascertain the effect of hyperdiffusion. Force balance spectra are considered to ensure the force balances expected in the core are not perturbed. Using this holistic approach, the overall effect of hyperdiffusion is explored in order to determine the optimal method.

A pseudopotential multiphase lattice Boltzmann method with adaptive mesh refinement (AMR) and large density ratio

Wandong Zhao[1,2], Ralf Deiterding[2], Keertan Maskey[2], Mikael Grondeau[2], Jian-han Liang[1]

[1]Science and Technology on Scramjet Laboratory, College of Aerospace Science and Engineering, National University of Defense Technology, Changsha 410073, China
[2]Aerodynamics and Flight Mechanics Research Group, University of Southampton, Boldrewood Innovation Campus, Southampton SO16 7QF, United Kingdom

The pseudopotential multiphase lattice Boltzmann method (LBM) is a diffusion model, which often suffers from insufficient resolution in the phase interface. The spurious velocity between the interface is also linked with the width of the interface, especially for the multiphase flows with a high-density ratio. Consequently, the large deformation of bubbles or droplets is quite difficult to handle. In the current study, the adaptive mesh refinement method and parallel calculation of the pseudopotential LBM are developed, maintaining a high mesh resolution in the interface and low spurious velocity even with a high-density ratio of 1000. The AMR-LBM technology is introduced comprehensively, especially for the multiphase model with a pseudopotential force. The thermodynamic consistency of the two-phase model and the surface tension of a single droplet are numerically tested with different AMR levels. The results demonstrate that the thermodynamic consistency is in agreement with the Maxwell resolution with an increasing mesh resolution, especially for a multiphase model having a high-density ratio. The surface tension of the droplet is collapsed as one with an increasing mesh resolution. The AMR-LBM solver in the current study enhances the ability of the pseudopotential multiphase LBM in more complicity flows even with a large shape deformation of droplets or bubbles in an engineering application.

Unsteady wing loading from tufts

Francis De Voogt and Bharath Ganapathisubramani

University of Southampton

Unsteady separated flow is present in many applications from cars to wind turbines. During the design phase simulations and wind tunnel experiments typically can only provide limited information due to limitations by each method. The use of tufts to validate the surface flow can be extended to provide quantitative information. Tuft surface flow visualisation can be performed on small to large objects offering the ability to investigate full scale products. In the current investigation use is made of concurrent tuft surface flow visualisation and force measurements on a wing to illustrate the ability of data driven methods to directly estimate the unsteady aerodynamic loading based on surface tufts. An investigation based on 3D URANS simulations of a NACA0012 wing showed a similar frequency content for the surface flow changes and the lift force fluctuations. A linear approach has been constructed which can be utilised to estimate the lift fluctuations based on the POD coefficients of the tuft angles, a non-linear extension to this method based on the universal approximation theorem showed further improvements for the estimated lift fluctuations. Lastly experiments have been conducted with tufts on a NACA0012 wing which were recorded at 120 fps. The wing was positioned at an angle of attack and Reynolds number combination which produced highly unsteady surface flow, during which forces were recorded. With the use of a two-layer neural network it has been found that the lift and moment fluctuations can be estimated at each instance based on the tuft angles.

New understanding of heat transfer deterioration for supercritical fluid flows

Zewen Zou and Shuisheng He

The University of Sheffield

When the temperature of a fluid rises above its critical value, the thermophysical properties of the fluid undergo drastic changes without phase change. This characteristic makes supercritical fluid an efficient working fluid in a variety of engineering systems. Extensive experimental and numerical work has been done to investigate the complex heat transfer and turbulence characteristics of such fluids, and pertinent correlations and modelling methods have been developed to describe the flow and thermal physics of such fluid. The insights gained from these studies serve as good references and guidelines for the design of engineering systems.

Recently a so-called apparent Reynolds number ideal has been proposed to explain laminarisation of a turbulent flow caused by an idealized non-uniform streamwise body force. This concept was further studied in recent publications to be applied to real flows, including a heated upward air flow and a heated flow of CO2 at supercritical pressure. In both cases, the theory explains the turbulence behavior well. In this study, we extend this approach (which was only used to explain turbulence behavior) to describe the heat transfer deterioration process and attempt to establish a Nusselt number correlation for mixed convection. We first derive the correlation using the ARN concept, and then numerically test it on some heated flows using Ansys Fluent. Following these tests, the understanding/correlation will be applied to supercritical fluid flows with heating. The current study will contribute to a better understanding of the heat transfer characteristics of supercritical fluid flows.

Aerodynamic performance of flatback aerofoils for vertical axis wind turbine

Khalid M Alzahrani, Jee Loong Hee, Derek Ingham, Lin Ma and Mohammed Pourkashanian

ENERGY 2050, Department of Mechanical Engineering, University of Sheffield

Although horizontal axis wind turbines (HAWTs) dominate large-scale wind power generations, the H-type vertical axis turbine (VAWT) demonstrates a substantial advantage for specific conditions, such as regions with rapidly varying wind directions e.g., in urban areas and small-scale applications. The ability to self-start and increased efficiency are the main considerations in improving the performance of VAWTs. However, investigations into the aerodynamics of VAWTs is a challenge due to the complexity of the flow phenomena around the turbine, and this is an active area of research. With the aid of CFD, the aerodynamic performance of a three-bladed VAWT was examined using a modified aerofoil, namely the flatback aerofoil (also called the blunt trailing edge aerofoil). Flatback aerofoils were originally developed for use in HAWTs and has been demonstrated to greatly improve the lift characteristics of the aerofoil. However, investigation of their performance in H-type VAWTs has not been presented in depth in the literature. Therefore, 2D unsteady Reynolds-averaged Navier-Stokes (URANS) simulations were carried out to investigate the flow physics around a VAWT fitted with a flatback aerofoil. Different profiles of the flatback blades (varying thicknesses at the aft half of the blade) were considered and their aerodynamic performance compared with that of a NACA0018 aerofoils (which was served as a baseline). Both turbines were evaluated by dynamic analysis of the instantaneous moment coefficients, power curves, flow field characteristics, self-starting ability and the vorticity fields surrounding the turbine. The turbine with flatback blades produced a higher power coefficient over a wide range of tip speed ratios (TSRs) than the baseline turbine while maintaining the self-starting capability. Also, with further modification to the trailing edges, an improvement in self-starting performance and a small negative effect at high TSRs were observed.

Electromagnetic effects on the stability on scrape-off layer plasmas

George McGilvray

University of Leeds

Previous research has investigated various aspects of the linear instability of plasma in a tokamaks, exploiting the analogy with Rayleigh-Bénard convection. Understanding the complex dynamics in a plasma requires several assumptions which may not necessarily be representative of actual plasma instabilities in a tokamak. In particular the standard approach in this research is to assume an electrostatic plasma. Our research focuses on how the electromagnetic effects affect the stability of the plasma. We show our preliminary work including the setting up of the electromagnetic plasma linear stability problem, some of our analysis of a reduced problem and a discussion of our plans for future work including numerical simulations and the possibility of comparing our results to experimental data.

A resolvent-based variational method for statistical prediction of channel flow

Thomas Burton, Sean Symon and Davide Lasagna

University of Southampton

Low-dimensional chaotic dynamics, that is dynamics on a strange attractor, can be viewed as an exploration of a dense set of invariant periodic solutions, known as Unstable Periodic Orbits (UPOs), which form a "skeleton" of the attractor itself. Taking suitable ensemble averages of these UPOs, known as a cycle expansion, can efficiently reconstruct important features of the dynamics (the Lyapunov exponent, escape rate, etc.). In addition, a sufficiently long period UPO can act as a surrogate for the cycle expansion providing useful statistical predictions of the long-time dynamics on its own. In turbulent fluid flows invariant periodic solutions known as Exact Coherent Structures (ECSs) have been found via LES/DNS simulations. There is some evidence that these structures can serve an analogous purpose to UPOs for the high-dimensional chaos of turbulent flow.

In the work presented, a variational method is used to drive periodic flow fields towards ECSs present in the turbulent state-space. The resulting periodic flow does not have to be a true solution of the Navier-Stokes equations to resemble ECSs making up the space. Therefore an optimisation approach is employed, rather than a root-finding approach. This optimisation is formulated in terms of Resolvent modes which are an efficient basis for the velocity field. The Resolvent modes can be computed from the linearised Navier-Stokes equations assuming the mean flow is known a priori.

The methodology has been developed for a streamwise independent Rotating-Plane Couette Flow (a Taylor-Couette flow in the limit of a small gap between the rotating cylinders). Special attention is given to the simultaneous enforcement of incompressibility and the no-slip condition at the boundaries of the flow.

Internal and external dynamics of coalescing non-isothermal droplets

Yatin Darbar[1], Mark Wilson[2], Harvey Thompson[2], David Harbottle[3] and Thomas Sykes[4]

[1]CDT in Fluid Dynamics University of Leeds.
[2]School of Mechanical Engineering, University of Leeds
[3]School of Chemical and Process Engineering, University of Leeds
[4]Department of Engineering Science, University of Oxford

Droplet coalescence is a crucial component of emerging technologies including inkjet printing of bio-materials, lab-on-a-chip microfluidic systems and environmentally-friendly sprays. In these applications, efficient mixing is required, especially where a chemical reaction or biological process is instigated. However, mechanisms of mixing between coalescing droplets remain poorly understood. This work focuses on understanding and elucidating this by means of numerical simulation.

The novelty of this project lies within simulating transport effects, such heat transfer and chemical reactions, which will require accurate capturing of the intricate free surface and internal interface motion. Additionally, capturing the motion of the contact line in situations where a substrate is present is imperative. Together, these will allow for an improved numerical description of the mixing within coalescing droplets.

Using the latest version of OpenFOAM, the first necessary step towards obtaining reliable simulations is to examine the interfacial resolution and numerical schemes needed to capture the free surface dynamics of coalescence. This was achieved by comparison with experimental data within the literature. The numerical modelling of chemical reactions has been advanced towards by incorporating the effects of heat transfer into the numerical scheme. This facilitated an exploration of thermal transport effects within droplet coalescence. Validation of these methods against an analytic solution confirmed heat transfer over the interface is captured accurately. This numerical model can now be extended to incorporate temperature dependent material properties and chemical reactions. Once established, experimental data will be used to verify the results.

Fusion technology: Experiments with digital twinning

Dominykas Buta

University of Leeds

It is becoming widely accepted that crucial to the design and qualification of technology for a fusion reactor will be the development of highly advanced coupled simulations and digital-design methods. Development of virtual engineering capabilities to provide an automated multi-physics simulation workflow for enhancing understanding and ultimately predicting the behaviour of fusion reactor systems is of interest. The driving concept to this is digital twinning. By coupling real-world data with virtual modelling, it would enable extensive and data-rich virtual testing and qualification of a close digital representation of a physical asset under conditions that currently are only feasible within costly physical tests of a complete fusion device. However, first a high-quality real-world data source is required to validate and then to enhance virtual modelling. UK Atomic Energy Authority (UKAEA) in commitment to realise the digital twinning concept are currently building a unique components loading device named CHIMERA (combined heating and magnetic research apparatus), objective of which is to perform experiments of varying complexity and realism on component mock-ups of a fusion device, to provide high volume of suitable high-quality data. Secondly, a high-order numerical methodology capable of accurately capturing multi-scale physics experienced within fusion reactors and that would also be a candidate to achieve real-time predictions is desired. Although a suitable numerical methodology of interest exists, namely the spectral/hp element method, it is still relatively novel and requires extensive validation studies to be carried out. This work selects Nektar++ software package, an open-source cross-platform spectral/hp element framework, to carry out a series of validation studies on fusion relevant fluid dynamics problem, namely the classical vertical slot problem, that will ultimately lead to more validation studies of multi-physics models at CHIMERA facilities.

COVID-19 transmission risks associated with environmental contamination in workplace and public toilets

Ciara Higham[1], Louise Fletcher[2], Catherine Noakes[3], Martín López-García[4]

[1]CDT Fluid Dynamics, University of Leeds
[2]School of Civil Engineering, University of Leeds
[3]School of Mathematics, University of Leeds

Shared toilets, both in the workplace and public toilets, are a known facilitator of disease transmission. The current COVID-19 pandemic has illustrated the absolute need to reduce the risk of disease transmission in such an essential and highly frequented environment. There are two key mechanisms of disease transmission when an infected individual uses a toilet: through respiratory emission and through the toilet plume generated when flushing the toilet. When flushing the toilet, up to 145,000 droplets can be produced of varying sizes [1]. Both types of emission release pathogens into the environment which may be infectious. The pathogens will be released in larger droplets that settle on surfaces quickly due to gravity and smaller ones which evaporate and can remain airborne for hours, eventually settling on surfaces. When a susceptible individual uses the toilet they can subsequently become infected. This can happen when the susceptible individual inhales the airborne pathogen or when they touch contaminated surfaces and then touch eyes/nose/mouth. This project focuses specifically on the SARS-CoV-2 virus but is applicable to other pathogens. Combining experimental work, Computational Fluid Dynamics (CFD) and mathematical risk modelling will provide an insight into the likelihood of disease transmission in shared toilets. The combination of experimental and CFD work will quantify the exposure an individual has to the virus which can then be fed into a dose-response model. The effectiveness of mitigation strategies such as ventilation, far UV-C and room layout will be analysed using CFD to determine the most appropriate measures. This insight can then be fed back to health agencies to inform policies.

[1]D. L. Johnson et al.," Lifting the lid on toilet plume aerosol: A literature review with suggestions for future research, American Journal of Infection Control, DOI:10.1016/j.ajic.2012.04.330

Investigation and evaluation of scalable liquid-metal MHD solvers for fusion breeder blanket applications

Rupert W Eardley, Aleksander J Dubas and Andrew Davis

UKAEA (United Kingdom Atomic Energy Authority)

Fusion holds great potential as a clean, sustainable power source for the future, with magnetic confinement fusion devices known as tokamaks likely to be the first fusion technology to become commercially viable, such as the UK's STEP demonstration power plant. A key component of these devices for the generation of net power and closing the fuel cycle is the breeder blanket. Some designs rely on the flow of a liquid metal such as lithium, providing a medium for producing tritium from the reaction of lithium with neutrons produced by the reactor. Liquid-metal flows in the presence of strong magnetic fields are governed by magnetohydrodynamics (MHD), a coupling of fluid mechanics to electromagnetism, due to the flow of currents in the fluid which exert a Lorentz force on the fluid with feedback strongly coupling the magnetic field and the flow. Computational modelling of these flows is a key part of understanding these complex systems, with analytic solutions difficult to obtain beyond the simplest cases due to the non-linearity of the MHD equations. While codes for solving these problems exist, a major challenge is tackling complex multiphysics problems in order to study large integrated components of fusion devices, requiring highly scaleable open-source software that can be incorporated into multiphysics packages such as MOOSE. This work outlines methods of introducing MHD effects into the Navier-Stokes equations in order to study incompressible liquid-metal duct flows. An investigation into existing liquid-metal MHD solvers compares the different formulations and the validity of their implementations, as well as assessing their efficiency in terms of parallel scaling. This research provides an understanding of available codes for highly scalable MHD simulation capable of interfacing with other physics, advancing the path towards an efficient solution to the modelling of liquid-metal flows in magnetic confinement fusion devices.

Perspectives of women in fluid dynamics

University of Nottingham

Mark Jabbal

The purpose of this work is to summarise key responses from women working in fluid dynamics, in academia and industry, on perceived barriers in the field and opportunities to improve equity. These responses have been collected almost weekly since 2018 as part of a wider social media initiative, '@AeroWomen' (https://twitter.com/AeroWomen), which to-date has featured more than 180 women, with the aim of highlighting their contributions in fluid dynamics through published Q&As. In addition, responses from almost 100 women has been obtained and anonymised on issues in the fluid dynamics field including: career barriers faced (tinyurl.com/y9tpa5sn), obstacles to newcomers (tinyurl.com/y7t7ahp2), and supporting career progression (tinyurl.com/ya4eh2zx). In addition to providing qualitative summaries on these key issues, further work to analyse responses more in-depth to provide quantitative summaries will be undertaken and presented as part of this work.

Author	Paper
A.F. Abusewal	107
N. Afify	P15
R. Agrawal	72
T. Ala	P17
A.I. Aleksyuk	35
Y. Almalki	47
A. Alsenafi	P18
K.M. Alzahrani	P28
J. Anderson	66
E. Andrews	14
A. Arslan	32
Y.D. Baba	2
H. Barrio-Zhang	123
C. Beaume	34
V. Bellos	71
J. Bennett	P19
O. Bidar	103
J.C. Bilbao-Ludena	135
G. Booth	121
T. Boulafentis	166
H. Broadley	24
B. Bugeat	63
D. Bullamore	148
D. Burnett	111
T. Burton	P30
D. Buta	P32
E. Butler	P21
D.W. Carter	15
J. Castelino	94
C.F. Castro-Faccetti	163
A. Chakra	3
R. Chatterjee	165
B. Chen	129
K. Chinembiri	P36
S. Chu	96
M. Cicolin	106
R. Cimpeanu	91
A. Cleary	128
D. Clements	126
A. Court	78
M. Cox	44
C. Cuciumita	57
H.M. D'Ambrosio	145
Y. Darbar	P31
B. Davy	P24
F. Dent	85
C. Duguid	29
Craig Duguid	Thesis Prize
R.W. Eardley	P34
A.J. Edwards	36
I. El Mellas	21

Author	Paper
M. Engel	154
M. Falcone	140
M. Fatehifar	105
G. Fedrizzi	149
J. Finneran	68
M. Fiori	133
D. Frank	138
J. Frank	P4
S. Gambuzza	120
R. Gilbert	10
F. Gori	11
E. R. Gutierrez	65
M. Hajihassanpour	27
N. Hanevy	13
A.B. Harish	160
J. He	125
C.E. Heaney	162
Y. Hengster	169
C. Higham	P33
J.M. Hill	16
J. Hirst	127
C. Hocking	67
A. Hodgkin	9
Julien Hoessler	KN-1
G. Huang	41
J. Huang	89
G. Hunter-Brown	101
V.C. Ibezim	113
M. Jabbal	P35
S. Jackson	131
C. Jane	12
T.O. Jelly	108
A. Jenkins	P14
E.M. Jolley	130
P. Kandangwa	104
K. Kechagidis	45
J. Keeler	124
S. Khodaparast	90
S. Kose	84
A. Kumar	46
M. Kurt	83
M. Lahooti	122
S. Lalloz	76
A. Lang	P2
I. Latimer	P16
M. Lellep	61
P. Lewin-Jones	60
L. Li	117
K. Li	70
H. Liaquet	80
H.D. Lim	4
B. Liu	153

Author	Paper
M. MacRaild	55
M. Magnini	86
O. Maklad	115
C. Mallorie	167
S. Mandhani	22
J.-P. Marx	141
K.K. Maskey	110
E. Masoudi	8
D. McCaughey	40
G. McGilvray	P29
G. McHale	20
B. McLaughlin	75
D. Mesa	100
M. Mesgarpour	P23
L.M. Mills	64
F. Mirghaderi	168
F.A. Morris	144
L. Morrow	147
P. Mudiyanselage	77
A. Naga	23
S. Nicholas	54
Z. Nissar	56
Catherine Noakes	KN-2
H. Ong	P5
A. Othman	164
B. Owen	102
S. Owens	30
J.R. Panter	88
O. Paulin	116
M. Pelizzari	P6
J. Perez	38
D. Pickles	81
A. Potherat	48
T. Preskett	73
R. Quadros	79
I. Qureshi	157
R. Raj	33
C.R. Rees-Zimmerman	7
Q. Ren	P10
T. Rich	28
A. Roy	51
K. Sabnis	74
M. S. Sadullah	50
A. Saxena	137
R.K. Schlander	87
C.S. Skene	109
C. Stafford	82
C. Stafford	152
J. Stout	P3
Y. Sun	119
M.L. Taccari	114
C. Thomas	11

Author	Paper
C. Thompson	49
U. Thorat	92
K. Thota	139
M.R. Threadgold	7
A.M. Tomy	17
A. Tully	97
T.H. Un	99
C. Vaquero-Stainer	155
R. Vilumbrales-Garcia	69
G. Vivarelli	151
F. Voogt	P26
C. Vouriot	18
Nathalie Vriend	KN-3
G. Walton	52
H. Kusumaatmaja	6
W. Wang	161
J. Wang	19
D.D. Wangsawijaya	25
C. Wilk	5
Y.H. Wong	53
B. Yang	P8
M.Y. Yazici	95
G. Ye	58
Q. Yu	P1
G. Zampino	132
L. Zhang	112
J. Zhang	146
C. Zhang	P37
W.B. Zimmerman	59
Z. Zou	P27
A. Zurman-Nasution	93