38th Scottish Fluid Mechanics Meeting

28th May 2025

Glasgow

Contents

About

SFMM38

The Scottish Fluid Mechanics Meeting, now in its 38th edition, is an annual gathering that brings together researchers working in the field of fluid mechanics from Scotland and beyond.

This informal event welcomes contributions from all areas of study and provides the perfect forum to discuss and engage with cutting-edge research being carried out in Scottish universities and other institutions. The event is particularly designed to give PhD students and early career researchers the opportunity to present their work and receive constructive feedback, but contributions are also welcome from scientists at all career stages, from PDRAs to senior professors.

Organising committee

Lead organisers

Dr Matthew Durey
Dr Katarzyna Kowal

Committee members

Dr Hannah-May D'Ambrosio
Dr Wrik Mallik

On-the-day assistants

Abdulrahman Alenezi
Hessah Almaaz
Mariam Al Mudarra
Tanisha Kumari
Ijuptil Joseph Kwajighu
Geng Qiao
Ke Wang
Jiaming Zhang

IT assistant

Haolin Yang

Sponsors

Dantec Dynamics pioneers the development and distribution of cutting-edge integrated measurement systems catering to diagnostics and research in fluid mechanics, solid mechanics, microfluidics, spray analysis, and combustion technology. Renowned globally, our clientele entrusts our user-friendly systems and unparalleled application support to drive the forefront of discovery, innovation, quality control, and non-destructive testing.

Dantec Dynamics – Turn Measurements Into Knowledge

www.dantecdynamics.com

LaVision is the leading supplier of laser imaging systems in top selling application fields like fluid mechanics (aerodynamics, microfluidics), combustion (automotive, power generation) as well as spray and particle diagnostics (engines, pharma). Due to their non-intrusive nature our optical instruments offer unique capabilities for multi-parameter flow measurements with high spatial and temporal resolution. Our measurement equipment is used in well known R&D labs all over the world. Largely customer oriented, we offer user-friendly, reliable and high quality products tailored to the needs of our customers.

www.lavision.de/en/

Programme

8:30—9:50	**Registration, breakfast, poster session**	
9:50—10:00	**Welcome and introduction**	
10:00—12:00	**Session 1**	**Chair: David Rees Jones (St Andrews)**
10:00—10:15	**Multiscale modelling of fluid and drug transportation in malignant tumours** Andrew Brown, Raimondo Penta *School of Mathematics and Statistics, University of Glasgow*	
10:15—10:30	**Solute transport in the oscillatory flow of cerebrospinal fluid** Alannah Neff *School of Mathematics, University of Edinburgh*	
10:30—10:45	**Mass transport induced by infragravity wave** Zhiling Liao, Qingping Zou *The Lyell Centre for Earth and Marine Science and Technology, Institute for Infrastructure and Environment, Heriot-Watt University*	
10:45—11:00	**Modelling fluid-soil regime transitions in the sedimentation of clay suspensions** Yuan Yi, Alan Cuthbertson, Tom Eaves *School of Science and Engineering, University of Dundee*	
11:00—11:15	**Icequakes at the nanoscale: inducing heterogeneous ice nucleation via vibration** Pengxu Chen, Rohit Pillaia, Saikat Datta *School of Engineering, University of Edinburgh and Faculty of Science and Engineering, Swansea University*	
11:15—11:30	**Force balances characteristic of different dynamo regimes** Ayesha Sarwar, Robert Teed, Radostin Simitev *School of Mathematics and Statistics, University of Glasgow*	
11:30—11:45	**Topology-preserving finite element methods for magnetic relaxation** Mingdong He, Patrick E Farrell, Kaibo Hu, Boris D Andrews *Mathematical Institute, University of Oxford and School of Mathematical Sciences, University of Edinburgh*	
11:45—12:00	**Thermo-electrohydrodynamic convection in a cylindrical annulus during TEXUS57th sounding rocket flight: Numerical simulation** Peter Szabo, Olivier Crumeyrolle, Antoine Meyer, Christoph Egbers *Department of Aerodynamics and Fluid Mechanics, Brandenburg University of Technology and Laboratoire Ondes et Milieux Complexes (LOMC), Université Le Havre Normandie*	
12:00—12:10	**Group photo**	
12:10—13:15	**Lunch and poster session**	

Programme

13:15—15:00	**Session 2**	**Chair: Alex Wray (Strathclyde)**
13:15—13:30	**Hydrodynamic characterisation of produced water in conventional deoiling hydrocyclones** Okwunna Maryjane Ekechukwu, Taimoor Asim *School of Computing, Robert Gordon University*	
13:30—13:45	**Electrohydrodynamic interactions of a pair of leaky dielectric drops** Michael McDougall, Stephen K Wilson, Debasish Das *Department of Mathematics and Statistics, University of Glasgow and Department of Mathematical Sciences, University of Bath*	
13:45—14:00	**Investigating advancing contact lines for microdroplets on circular micropillared surfaces** Janice To, Khellil Sefiane, Rodrigo Ledesma-Aguilar, Daniel Orejon *Institute for Multiscale Thermofluids, School of Engineering*	
14:00—14:15	**Particle deposition from an unpinned particle-laden sessile droplet undergoing one-sided evaporation** Henry T Sharp, Stephen K Wilson, Alexander W Wray *Department of Mathematics and Statistics, University of Strathclyde and Department of Mathematical Sciences, University of Bath*	
14:15—14:30	**The role of the diffusive boundary layer in deposition from sessile droplets** Alexander W Wray, Hannah-May D'Ambrosio, Madeleine R Moore *Department of Mathematics and Statistics, University of Strathclyde; School of Mathematics and Statistics, University of Glasgow and Department of Mathematical Sciences, Loughborough University*	
14:30—14:45	**Non-porous viscous fingering of a gravity-driven free-surface flow** Haolin Yang, Katarzyna N Kowal *School of Mathematics and Statistics, University of Glasgow*	
14:45—15:00	**The effect of capillary forces on two-layer flow down an incline** Hannah-May D'Ambrosio, Katarzyna N Kowal *School of Mathematics and Statistics, University of Glasgow*	

15:00—15:30	**Coffee break**	

15:30—17:15	**Session 3**	**Chair: Tom Eaves (Dundee)**
15:30—15:45	**Sports aerodynamics: an overview of activities at Heriot-Watt University** Fabio Malizia, Xiaoqi Hu, Bert Blocken *School of Engineering & Physical Sciences, Heriot-Watt University*	
15:45—15:45	**Autoencoders for Lagrangian Computational Fluid Dynamics** Joseph O'Connor *Edinburgh Parallel Computing Centre, University of Edinburgh*	

16:00—16:15	**On-the-fly Lagrangian averaging** Abhijeet Minz, Lois E. Baker, Jacques Vanneste, Hossein A Kafiabad *School of Mathematics and Maxwell Institute for Mathematical Sciences, University of Edinburgh and Department of Mathematical Sciences, Durham University*
16:15—16:30	**Spatial tiling and localized shadowing in 2D Kolmogorov flow** Dmitriy Zhigunov, Jacob Page *School of Mathematics and Maxwell Institute for Mathematical Sciences, University of Edinburgh*
16:30—16:45	**Purely elastic turbulence in channel flows** Damiano Capocci, Moritz Linkmann, Alexander Morozov *School of Physics and Astronomy, University of Edinburgh*
16:45—17:00	**Transition to turbulence in the Stokes boundary layer: edge states and the periodic self-sustaining process** Jorge Sandoval, Tom Eaves *School of Science and Engineering, University of Dundee*
17:00—17:15	**Impact of steam on the thermodiffusive instability in hydrogen premixed flames** Sofiane Al Kassar, Hunyan Zubair, Innes Cameron, Khushboo Pandey, Antonio Attili *School of Engineering, Institute for Multiscale Thermofluids, University of Edinburgh*
17:15-17:20	**Closing remarks**

Posters

1. **Controlling droplet size density during dropwise condensation on silicone oil grafted surfaces**
   Anam Abbas, Zafar Iqbal, Gary G Wells, Glen McHale, Khellil Sefiane, Daniel Orejon
   School of Engineering, Institute for Multiscale Thermofluids and Institute for Energy Systems, University of Edinburgh; Department of Mechanical Engineering, University of Engineering and Technology, Lahore and International Institute for Carbon-Neutral Energy Research (WPI-I2CNER), Kyushu University
2. **Effect of free-stream turbulence on the trajectories of free-falling discs**
   Panagiotis Alexandrou, Chandan Bose, Ignazio M Viola, Anotnio Attili
   School of Engineering, Institute for Multiscale Thermofluids and Institute for Energy Systems, University of Edinburgh and Aerospace Engineering, University of Birmingham
3. **Rayleigh-Bénard convection in the weakly elastic regime: insights from water-based liquid bridges**
   Ahmed Aljanadi, Monica Oliveira, Marcello Lappa
   James Weir Fluids Laboratory, The Department of Mechanical and Aerospace Engineering, University of Strathclyde
4. **Pulsatile fluid flow through porous membranes**
   Hessah Almaaz, Matthew Durey, Katarzyna N Kowal
   School of Mathematics & Statistics, University of Glasgow
5. **A model for walking droplets over submerged barriers**
   Zubaydah Alotaibi, Katarzyna N Kowal, Matthew Durey
   School of Mathematics and Statistics, University of Glasgow
6. **Convolutional Neural Networks (CNNs) for turbulent combustion modelling**
   Dikran Bakkalian Julian, Geveen Arumapperuma, Antonio Attili
   School of Engineering, Institute for Multiscale Thermofluids, University of Edinburgh
7. **Convection and dynamo action in rotating spherical shells with different radius ratios**
   Francesca Coke, Rob Teed
   School of Mathematics and Statistics, University of Glasgow
8. **Simulations of free-falling plates in perturbed flows**
   Katrina N Connolly, Panagiotis Alexandrou, Chandan Bose, Ignazio M Viola, Anotnio Attili
   School of Engineering, Institute for Multiscale Thermofluids and Institute for Energy Systems, University of Edinburgh and Aerospace Engineering, University of Birmingham
9. **Evaporation of a droplet on a porous substrate**
   David Craig, Stephen K Wilson, Alexander W Wray, Khellil Sefiane
   Department of Mathematics and Statistics, University of Strathclyde; Department of Mathematical Sciences, University of Bath and School of Engineering, Institute for Multiscale Thermofluids, University of Edinburgh

10. **Modelling gas-granular flow due to an impinging plume in the RECAP framework**
Jack L Davies, Andrea Cammarano, Hossein Zare-Behtash, Kevin Worrall
James Watt School of Engineering, University of Glasgow
11. **Direct Numerical Simulation of evaporating droplet populations**
Debarshi Debnath, Joseph J Kilbride, George Karapetsas, David Scott, Daniel Orejon, Khellil Sefiane, Prashant Valluri
School of Engineering, University of Edinburgh; School of Physics, University of Edinburgh; Department of Chemical Engineering, Aristotle University of Thessaloniki and Edinburgh Parallel Computing Centre, University of Edinburgh
12. **Drying and filtration effects on the bheological Behavior of aqueous PEO solutions**
Itzia M García, John J Liggat, Mónica S N Oliveira
Department of Mechanical and Aerospace Engineering and Department of Pure and Applied Chemistry, University of Strathclyde
13. **Magnetoconvection and transitions in Earth-like dynamos**
Luke Gostelow, Robert Teed
School of Mathematics and Statistics, University of Glasgow
14. **Investigating the role of circulating tumour cell softness in inertial microfluidic devices**
Roslyn Hay, Timm Krüger, Benjamin Owen
School of Engineering, University of Edinburgh
15. **Single molecule dynamics in microfluidic flows: from homogeneous extension to mixed kinematics**
Arezoo Khakpour, Zhibo Li, Samatha Robinson, Olivia du Roure, Emad Chaparian, Anke Lindner, Mónica S N Oliveira
James Weir Fluids Laboratory, Department of Mechanical and Aerospace Engineering, University of Strathclyde and Centre National de la Recherche Scientifique (CNRS), ESPCI Paris PSL
16. **Mathematical modelling of ice-sheet dynamics**
Tanisha Kumari, Matthew Durey, Peter Stewart, Katarzyna N Kowal
School of Mathematics and Statistics, University of Glasgow
17. **Physics-based models for large-eddy simulations of (un)steady magnetohydrodynamic turbulence**
Asif Nawaz, Damiano Capocci, Moritz Linkmann
School of Mathematics and Maxwell Institute for Mathematical Sciences, University of Edinburgh and SUPA, School of Physics and Astronomy, University of Edinburgh
18. **Investigation of biomimetic riblet structures in a turbulent channel flow**
Antreas Papasavvas, Panagiotis Alexandrou, Pasquale E Lapenna, Antonio Attili
School of Engineering, Institute for Multiscale Thermofluids, University of Edinburgh and Department of Mechanical Engineering and Aerospace Sapienza, University of Rome

19. **Cavitation abrasive surface engineering (CASF) process for sustainable titanium alpha case removal**
Elangovan Parameswaran, Evgenia Yakushina, Xichun Luo, Daniel G Sanders
Department of Design, Manufacture and Engineering Management, National Manufacturing Institute Scotland and Advanced Forming Research Centre, University of Strathclyde
Department of Mechanical Engineering University of Washington
20. **Influence of lower limb amputation on the cardiovascular system – a CFD investigation**
Sava Popescu, Arjan Buis, Asimina Kazakidi
Department of Bioengineering, University of Strathclyde
21. **Numerical simulation and validation of concentration polarisation effects of sucrose aqueous solutions in nanofiltration membranes**
Aniol Puigdefabregas Nogueras, Asimina Kazakidi, Emmanouil H Papaioannou
Biomedical Engineering, University of Strathclyde and School of Engineering, Lancaster University
22. **Propeller-wing aerodynamic interactions**
Geng Qiao, George N. Barakos
CFD Laboratory, School of Engineering, University of Glasgow
23. **Flow characterisation of oscillating jet nozzles for wellbore descaling**
Mubaraq Adewale Razaq, Okwunna Maryjane Ekechukwu, Khaliq Ur Rahman, Taimoor Asim, Murphy Lai
School of Computing, Robert Gordon University
24. **Evaporation of an elongated droplet**
Lauren B Schofield, Alexander W Wray, Debasish Das, Stephen K Wilson
Department of Mathematics and Statistics, University of Strathclyde and Department of Mathematical Sciences, University of Bath
25. **Physics-informed neural networks for predicting Rhodamine diffusion coefficient**
Quinn Stein, Humphrey Yiu, Ali Ozel
School of Engineering and Physical Sciences, Heriot-Watt University
26. **Ferrofluid emulsion in shear flow: Effects of viscosity ratio and magnetic field**
Ayaz Sultan, Paolo Capobianchi, Mónica S N Oliveira
James Weir Fluids Laboratory, Department of Mechanical and Aerospace Engineering, University of Strathclyde
27. **Pore- and core-scale observation of gas-water intermittent flow in natural porous media: Unravelling intermediate region between capillary and viscous dominance**
Amin Taghavinejad, Azibayam Josiah Amabogha, Yihuai Zhang
School of Engineering, University of Glasgow

28. **Thermo-hydraulic performance analysis of a hybrid wavy bifurcated microchannel heat sink**
Evans J Udom, Marcello Lappa
Department of Mechanical and Aerospace Engineering, University of Strathclyde
29. **Vibro-fluidization of lunar regolith simulant in hourglass and inclined pipe configurations**
Peter Watson, Sebastien V Bonnieu, Ali Anwar, Marcello Lappa
Department of Mechanical and Aerospace Engineering, University of Strathclyde and European Space agency, European Space Research and Technology Centre
30. **Parallelised convolutions for generative models of plasma turbulence**
Josh Williams, Jony Castagna
Hartree Centre, STFC Daresbury Laboratory
31. **Quantifying the mechanical properties of biomolecular condensates**
Thomas A Williamson, Jack O Law, Thomas Stevenson, Carl M Jones, Sushma N Grellscheid, Halim Kusumaatmaja
School of Engineering, Institute for Multiscale Thermofluids, University of Edinburgh; Department of Biological Sciences, University of Bergen and Department of Biosciences, Durham University
32. **A biphasic flow model in idealised and engineered microvessels**
Tanchanok Wisitponchai, Laure Vidal-Roussel, Junxi Wu, Asimina Kazakidi
Department of Biomedical Engineering, University of Strathclyde
33. **Resonant triads of gravity waves in confined basins**
Jie Yang, Katarzyna Kowal and Matthew Durey
School of Mathematics & Statistics, University of Glasgow
34. **Adaptive mesh-free CFD modelling for rotorcraft**
Tao Zhang, George Barakos
James Watt School of Engineering, University of Glasgow

Information about talks and posters

Talks

- Oral presentations will be 12 minutes in length with an additional 3 minutes for discussion and transition from one speaker to the next.
- It is possible to use the computer provided in the room or your own.

Posters

- Posters must be A0-sized (841mm x 1189mm) in portrait orientation.

Getting to the venue

Venue and travel

SFMM 2025 will be held at the **Advanced Research Centre** (Seminar Suite and Atrium) at the main campus of the University of Glasgow. The Seminar Suite and Atrium are on the ground floor by the main entrance. Signs will be in place to direct you to SFMM.

There are excellent transport links between the City Centre and the West End via the Subway and bus services.

Travel from Glasgow International Airport

At the Airport take First 500 Glasgow Shuttle to Glasgow City Centre from Stance 1. Get off at Queen Street bus stop. Walk for 1 min to Glasgow Subway Queen Street station. Take the subway to Hillhead station. Walk for 5 minutes to the Advanced Research Centre.

Travel from Glasgow Central Rail Station

Walk for 5 min to Glasgow Subway St Enoch station. Take the subway to Hillhead station. Walk for 5 minutes to the Advanced Research Centre.

Travel from Glasgow Queen Street Rail Station

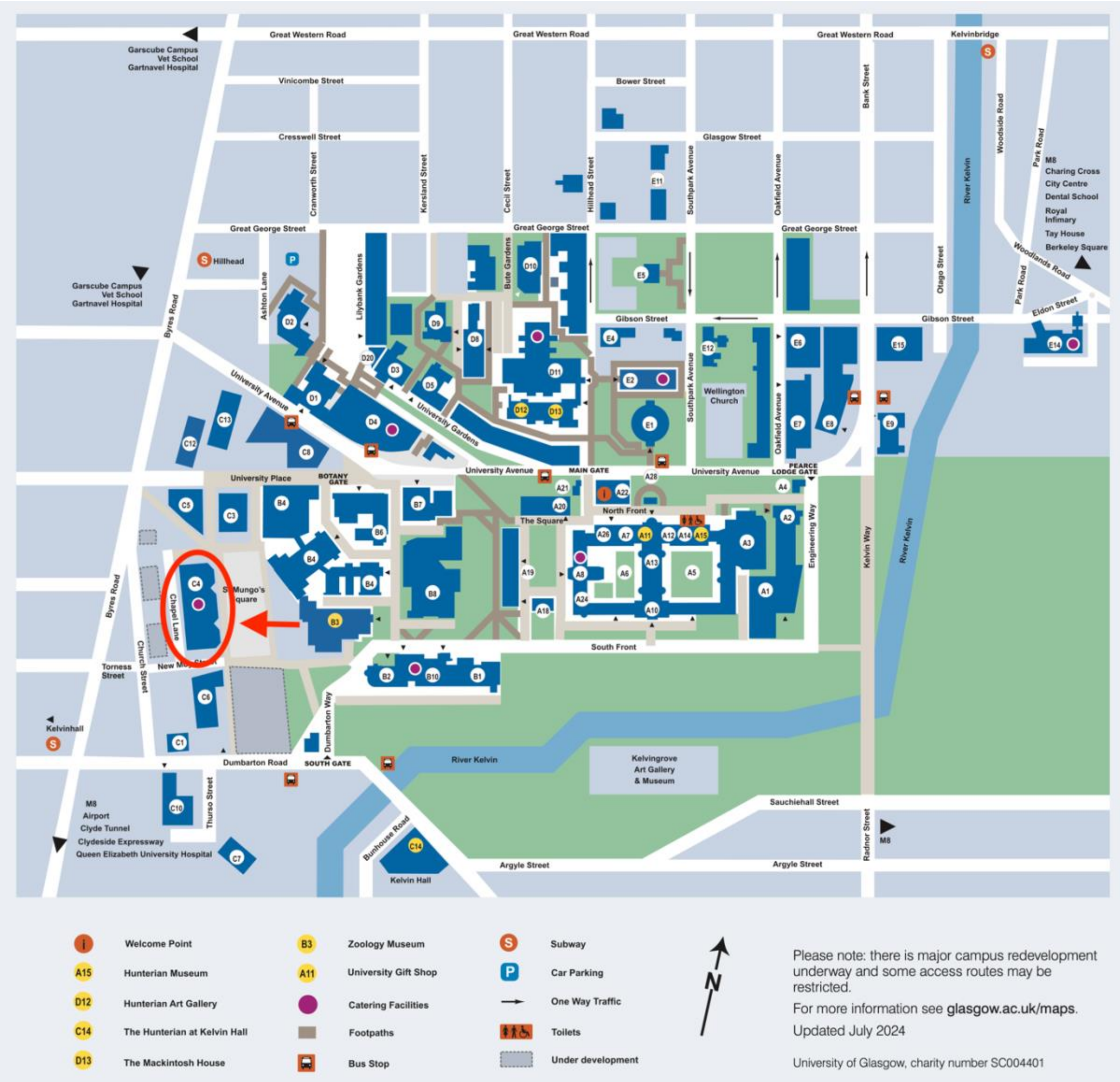
Walk for 2 minutes to Glasgow Subway Buchanan Street station. Take the subway to Hillhead station. Walk for 5 minutes to the Advanced Research Centre.

Travel by taxi

The address of your destination is

The Mazumdar-Shaw Advanced Research Centre,
11 Chapel Lane,
University of Glasgow,
Glasgow G11 6EW

The **Advanced Research Centre**, where SFMM is taking place, is marked with a red circle and arrow in the map below. The arrow points to the main entrance of the building.



Abstracts

Oral presentations

38th Scottish Fluid Mechanics Meeting

Multiscale modelling of fluid and drug transportation in malignant tumours

Andrew Brown and Raimondo Penta
School of Mathematics and Statistics
University of Glasgow

28th May 2025

Abstract

We create a system of differential equations for coupled fluid and drug transportation across three physical domains present in vascularised (malignant) tissue: the interstitial mass, capillary vessels and lymphatic vessels. Beginning from the mass and momentum balance equations in each physical domain, which is geometrically characterised by the mean inter-capillary distance (*microscale*). Kedem - Katchalsky equations[1] are used to account for fluid and drug exchange across the vessel walls. Employing asymptotic homogenisation[2] we can formulate a triple coupled porous medium problem characterised, by Darcy's law,[2] at the tumours characteristic length (*macroscale*). This is done under assumptions of continuity and local periodicity which allows for macroscale variations of the microstructure: accounting for spatial heterogeneity of the angiogenic nature of the capillary and lymphatic networks. Our homogenisation also creates a triple coupled advection-diffusion-reaction model, where the advection operator is dependent on the blood convection and diffusion-reaction operators are dependent on macroscopic variations of the microstructure. We demonstrate the validity of our model by deriving other models with a number of simplifications and discuss the potential our model has for numerical computation, by solving differential cell problems dependent on the microstructure. Our homogenised model treats the fluxes as source terms, highlighting the importance of fluid and drug exchange at the macroscale. We use the isotropic case of this model to find analytic solutions for pressure and fluid velocity, then perform a parametric analysis on several physiological parameters. Analysis reveals there is some optimal pressure and velocity of the fluid dependent on the permeability of the lymphatic vessels, which is of great interest for drug transportation, and in general we seek an optimal permeability. The overall purpose of this work is to create more general theoretical models further developing our understanding of the lymphatic vessels in fluid and drug transportation: helping us to improve our anti-cancer strategies.

References

[1] Wang, J., Dlamini, D. S., Mishra, A. K., Pendergast, M. T. M., Wong, M. C., Mamba, B. B., Freger, V., Verliefde, A. R., and Hoek, E. M., "A critical review of transport through osmotic membranes," *Journal of membrane science*, Vol. 454, 2014, pp. 516–537.

[2] Penta, R. and Gerisch, A., "An introduction to asymptotic homogenization," *Multiscale models in mechano and tumor biology: Modeling, homogenization, and applications*, Springer, 2018, pp. 1–26.

38^{th} Scottish Fluid Mechanics Meeting

Solute transport in the oscillatory flow of cerebrospinal fluid

Alannah Neff
School of Mathematics, University of Edinburgh
James Clerk Maxwell Building, Peter Guthrie Tait Road, Kings Buildings, Edinburgh, EH9 3FD

28^{th} May 2025

Abstract

Cerebrospinal fluid (CSF) is a clear, Newtonian fluid that envelops the brain and spinal cord within the subarachnoid space (SAS). The CSF experiences pulsations during the cardiac cycle due to fluctuations in brain volume: systole increases brain volume, pushing CSF into the spinal SAS, while diastole reduces brain volume, allowing CSF to return to the cranial SAS [1]. This motion is believed to be crucial for nutrient transport and clearing harmful substances, such as toxins and toxic proteins. Additionally, CSF provides a pathway for drug delivery through intrathecal administration [2]. This study focuses on understanding solute transport in CSF for enhancing metabolic waste clearance and drug delivery.

In this study, we develop a simplified model of CSF flow and solute transport. Dimensional analysis confirms that CSF flow significantly influences solute transport in the cranial SAS. We derive an analytical solution for the fluid flow at the leading order (oscillatory flow) and the steady component of the first-order flow. Solute transport is modelled using an advection-diffusion equation, and we derive the long-term transport equation, solving it numerically. The advection term in this equation is the mean Lagrangian velocity of the fluid, which includes steady streaming and Stokes drift caused by oscillations (as shown in Figure 1).

We examine solute transport in two scenarios: (i) drug entry via the spinal canal, estimating delivery time to the brain, and (ii) clearance of toxic proteins from perivascular pathways, assessing SAS efficiency. This approach allows us to understand the fundamental mechanisms of solute transport in the CSF and provides a framework for investigating more realistic domain geometries and displacements.

References

[1] Adams, Ayodeji & Kuijf, Hugo & Viergever, Max & Luijten, Peter & Zwanenburg, Jaco. (2018). Quantifying cardiac-induced brain tissue expansion using DENSE, NMR in Biomedicine, 32.

[2] Kelley, Douglas & Thomas, John. (2022). Cerebrospinal Fluid Flow, Annual Review of FluidMechanics., 55.

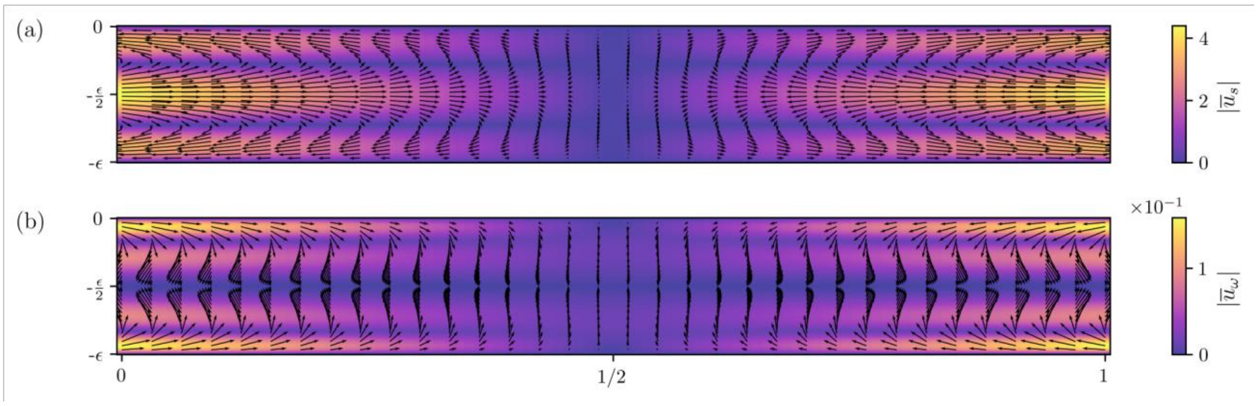


Figure 1: (a) Steady streaming and (b) Stokes drift flows arising in oscillatory flow of CSF.

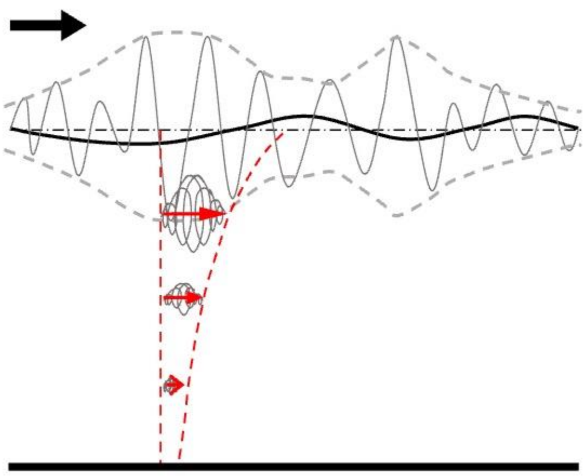
38th Scottish Fluid Mechanics Meeting
Mass transport induced by infragravity wave

Zhiling Liao, Qingping Zou
The Lyell Centre for Earth and Marine Science and Technology, Institute for Infrastructure and Environment, Heriot-Watt University
Edinburgh EH14 4AS, UK

28th May 2025

Abstract

Mass transport induced by group-forced subharmonic waves (infragravity waves) is investigated in the present study. A theoretical solution for subharmonic waves' kinematic contributions to fourth-order mass transport and drifting velocity has been proposed for any depth and bandwidth for the first time. The solution is validated using particle-tracking simulations driven by the flow field generated by the SWASH numerical model. The subharmonic-induced mass transport solution is dependent on a weighted sum of the subharmonic velocity variance spectrum and velocity skewness bispectrum. For narrow-banded waves with long wave group relative to depth, the weightings become independent of spectral components, and the solution is recovered in the time domain. Two mechanisms contributing to mass transport were identified: a forward drift resulting from self-interaction similar to Stokes drift, and a depth-decaying backward drift induced by negative subharmonic velocity skewness due to the anti-phase coupling between subharmonics and wave groups. For narrow-banded waves the forward transport surpasses the backward transport for kh<0.72. For other waves, the critical kh for this phenomenon decreases as wave period, bandwidth and bed slope increases. At greater depths or steeper bed slopes, near-surface backward transport predominates over forward transport; at shallower depths or gentler slopes, forward transport is dominant throughout the water column. Although smaller than Stokes transport, the subharmonic wave-induced mass transport can affect the long-term trajectory of a particle. This study provides the first evidence and insight for the influences of group-forced subharmonics on vertically varying mass transport from ocean surface to seabed in coastal environments.



38th Scottish Fluid Mechanics Meeting

Modelling Fluid-Soil Regime Transitions in the Sedimentation of Clay Suspensions

Yi Yuan, Alan Cuthbertson, Tom Eaves
School of Science and Engineering, University of Dundee
Nethergate, Dundee DD1 4HN

28th May 2025

Abstract

This study aims to develop a new physics-based model framework to analyse the sedimentation of clay suspensions under hydrostatic conditions, where the degree of compaction depends primarily on the initial suspended sediment mass. A two-phase sedimentation model[1], that simplifies the sedimentation process efficiently and accurately, has been calibrated and applied successfully to the full sedimentation process for clay suspensions, demonstrating consistency with new experimental results reported herein.

Two batches of clay suspensions were tested in cylinders to calibrate the relationships of clay properties (i.e. $\phi_{c1} = 0.094$ and $\phi_{c2} = 0.064$) in the model. Additional sedimentation tests were then conducted in a larger settling column for different clay suspensions seeded with tracer particles to assess hindered settling velocity, consolidation rates and structural development at different elevations in the clay sedimentation layer at different elapsed times (*Fig. 1 (a)*).

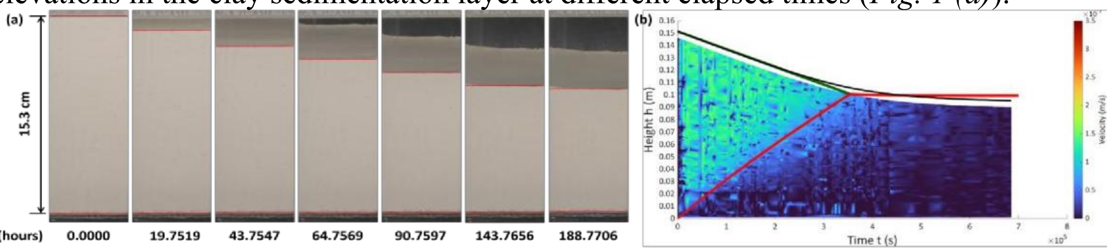


Fig. 1 (a) Images of clay sedimentation process with black tracers encapsulated in the structure and red lines showing the boundary of the clay solid. (b) Comparison of experimental measurements and model predictions. The solid black line is the measured fluid-sediment interface height, the colourmap shows the tracer particle velocity field. The red and green lines predicted gelling and fluid-sediment interfaces from the two-phase model.

The accuracy of the two-phase model was evaluated against experimental data (*Fig. 1 (b)*), with the predicted fluid-sediment interface agreeing well with measurement (i.e. maximum error < 4.3%). The predicted "gelling" interface (ascending red lines) also matched observed reduction in tracer velocities, indicating the transition from hindered settling to consolidation regimes. The model hindered settling velocity is primarily influenced by permeability, with the upper fluidsediment interface following a linear trajectory (green lines) under uniform mixing. In the subsequent consolidation regime, the interface height transitions smoothly due to the interaction of permeability and diffusion of particle network stress, forming a continuous curve at the top of the gelled layer, especially near the intersection of the gelling and fluid-sediment interface. Discrepancies between the two-phase model predictions and experimental data are attributed to insufficient initial mixing within the clay suspension, leading to concentration gradients, and/or clay flocculation processes, neither of which the two-phase model includes at this stage.

References

[1] Paterson, D.T., Eaves, T.S., Hewitt, D.R., Balmforth, N.J., & Martinez, D. M., "Flow-driven compaction of a fibrous porous medium," *Physical Review Fluid,* Vol. 4, No. 7, 2019.

38th Scottish Fluid Mechanics Meeting

Icequakes at the nanoscale: inducing heterogeneous ice nucleation via vibration

Pengxu Chen[a], Rohit Pillai[a], Saikat Datta[b]
[a]School of Engineering, The University of Edinburgh, EH9 3FB, Edinburgh
[b]Faculty of Science and Engineering, Swansea University, SA1 8EN, Swansea

28th May 2025

Abstract

Rapid ice nucleation in confined liquids is crucial for numerous applications across industries such as food technology,[1] cryopreservation,[2] and ice casting.[3] Despite their widespread usage, traditional methods to enhance nucleation encounter efficiency and practical limitations. An emerging alternative is the use of acoustic waves (AW) such as ultrasound, yet their underlying physical mechanisms remain debated. In this study, we employ brute-force molecular dynamics simulations to investigate how AW induce ice nucleation within confined nanoporous environments. Our simulations explore the effects of varying vibrational frequency and amplitude, revealing five distinct nucleation regimes. These regimes are integrated into a regime map, providing a correlation between vibrational parameters and nucleation outcomes. Our findings show ice nucleation is preceded by the emergence of ice-like clusters formed by the water molecules in the ice phase on the surface, driven largely by vibration-induced negative pressure. To extend the applicability of our findings beyond nanoscale contexts, we propose a generalised, strain-based criterion. This criterion underpins a scalable framework for controlling ice nucleation through targeted surface vibrations, offering potential for optimising industrial ice formation processes.

References

[1] Jia, G., Chen, Y., Sun, A., and Orlien, V., "Control of ice crystal nucleation and growth during the food freezing process," *Comprehensive Reviews in Food Science and Food Safety*, Vol. 21, No. 3, 2022, pp. 2433–2454.

[2] Murray, K. A. and Gibson, M. I., "Chemical approaches to cryopreservation," *Nature Reviews Chemistry*, Vol. 6, No. 8, 2022, pp. 579–593.

[3] Wegst, U. G., Kamm, P. H., Yin, K., and García-Moreno, F., "Freeze casting," *Nature Reviews Methods Primers*, Vol. 4, No. 1, 2024, pp. 28.

38^{th} Scottish Fluid Mechanics Meeting
Force balances characteristic of different dynamo regimes

Ayesha Sarwar, Dr. Robert Teed, Prof. Radostin Simitev
School of Mathematics and Statistics, University of Glasgow
University Place, Glasgow G12 8SQ, United Kingdom

28^{th} May 2025

Abstract

The geodynamo, which sustains the Earth's magnetic field through convectively-driven fluid motion in the outer core, is a fundamental, yet complex process that continues to challenge our understanding. The fluid flow is determined by the balance of forces in the momentum equation. These forces are inherently dependent on position, time, and scale. Recent studies have investigated this complexity through scale-dependent and position-dependent analyses.[1,2] A detailed scale-based force balance analysis was conducted by Schwaiger et al., in which a simplified regime diagram illustrates the transition from a quasi-geostrophic (QG) balance at zeroth order and a Magneto-Archimedean-Coriolis (MAC) balance at first order in dipolar regimes, to a QG balance at zeroth order and a Coriolis-Inertia-Archimedean (CIA) balance at first order in multipolar regimes.[3] We present three-dimensional dynamo simulations in different dynamical regimes to investigate the temporal evolution of forces and their hierarchy. Our time series do not robustly respect the QG-MAC and QG-CIA balances for the dipolar and multipolar dynamo cases, respectively, but these balances can be better respected when taking the curl of the forces, which removes the gradient components that are dynamically insignificant in incompressible fluids.

References

[1] Dormy, E., "Strong-field spherical dynamos," *Journal of Fluid Mechanics*, Vol. 789, 2016, pp. 500–513.

[2] Teed, R. J. and Dormy, E., "Solenoidal force balances in numerical dynamos," *Journal of Fluid Mechanics*, Vol. 964, 2023, pp. A26.

[3] Schwaiger, T., Gastine, T., and Aubert, J., "Force balance in numerical geodynamo simulations: a systematic study," *Geophysical Journal International*, Vol. 219, No. Supplement_1, 2019, pp. S101–S114.

38th Scottish Fluid Mechanics Meeting

Topology-preserving finite element methods for magnetic relaxation

Mingdong He[1], Patrick E. Farrell[1], Kaibo Hu[2], Boris D. Andrews[1]
1. Mathematical Institute, University of Oxford, UK
2. School of Mathematical Sciences, University of Edinburgh, UK

28th May 2025

Abstract

The Parker conjecture, which explores whether magnetic fields in perfectly conducting plasmas can develop tangential discontinuities during magnetic relaxation, remains an open question in astrophysics. Helicity conservation provides a topological barrier during relaxation, preventing topologically nontrivial initial data relaxing to trivial solutions; preserving this mechanism discretely over long time periods is therefore crucial for numerical simulation. This work presents an energy- and helicity-preserving finite element discretization for the magneto-frictional system

$$\partial_t \boldsymbol{B} + \operatorname{curl} \boldsymbol{E} = \boldsymbol{0}, \tag{1}$$
$$\boldsymbol{E} + \boldsymbol{u} \times \boldsymbol{B} = \boldsymbol{0}, \tag{2}$$
$$\boldsymbol{j} = \operatorname{curl} \boldsymbol{B}, \tag{3}$$
$$\boldsymbol{u} = \tau \boldsymbol{j} \times \boldsymbol{B}, \tag{4}$$

for investigating the Parker conjecture. The algorithm preserves a discrete version of the topological barrier and a discrete Arnold inequality $|\mathcal{H}| := |\int \boldsymbol{A} \cdot \boldsymbol{B}\, dx| \leq C^{-1} \int \boldsymbol{B} \cdot \boldsymbol{B}\, dx$. We also discuss extensions to domains with nontrivial topology.

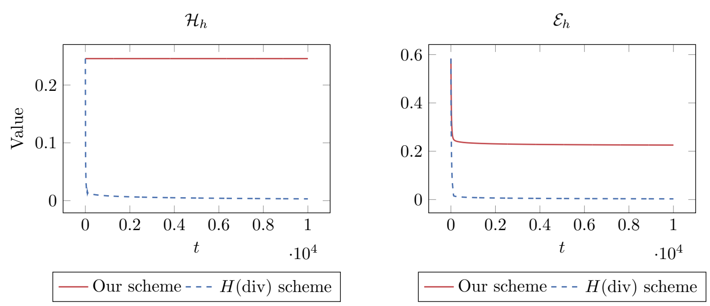


Figure 1: Comparison of helicity $\mathcal{H}_h$ and energy $\mathcal{E}_h$.

38th Scottish Fluid Mechanics Meeting

Thermo-electrohydrodynamic convection in a cylindrical annulus during TEXUS57$^{\text{th}}$ sounding rocket flight: Numerical simulation

P. S. B. Szabo[a], O. Crumeyrolle[b], A. Meyer[b] and C. Egbers[a]

[a]Department of Aerodynamics and Fluid Mechanics, Brandenburg University of Technology Cottbus-Senftenberg, Siemens-Halske-Ring 15a, 03046 Cottbus, Germany

[b] Normandie Universite, UNIHAVRE, CNRS UMR 6294, Laboratoire "Ondes et milieux complexes", 53, rue de Prony, Le Havre 76058, France

28th May 2025

Abstract

Since the early 1930s, it has been known from the study of Seftleben[1] that electric fields can influence thermal flow and heat transport in fluids. This phenomenon is driven by electrohydrodynamic force density, with this study focusing solely on the dielectrophoretic effect to induce convection. To investigate the resulting thermo-electrohydrodynamic convection in a differentially heated annulus, without interference from natural convection, the experiment was placed under microgravity conditions during a sounding rocket flight. The experimental parameter space was complemented by numerical simulations that captured the evolving convective structures between the inner and outer cylinders. The primary objective was to identify the helicoidal nature of the convective flow, initially reported by Travnikov *et al.*,[2] and to determine the onset of convection. Our simulations revealed competing helicoidal vortices with mode numbers 3 and 4, which are predominant for an aspect ratio of 0.5. The critical Rayleigh number was determined using linear regression. The transition from the basic state to convective flow suggests a supercritical pitchfork bifurcation.

References

[1] Senftleben, H., "Über den Einfluß von magnetischen und elektrischen Feldern auf den Wärmestrom in Gasen," *Zeitschrift für Physik*, Vol. 74, No. 11-12, 1932, pp. 757–769.

[2] Travnikov, V., Crumeyrolle, O., and Mutabazi, I., "Numerical Investigation of the Heat Transfer in Cylindrical Annulus with a Dielectric Fluid under Microgravity," *Physics of Fluids*, Vol. 27, No. 5, May 2015, pp. 054103.

38th Scottish Fluid Mechanics Meeting

Hydrodynamic Characterisation of Produced Water in Conventional Deoiling Hydrocyclones

Okwunna Maryjane Ekechukwu & Taimoor Asim

School of Computing, Engineering & Technology, Robert Gordon University, Aberdeen, United Kingdom.

28th May 2025

Abstract

Deoiling hydrocyclones are widely used in industry for the treatment of produced water, which is a mixture of oil and water, and is the primary byproduct of oil and gas reservoir operation [1]. Produced water is fed tangentially into conventional deoiling hydrocyclones, while the vortex finder inside aids in separating dispersed oil droplets from water [2], as shown in Figure 1. Efficient removal of oil droplets remains a major challenge faced by the industry, dictated by stringent regulatory requirements. To better understand the complexities involved in separating oil droplets from water, Computational Fluid Dynamics (CFD) based investigations have been carried out in the present study. Population Balance Model (PBM) has been used to numerically predict spatio-temporal distribution of oil droplets within the hydrocyclone. Discretized bin-based approach has been employed along with log-normal distribution of oil droplets, ranging from 5 – 80 μm in size. Droplets transport, coalescence and breakage phenomena are modelled across nine bins. Reynolds Stress Model (RSM) is used to capture anisotropic turbulent flow structures. Detailed hydrodynamic characterization of produced water and the separation efficiency of the hydrocyclone are the main objectives of this work. It has been observed that as the produced water propagates from inlet towards the underflow, the droplet size distribution shifts from smaller sizes (10-40 μm) to larger sizes (40-70 μm) due to droplets coalescence. It has also been observed that droplet-droplet coalescence is dominant over droplet breakage due to low turbulent energy dissipation rate within the hydrocyclone. The separation efficiency for a median oil droplet size of 28 μm was calculated to be 74%, which is agreement with the experimental findings reported by Young [3]. Findings of this study enhance our understanding of key mechanisms governing oil droplets separation in produced water and offer a platform for optimization of deoiling hydrocyclones' design and operation.

References

[1] Liu, Y., Lu, H., Li, Y., Xu, H., Pan, Z., Dai, P., Wang, H. and Yang, Q., 2021. A review of treatment technologies for produced water in offshore oil and gas fields. *Science of the Total Environment*, *775*, p.145485.

[2] Wang, D., Zhao, Z., Qiao, C., Yang, W., Huang, Y., McKay, P., Yang, D., Liu, Q. and Zeng, H., 2020. Techniques for treating slop oil in oil and gas industry: a short review. *Fuel*, *279*, p.118482.

[3] Young, G.A.B., Wakley, W.D., Taggart, D.L., Andrews, S.L. and Worrell, J.R., 1994. Oil-water separation using hydrocyclones: An experimental search for optimum dimensions. *Journal of petroleum science and engineering*, *11*(1), pp.37-50.

Appendix

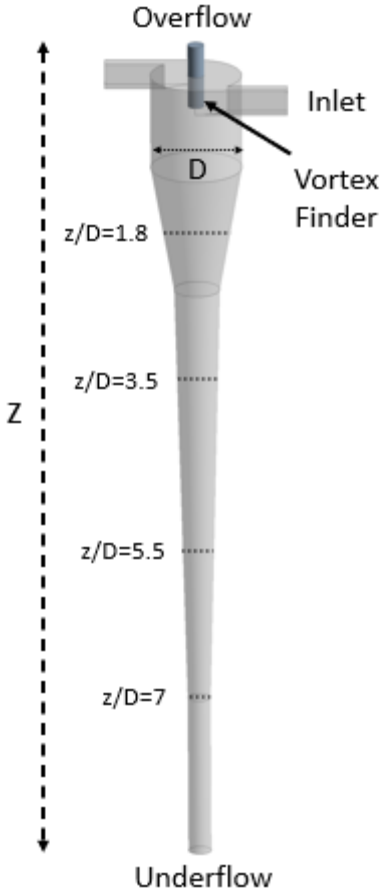


Figure 1. Conventional deoiling hydrocyclone

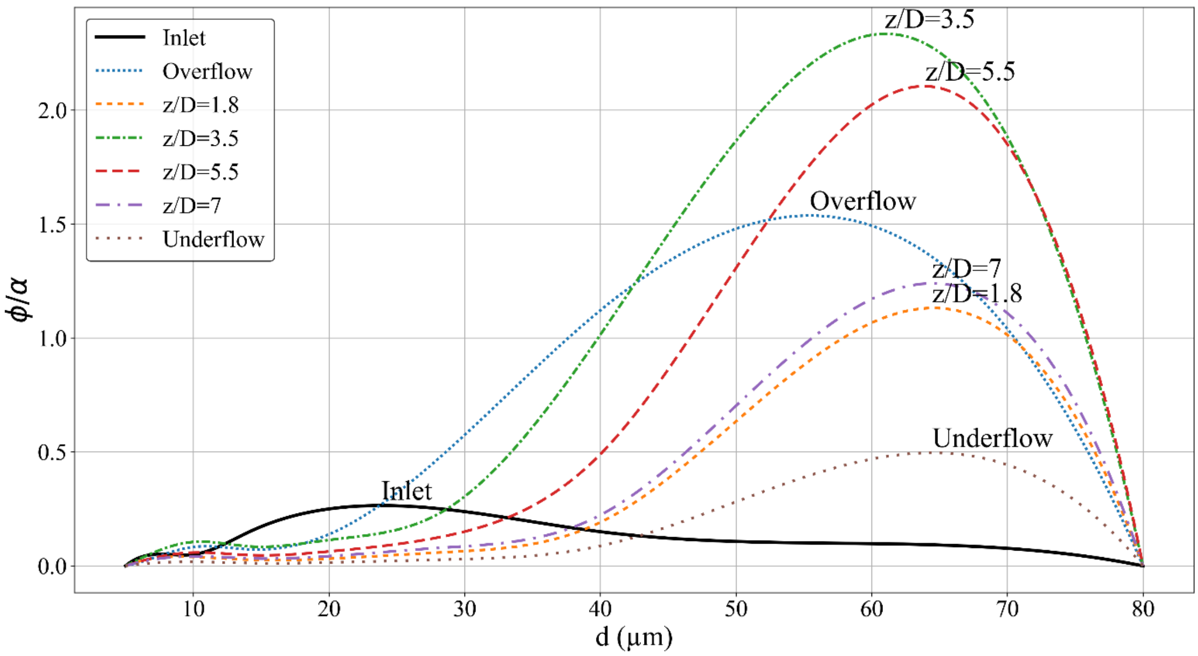


Figure 2. Particle size distribution profiles at different axial locations

38^{th} Scottish Fluid Mechanics Meeting

Electrohydrodynamic interactions of a pair of leaky dielectric drops

Michael A. McDougall, Stephen K. Wilson, Debasish Das
Department of Mathematics and Statistics, University of Strathclyde,
Livingstone Tower, 26 Richmond Street, Glasgow G1 1XH

28^{th} May 2025

Abstract

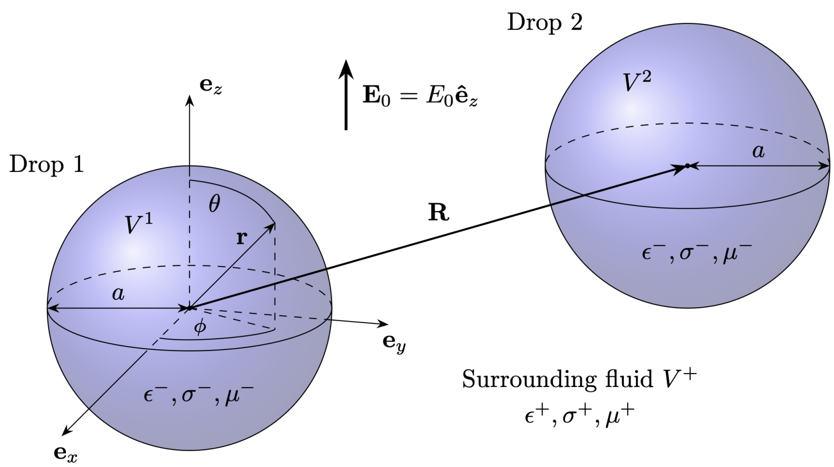


Figure 1: A pair of identical leaky dielectric drops in a uniform electric field

A weakly conducting (leaky dielectric) drop immersed in another fluid polarises in the presence of a uniform DC electric field. A surface charge density develops on the interface between the fluids which acts to deform the drop and shear both fluids into motion.[1] In the presence of a second identical drop, the dynamics of the first drop are modified due to electrohydrodynamic interactions; most importantly, the drops translate due to dielectrophoretic forces and hydrodynamic interactions. Building on the assumptions of the Taylor-Melcher leaky dielectric model,[2] we present a three-dimensional small deformation theory for a pair of widely-separated, leaky dielectric drops suspended in a leaky dielectric fluid medium, valid in the limit of high drop viscosity and surface tension. We derive the fluid flow inside and outside both drops, the rotational and translational velocity of the drops, and the drop shapes. The novelty of the present work lies in the retention of transient charge relaxation and convection by fluid flow in the charge transport equation. While these effects render the governing equations nonlinear and difficult to solve analytically, they are crucial if one wishes to capture the transition to Quincke rotation, a symmetry-breaking bifurcation whereby a drop can begin rotating in an electric field stronger than some critical value.[3] Our theory captures qualitative differences between the onset of Quincke rotation for a solid sphere and for a viscous drop, which displays hysteresis due to the deformed shape. Further, the onset of rotation for a pair of drops is found to be qualitatively different from that for an isolated drop due to symmetry-breaking electrohydrodynamic interactions and from that for a pair of solid spheres due to straining flows present only in drops.

References

[1] Taylor, G. I., "Studies in electrohydrodynamics I. The circulation produced by a drop in an electric field," *Proceedings of the Royal Society of London A.*, Vol. 291, No. 1425, 1966, pp. 159–166.

[2] Melcher, J. R. and Taylor, G. I., "Electrohydrodynamics: a review of the role of interfacial shear stresses," *Annual Review of Fluid Mechanics*, Vol. 1, 1969, pp. 111–146.

[3] Jones, T. B., "Quincke rotation of spheres," *IEEE Transactions on Industry Applications*, Vol. 20, No. 4, 1984, pp. 845–849.

38th Scottish Fluid Mechanics Meeting

Investigating Advancing Contact Lines for Microdroplets on Circular Micropillared Surfaces

Janice To, Khellil Sefiane, Rodrigo Ledesma-Aguilar, and Daniel Orejon

Institute for Multiscale Thermofluids, School of Engineering
The University of Edinburgh, Edinburgh EH9 3FD, Scotland, UK

28th May 2025

Abstract

Droplet dynamics on microstructured surfaces are vital for applications like condensation, biomedical diagnostics, and industrial coatings. Our study investigates how circular micropillars affect the advancing contact line dynamics of microdroplets using microgoniometry (**Fig. 1a**). Unlike traditional goniometers, our method uses a piezoelectric nozzle to dispense monodispersed droplets (15-60 µm), without the direct needle contact, mitigating needle-induced droplet shape distortion, that can compromise the accuracy of contact angle measurements [1].
We analyse droplets growing from hundredths of micrometres to a millimetre on circular micropillars with fixed height and lateral dimensions (10 µm), and inter-pillar spacings of 5-160 µm. Scanning Electron Microscopy (SEM) snapshots exemplifying the morphology of the micropillar arrays are shown in **Fig. 1b**. Key parameters examined include droplet contact angle, base radius, surface energy, contact line dynamics per jump, and jump count. We aim to correlate droplet-to-micropillar size ratio with jump strength, drawing analogies to drop evaporation studies. Our research enhances the understanding of the relationship between microstructure design and droplet wetting behaviour at comparable length scales, providing insights for optimising surface engineering in scientific and industrial contexts.

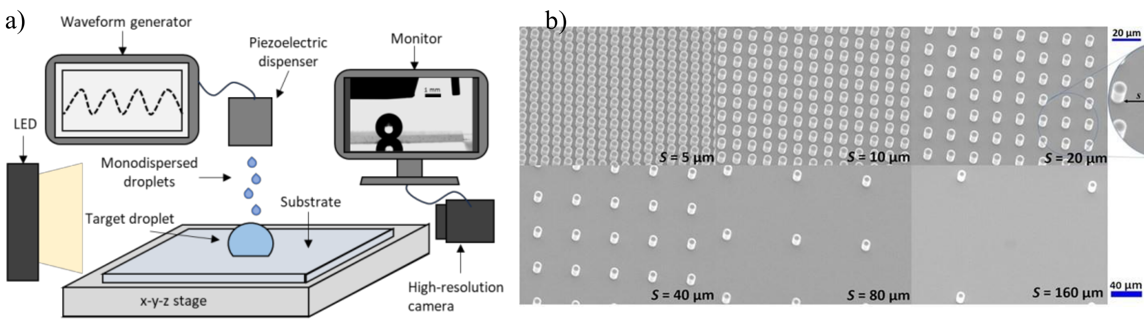


Fig 1. (a) Schematic of the microgoniometry setup. (b) SEM images of the surfaces at a 30° tilt and at the same magnification (500×), for h and d = 10 µm with s ranging from 5-160 µm. Inset magnification for s = 20 µm [2]. Fig. 1.b) has been reproduced under CC-BY 4.0 from Al-Balushi *et al.* [2].

References

[1] T. Huhtamäki et al., Nature Protocols, vol. 13, pp. 1521-1538, 2018.
[2] K. M. Al Balushi et al., Langmuir, vol. 39, no. 23, pp. 8323–8338, 2023

38^{th} Scottish Fluid Mechanics Meeting

Particle deposition from an unpinned particle-laden sessile droplet undergoing one-sided evaporation

Henry T. Sharp[1], Stephen K. Wilson[2,1], Alexander W. Wray[1]

[1]Department of Mathematics and Statistics, University of Strathclyde, Livingstone Tower, 26 Richmond Street, Glasgow, G1 1XH, United Kingdom

[2]Department of Mathematical Sciences, University of Bath, Claverton Down, Bath, BA2 7AY, United Kingdom

28^{th} May 2025

Abstract

The evaporation of a sessile droplet is a widely studied phenomenon due to its use in a variety of industrial applications, including inkjet printing. In many of these applications, droplets contain suspended particles, and the morphology of the deposit of particles left behind on the substrate is of considerable interest. In the case of inkjet printing for example, a uniform deposition of particles is typically required to produce a high-quality image.[1] When an evaporating droplet is surrounded entirely by vapour, or when the vapour moves rapidly away from the surface of the droplet, the liquid and gas phases decouple, and evaporation is entirely dependent on the liquid phase.[2] In this case, the associated thermodynamic disequilibrium at the liquid-gas interface is what drives evaporation and is described by the so-called “one-sided” model of evaporation.[2,3] This is in contrast with the widely-studied case of diffusion-limited evaporation in which evaporation depends on the transport of vapour in the gas phase.[4] A mathematical model is formulated and analysed that predicts the evolution of and the flow within a thin, unpinned, particle-laden, sessile droplet undergoing one-sided evaporation. In particular, this model is used to analyse the evolution of the concentration of suspended particles within such a droplet, and determine the density of particles per unit area deposited onto the substrate. It is shown that the flow within the droplet can undergo three distinct phases throughout the lifetime of the droplet. During the first phase, the flow within the droplet is entirely outwards. During the second phase, a stagnation point develops and advances from the centre of the droplet outwards. The flow within the droplet is simultaneously inwards behind the stagnation point and outwards in front of the stagnation point. During the third phase, the flow within the droplet is entirely inwards. The existence and duration of each phase is determined by the degree of thermodynamic disequilibrium at the liquid-gas interface. For example, for a certain range of values of thermodynamic disequilibrium, the first phase or the first and second phases do not occur. The density of particles per unit area deposited onto the substrate is determined by the degree of thermodynamic disequilibrium. A higher degree of thermodynamic disequilibrium corresponds to a longer first phase, and hence a longer period of outward flow. This leads to a higher density of particles being deposited near the contact line of the droplet compared to that of when the degree of thermodynamic disequilibrium is lower.

References

[1] Kuang, M., Wang, L., and Song, Y., "Controllable printing droplets for high-resolution patterns," *Advanced Materials*, Vol. 26, No. 40, 2014, pp. 6950–6958.

[2] Murisic, N. and Kondic, L., "On evaporation of sessile drops with moving contact lines," *Journal of Fluid Mechanics*, Vol. 679, 2011, pp. 219–246.

[3] Burelbach, J. P., Bankoff, S. G., and Davis, S. H., "Nonlinear stability of evaporating/condensing liquid films," *Journal of Fluid Mechanics*, Vol. 195, 1988, pp. 463–494.

[4] Wilson, S. K. and D'Ambrosio, H.-M., "Evaporation of sessile droplets," *Annual Review of Fluid Mechanics*, Vol. 55, 2023, pp. 481–509.

38th Scottish Fluid Mechanics Meeting

The role of the diffusive boundary layer in deposition from sessile droplets

A. W. Wray[1], H.-M. D'Ambrosio[2], and M. R. Moore[3]
[1]University of Strathclyde, Glasgow G1 1XH, UK
[2]University of Glasgow, Glasgow G12 8QQ, UK
[3]Loughborough University, LE11 3TU, UK

28th May 2025

Abstract

The evaporation of particle-laden sessile droplets is a problem of fundamental scientific and industrial interest, extensively studied through theoretical, numerical, and experimental approaches. The core mechanism underlying the dynamics of such droplets—namely that an evaporation-driven capillary flow drives the suspended particles to the contact line—is well-understood. Theoretical models have been developed that reproduce this behaviour. However, while these models qualitatively capture the essential mechanism, they often predict unphysical outcomes, such as complete transport of particles to a zero-width deposit at the contact line.

Some studies have addressed this limitation, but fully resolving the issue requires accounting for the subtle interplay of several mechanisms, including at least particle diffusion and particle-substrate adsorption. We develop an asymptotic model in the physically relevant limit of large Péclet number, and analyse how the resulting ring structure depends on the rates of diffusion and adsorption. Our findings are compared with numerical and experimental results. We also identify additional mechanisms that must be incorporated to obtain a fully quantitative match with experimental data.

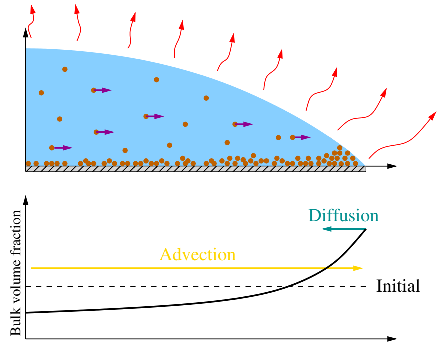


Figure 1: Schematic of a particle-laden sessile droplet evaporating on a substrate with adsorption. The advective flux towards the contact line induces a competing diffusive flux that opposes advection.

38th Scottish Fluid Mechanics Meeting

Non-porous viscous fingering of a gravity-driven free-surface flow

Haolin Yang, Katarzyna Kowal
School of Mathematics and Statistics , University of Glasgow
University Place, Glasgow, G12 8QQ

28th May 2025

Abstract

We investigate the dynamics and frontal instability of a viscous gravity current flowing over an inclined substrate that is prewetted with a dissimilar viscous fluid. To do so, we employ lubrication theory to model the flow of both layers of fluid by assuming that vertical viscous shear stresses provide the dominant resistance to the flow and that the effects of inertia and surface tension are negligible. Such free-surface flows are relevant across various domains, including geophysics, industry, and physiology, from the micro-scale of thin coating films and nasal drug delivery to the macro-scale of the flow of ice sheets. We solve the full system of partial differential equations numerically, revealing the emergence of two distinct local traveling wave solutions, each characterised by an intrinsic velocity. One of these velocities characterises the motion of the upper-layer intrusion front, while the other characterises the downstream flow of the lubricating film. We classify the flow into three scenarios depending on the relationship between these velocities. When both velocities are equal, the flow converges towards a global traveling wave solution over time. We perform a linear stability analysis of the flow in this scenario, revealing regions of instability across parameter space. We find that the flow is prone to a fingering instability above a critical viscosity ratio. Specifically, when a low-viscosity fluid spreads over another viscous fluid of sufficiently high viscosity, it becomes prone to instability, reminiscent of traditional viscous fingering phenomena observed in porous media. Furthermore, we observe that the critical viscosity ratio increases with the density difference between the two fluid layers, while sufficiently high density differences suppress the instability entirely.

38^{th} Scottish Fluid Mechanics Meeting

The effect of capillary forces on two-layer flow down an incline

Hannah-May D'Ambrosio, Katarzyna Kowal
School of Mathematics and Statistics, University of Glasgow
University Place, Glasgow G12 8QQ, UK

28^{th} May 2025

Abstract

Gravity-driven flow is ubiquitous in nature, industry, and biology, from the large scales of geophysical applications such as the flow of lava and ice sheets, to the small scales of industrial and medical applications, including coating processes and nasal drug delivery. The majority of previous theoretical studies of single-layer and two-layer viscous gravity currents assume a dominant balance of viscous and gravitational forces, which frequently admit similarity solutions. Whilst these models are applicable in the bulk, key assumptions are expected to break down near the front of the viscous gravity currents. One of these involves capillary forces, which are typically neglected in the bulk, but become important near the front where free surface gradients are large. In this talk, we explore a reduced-order model that incorporates gravitational, viscous, and capillary forces to describe the evolution of two-layer flow down an incline near the front, as depicted in Figure 1. We determine predictions for the thickness of the current and show that capillary forces act to reduce spreading, and hence the extent of the current, relative to the predictions of models which ignore this effect.

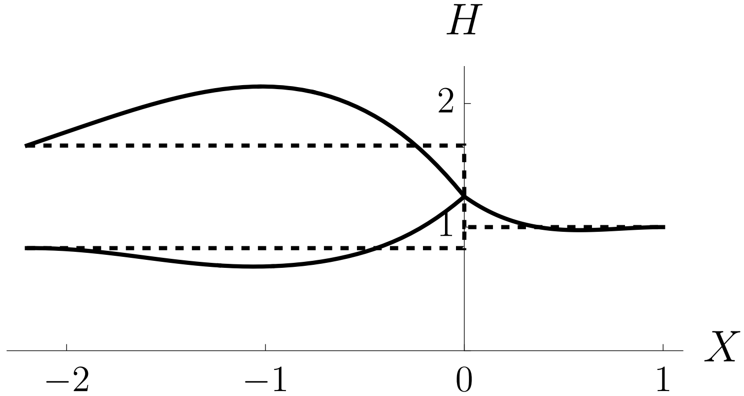


Figure 1: Travelling wave solutions for the thickness H of the fluid near the front $X = x - x_N$. The dashed lines indicate the similarity solutions when capillary forces are neglected.

38th Scottish Fluid Mechanics Meeting

Sports aerodynamics: an overview of activities at Heriot-Watt University

F. Malizia, X. Hu, B. Blocken
School of Engineering & Physical Sciences, Heriot-Watt University
Edinburgh EH14 4AS, UK

28th May 2025

Abstract

Sports aerodynamics is a branch of sports engineering that plays a significant role in improving the performance of athletes, mainly in speed sports [1]. In addition, it can play an important role in improving the fairness of competitions. For example, new rules imposing a greater distance between vehicles and cyclists have been created [2] as a result of aerodynamic studies [3,4].
Current research at Heriot-Watt University focuses on the aerodynamics of cyclists (Fig. 1a) cycling components such as wheels, and runners (Fig. 1b), using computational fluid dynamic simulations (CFD). The aim is to create guidelines on how to perform CFD simulations in sports aerodynamics, increase the performance of athletes and possibly help them break records, and help sports governing bodies improve current regulations in terms of fairness.

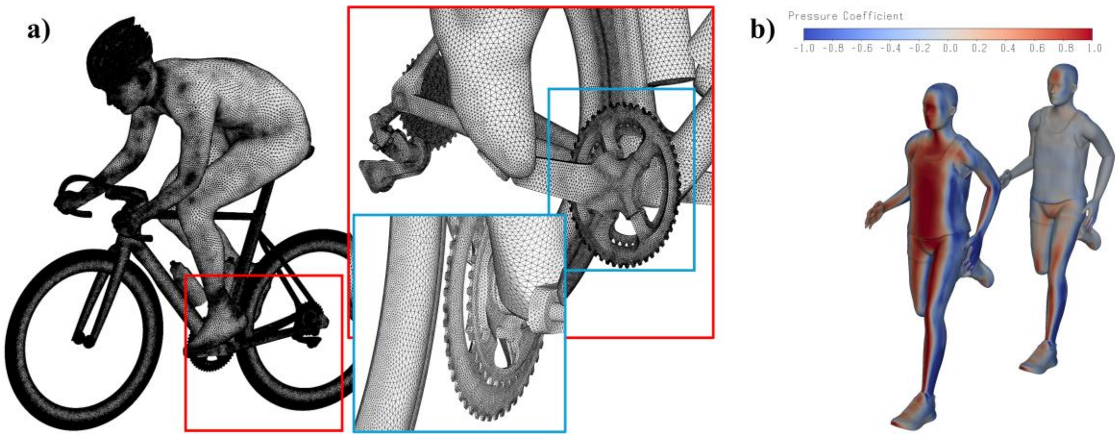


Figure 1. (a) computational grid of a cyclist on a bicycle, with close-up views; (b) Simulation of two drafting runners

References

[1] Alam, F., Chowdhury, H., Moria, H., 2019. A review on aerodynamics and hydrodynamics in sports, in: Energy Procedia. Elsevier Ltd, pp. 798–805.
[2] UCI. (2023, January 16). UCI Regulations for road cycling: changes for time trials. https://www.uci.org/pressrelease/uci-regulations-for-road-cycling-changes-for-time-trials/1QmN4gRxrsgeQtgahAAfXr (accessed on 01/05/2025)
[3] Blocken, B., & Toparlar, Y. (2015). A following car influences cyclist drag: CFD simulations and wind tunnel measurements. JWEIA, 145(0), 178–186.
[4] Blocken, B., & Malizia, F. (2024). How much can roof-mounted bicycles on a following team car reduce cyclist drag? JWEIA, 249.

38^{th} Scottish Fluid Mechanics Meeting
Autoencoders for Lagrangian Computational Fluid Dynamics

Joseph O'Connor
EPCC, University of Edinburgh
Bayes Centre, 47 Potterrow, Edinburgh, EH8 9BT, UK

28^{th} May 2025

Abstract

Lagrangian computational fluid dynamics (CFD) methods, such as smoothed particle hydrodynamics, offer an alternative to traditional mesh-based (Eulerian) approaches (e.g. finite volume). The key advantage of Lagrangian methods lies in their meshless formulation, making them well suited to problems where mesh-based methods struggle (e.g. highly deforming interfaces). However, while mesh-based CFD methods have benefited from significant advances thanks to their integration with data-driven techniques, Lagrangian methods have fallen behind. This is primarily due to the unique characteristics of the Lagrangian formulation (e.g. unstructured and dynamic particle distributions), making it impossible to apply popular data-driven techniques that have been widely applied to mesh-based CFD methods (e.g. convolutional neural networks). In light of the above, this work aims to develop the first autoencoder architecture for Lagrangian CFD. Drawing inspiration from computer vision (e.g. point cloud classification), particle distributions are compressed to a reduced latent space, before reconstructing the original distribution. For the encoder, the widely-used PointNet architecture is explored, along with graph neural networks that can capture local structure. For the decoder, classical multi-layer perceptron architectures are investigated, as well as more flexible diffusion-based models. Among other applications, this work opens the door to reduced-order models for Lagrangian CFD – where the dynamics are propagated in the latent space, while the autoencoder facilitates switching between the low- and high-dimensional representations – as has already been demonstrated for traditional mesh-based CFD methods.[1]

References

[1] Kičić, I., Vlachas, P. R., Arampatzis, G., Chatzimanolakis, M., Guibas, L., and Koumoutsakos, P., "Adaptive Learning of Effective Dynamics for Online Modeling of Complex Systems," *Comput. Methods Appl. Mech. Eng.*, Vol. 415, 2023, pp. 116204.

38th Scottish Fluid Mechanics Meeting

On-the-fly Lagrangian averaging

Abhijeet Minz[1], Lois E. Baker[1], Jacques Vanneste[1], Hossein A. Kafiabad[2]

[1]Department of Mathematical Sciences, Durham University, Durham DH1 3LE, UK

[2]School of Mathematics and Maxwell Institute for Mathematical Sciences, University of Edinburgh, Edinburgh EH9 3FD, UK

28th May 2025

Abstract

Many fluid dynamical phenomena involve temporal and spatial scales that cannot be fully resolved by measuring instruments or numerical simulations. Their modelling requires coarse graining, typically in the form of temporal or spatial averaging of the equations of motion, and the parameterisation of the impact of the unresolved scales. Eulerian averaging, in which flow variables are averaged at fixed spatial locations, is widely used. The alternative of Lagrangian averaging, which involves averaging flow variables along the particle trajectories, has benefits because it respects the advective structure of the equations modelling fluid motion.

The numerical computation of Lagrangian (time) averages from simulation data is challenging, however. It can be carried out by tracking a large number of particles or, following a recent approach, by solving a dedicated set of partial differential equations (PDEs). Both approaches are computationally demanding because they require an entirely new computation for each time at which the Lagrangian mean fields are desired.

We overcome this drawback by developing a PDE-based method that delivers Lagrangian mean fields for all times through the single solution of evolutionary PDEs. This allows for an on-the-fly implementation, in which Lagrangian averages are computed along with the dynamical variables. This is made possible by the use of a special class of temporal filters whose kernels are sums of exponential functions.[1] We implement these in the rotating shallow-water model and demonstrate their effectiveness at filtering out large-amplitude Poincar´e waves.

References

[1] Minz, A., Baker, L. E., Kafiabad, H. A., and Vanneste, J., “The exponential Lagrangian mean,” *arXiv preprint arXiv:2406.18243*, 2024.

38th Scottish Fluid Mechanics Meeting

Spatial tiling and localized shadowing in 2D Kolmogorov flow

Dmitriy Zhigunov and Jacob Page
School of Mathematics and Maxwell Institute for Mathematical Sciences,
University of Edinburgh
Edinburgh, EH9 3FD, UK

28th May 2025

Abstract

The dynamical systems picture of turbulence succeeds both in describing physical mechanisms[1] and in reconstructing statistics of turbulent flows.[2] However, studies have largely focused on small domains, dominated by a single length scale. Many real-world systems, such as the oceans and the atmosphere, exist in extended domains with many physical processes occurring simultaneously. Likewise, two-dimensional Kolmogorov flow on large ($2n\pi \times 2m\pi$) domains is composed of spatially localized regions that qualitatively resemble high dissipation states on the smaller, more standard, $2\pi \times 2\pi$ domain. We develop an optimization-based, multi-shooting method to spatially tile existing exact solutions (e.g. periodic orbits, traveling waves) from the $2\pi \times 2\pi$ domain, leading to new, large-domain, solutions which contain two or more regions with largely independent dynamics. Furthermore, we demonstrate that similar computational approaches can shed light on the mechanics of spatially localized shadowing events of exact solutions.

References

[1] Brand, E. and Gibson, J. F., "A doubly localized equilibrium solution of plane Couette flow," *Journal of Fluid Mechanics*, Vol. 750, 2014, pp. R3.

[2] Page, J., Norgaard, P., Brenner, M. P., and Kerswell, R. R., "Recurrent flow patterns as a basis for two-dimensional turbulence: Predicting statistics from structures," *Proceedings of the National Academy of Sciences*, Vol. 121, No. 23, 2024, pp. e2320007121.

38^{th} Scottish Fluid Mechanics Meeting
Purely elastic turbulence in channel flows

Damiano Capocci, Moritz Linkmann and Alexander Morozov
School of Physics and Astronomy, University of Edinburgh
Mayfield Road, Edinburgh EH9 3JZ

28^{th} May 2025

Abstract

Turbulence production in wall-bounded parallel shear flows of Newtonian fluids is predominantly located near the bounding walls. In contrast, our recent work[1] demonstrates that in purely elastic turbulence, production is concentrated along the center plane of the channel, away from the walls. This raises the question of what type of production occurs in turbulent flows of non-Newtonian fluids at finite Reynolds numbers, specifically in elasto-inertial turbulence.

In this study, we focus on the zero Reynolds number case in the well-diluted polymer regime. Through a systematic investigation of the dependence on the Weissenberg number (Wi) via direct numerical simulations, we observe the emergence of chaotic motion along the channel midplane, characterised by a rich variety of dynamical states. We further analyse the Wi dependence of velocity statistics and gradient dynamics, revealing a degree of intermittency even in inertialess configurations.

References

[1] Lellep, M., Linkmann, M., and Morozov, A., "Purely elastic turbulence in pressure-driven channel flows," *Proceedings of the National Academy of Sciences*, Vol. 121, No. 9, 2024, pp. e2318851121.

38th Scottish Fluid Mechanics Meeting

Transition to Turbulence in the Stokes Boundary Layer: Edge States and the Periodic Self-Sustaining Process

Jorge Sandoval, Tom Eaves
School of Science and Engineering, University of Dundee, Dundee, DD1 4HR, UK

28th May 2025

Abstract

The Stokes boundary layer is an oscillatory flow above an infinite plate, with oscillations driven either by (1) a transverse sinusoidal motion of the plate or (2) a sinusoidal applied pressure gradient. Beyond a critical Reynolds number of 2511, the laminar solution of the Stokes boundary layer is susceptible to linear instability. However, this instability is subcritical given that turbulence is observed for Reynolds numbers above approximately 700, despite the flow being linearly stable in this range.

The state space of a subcritical flow consists of two basins of attraction: that of the laminar flow and that of turbulence. A saddle point separates these basins, termed the ‘edge state’, and its codimension-one stable manifold termed the ‘edge manifold’, or simply ‘edge’. The edge states may be found by ‘edge tracking’, an iterative procedure in which the trajectories of initial conditions on either side of the edge are computed and bisected.

Edge states in the Stokes boundary layer are composed of vortical structures of the same nature as canonical shear flows, namely streaks, rolls and waves. For non-oscillating shear flows, these structures are known to coexist and mutually balance through a mechanism known as the Self-Sustained Process. However, in the Stokes boundary layer, these structures are inherently periodic and utilise the oscillating base flow in a novel way. Structures migrate upwards to align with the location of the maximum shear of the laminar velocity profile, and large-scale rolls form to periodically create new streaks near the wall. This talk will present these edge states in the Stokes boundary layer, compare them to their known non-oscillatory counterparts, and provide insights about their effects on momentum and energy transport among structures near the wall.

38th Scottish Fluid Mechanics Meeting

Impact of steam on the thermodiffusive instability in hydrogen premixed flames

Sofiane Al Kassar[1], Hunyan Zubair[1], Innes Cameron[1], Khushboo Pandey[1], Antonio Attili[1]

[1]Institute for Multiscale Thermofluids, School of Engineering, The University of Edinburgh, Edinburgh EH9 3FD, United Kingdom

28th May 2025

Abstract

Hydrogen is a promising sustainable alternative to natural gas for electricity generation. Steam injection offers additional benefits in premixed gas turbine combustion, yet its influence on thermodiffusive instability, particularly significant in hydrogen flames,[1,2] remains insufficiently explored. Water can be added to the gas mixture to lower the flame temperature and modify the NOx formation pathways, thereby reducing NOx emissions and the risk of flashback.[3] However, the presence of steam also modifies key physical and chemical properties, affecting the behaviour of the instability.[4]

In this study, a series of 2D Direct Numerical Simulations (DNS) were conducted to investigate the effects of steam addition on the thermodiffusive instability in lean premixed hydrogen flames under gas-turbine conditions (20 atm and 700 K). Both the linear and non-linear regimes of the instability were examined. Results indicate that adding steam at a constant equivalence ratio simultaneously amplifies thermodiffusive instability while attenuating hydrodynamic instability, leading to a peak growth rate at approximately 13% steam by mass. Similarly, the normalised consumption speed exhibited a maximum at this same steam concentration, reflecting a consistent trend in the long-term flame dynamics.

References

[1] Berger, L., Attili, A., and Pitsch, H., *Combustion and Flame*, Vol. 240, 2022, pp. 111935.

[2] Berger, L., Attili, A., and Pitsch, H., *Combustion and Flame*, Vol. 240, 2022, pp. 111936.

[3] Göke, S., Füri, M., Bourque, G., Bobusch, B., Göckeler, K., Krüger, O., Schimek, S., Terhaar, S., and Paschereit, C. O., *Fuel Processing Technology*, Vol. 107, 2013, pp. 14–22.

[4] Howarth, T. L., Day, M. S., Pitsch, H., and Aspden, A. J., *Proceedings of the Combustion Institute*, Vol. 40, Jan. 2024, pp. 105429.

Abstracts

Posters

38th Scottish Fluid Mechanics Meeting

Controlling Droplet Size Density during Dropwise Condensation on Silicone Oil Grafted Surfaces

Anam Abbas[1,2], Zafar Iqbal[3], Gary G. Wells[1], Glen McHale[1], Khellil Sefiane[1], Daniel Orejon[1,4]

[1] School of Engineering, Institute for Multiscale Thermofluids, The University of Edinburgh, Edinburgh EH9 3FD, Scotland, UK

[2] Department of Mechanical Engineering, University of Engineering and Technology, Lahore, Pakistan

[3] School of Engineering, Institute for Energy Systems, The University of Edinburgh, Edinburgh EH9 3FD, Scotland, UK

[4] International Institute for Carbon-Neutral Energy Research (WPI-I2CNER), Kyushu University, 744 Motooka, Nishi-ku, Fukuoka 819-0395, Japan

28th May 2025

Abstract

Condensation is a crucial process playing a substantial role phenomenon in many engineering and industrial applications. To this end, surface wettability has a significant impact on the type of condensation mechanism ensuing, allowing for two main condensation modes named dropwise and filmwise condensation. Studies have shown that dropwise condensation can yield heat transfer rates up to 6–8 times higher than those achieved via filmwise condensation[1]. As a result, a variety of surface modification techniques have been explored so to promote dropwise condensation. One such method involves applying low surface energy coatings to create surfaces with low contact angle hysteresis (CAH)[2], which facilitate rapid shedding of smaller droplets. In this work, we use silicone oil grafting method to convert inherently hydrophilic silicon substrates into hydrophobic surfaces. Surfaces fabricated using our optimized fabrication parameters[2], demonstrate desirable wetting characteristics, with high contact angles (CA ≈ 107°) and extremely low contact angle hysteresis (CAH) (1°-2°), which are comparable to Slippery Omniphobic Covalently Attached Liquid (SOCAL) surfaces or Slippery Lubricant Infused Porous Surfaces (SLIPS)[3]. On one hand, dropwise condensation is promoted on all silicone oil grafted surfaces regardless of the grafted oil's viscosity and/or number of layers. On the other hand, the droplet shedding radii as well as departure times are significantly influenced by the grafting parameters. Surfaces grafted with "high" viscosity oil (100 cSt) have shown departure of smaller droplets (< 350 μm) within 15-20 minutes as compared to those grafted with low viscosity oil grafted where droplets start shedding after approximately an hour with sizes in the order of 800 μm. We further demonstrate that droplet shedding behaviour correlates to their respective CAH with 100 cSt oil grafted surfaces minimizing droplet pinning as well as enhancing shedding efficiency. These findings provide a framework to select the suitable grafting parameters for creating low CAH surfaces that promote dropwise condensation. Furthermore, based on the fabrication parameters, we can control the droplet size distribution and growth dynamics, hence influencing overall condensation heat transfer performance.

References

1. Miljkovic, N., Enright, R., Nam, Y., Lopez, K., Dou, N., Sack, J., and Wang, E.N., Jumping Droplet-Enhanced Condensation on Scalable Superhydrophobic Nanostructured Surfaces, Nano Letters, 13(1), pp. 179-187, 2013.
2. Abbas, A., Wells, G.G., McHale, G., Sefiane, K., and Orejon, D., Silicone Oil-Grafted Low Hysteresis Water-Repellent Surfaces, *ACS Applied Materials & Interfaces*, 15(8), pp. 1128111295, 2023.
3. Armstrong, S., McHale, G., Ledesma-Aguilar, R., and Wells, G.G., Pinning-Free Evaporation of Sessile Droplets of Water from Solid Surfaces, *Langmuir*, 35(8), pp. 2989-2996, 2019.

38th Scottish Fluid Mechanics Meeting

Effect of free-stream turbulence on the trajectories of free-falling discs

P. Alexandrou[1], C. Bose[2], I. M. Viola[1] A. Attili[1]
[1] University of Edinburgh, EH9 3FD, United Kingdom
[2] University of Birmingham, B12 2TT, United Kingdom

28th May 2025

Abstract

The trajectories of free-falling discs in turbulence are investigated. Different free-falling regimes depend on the Archimedes number ($Ar = U_g d/\nu$), a measure of the ratio of gravitational forces against viscous forces based on the gravitational velocity $U_g = \sqrt{((\rho^* - 1)tg)}$, where g is the gravitational acceleration, ν is the kinematic viscosity, and ρ^* is the ratio between the disc and fluid densities. They also depend on the non-dimensional moment of inertia $I^* = \pi\rho^* t/64d$, where t is the disc thickness. The background turbulence is characterised by the Taylor microscale Reynolds number (Re_λ) and the ratio of the integral length scale to the disc diameter, l/d. We investigate discs in the transitional regime ($Ar = 800$ and $I^* = 0.005$), interacting with turbulence characterised by $l/d = 6, 4$ and $Re_\lambda = 50$. Turbulence is statistically stationary in time, decays in the streamwise direction, and is homogeneous and isotropic in the spanwise directions. Turbulence is superimposed on a constant mean inlet velocity equal to the average terminal velocity in quiescent conditions (U_t), using the forward stepwise method.[1] For the same Re_λ, the $l/d = 6$ case exhibits an average terminal velocity 10% lower than a disc falling in quiescent flow, while for the $l/d = 4$ case it is only 5% lower. This shows that low-intensity turbulence is enough to alter the free-fall characteristics of a disc in this regime.

References

[1] Xie, Z.-T. and Castro, I. P., "Efficient generation of inflow conditions for large eddy simulation of street-scale flows," *Flow, Turbulence and Combustion*, Vol. 81, No. 3, 2008, pp. 449–470.

38^{th} Scottish Fluid Mechanics Meeting

Rayleigh-Bénard Convection in the Weakly Elastic Regime: Insights from Water-Based Liquid Bridges

Ahmed Aljanadi, Monica Oliveira and Marcello Lappa
James Weir Fluids Laboratory,
The Department of Mechanical and Aerospace Engineering,
University of Strathclyde
James Weir Building, 75 Montrose Street, Glasgow, G1 1XJ, UK

28th May 2025

Abstract

This study investigates the weakly elastic regime (WER) of Rayleigh-Bénard convection, a domain that remains significantly underexplored in comparison to its extensively studied Newtonian counterpart. Of particular interest is the nature of the primary flow bifurcation from the purely diffusive, motionless state. In the WER, the first bifurcation is shown to be stationary, a feature that stands in contrast to certain regimes observed in strongly elastic fluids. The present research relies on the liquid bridge configuration, a classical benchmark problem for understanding thermocapillary and buoyancy-driven convection in microgravity and terrestrial conditions. The scarcity of experimental data on viscoelastic convection in this weakly elastic domain, especially in geometries such as liquid bridges, has limited the development of theoretical and numerical models. To address this gap, this study introduces a novel microscale experimental framework, wherein laser-cut diagnostics are employed to illuminate tracer particles suspended in the fluid. This high-resolution visualization technique allows for a precise reconstruction of the flow field, thereby facilitating a deeper understanding of convective patterns and instability mechanisms. Building upon prior investigations that were largely restricted to Newtonian fluids such as pure water [1], this research expands the parameter space by incorporating viscoelastic fluids with weak elasticity. These fluids exhibit subtle, yet crucial, deviations from Newtonian behavior. The ensuing results aim to enrich our understanding of the convective instability dynamics in WER viscoelastic media, offering new insights into the transition pathways, symmetry-breaking mechanisms, and the influence of elastic stresses on pattern formation. By rigorously characterizing the bifurcation structure and flow topology, this work not only extends the existing theoretical framework but also lays the experimental groundwork necessary for validating numerical simulations. Ultimately, this research aspires to bridge the gap between theoretical predictions and empirical observations in the emerging field of viscoelastic thermal convection, with implications spanning fluid dynamics, polymeric material processing, and soft matter physics.

References

[1] Aljanadi A., Oliveira M.S.N., Lappa M., (2024), “Gravitational modes of convection in water liquid bridges”, *Physics of Fluids*, 36(5), 054113 (22 pages).

38th Scottish Fluid Mechanics Meeting

Pulsatile fluid flow through porous membranes

Hessah Almaaz, Matthew Durey and Katarzyna Kowal
School of Mathematics & Statistics, University of Glasgow
Glasgow G12 8QQ, UK

28th May 2025

Abstract

In a number of physiological and industrial systems, particles are transported and filtered through permeable membranes. The net filtration depends on the form of the fluid delivery, such as whether the flow is steady or pulsatile. We explore the effects of fluid pulsation on the transport of particles through porous hollow fibres, which are typically long and thin. Despite initial indications that fluid pulsation may enhance filtration efficiency, the underlying mechanisms are not well understood. We aim to bridge this gap by developing a fluid-mechanical model of the viscous flow that accounts for rapid changes in the fluid flux between pulses. Our model has been developed for an idealised single fibre, assuming that the membrane is a uniform porous medium. We derive an evolution equation for the fluid flux, which we examine using numerical computation and asymptotic analysis. Our theoretical framework is extendable to various geometries relevant to different applications and gives insight into the extent to which fluid pulsation offers an efficient filtration mechanism.

38th Scottish Fluid Mechanics Meeting

A Model for Walking Droplets over Submerged Barriers

Zubaydah Alotaibi, Katarzyna N. Kowal & Matthew Durey
School of Mathematics and Statistics, University of Glasgow
University Avenue,
Glasgow, G12 8QQ

28th May 2025

Abstract

Small droplets bouncing and self-propelling across the surface of a vibrating liquid bath can behave like quantum particles.[1] In this research, we study how variations in the liquid depth affect the movement of these droplets. Specifically, we model the influence of submerged barriers on the wave field by varying the amplitude and slope of the waves generated by the bouncing droplet. By combining mathematical methods from dynamical systems with numerical simulation, we analyse how droplets stay confined in corrals, tunnel across barriers and interact with obstacles. Our model provides insight into how a classical wave-particle system can give rise to quantum-like behaviour.

References

[1] Bush, J. W. M. and Oza, A. U., "Hydrodynamic quantum analogs," *Rep. Prog. Phys.*, Vol. 84, 2021, pp. 017001.

38^{th} Scottish Fluid Mechanics Meeting
Convolutional Neural Networks (CNNs) for turbulent combustion modelling

Dikran Bakkalian Julian, Geveen Arumapperuma and Antonio Attili

School of Engineering, Institute for Multiscale Thermofluids, University of Edinburgh, Edinburgh, EH9 3FD, United Kingdom

28^{th} May 2025

Abstract

Accurately predicting the formation of pollutants such as polycyclic aromatic hydrocarbons (PAH) remains a significant challenge, as this is governed by complex, multi-scale interactions between turbulence and chemical kinetics.[1,2] While effective at capturing large-scale turbulent structures, conventional Large Eddy Simulations (LES) struggle to resolve the non-linear filtered terms that emerge from the combustion process.[3]

This study investigates the use of convolutional neural networks (CNNs) to enhance the modelling accuracy of LES for turbulent combustion. CNNs are trained using high-fidelity Direct Numerical Simulation (DNS) data obtained by Arumapperuma and Attili[4] to estimate the PAH mass fraction ($\widetilde{Y}_{\mathrm{PAH}}$) in a non-premixed, turbulent ethylene flame.

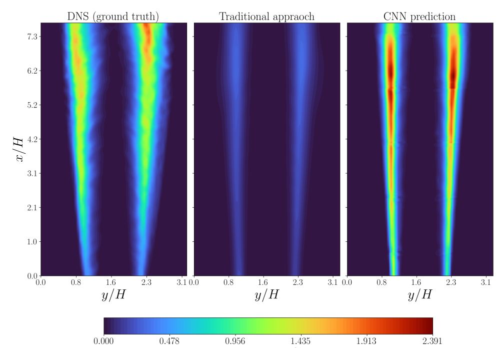


Figure 1: *2D slice of $\widetilde{Y}_{\mathrm{PAH}}$. Left: DNS data. Centre: flamelet-based prediction. Right: CNN prediction. The PAH was averaged across all time steps and the entire z-direction in all three cases.*

After validating the CNN *a priori* on DNS-derived input data, the models are deployed *a posteriori* to predict the PAH mass fraction in LES. The results are compared against traditional LES flamelet-based estimations.[1]

The results reveal that the CNN-based $\widetilde{Y}_{\mathrm{PAH}}$ predictions align much more closely with the DNS than conventional LES approaches. Compared to the DNS, LES estimations underpredict the PAH mass fraction by approximately one order of magnitude in all simulations. In contrast, CNN models overpredict $\widetilde{Y}_{\mathrm{PAH}}$ by a factor of 1.5.

References

[1] Mueller, M. E. and Pitsch, H., "LES model for sooting turbulent nonpremixed flames," *Combustion and Flame*, Vol. 159, No. 6, 2012, pp. 2166–2180.

[2] Arumapperuma, G., Tang, Y., Attili, A., and Han, W., "Spectral analysis of soot dynamics in an aero-engine model combustor," *Proceedings of the Combustion Institute*, 2024.

[3] Pitsch, H., "Large-Eddy Simulation of Turbulent Combustion," *Annual Review of Fluid Mechanics*, Vol. 38, 2006, pp. 453–482.

[4] Arumapperuma, G. and Attili, A., "Analysis and Modelling of PAHs in Turbulent Non-Premixed Jet Flames," *Proceedings of the 12th European Combustion Meeting*, 2025.

38^{th} Scottish Fluid Mechanics Meeting
Convection and Dynamo Action in Rotating Spherical Shells with different Radius Ratios

Francesca Coke and Rob Teed
School of Mathematics and Statistics, University of Glasgow
University Avenue, Glasgow, G12 8QQ

28^{th} May 2025

Abstract

The Earth's outer core is a spherical shell where convection drives a dynamo generating the geomagnetic field. We aim to model dynamos relevant to the Earth's core in the past, present and future by varying the core size. In this poster we develop a mathematical and numerical formulation for this problem, allowing for the calculation of critical Rayleigh numbers (determining the onset of convective flows) within a Boussinesq fluid, and how they depend on the core size, defined by $\chi = r_i/r_o$ where r_i is the inner core radius and r_o is the outer core radius. The dependence on the Ekman number (controlling the effect of rotation) is also considered and these results are confirmed by comparing with existing literature.[1] We then use an existing dynamo code[2] to study the nonlinear dynamo problem when varying χ, using supercriticality of the critical Rayleigh number to determine when dynamos are produced. This aims to understand how known regimes at $\chi = 0.35$ (present day) are affected when varying χ (past/future).

References

[1] Al-Shamali, F., Heimpel, M., and Aurnou, J., "Varying the Spherical Shell Geometry in Rotating Thermal Convection," *Taylor and Francis*, 2004.

[2] Willis, A. P., Sreenivasan, B., and Gubbins, D., "Thermal core–mantle interaction: Exploring regimes for 'locked' dynamo action," *Physics of the Earth and Planetary Interiors*, Vol. 165, No. 1, 2007, pp. 83–92.

38th Scottish Fluid Mechanics Meeting

Simulations of Free-Falling Plates in Perturbed Flows

K. N. Connolly[1], P. Alexandrou[1], C. Bose [2], I. M. Viola[3], A. Attili[1]
[1] School of Engineering, Institute for Multiscale Thermofluids, University of Edinburgh, EH9 3FD, United Kingdom
[2] Aerospace Engineering, University of Birmingham, Birmingham, B15 2TT, United Kingdom
[3] School of Engineering, Institute for Energy Systems, University of Edinburgh, EH9 3FD, United Kingdom

28th May 2025

Abstract

This study investigates the effect of transverse periodic gusts on the flight dynamics of a two-dimensional (2D) free-falling plate. The physical problem is defined by the plate to fluid density ratio, $\rho^* = \rho_p/\rho_f$, the plate diameter to height ratio, $\chi = d/h$, and the Galilei number (Ga). The latter is a measure of the ratio of the gravitational and viscous forces, based on the gravitational velocity $U_g = \sqrt{(\rho^* - 1)dg}$, where g is the gravitational acceleration. We performed numerical simulations for plates of $\chi = 20$, with $\rho^* = 5$ and $Ga = 0.707$ and 10. For all cases, the plate falls steadily when the gust is not applied. We first consider a reference case of a free-falling plate in quiescent flow until it reaches terminal velocity. To define the response frequency $f_r = 1/t_r$, we apply an impulsive horizontal gust and identify the response timescale t_r as the time when the plate's angular velocity is within 1% of its pre-gust value. Then, several cases are simulated with periodic horizontal gusts of velocity $v_x(t) = -A_G \cos(2\pi f t)$ where f is the frequency of the gust and $1 \leq A_G/U_g \leq 4$. Two main observations can be made: i) The range of horizontal motion depends on the frequency of the imposed gust. ii) The mean vertical velocity of the plate is up to 9% slower for lower frequencies or 51% for higher frequencies due to the plate rotating to a vertical position.

38^{th} Scottish Fluid Mechanics Meeting

Evaporation of a Droplet on a Porous Substrate

David Craig[1], Stephen K. Wilson[2,1], Alexander W. Wray[1], and Khellil Sefiane[3]

[1]Department of Mathematics and Statistics, University of Strathclyde, Livingstone Tower, 26 Richmond St, Glasgow G1 1XH, UK.

[2]Department of Mathematical Sciences, University of Bath, Claverton Down, Bath BA2 7AY, UK.

[3] School of Engineering, Institute for Multiscale Thermofluids, University of Edinburgh, The King's Buildings, Mayfield Road, Edinburgh EH9 3JL, UK.

28^{th} May 2025

Abstract

The majority of previous work on the evaporation of particle-laden sessile droplets has focused on the situation in which the substrate is solid. However, in many applications, such as printing onto paper, the substrate is porous and the imbibition of liquid into the substrate may play a significant role in the evolution, lifetime of, and deposition from a particle-laden droplet. Building on recent work on droplet evaporation on a solid substrate[1] we develop a mathematical model based on lubrication theory for the evolution of an evaporating droplet on a porous substrate. Specifically, we develop a model for the evolution of, the flow within, and the deposition from a thin, sessile droplet undergoing simultaneous evaporation and imbibition on an initially dry porous substrate in a variety of modes. Motivated by the wide range of practical applications (such as inkjet printing and disease diagnostics) which would benefit from theoretical insight into the formation of various final deposit patterns, we show that, on one hand, in the regime in which radial advection dominates, single contact-line ring and continuous deposits are formed in the CR and CA modes, respectively. On the other hand, we show that in the regime in which both radial and axial advection dominates, the presence of imbibition suppresses the formation of a ring deposit in the CR mode.

References

[1] Wilson, S. K. and D'Ambrosio, H.-M., "Evaporation of Sessile Droplets," *Annual Review of Fluid Mechanics*, Vol. 55, 2023, pp. 481–509.

38th Scottish Fluid Mechanics Meeting

Modelling Gas-Granular Flow due to an Impinging Plume in the *RECAP* Framework

Jack L. Davies, Andrea Cammarano, Hossein Zare-Behtash, Kevin Worrall
James Watt School of Engineering, University of Glasgow
University Avenue, Glasgow, G12 8QQ

28th May 2025

Abstract

Predictive modelling of spacecraft rocket exhaust plume-surface interactions (PSI) is vital to the advancement of a rapidly maturing field where gathering relevant experimental data is challenging[1].[2] An accurate numerical model must represent a multiphase, multispecies flow of heterogeneous density, incorporating complex shock structures and unsteady flow phenomena[3].[4] In this work, the authors simulate the gas-granular flow which occurs when an impinging exhaust plume strikes a dusty surface using the *Rocket Exhaust Coupled Analysis of Plumes (RECAP)* Navier-Stokes (NS) - discrete element method (DEM) framework: the evolution of the gas phase was solved using *ANSYS Fluent*, with the reciprocal behaviour of the soil phase modelled using *ANSYS Rocky*. A fully transient gas-granular flow solver is achieved through the two-way coupling of the phase solutions. The *RECAP* methodology successfully predicts the time-evolution of a soil bed subjected to an impinging hypersonic plume, capturing both the aerodynamic and soil dynamic environments. Preliminary results exhibit qualitative similarity to flow structures and soil craters observed during Martian spacecraft near-surface operations, but refinement of both the soil model and coupling forces is required to produce more realistic gas-granular flow behaviour.

References

[1] Watkins, R., Metzger, P. T., Mehta, M., Han, D., Prem, P., Sibille, L., Dove, A., Jolliff, B., Daniel P. Moriarty, I., Barker, D. C., Patrick, E., Kuhns, M., Laine, M., and Radley, C. F., "Understanding and Mitigating Plume Effects During Powered Descents on the Moon and Mars," *Bulletin of the AAS*, Vol. 53, 2021.

[2] Chambers, W. A., "An Overview of Plume-Surface Interaction Research," 2023.

[3] Capecelatro, J., "Modeling high-speed gas–particle flows relevant to spacecraft landings," *International Journal of Multiphase Flow*, Vol. 150, 2022.

[4] Metzger, P. T., "Dust Transport and its Effects due to Landing Spacecraft," 2020.

38th Scottish Fluid Mechanics Meeting

Direct Numerical Simulation of Evaporating Droplet Populations

Debarshi Debnath[1], Joseph J Kilbride [2], George Karapetsas[3], David Scott [4], Daniel Orejon[1], Khellil Sefiane[1], and Prashant Valluri[1]

[1]School of Engineering, University of Edinburgh, Edinburgh EH9 3FB, UK

[2]School of Physics, University of Edinburgh, Edinburgh, UK

[3]Department of Chemical Engineering, Aristotle University of Thessaloniki, Greece

[4]Edinburgh Parallel Computing Centre, University of Edinburgh, Edinburgh, UK

28th May 2025

Abstract

The intricate behavior of evaporating droplet populations is vital across various fields such as inkjet printing, microelectronics cooling, disease detection, and many more. To gain a comprehensive physical understanding of the interaction between evaporating droplets, we developed a model based on the diffuse interface method. Using our in-house finite volume code*, the Cahn-Hilliard equation is solved along with energy, mass, and momentum conservation equations.[1] A diffusion-limited evaporation model is used for phase change. When droplets evaporate in the vicinity of other droplets, interactions occur through the vapor phase. This phenomenon is frequently termed vapour shielding.[2] To understand this phenomenon, we performed high-resolution 3-D simulations of various droplet array patterns. The simulations covered a range of pinned droplet arrays, ranging from configurations with four droplets to those with up to 100 droplets. Our simulation reliably forecasts both the accumulation of vapor and the diminished evaporation rate of droplets within the array. We further investigated the symmetry breaking of the droplets located on the edges of the arrays because of the asymmetric evaporation. Moreover, we illustrate how the initial contact angles of the droplets within the arrays influence their evaporation behavior.

References

[1] Sàenz, P. J., Sefiane, K., Kim, J., Matar, O. K., and Valluri, P., “Evaporation of sessile drops: a three-dimensional approach,” *Journal of Fluid Mechanics*, Vol. 772, Jun. 2015, pp. 705–739, Publisher: Cambridge University Press.

[2] Wray, A. W., Duffy, B. R., and Wilson, S. K., “Competitive evaporation of multiple sessile droplets,” *Journal of Fluid Mechanics*, Vol. 884, Feb. 2020, pp. A45, Publisher: Cambridge University Press.

*https://sourceforge.net/projects/tpls/

38th Scottish Fluid Mechanics Meeting

Drying and Filtration Effects on the Rheological Behavior of Aqueous PEO Solutions

Itzia M. García[1], John J. Liggat[2], Mónica S.N. Oliveira[1]
[1]Department of Mechanical and Aerospace Engineering, [2]Department of Pure and Applied Chemistry
University of Strathclyde, Glasgow, UK

28th May 2025

Abstract

The effect of preparation protocol on the rheological behaviour of aqueous poly(ethylene oxide) (PEO) solutions was studied for different concentrations spanning the dilute and semi-dilute regimes, using PEO with a high molecular weight of 8×10^6 g/mol.[1] Solutions were prepared from the same as-supplied polymer, either directly or after oven drying. The low concentration solutions exhibit a Boger fluid like behaviour, with nearly constant viscosity for the range of shear rates tested, with the shear thinning behaviour becoming more pronounced for higher concentrations. For the latter, drying prior to dissolution led to reduced viscosity and relaxation time, measured using a DRH-2 rotational rheometer and a capillary breakup extensional rheometer (CaBER), respectively. Filtration experiments further showed a decrease in viscosities for as-supplied PEO solutions, which can be associated with the removal of clusters, which are known to form in water solutions. This behaviour suggests that the as-supplied polymer may have a greater tendency to form such structures in solution compared to the dried material.[2–4] These findings highlight the significant impact of polymer pre-treatment on the rheological behaviour of PEO solutions and contribute to the understanding of how physical processing influences clustering phenomena in aqueous systems.

References

[1] W. Graessley, "Polymer chain dimensions and the dependence of viscoelastic properties on concentration, molecular weight and solvent power," *Polymer*, Vol. 21, No. 3, 1980, pp. 258–262.

[2] D. L. Ho, B. Hammouda, and S. R. Kline, "Clustering of poly(ethylene oxide) in water revisited," Journal of Polymer Science Part B Polymer Physics, Vol. 41, No. 1, 2002 pp. 135–138.

[3] M. Duval and D. Sarazin, "Identification of the formation of aggregates in PEO solutions," *Polymer*, Vol. 41, No. 7, 2000, pp. 2711–2716.

[4] B. Hammouda, D. L. Ho, and S. Kline, "Insight into Clustering in Poly(ethylene oxide) Solutions," *Macromolecules*, Vol. 37, No. 18, 2004, pp. 6932–6937.

38^{th} Scottish Fluid Mechanics Meeting

Magnetoconvection and transitions in Earth-like dynamos

Luke Gostelow & Robert Teed
School of Mathematics and Statistics, University of Glasgow

28^{th} May 2025

Abstract

Planetary dynamo models exhibit a wide variety of possible solutions which are often characterised by their dominant force balance. In Earth's fluid outer core, a balance between the Lorentz, buoyancy, and Coriolis forces (known as a MAC - Magnetic-Archimedian-Coriolis balance) is predicted. This class of solution exhibits strong dipolar magnetic fields, whose presence can break the Taylor-Proudman constraint arising from Earth's rapid rotation. Hydrodynamic (or kinematic) simulations at low driving with otherwise similar parameters will produce velocity fields that, except near boundaries, are (essentially) invariant along the axis of rotation. As convective forcing increases in these models, this vertical anisotropy could also be broken by inertial forces; however, the solution class that then emerges (known as a CI(nertia)A balance) is generally dominated instead by kinetic energy, producing weaker multipolar magnetic fields, in stark contrast to the MAC class.

In this study, we investigate the solution space of models in which the magnetic field strength is constrained by an alternative boundary condition. We demonstrate that our magnetoconvection model still produces the known solution classes but also extends them, with intermediate solutions arising, particularly between the V(iscous)AC (which appears near onset) and MAC classes. We argue that examining the structure of the velocity and magnetic fields, as well as the force balance, in these simulations, can provide insight into which dynamo regimes and field structures can emerge in Earth and planetary contexts, when true physical parameters are still far out of the range of even the best current computers.

38th Scottish Fluid Mechanics Meeting

Investigating the role of Circulating Tumour Cell Softness in Inertial Microfluidic Devices

Roslyn Hay, Timm Krüger, Benjamin Owen

School of Engineering, University of Edinburgh

King's Building Campus, Edinburgh, EH9 3JL

28th May 2025

Abstract

Circulating tumour cells (CTCs) are cells that detach from the primary tumour, enter the bloodstream and can initiate secondary tumours. CTCs also act as an early biomarker for cancer detection, having the potential to improve survival rates; however, their rarity in blood makes identification challenging. Inertial microfluidic devices have shown potential in separating a lysed blood sample of CTCs and WBCs, with larger cells (CTCs) migrating laterally into the target stream faster.[1] However, clinical use, such as cell analysis and disease diagnostics, remains limited due to an incomplete understanding of the underlying physics and the role of cell softness, preventing full separation. This work uses numerical modelling to investigate how CTC softness — varying with cancer type and stage — affects separation efficiency. Here we show that the softness of the CTC influences not only its own migration behaviour but also the migration behaviour of WBCs into the target stream. Softer CTCs travel farther downstream before they migrate into the target stream, with a higher axial and lower lateral velocity. Additionally, when a stiffer CTC is present, WBCs tend to enter the target stream farther upstream. This study highlights the need to consider cell softness in the design of inertial microfluidic devices for optimised CTC separation.

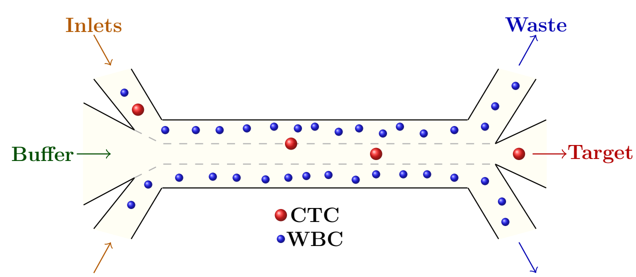


Figure 1: Schematic of Inertial Microfluidic Device: Lysed blood sample inserted through the inlets, CTCs leave via the target stream and WBCs leave via the waste streams.

References

[1] Zhou, J., Kulasinghe, A., Bogseth, A., O'Byrne, K., Punyadeera, C., and Papautsky, I., "Isolation of circulating tumor cells in non-small-cell-lung-cancer patients using a multi-flow microfluidic channel," *Microsyst Nanoeng*, Vol. 5, No. 1, 2019, pp. 8.

38th Scottish Fluid Mechanics Meeting

Single molecule dynamics in microfluidic flows: from homogeneous extension to mixed kinematics

Arezoo Khakpour[1], Zhibo Li[2], Samatha Robinson[2], Olivia du Roure[2], Emad Chaparian[1], Anke Lindner[2], Mónica S. N. Oliveira[1]

[1]James Weir Fluids Laboratory, Department of Mechanical and Aerospace Engineering, University of Strathclyde
Glasgow G1 1XJ, UK
[2]PMMH, CNRS, ESPCI Paris PSL,
F-75005, Paris, France

28th May 2025

Abstract

Understanding polymer dynamics at the microscale is critical for the design and optimisation of microfluidic systems in biotechnology and soft matter applications. We investigate the behaviour of individual model molecules (T4 DNA) in microfluidic flows using a combination of high-resolution fluorescence microscopy and Brownian dynamics simulations, which allows us to observe a wide range of dynamic behaviour, including morphological transitions. We first utilise a microfluidic platform in the form of a converging-diverging channel specifically designed to generate a well-defined and homogeneous strain field [1]. The optimised microchannel design enables consistent extensional flow, closely reproducing theoretical predictions for molecular deformation based on Hencky strain. We investigate the effects of history at varying Weissenberg numbers on the dynamic behaviour and molecular deformation. We then explore more complex structured pillar array configurations with spatially varying kinematics that can be tuned by changing the orientation of the flow relative to the pillars. This allows us to probe the effects of changing flow type on the molecular response. The results provide insights into flow-structure interactions at the single-molecule level that have implications for the development of advanced microfluidic platforms for DNA analysis and polymer processing.

References

[1] Liu et al., “Optimised hyperbolic microchannels for the mechanical characterisation of bio-particles” *Soft Matter*, 16, 2020, 9844-9856.

38^{th} Scottish Fluid Mechanics Meeting

Mathematical modelling of ice-sheet dynamics

Tanisha Kumari, Matthew Durey, Peter Stewart, Katarzyna Kowal
School of Mathematics and Statistics, University of Glasgow
Glasgow, G12 8QQ

28^{th} May 2025

Abstract

This study involves mathematical modelling of marine ice sheets, such as those of Antarctica and Greenland. Specifically, we seek to bridge the gap in understanding the role of subglacial till (water-saturated subglacial sediment) on the large-scale dynamics of ice sheets. Over large time and length scales, such as those of Antarctic ice, for example, ice behaves as a viscous fluid owing to the tremendous pressure gradients within the ice, which cause it to slowly flow like a viscous fluid. As such, we can apply fluid-mechanical principles to understand the flow of ice sheets. A similar principle applies to subglacial till on sufficiently large length scales. We model both the ice and till as thin films of viscous fluid that spread over rigid bedrock and into the ocean. Upstream, the ice sheet is in contact with the bedrock. As it flows towards the ocean, the ice sheet detaches from the bedrock at the grounding line, beyond which it feeds into a freely floating ice shelf. We assume that the dominant resistance in the grounded ice sheet arises due to vertical shear stresses, in line with the assumptions of the shallow ice approximation,[1] while the dominant resistance in the freely floating ice shelf is due to viscous extensional stress, in line with the assumptions of the shallow shelf approximation.[1] We also assume that the dominant resistance in the subglacial till is due to the vertical shear stress. While the rheology of ice and till is non-Newtonian, much of the underlying physical principles can be understood using a Newtonian rheology,[2] which is where we begin. Curiously, geophysical data indicates that till accumulates at the grounding line, stabilising it against retreat. We examine why these wedges form and their effect on the dynamics of the overlying ice.

References

[1] Schoof, C. and Hewitt, I. J., "Ice-sheet dynamics," *Annu. Rev. Fluid Mech.*, Vol. 45, 2013, pp. 217–239.

[2] Robinson, R. A. V., Huppert, H. E., and Worster, M. G., "Dynamics of viscous grounding lines," *J. Fluid Mech.*, Vol. 648, 2010, pp. 363–380.

38th Scottish Fluid Mechanics Meeting

Physics-based models for large-eddy simulations of (un)steady magnetohydrodynamic turbulence

Asif Nawaz*, Damiano Capocci†, Moritz Linkmann*

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

†SUPA, School of Physics and Astronomy, University of Edinburgh

28th May 2025

Abstract

For Navier-Stokes turbulence, contributions to the energy cascade from e.g. vortex stretching can be identified by an exact decomposition of the energy flux.[1] This results in a quantification of the relative contributions of such effects to the kinetic energy cascade by direct numerical simulation, and provides guidance for physics-informed LES modelling. LES modelling is of particular importance for magnetohydrodynamic (MHD) turbulence due to the extreme Reynolds numbers typical for plasma flows in astrophysics and nuclear fusion.[2] The provision of better models that capture the main physical mechanisms that govern MHD cascades would therefore be a major advance for computational research in the field. For MHD LES, difficulties arise due to interactions between flow and magnetic field,[2] and the unsteady nature of many fundamental problems in MHD, such as the kinematic and nonlinear dynamo. A key issue is that LES models typically underestimate the magnetic field amplification by a turbulent flow,[3] for instance, the turbulent small-scale dynamo is not captured adequately by current LES models in neither its kinematic, nonlinear or saturated (i.e. statistically steady) phases.

The decomposition formalism was recently extended to MHD, which led to the identification and quantifcation of the physical processes governing the energy cascade.[4] Here, we follow up on this work and formulate fundamental requirements of physical consistency that MHD LES model must have. Moreover, we devise and test LES models based thereon to provide new physics-informed modelling approaches for MHD turbulence and in particular the small-scale dynamo. We find that LES models that explicitly incorporate terms representative of current-sheet thinning and its back-reaction on the flow result in higher dynamo growth rates and nonlinear magnetic energy saturation levels.

References

[1] Johnson, P. L., "Energy transfer from large to small scales in turbulence by multiscale nonlinear strain and vorticity interactions," *Physical review letters*, Vol. 124, No. 10, 2020, pp. 104501.

[2] Miesch, M., Matthaeus, W., Brandenburg, A., Petrosyan, A., Pouquet, A., Cambon, C., Jenko, F., Uzdensky, D., Stone, J., Tobias, S., et al., "Large-eddy simulations of magnetohydrodynamic turbulence in heliophysics and astrophysics," *Space Science Reviews*, Vol. 194, 2015, pp. 97–137.

[3] Haugen, N. E. L. and Brandenburg, A., "Hydrodynamic and hydromagnetic energy spectra from large eddy simulations," *Physics of Fluids*, Vol. 18, No. 7, 2006.

[4] Capocci, D., Johnson, P. L., Oughton, S., Biferale, L., and Linkmann, M., "Energy flux decomposition in magnetohydrodynamic turbulence," *Journal of Plasma Physics*, Vol. 91, No. 1, 2025, pp. E11.

38^{th} Scottish Fluid Mechanics Meeting
Investigation of biomimetic riblet structures in a turbulent channel flow

Antreas Papasavvas [1], Panagiotis Alexandrou [1]
Pasquale Edurado Lapenna [2], Antonio Attili[1]
1. School of Engineering, Institute for Multiscale Thermofluids
University of Edinburgh, Edinburgh, EH9 3FD, United Kingdom
2. Department of Mechanical Engineering and Aerospace
Sapienza University of Rome, Via Eudossiana 18, 00184 Rome, Italy

28^{th} May 2025

Abstract

Riblet structures are a very actively researched topic as they can effectively reduce the skin friction drag acting on a surface. Multiple experiments have already been carried out for sawtooth, bladed and scalloped riblets, while other geometries are either neglected or no recent experiments were conducted. This project investigates the effects of low aspect ratio riblet structures in a turbulent channel flow near the wall and far away from the wall with the goal of understanding the flow behaviour for these surface profiles and build upon the existing literature. The simulations were completed on the Archer2 supercomputer using the Direct Numerical Simulation method. A smooth channel simulation is set up initially, which is then used as a reference to compare two rough channel simulations. For the rough wall cases, the surface profiles are a regular smooth sinewave and the absolute value of sinewave, which is a close approximation to the well established scalloped profile. The results of the simulations were extracted using statistical analysis and 3D visualization tools, following the well established theory of turbulence. The investigation revealed that the bulk flow remains unaffected by the riblets, while closer to the wall there are notable differences between the smooth and rough channels, particularly for the absolute sinewave channel that appears to be more effective compared to the larger, smoother regular sinewave channel. The results provide valuable insight into rough channel flows and allowing future researchers to use the data for the development of more simple models for used in Reynolds Averaged Navier Stokes (RANS) and Large Eddy Simulations (LES).

38th Scottish Fluid Mechanics Meeting

Cavitation abrasive surface engineering (CASF) process for sustainable titanium alpha case removal

Elangovan Parameswaran[1], Evgenia Yakushina[1], Xichun Luo[1], Daniel G. Sanders[2]

[1]Department of Design, Manufacture and Engineering Management, National Manufacturing Institute Scotland and Advanced Forming Research Centre, University of Strathclyde, Glasgow, UK

[2]Department of Mechanical Engineering, University of Washington, Washington, USA

Abstract

Superplastic forming (SPF) is widely used in aerospace industries using titanium alloy (Ti-6Al-4V) to manufacture complex parts using sheet metal, plate, and multi-sheet sandwich structures. The SPF process is usually carried out at elevated temperatures above 900°C. At these high temperatures, the presence of oxygen can lead to the formation of an alpha case, a hard and brittle layer that can serve as a prime location for crack initiation if not properly removed.

Industries use chemical etching process to remove the alpha case, but this method poses serious health risks due to the usage of hazardous chemicals like hydrofluoric and nitric acids. Additionally, disposing of used chemicals faces significant environmental challenges, highlighting the need for safer, and sustainable alternatives.

In this paper, a sustainable surface engineering process, cavitation abrasive surface finishing (CASF) was studied to remove the alpha case from heat treated titanium test samples. Initial experiments were conducted using high-speed camera to analyse the flow characteristics of cavitation cloud generated during the CASF process. Detailed experiments were conducted to understand the effect of individual and combined process variables for the effective alpha case removal. Further, a preliminary computational fluid model was developed using ANSYS Fluent to analyse the flow characteristics using different turbulence models. The variants of K-epsilon (k-ε) and the K-omega (k−ω) models were computed. The CFD results were then compared with experimental flow behaviour, and it reveals that the K-epsilon RNG model was effective in predicting the turbulence flow behaviour.

Keywords: TITANIUM, SPF, CASF, CAVITATION, ALPHA CASE REMOVAL, ANSYS FLUENT

References

[1] Miguel G. Coussirat, Flavio H. Moll and Alfred Fontanals, CAPABILITY OF THE PRESENT CAVITATING AND TURBULENCE MODELS FOR CONFINED FLOW SIMULATIONS Mecánica Computacional Vol XXXIV, págs. 1989-2007.

[2] Nagalingam, A.P., V.C. Thiruchelvam, and S.H. Yeo, A novel hydrodynamic cavitation abrasive technique for internal surface finishing. Journal of Manufacturing Processes, 2019. 46: p. 44-58.

[3] Soyama, H. and D. Sanders, Use of an Abrasive Water Cavitating Jet and Peening Process to Improve the Fatigue Strength of Titanium Alloy 6Al-4V Manufactured by the Electron Beam Powder Bed Melting (EBPB) Additive Manufacturing Method. Jom, 2019. 71(12): p. 4311-4318.

[4] Nagalingam, A.P. and S.H. Yeo, Surface finishing of additively manufactured Inconel 625 complex internal channels: A case study using a multi-jet hydrodynamic approach. Additive Manufacturing, 2020.

38th Scottish Fluid Mechanics Meeting

Influence of lower limb amputation on the cardiovascular system – a CFD investigation

Sava Popescu, Arjan Buis, Asimina Kazakidi
Department of Bioengineering, University of Strathclyde
106 Rottenrow, Glasgow, G4 0NW

28th May 2025

Abstract

Following amputation, patients face significantly higher cardiovascular risk compared to the general population, still the underlying mechanisms that cause this remain poorly characterised [1]. With a cardiovascular system already compromised in most cases by diabetes or peripheral arterial disease – the main causes of amputation – limb loss itself introduces systemic vascular changes which can aggravate the issue. Altered vascular geometry and disrupted flow patterns can have a negative effect on arterial stiffness, blood pressure and reduce arterial perfusion efficiency [2]. The aim of this study is utilising computational fluid dynamics (CFD) to simulate blood flow in a high-resolution anatomically accurate 3D model of the lower limb vascular tree. The vascular geometry is reconstructed from CT angiograms acquired from a healthy individual. By performing plane cuts on the healthy model, "synthetic amputations" are achieved to mimic the surgical outcome with models corresponding to different types and severities of amputation both in the unilateral and bilateral cases. A range of flow conditions can be simulated within the CFD model to capture the complex haemodynamic changes associated with lower limb amputation. By modelling bilateral amputation, the current study is able to elucidate the influence on cardiac output, the asymmetric distribution of flow, and the increased haemodynamic burden placed on proximal and contralateral blood vessels. Additionally, conditions corresponding to different pathologies can be incorporate in the flow conditions of the model to examine how common comorbidities can aggravate circulatory strain in the post-amputation phase. Potential applications of the results of the study include the design of prosthetic and assistive devices, surgical decision-making or personalised rehabilitation planning, all aiming to mitigate adverse circulatory effects and support long-term cardiovascular health.

References

[1] Mundell, B.F., Luetmer, M.T., Kremers, H.M., Visscher, S., Hoppe, K.M. and Kaufman, K.R. (2018). The risk of major cardiovascular events for adults with transfemoral amputation. *Journal of NeuroEngineering and Rehabilitation*, 15(S1). doi:https://doi.org/10.1186/s12984-018-0400-0.

[2] Obeid, H., Bikia, V., Segers, P., Pare, M., Boutouyrie, P., Nikos Stergiopulos and Mohsen Agharazii (2024). Impact of arterial system alterations due to amputation on arterial stiffness and hemodynamics: a numerical study. *Scientific Reports*, [online] 14(1). doi:https://doi.org/10.1038/s41598-024-75881-5.

38th Scottish Fluid Mechanics Meeting

Numerical Simulation and Validation of Concentration Polarisation Effects of Sucrose Aqueous Solutions in Nanofiltration Membranes

A. Puigdefabregas Nogueras[1], A. Kazakidi[1], E. H. Papaioannou[2]
[1]University of Strathclyde, Glasgow, UK
[2]Lancaster University, Lancaster, UK

28th May 2025

Abstract

Introduction

Filtration of organic molecules from aqueous solutions is extensively utilised in food processing, water treatment and pharmaceutical industries[1], where efficient separation and concentration methods are critical. Pressure-driven nanofiltration (NF) processes are employed for separating organic molecules from aqueous solutions, though they are susceptible to concentration polarisation (CP). CP arises from solute accumulation on the retentate side of the membrane, increasing osmotic pressure ($\Delta\pi$) and counteracting the transmembrane pressure (TMP), thereby reducing permeate flux[2]. The appropriate tuning of TMP and crossflow velocity (CFV) can mitigate flux decline by reducing solute back-transport. Computational fluid dynamics (CFD) offers a cost-effective approach to model and visualize the CP dynamics and predict membrane performance during NF, under various operating conditions [2]. This study develops a CFD model to simulate the behaviour of a commercial NF membrane processing organic molecule solutions, addressing CP effects under varying operational conditions.

Experimental/methodology

A commercially available flat-sheet polyamide-based NF membrane (NF270, Dow Filmtec) was tested experimentally in a crossflow configuration of a prototype scale equipment composed of a jacketed thermostatic feed tank (30L), a permeate tank and a pump under constant temperature (30±2°C), crossflow rates of 500-1500ml/min and pressures of 2-7 bar. The membrane was compacted with deionised water (DI) at a constant pressure of 7 bar, until the membrane flux was stable. A sucrose aqueous solution at 1% w/v was used. A two-dimensional, steady flow CFD model was developed using the commercially available software COMSOL Multiphysics (Version 6.2.0.290, COMSOL inc.). The dimensions of the 2D model was created for the flat-sheet module, 75x30x1 (mm), and a membrane thickness of 0.25mm was assumed. The membrane was modelled using the Brinkman equations for porous media flow, while the main channel was governed by the Navier-Stokes equations.

Additionally, the performance of the CFD model was compared against the results obtained by Geraldes et al.[2], to validate this new approach for cases of extreme CP conditions.

Results and Discussion

The experimental fluxes for 1% sucrose solutions at both 500 and 1500 ml/min increased linearly with transmembrane pressure (TMP) and were predicted accurately by the CFD model (maximum absolute error was less than 4% in all cases). The performance of the current CFD model against the results from Geraldes et al.[2] showed a slightly improved accuracy compared to their predictive model for their experimental conditions, with a maximum error of 9.5% for the case of high CFV when compared to their experiments. This suggests that while the model has improved accuracy, it might not fully quantify the effects of CFV in understanding the CP layer contribution in flux decline.

Conclusions

A CFD model was developed to predict the flux decline due to CP near a pressure-driven flat-sheet membrane. The simulations showed good agreement with our experimental results for all TMPs. Testing the present CFD model against the available limited literature data, showed an improvement in respect to its prediction capabilities. These initial CFD results are promising and contribute to the understanding of NF processes CP and mass transfer.

References

1. Luo, J., Guo, S., Wu, Y., & Wan, Y. (2018). Separation of Sucrose and Reducing Sugar in Cane Molasses by Nanofiltration. Food and Bioprocess Technology, 11(5), 913–925.
2. Geraldes, V., Semião, V., & Norberta De Pinho, M. (2001). Flow and mass transfer modelling of nanofiltration. In Journal of Membrane Science (Vol. 191).

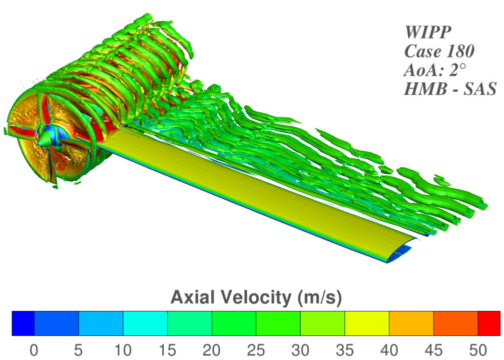
38^{th} Scottish Fluid Mechanics Meeting
Propeller-Wing Aerodynamic Interactions

Geng Qiao and George N. Barakos
CFD Laboratory, School of Engineering, University of Glasgow
Scotland, UK

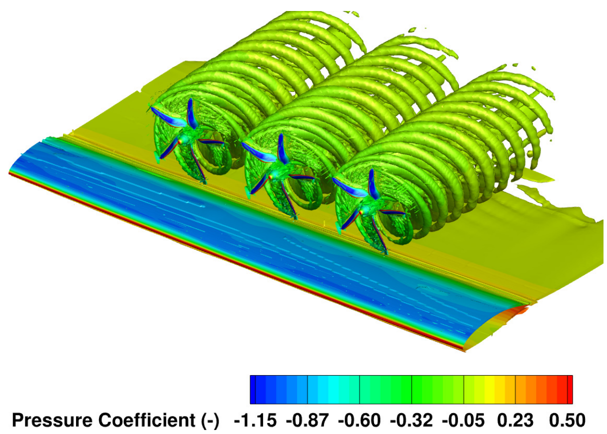
28^{th} May 2025

Abstract

In an effort to bridge the knowledge gap in propeller/wing interactions, tests on wingtip-mounted propellers (WIPP) were conducted in the Lockheed Martin low-speed wind tunnel (LSWT).[1] The flow visulisation of the TMP and the OTWDP propulsion systems using the high-fidelity CFD code HMB3 are presented in Figure 1.



(a) Wingtip-mounted propeller



(b) Flow visualisation OTWDP system at $\mu = 0.42$.

Figure 1: Flow visualisation of wingtip-mounted propeller and three-propeller, installed distributed propulsion systems.

References

[1] Hooker, J. R., Wick, A., Ginn, S. R., Walker, J., and Schiltgen, B. T., "Overview of Low Speed Wind Tunnel Testing Conducted on a Wingtip Mounted Propeller for the Workshop for Integrated Propeller Prediction," *AIAA AVIATION 2020 FORUM*, 2020, p. 2673.

38th Scottish Fluid Mechanics Meeting

Flow Characterisation of Oscillating Jet Nozzles for Wellbore Descaling

Mubaraq Adewale Razaq, Okwunna Maryjane Ekechukwu, Khaliq Ur Rahman, Taimoor Asim & Murphy Lai

School of Computing, Engineering & Technology, Robert Gordon University, Aberdeen, United Kingdom.

28th May 2025

Abstract

Wellbore descaling and cleanup are vital processes in drilling, well completion and Enhanced Oil Recovery (EOR) for flow assurance and well longevity.[1] Wellbore surface fouling and material build-up pose significant challenges, leading to reduced operational efficiency, increased maintenance costs and operational downtime.[2] Conventional jet cleaning methods, based on straight jets, are limited by poor area coverage and long operational time. These limitations give rise to a need for energy-efficient solutions with superior wellbore surface cleaning. Oscillating Jet Nozzles (OJNs) have emerged as a promising alternative due to their capability of delivering oscillatory jets that enhance the impact force on the target surface, thus improving cleaning performance.[3] This study presents a detailed investigation into flow characteristics and performance of an oscillating jet nozzle through an integrated computational and experimental approach. A numerical model was developed using Computational Fluid Dynamics (CFD) based solvers to simulate the internal flow behaviour within a passive OJN. Results obtained reveal dynamic jet attachment due to Coanda effect, leading to flow oscillations, that contribute towards enhanced surface coverage, as shown in Figure 1. It has also been observed that an increase in injected flowrate from 29-58 L/min results in a corresponding increase in impact force from 10-44N. The numerical results are in close agreement with the experiment results, as shown in Figure 2. The findings confirm the effectiveness of OJNs in enhanced wellbore descaling while reducing the operational carbon footprint and the lead time, and conserving water.

References

[1] LONGVAL, R. et al., 2024. An Overview of Silica Scaling Reduction Technologies in the Geothermal MarketEnergies, 17(19), pp. 4825.

[2] AL-RUBAII, M., AL-SHARGABI, M. and AL-SHEHRI, D., 2023. Hole Cleaning during Drilling Oil and Gas Wells: A Review for Hole-Cleaning Chemistry and Engineering Parameters*Advances in Materials Science and Engineering,* 2023(1), pp. 6688500

[3] BAGHAEI, M. and BERGADA, J.M., 2020. Fluidic oscillators, the effect of some design modifications. Applied Sciences, 10(6), pp. 2105.

Appendix

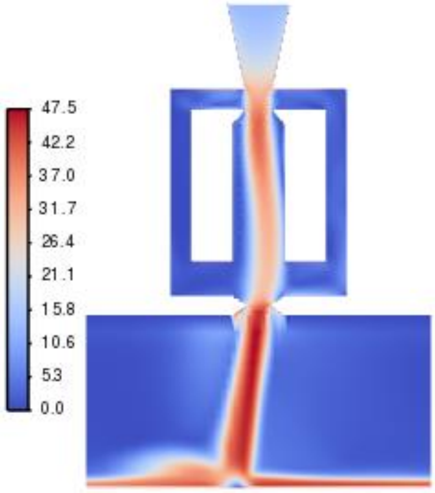


Figure 1. Flow velocity magnitude (m/s)

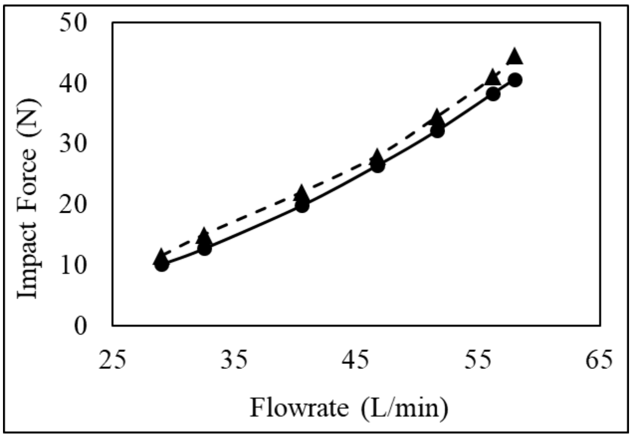


Figure 2. Jet Impact force (N)

38th Scottish Fluid Mechanics Meeting

Evaporation of an Elongated Droplet

Lauren B. Schofield[1], Alexander W. Wray[1], Debasish Das[1], and Stephen K. Wilson[2,1]

[1]Department of Mathematics and Statistics, University of Strathclyde, 26 Richmond St, Glasgow, G1 1XH

[2]Department of Mathematical Sciences, University of Bath, Claverton Down, Bath, BA2 7AY

28th May 2025

Abstract

The final deposit patterns left behind on a substrate by evaporating particle-laden droplets are of great interest due to applications such as inkjet printing in electronics, where they are, for example, used in the manufacture of conductors and electrodes.[1] Most existing studies of small sessile droplets examine the situation in which the footprint of the droplet is circular, and hence the free surface of the droplet forms a segment of a spherical cap. In the present work we examine the evaporation of an elongated, particle-laden droplet for which the footprint of the droplet is a rectangle whose length is much greater than its width. We investigate how the ratio of these lengths affects the evaporation dynamics of the droplet, including how they influence its lifetime and the final deposition of particles. Building on work concerning the evaporation of two-dimensional droplets, we use the method of matched asymptotic expansions to formulate outer and inner problems for the total evaporative flux. The resulting solution for the total evaporative flux allows us to determine the evolution, and hence the lifetime, of the droplet in various different modes of evaporation. In particular, we consider four evaporation modes, namely the constant-radius, constant-angle, stick-slide, and stick-jump modes. In the case in which the contact line is pinned (i.e. in the constantradius mode), we analyse the flow and particle transport within the droplet. In particular, we find that the flow transports all particles toward the contact line, leading to a final deposit concentrated at the edges, corresponding to a rectangular coffee-ring.

References

[1] Szczech, J., Megaridis, C., Gamota, D., and Zhang, J., "Fine-line conductor manufacturing using drop-on demand PZT printing technology," *IEEE Transactions on Electronics Packaging Manufacturing*, Vol. 25, No. 1, 2002, pp. 26–33.

38^{th} Scottish Fluid Mechanics Meeting

Physics-Informed Neural Networks for Predicting Rhodamine Diffusion Coefficient

Quinn Stein, Humphrey Yiu, Ali Ozel
School of Engineering and Physical Sciences, Heriot-Watt University
Edinburgh EH14 4AS

28^{th} May 2025

Abstract

Accurate measurement of drug diffusion coefficients is essential to optimize drug delivery systems. We focus on Rhodamine as a model compound to simulate drug behaviour and use its widely documented diffusive behaviour as a benchmark for *in silico* studies. In this study, we developed an experimental setup to track Rhodamine diffusion in water, generating spatio-temporal concentration data. These images are input for a physics-informed neural network (PINN) [1] to inversely solve diffusion PDE determining the Rhodamine diffusion coefficient. Figure a and b show the digital image of the Rhodamine concentration front at initial condition and $t = 2\,\mathrm{s}$, respectively. Figure c shows the corresponding PINN predictions of the Rhodamine concentration with the error map (Figure d). The predicted diffusion coefficient of $\mathrm{D} = 3.7 \times 10^{-10}\,\mathrm{m}^2\,\mathrm{s}^{-1}$ for rhodamine in water, which is in good agreement with the range of values previously reported [2]. This experimental and computational framework provides a platform for diffusion coefficient determination without requiring extensive calibration experiments. Future work will extend this methodology to track Rhodamine binding with proteins, employing an enhanced PINN architecture coupled with reaction-diffusion equations solving for diffusion coefficients, reaction rate, and reaction order.

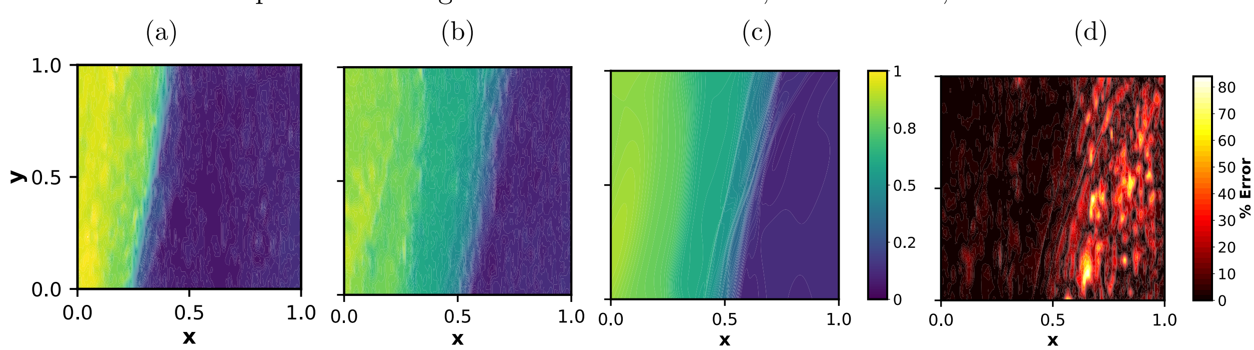


Figure 1: Rhodamine concentration colormaps: (a) Digital image at $t = 0$; (b) Digital image at $2\,\mathrm{s}$; (c) PINN predictions at $t = 2\,\mathrm{s}$; (d) Percent error between ground truth and PINN predictions at $t = 2\,\mathrm{s}$.

References

[1] Raissi, M. et al., *J. Comput. Phys.*, Vol. 378, 2019, pp. 686–707.

[2] Gendron, P.-O. et al., *J. Fluoresc.*, Vol. 18, No. 6, 2008, pp. 1093–1101.

38th Scottish Fluid Mechanics Meeting

Ferrofluid emulsion in shear flow: Effects of viscosity ratio and magnetic field

A. Sultan, P. Capobianchi, M.S.N Oliveira
James Weir Fluids Laboratory, Department of Mechanical and Aerospace Engineering, University of Strathclyde
75 Montrose Street, Glasgow, G1 1XJ

28th May 2025

Abstract

This study investigates the dynamics and rheology of ferrofluid droplet emulsions under simple shear flow and uniform magnetic field. Specifically, we explore the role of viscosity ratio, defined as the ratio between droplet and matrix fluid viscosities, using a Volume of Fluid-Level Set method implemented in OpenFOAM[1,2]. We validate our results in term of the effect of capillary number on the droplet deformation and orientation against existing numerical and experimental literature and then further extend it to examine the effect of magnetic field at different Bond numbers. Magnetic stresses are found to deform and orient the droplets along the magnetic field direction for all viscosity ratios tested, with larger viscosity ratios leading to less deformed droplets. Finally, we evaluate the emulsion's bulk rheological properties using the approach of Capobianchi et al.[3], and using a direct integration of the viscous stress. Magnetothickening behavior was observed for all viscosity ratios tested, with the emulsion's effective (bulk) viscosity increasing for increasing Bond number.

References

[1] Capobianchi, Paolo, Marcello Lappa, and Mónica SN Oliveira. "Deformation of a ferrofluid droplet in a simple shear flow under the effect of a constant magnetic field." *Computers & Fluids*, Vol. 173, 2018, pp. 313-323.

[2] Yamamoto, Takuya, Yasunori Okano, and Sadik Dost. "Validation of the S-CLSVOF method with the density-scaled balanced continuum surface force model in multiphase systems coupled with thermocapillary flows." *International Journal for Numerical Methods in Fluids*, Vol. 83, No. 3, 2017, pp. 223-244.

[3] Capobianchi, Paolo, Marcello Lappa, Mónica SN Oliveira, and Fernando T. Pinho. "Shear rheology of a dilute emulsion of ferrofluid droplets dispersed in a nonmagnetizable carrier fluid under the influence of a uniform magnetic field." *Journal of Rheology*, Vol. 65, No. 5, 2021, pp. 925-941.

38th Scottish Fluid Mechanics Meeting

Pore- and Core-Scale Observation of Gas-Water Intermittent Flow in Natural Porous Media: Unravelling Intermediate Region Between Capillary and Viscous Dominance

Amin Taghavinejad, Azibayam Josiah Amabogha, Yihuai Zhang
School of Engineering, University of Glasgow G12 8QQ, Glasgow, UK

28th May 2025

Abstract

Two-phase flow in porous media, neglecting gravitational effects, is governed by capillary and viscous forces. Defining capillary number (Ca) as the ratio of viscous to capillary forces, in very low Ca values, flow is capillary dominated, while a viscous-dominated flow prevails in higher Ca close to 1. In the transitional region between these extremes, a flow regime is found with intermittent pore space occupancy by the two (wetting and non-wetting) phases [1]. Compared to oil-water flow in natural porous media, gas-water dynamics in intermittent flow is less explored, limited to studies on nitrogen-brine with pore-scale imaging [2]. In this study, we carried out pore-scale synchrotron CT imaging and differential pressure recoding for CO_2-brine and H_2brine systems, which were co-injected into core samples. The results suggest that onset of intermittency occurs at Ca ~10^{-9}-10^{-7} in gas-water systems, comparable to that of oil-water systems (Ca~10^{-6}-10^{-5}). Pore-scale imaging reveals that intermittent flow pathways stabilise after a initial development period. This indicates that although pore volume fraction intermittently occupied by the two phases grows with Ca ($\propto$ total velocity), this development halts beyond a certain threshold Ca. We theoretically demonstrate that this threshold capillary number corresponds to the maximum interfacial energy. Experimental observations from two carbonate/CO_2/brine and sandstone/H_2/brine systems of this research support this theoretical threshold Ca. Results combined with pressure recordings show that gas-water intermittent flow is of two stages: 1) developing intermittent flow (DIF), and 2) Stabilized intermittent flow (SIF). DIF exhibits a nonlinear, non-Darcian behavior of pressure gradient (∇P) with Ca, whereas that of SIF proves Darcian. Considering Darcy flow, ∇P=Ca holds true in SIF. In contrast, pressurerate behaviour in DIF follows $\nabla P = Ca^a$ with ‘a’ exponent ranging between 0.05 and 0.3, depending on the co-injected gas-brine fraction.

References

[1] Y. Gao, A. Qaseminejad Raeini, M. J. Blunt, and B. Bijeljic, “Pore occupancy, relative permeability and flow intermittency measurements using X-ray micro-tomography in a complex carbonate,” *Adv Water Resour*, vol. 129, pp. 56–69, Jul. 2019, doi: 10.1016/J.ADVWATRES.2019.04.007.

[2] C. Spurin, T. Bultreys, B. Bijeljic, M. J. Blunt, and S. Krevor, “Intermittent fluid connectivity during two-phase flow in a heterogeneous carbonate rock,” *Phys Rev E*, vol. 100, no. 4, p. 043103, Oct. 2019, doi: 10.1103/PHYSREVE.100.043103/FIGURES/9/MEDIUM.

38th Scottish Fluid Mechanics Meeting

Thermo-hydraulic performance analysis of a hybrid Wavy Bifurcated Microchannel Heat Sink

Evans Joel Udom, Marcello Lappa
Department of Mechanical and Aerospace Engineering, University of Strathclyde
Glasgow, UK

28th May 2025

Abstract

A numerical investigation into the thermo-hydraulic performance of a hybrid “bifurcated-wavy” microchannel heat sink (MCHS) has been carried out. The choice of this specific geometry is motivated by the observation that “bifurcations” and “wavy geometries” have often been employed separately in the literature as relevant strategies to enhance the thermal performance of MCHS.[1,2] In particular, the present study relies on the direct numerical solution of the governing equations for mass, momentum, and energy conservation. The performance of the hybrid MCHS configuration is evaluated by varying several key parameters, including the number and position of internal geometrical bifurcations and the curvature of the sinusoidal channel itself. Specifically, the curvature angle spans the range between 30° and 45°, while bifurcation barriers are placed at distances of 10 mm, 20 mm, and 25 mm from the microchannel inlet. The interplay between heat transfer efficiency and frictional losses also constitutes an integral part of the analysis, which considers increasing values of the Reynolds number up to 1500. Particular attention is also paid to the onset of fluid dynamic instabilities and the transition to soft turbulence. The results demonstrate that the Nusselt number increases with channel waviness, and that the wavy channel with two internal bifurcations exhibits superior thermal performance compared to other configurations. Fluid dynamic instabilities emerge due to the continuously varying curvature of the channel walls, generating vorticity that enhances fluid mixing and thereby improves vertical heat transfer. The associated frequency spectra become increasingly complex with rising Reynolds number, extending towards higher frequencies, and exhibiting greater magnitudes in the velocity components involved. However, the introduction of bifurcation plates tends to postpone the onset of these instabilities to higher Reynolds numbers (Re > 900).

References

[1] Sui Y., Teo C., Lee P., Chew Y., Shu C., Fluid flow and heat transfer in wavy microchannels, Int. J. Heat Mass Transf., 53, 2010, 2760–2772.
[2] Udom, E.J. and Lappa, M., "Comparative analysis of microchannel heat sinks for different values of the Prandtl and Reynolds numbers", International Journal of Numerical Methods for Heat & Fluid Flow, Vol. ahead-of-print No. ahead-of-print, 2025.

38th Scottish Fluid Mechanics Meeting

Vibro-Fluidization of Lunar Regolith Simulant in Hourglass and Inclined Pipe Configurations

Peter Watson[1*], Sebastien Vincent Bonnieu[2], Ali Anwar[1], Marcello Lappa[1]
[1]Department of Mechanical and Aerospace Engineering, University of Strathclyde, James Weir Building, 75 Montrose Street, Glasgow, G1 1XJ, UK
[2]European Space agency, European Space Research and Technology Centre Keplerlaan 1 2201 AZ Noordwijk, Netherland

28th May 2025

Abstract

Under the umbrella of the ISRU (In-Situ Resource Utilization) approach, we investigate the application of vibrations as a means to excavate and transport lunar regolith in the lunar environment. Shaking is applied to both hourglass-shaped and inclined pipe vessels containing regolith simulants, in a variety of directions and intensities, to determine the optimal conditions for regolith manipulation. This is achieved through an exhaustive parametric study, wherein one variable is altered at a time while keeping the other experimental parameters constant.

In the hourglass configuration, vertical and horizontal vibrations are applied to fixed quantities of simulant material, with acceleration and frequency values ranging from $\Gamma = 1$ to 10 and $f = 20$ to 500 Hz, respectively. The resulting effects on the mass flow rate of material discharged through the vessel's orifice are recorded and analyzed. A similar methodology is employed for an inclined acrylic pipe, 0.5 meters in length. In this case, the pipe's inclination is varied between 5° and 30° from the horizontal, while the vibration direction is progressively rotated from 0° to 30° relative to the vertical.

These experiments reveal a wide spectrum of flow regimes for both configurations. We demonstrate that, under specific conditions, vibrations can enhance the flow of material through the hourglass by disrupting flow-limiting arch formations, thereby improving discharge rates compared to the unforced (non-vibrated) case. Conversely, other conditions significantly hinder or even completely suppress flow. The same duality is observed in the inclined pipe experiments: certain parameter combinations enable effective transport of particles upward, against gravity, in a laminar regime, while others induce chaotic motion that degrades transport efficiency. The findings are interpreted through a focused lens: identifying the optimal operational parameters for efficient regolith transport.

Ultimately, this work provides essential insights for the engineering design of advanced ISRU technologies, with the goal of elevating their Technology Readiness Levels (TRLs) for practical deployment in future lunar missions.

38th Scottish Fluid Mechanics Meeting

Parallelised convolutions for generative models of plasma turbulence

Josh Williams, Jony Castagna
Hartree Centre, STFC Daresbury Laboratory, Daresbury, UK

28th May 2025

Abstract

Simulating plasma turbulence in the edge region of tokamak fusion reactors is challenging due to lack of RANS and LES models able to capture the strong inverse cascade of energy injected at small scales by the magnetized particles. Previously, we have used generative adversarial networks (GANs) to reconstruct DNS plasma flowfields in large eddy simulations.[1] However, when training large GANs on high resolution images, the model and its activations no longer fit on a single GPU. To overcome GPU memory limitations for training GANs on high resolution flowfields, we aim to leverage spatial parallelism through the Livermore Big Artficial Neural Network (LBANN) framework.[2] This essentially performs domain-decomposition for the convolutional layers in a GAN, allowing one image to be spread across several GPUs. We examine the strong and weak scalability of the hybrid spatial-data-parallel learning approach. Alongside performance results, we will demonstrate the GAN's ability to reconstruct DNS flow fields given filtered DNS data.

References

[1] Castagna, J., Schiavello, F., Zanisi, L., and Williams, J., "StyleGAN as an AI deconvolution operator for large eddy simulations of turbulent plasma equations in BOUT++," *Physics of Plasmas*, Vol. 31, No. 3, 2024.

[2] Dryden, N., Maruyama, N., Benson, T., Moon, T., Snir, M., and Van Essen, B., "Improving strong-scaling of CNN training by exploiting finer-grained parallelism," *2019 IEEE International Parallel and Distributed Processing Symposium (IPDPS)*, IEEE, 2019, pp. 210–220.

38th Scottish Fluid Mechanics Meeting

Quantifying the Mechanical Properties of Biomolecular Condensates

Thomas A. Williamson[1], Jack O. Law[2], Thomas Stevenson[2], Carl M. Jones[2], Sushma N. Grellscheid[2], and Halim Kusumaatmaja[1]

[1]Institute for Multiscale Thermofluids, School of Engineering, University of Edinburgh, Edinburgh, UK
[2]Computational Biology Unit and Department of Biological Sciences, University of Bergen, Bergen, Norway
[3]Department of Biosciences, Durham University, Durham, UK

28th May 2025

Abstract

Biomolecular condensates are viscoelastic droplets, rich in specific biomolecules, which play many important roles in sub-cellular organisation such as regulating stress response. There is growing recognition that the mechanical properties of condensates govern their macroscopic behaviour, which is often substantially different from that of purely viscous droplets. However, until recently, it has been difficult to measure these properties at scale.

We demonstrate a flicker spectroscopy method for measuring two key mechanical properties of thousands of condensates in live cells, and present *FlickerPrint*, an open-source Python package for handling the computational requirements of the method. We find that the fluctuation spectra of stress granules (a type of condensate) in live cells cannot be adequately fitted with an interfacial tension-only model, as expected for simple Newtonian liquids. Instead, the measured fluctuation spectra require an additional contribution, which we attribute to elastic bending deformation. We then show that at the population level, interfacial tension and bending rigidity span several orders of magnitude so cannot be accurately determined by observing only a small number of condensates. However, the mean behaviour of these properties across a population of stress granules can be used to distinguish between granules containing different ratios of constituent proteins.

In addition, we have preliminary results which suggest that stress granules display broken detailed balance in the amplitudes of the fluctuation modes, which we believe is a signature of the liquid-to-solid ageing transition which many condensates undergo.

38th Scottish Fluid Mechanics Meeting

A biphasic flow model in idealised and engineered microvessels

Tanchanok Wisitponchai, Laure Vidal-Roussel, Junxi Wu, Asimina Kazakidi
Department of Biomedical Engineering, University of Strathclyde, Glasgow G4 0NW, UK

28th May 2025

Abstract

The relationship between haemorheology and clinical conditions such as ischemia or cancer remains poorly understood, due in part to microvascular variability and limitations of both in vivo and in vitro research methods[1]. The red blood cell (RBC) distribution within microvascular networks is affected by the formation of a cell-free layer (CFL), which develops adjacent to the vessel wall and envelops the central RBC-rich region. This study investigates computationally blood distribution and mechanical responses in a two-phase continuum-based numerical model consisting of the RBC phase dispersed in the CFL phase. The model was developed in a symmetrical 3D Y-shaped bifurcation and applied to a planar capillary network reconstructed from images of bio-artificial microvessels. With a parent diameter of 20 μm, the geometries were meshed using polyhedral and prismatic elements in Star-CCM+ and simulated the two phases using the Eulerian Multiphase (EMP) method. Each phase was governed by the Navier-Stokes equations with different rheological properties. Results indicated that symmetrical annular CFL formed due to lateral drag and perpendicular lift forces at the RBC-CFL interface, driven by high slip velocity near vessel walls. These forces diminished at bifurcation junctions but intensified downstream, pushing RBC phase toward the center and promoting CFL phase formation. Although asymmetry developed near bifurcation, symmetry gradually recovered at a long distance downstream of the bifurcation. These findings align with experimental data[2]. Moreover, complex capillary geometry (asymmetrical bifurcations, tortuosity, short inter-bifurcation distance) resulted in uneven forces and RBC distribution. Flow velocity was the primary determinant of RBC distribution and WSS patterns in engineered capillaries, whereas pressure had minimal impact on RBC distribution at low velocities. Future studies could couple RBC dynamics with oxygen diffusion models to assess impacts on tissue perfusion.

References

[1] Alexy et al, “Physical properties of blood and their relationship to clinical conditions”, Front Physiol, 13:906768, 2022.
[2] Ng, Y.C., et al., 2016. American Journal of Physiology-Heart and Circulatory Physiology, 311(2):H487-H497.

.

38th Scottish Fluid Mechanics Meeting

Resonant triads of gravity waves in confined basins

Jie Yang, Katarzyna Kowal and Matthew Durey
School of Mathematics & Statistics, University of Glasgow
Glasgow G12 8QQ, UK

28th May 2025

Abstract

Confining a liquid to a partially filled basin can lead to the spontaneous formation of resonant triads of gravity waves, in which three standing wave modes interact and continuously exchange energy. This resonance mechanism induces waves of relatively large magnitude, representing a potential hazard in many industrial settings, such as the transport of liquid cargo. However, the formation of these triads depends critically on the dimensions of the container and the depth to which it is filled, making resonant triads challenging to identify and analyse.

I will present the results of a combined numerical and theoretical investigation into the existence and evolution of resonant triads in a wide class of confining basin. Particular attention will be paid to determining the critical container geometry and relative efficiency of energy exchange for different triad interactions. The results of this study may be used as a basis for developing strategies to mitigate hazardous resonances in industrial settings.

38th Scottish Fluid Mechanics Meeting

Adaptive mesh-free CFD modelling for rotorcraft

Tao Zhang, George Barakos
James Watt School of Engineering, University of Glasgow

28th May 2025

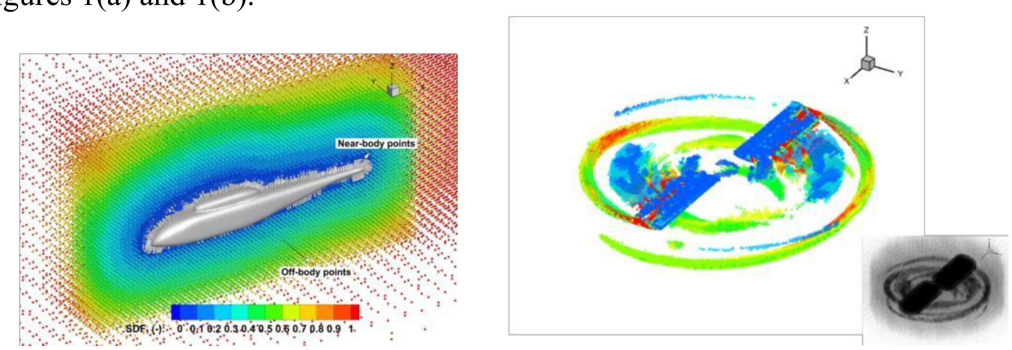
Abstract

This work presents the development of a novel adaptive, mesh-free, and largely automated CFD workflow within the Navier-Stokes framework, and its applications in rotorcraft flow modelling.

We will first introduce the development of a collocation-based Differential Quadrature (DQ) discretisation[1] of the compressible RANS equations. This approach reduces the need for mesh elements with complex connectivities to simple point clouds with a generalised Finite Difference concept. We will introduce the key principles, including least-squares approximations, collocation search, and weight balancing, and demonstrate the scheme's compatibility with classic mesh-based schemes within the Navier-Stokes framework.

We will then introduce a novel point cloud generation approach[2] for this meshfree CFD scheme. This approach incorporates the Signed Distance Function to guide point generation for arbitrary geometries, and adopts Cartesian points for efficient adaptive modelling.

This meshfree CFD framework will be demonstrated with various 2D/3D benchmark cases, especially with applications in rotorcraft flow modelling, such as helicopter fuselage and rotor flows in Figures 1(a) and 1(b).



(a) Point cloud for meshfree CFD simulation of the ROBIN fuselage shape.

(b) Adaptive simulation of the Caradonna-Tung rotor flow.

Figure 1: Applications of adaptive mesh-free CFD modelling in rotorcraft.

References

[1] Zhang, T. and Barakos, G. N., "Assessment of implicit adaptive mesh-free CFD modelling," *International Journal for Numerical Methods in Fluids*, Vol. 96, No. 5, 2024, pp. 670–700.

[2] Zhang, T. and Barakos, G. N., "SDF-Guided Point Cloud Generation Framework for MeshFree CFD," *International Journal for Numerical Methods in Fluids*, 2025.