37th Scottish Fluid Mechanics Meeting

29 May 2024

Edinburgh

THE UNIVERSITY OF EDINBURGH

LAVISION
FOCUS ON IMAGING

DANTEC DYNAMICS

Contents

About

SFMM37

The Scottish Fluid Mechanics Meeting, now in its 37th 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.

Conference organisers

Antonio Attili

Livio Gibelli

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, posters**

9:50–10:00 **Welcome and introduction**

10:00–12:00 **Session 1**

10:00–10:15 **Active Darcy's Law**
Ryan R. Keogh, Timofey Kozhukhov, Kristian Thijssen, Tyler N Shendruk
School of Physics and Astronomy, The University of Edinburgh

10:15–10:30 **Instabilities of thin-film flow over a spinning disk**
Laura Milne, Alexander W Wray, Omar K Matar, Marc Pradas, Stephen K Wilson
Department of Mathematics and Statistics, University of Strathclyde; Department of Chemical Engineering, Imperial College London; School of Mathematics and Statistics, The Open University

10:30–10:45 **CFD Simulation of Fast Fluidized Bed Carbonator for CO2 Capture Process using the Eulerian-Eulerian Model**
Darlington Amadi, Aditya Karnik, Mamdud Hossain
School of Engineering, Robert Gordon University

10:45–11:00 **Experimental modelling of rock dumping through vertical and inclined pipes**
Otto Neshamar, Alan Cuthbertson, Øyvind Thiem, Peter Davies
School of Science and Engineering, University of Dundee

11:00–11:15 **Possible ubiquitous nature of solitary waves in the stable atmospheric boundary layer**
Philip S. Anderson
Scottish Association for Marine Science, Oban

11:15–11:30 **A unifying heat transport model for magnetoconvection**
Matthew McCormack, Andrei Teimurazov, Olga Shishkina, Moritz Linkmann
School of Mathematics, The University of Edinburgh; Max Planck Institute for Dynamics and Self-Organization, Göttingen

11:30–11:45 **Scattering of surface waves by ocean currents**
Han Wang, Ana B. Villas Bôas, Jacques Vanneste, William R. Young
School of Mathematics, The University of Edinburgh; Department of Geophysics, Colorado School of Mines; Scripps Institution of Oceanography, University of California at San Diego

11:45–12:00 **The energetics of pilot-wave hydrodynamics**
Matthew Durey, John W. M. Bush
School of Mathematics and Statistics, University of Glasgow

12:00–12:10 **Group photo**

12:10–13:30 **Lunch and Posters**

13:30–15:00 **Session 2**

13:30–13:45 **Post-impact gas-cushioning in liquid-solid impacts**
Peter Hicks - *School of Engineering, University of Aberdeen*

13:45–14:00 **Effect of substrate wettability on the heat transfer coefficient in pool boiling**
Giada Minozzi, Alessio D. Lavino, Edward R. Smith, Tassos Karayiannis, Khellil Sefiane, Omar K. Matar, David Scott, Timm Krüger, Prashant Valluri
School of Engineering, The University of Edinburgh; Department of Chemical Engineering, Imperial College London; Department of Aerospace and Mechanical Engineering, Brunel University London

14:00–14:15 **Simulating collective bacterial swarming in sparse systems**
Francois de Tournemire, Kristian Thijssen, Gavin Melaugh, Tyler Shendruk
School of Physics and Astronomy, The University of Edinburgh

14:15–14:30 **The Role of Activity Patterning in Active Nematic Flow Transitions**
Alexander J.H. Houston, Nigel J. Mottram
School of Mathematics and Statistics, University of Glasgow

14:30–14:45 **Drug inhalation simulations in patient airways with two-way and four-way coupling**
Josh Williams, Uwe Wolfram, Ali Ozel
Hartree Centre, STFC Daresbury Laboratory; Institute for Material Science and Engineering, TU Clausthal; School of Engineering and Physical Sciences, Heriot-Watt University

14:45–15:00 **Modelling ferrofluid emulsions using a bulk stress approach**
Paolo Capobianchi, Ghulam Sultan, Marcello Lappa, Fernando T. Pinho, Mónica S.N. Oliveira
University of Strathclyde; University of Porto

15:00–15:30 **Coffee and Posters**

15:30–17:15 **Session 3**

15:30–15:45 **Reconstructing ocean's density stratification from surface data**
Anirban Guha
School of Science and Engineering, University of Dundee

15:45–16:00 **A 2D model for the drainage of melt ponds on Arctic sea ice**
Russell Campbell, David Rees Jones, David Dritschel
School of Mathematics and Statistics, University of St Andrews

16:00–16:15 **Simulation of the transient dissolution of a spherical particle using CFD**
Mahsa Hassanpour, Sina Haeri, Stacie Tibos
School of Engineering, The University of Edinburgh; PepsiCo International's R&D center, Leicester

16:15–16:30 **Dynamics of flow around wind-dispersed dandelion-inspired polyamide flyers**
Soumarup Bhattacharyya, Bappa Mitra, Marc Desmulliez, Ignazio Maria Viola
School of Engineering, The University of Edinburgh; School of Engineering & Physical Sciences, Heriot-Watt University

16:30–16:45 **Evidence of seabed currents enhancements in abyssal tropical ocean driven by the surface mesoscale eddies**
Dmitry Aleynik, Andrew Dale
Scottish Association for Marine Science, Oban

16:45–17:00 **The effect of contact line motion on the deposition of particles from an evaporating sessile droplet**
Hannah-May D'Ambrosio, Stephen K. Wilson, Alexander W. Wray
Department of Mathematics and Statistics, University of Strathclyde

17:00–17:15 **Dropwise Condensation on Silicone Oil Grafted Surfaces**
Anam Abbas, Gary G. Wells, Glen McHale, Khellil Sefiane, Daniel Orejon
School of Engineering, The University of Edinburgh; Department of Mechanical Engineering, University of Engineering and Technology, Lahore; International Institute for Carbon-Neutral Energy Research (WPI-I2CNER), Kyushu University

17:15–17:20 **Closing remarks**

Posters

1. **Flow and Jamming of Granular Material in TPMS structures**
   Aashish Kumar Gupta, Sina Haeri
   School of Engineering, The University of Edinburgh
2. **Development and validation of a cavitation erosion model**
   Adriano Evangelisti, Giuliano Agati, Domenico Borello, Paolo Venturini, Paolo Capobianchi
   Sapienza University of Rome; University of Strathclyde
3. **Investigation of three-dimensional effects of an isolated barnacle on a tidal turbine blade secion**
   Alexander McIntosh, Lada Murdoch, Richard Green, Angela Busse
   James Watt School of Engineering, University of Glasgow
4. **Numerical Modelling of Nanofiltration Membranes Using Polyethylene Glycol Solutions**
   Aniol Puigdefabregas Nogueras, E. H. Papaioannou, A. Kazakidi
   University of Strathclyde; Lancaster University
5. **Numerical investigation of pulsating flow around a circular cylinder**
   Brains Vasco, Oleksandr Zhdanov, Angela Busse
   James Watt School of Engineering, University of Glasgow
6. **Evaporation of droplets on porous substrates**
   David Craig, Stephen K. Wilson, Alexander W. Wray, Khellil Sefiane
   Department of Mathematics and Statistics , University of Strathclyde; School of Engineering , The University of Edinburgh
7. **Evaporation-induced translation of Multiple Binary Droplets**
   Debarshi Debnath, Anna Malachtari, Geroge Karapetsas, Daniel Orejon, Khellil Sefiane, Alidad Amirfazli, Prashant Valluri
   School of Engineering, The University of Edinburgh; Department of Chemical Engineering, Aristotle University of Thessaloniki; Department of Mechanical Engineering, York University, Toronto
8. **On the flow around 3D porous disks at incidence**
   Doudou Huang, Chandan Bose, Ignazio Maria Viola
   School of Engineering, The University of Edinburgh; College of Engineering and Physical Sciences, University of Birmingham
9. **Advanced Thermophysical Properties Determination within CCUS Operations: Thermodynamic Modelling for Phase Equilibrium in CO2-rich mixtures**
   Elahe Rostaminikoo, Edris Joonaki, Hamid Reza Nasriani
   School of Engineering and Computing, University of Central Lancashire; TÜV SÜD UK National Engineering Laboratory
10. **Stress dependent permeability of granular polymer material as a temporary plug in subsurface reservoirs**
    Feng Zhao, Yukie Tanino, Jianchun Guo, Amer Syed
    School of Oil & Natural Gas Engineering, Southwest Petroleum University, Chengdu; School of Engineering, University of Aberdeen; School of Engineering, The University of Edinburgh
11. **Ferrohydrodynamics of Ferrofluid-gaps for Direct Drive Wind Turbine Generators**
    Fergus Hall, Alasdair McDonald, Markus Mueller
    School of Engineering, The University of Edinburgh
12. **Numerical experiments to assess the dynamics of sediment particle entrainment**
    Gaston Latessa, Angela Busse, Maggie Creed, Jin Sun
    James Watt School of Engineering, University of Glasgow
13. **Generation of quasi-2D isolated spanwise vortex gusts**
    Geethanjali Pavar, T. Bruce, B. Peterson, A. McDonald
    School of Engineering, The University of Edinburgh
14. **Machine learning based PAH modelling for LES of turbulent non-premixed flames**
    Geveen Arumapperuma, Oliver Bladek, Antonio Attili
    School of Engineering, The University of Edinburgh

15. **Non-porous viscous fingering of a gravity-driven freesurface flow**
Haolin Yang, Nigel Mottram, Katarzyna Kowal
School of Mathematics and Statistics, University of Glasgow

16. **Evolution of an Evaporating Sessile Droplet According to the One-Sided Model**
Henry T. Sharp, Stephen K. Wilson, Alexander W. Wray
Department of Mathematics and Statistics, University of Strathclyde

17. **A single-phase Level-Set method for unsteady free surface flows: geometric-based reinitialisation and physically-consistent dynamic boundary conditions**
Jorge Sandoval, Cristián Escauriaza, David Ingram
School of Engineering, The University of Edinburgh

18. **Experimental investigation of flow boiling performance of microchannel with different hydraulic diameter and aspect ratio**
Luwen Qin, Arif Widyatama, Julio Cesar Passos, Khellil Sefiane, Daniel Orejon
School of Engineering, The University of Edinburgh; Department of Mechanical Engineering, Federal University of Santa Catarina, Florianopolis, SC, Brazil

19. **Thermodiffusive Instabilities in Ammonia/Hydrogen Laminar Flames**
Markos-Gerasimos Viagkinis, Sofiane Al-Kassar, Antonio Attili
School of Engineering, The University of Edinburgh

20. **Ocean wave-induced microplastic transport in the presence of the Basset-Boussinesq history force**
Mary Eby, Cathal Cummins
Department of Mathematics, Heriot-Watt University; School of Energy, Geoscience, Infrastructure and Society Heriot-Watt University

21. **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

22. **The Influence of Inlet, Headland, and Tidal Current Relative Positioning on the Formations of Ebb Tidal Jet and Eddy at Tidal Inlets**
Munawir Pratama, Vengatesan Venugopal
School of Engineering, The University of Edinburgh

23. **Numerical turbulent viscosity estimation**
Nguyen Q. Chien
School of Engineering, The University of Edinburgh

24. **Surface Vibration Effects on Nanoscale Ice Nucleation**
Pengxu Chen, Rohit Pillai, Saikat Datta
School of Engineering, The University of Edinburgh

25. **Experimental Studies on the Hydraulic Behaviour of Nigerian Soils for Efficient Water Management**
Peter Uloho Osame, Taimoor Asim, Sheikh Zahidul Islam, Dallia Ali
School of Engineering, Robert Gordon University

26. **Numerical Simulation of Inertial Particle Motion in Heterogeneous Suspensions**
Qi (Charles) Zhou, Benjamin Owen, Timm Krüger
School of Engineering, The University of Edinburgh

27. **Nucleate pool boiling of carbon dioxide on a vertical flat copper surface in confined spaces**
Rafael B. A. Passarella, Arthur K. A. Araújo, Adonis Menezes, Júlio C. Passos
Department of Mechanical Engineering, Federal University of Santa Catarina, Florianópolis, SC, Brazil

28. **Multiphase Flow Measurement of Wet Gas Flow Using Machine Learning Modelling Algorithms**
Seyedahmad Hosseini, Gabriele Chinello, Gordon Lindsay, Sheila Smith, Don McGlinchey
Department of Mechanical Engineering, Glasgow Caledonian University; TÜV SÜD UK National Engineering Laboratory; Department of Applied Science, Glasgow Caledonian University

29. **Molecular kinetic modelling of non-equilibrium evaporative flows**
Shaokang Li, Livio Gibelli, Yonghao Zhang
School of Engineering, The University of Edinburgh; Centre for Interdisciplinary Research in Fluids, Institute of Mechanics, Chinese Academy of Sciences, Beijing

30. **Two-phase blood flow downstream of a 3D bifurcation**
Tanchanok Wisitponchai, Junxi Wu, Asimina Kazakidi
Department of Biomedical Engineering, University of Strathclyde

31. **Mathematical modelling of ice-sheet dynamics**
    Tanisha Kumari, Matthew Durey, Peter Stewart, Katarzyna Kowal
    School of Mathematics and Statistics, University of Glasgow

32. **3D roughness effects on the aerodynamics of the outboard section of a wind turbine blade**
    Wasina Preamsakul, Oleksandr Zhdanov, Angela Busse
    James Watt School of Engineering, University of Glasgow

33. **Turbulent Hydrogen Flames at High Pressure**
    William Lauder, Sofiane Al-Kassar, Geveen Arumapperuma, Antonio Attili
    School of Engineering, The University of Edinburgh

34. **The interaction of waves with suspended seaweeds: wave attenuation and in-canopy hydrodynamics**
    Xinyi Zhang, Dominic van der A, Tom O'Donoghue
    School of Engineering, University of Aberdeen

35. **Decoupling Heat Transfer Mechanisms of Micrometre Droplets during Steady State Phase-Change**
    Xuecong Wang, Nenad Miljkovic, Daniel Orejon
    School of Engineering, The University of Edinburgh

36. **Exploring Critical size and temperature of size effect: Insights into Adsorption and Diffusion in Nanometer-Scale Confinement**
    Xueling Zhang, Qiang Ye, Yingfang Zhou
    School of Energy and Power Engineering, Zhengzhou University of Light Industry; School of Engineering, University of Aberdeen

37. **Fluid-Solid Flow Transition in Mixed (Sand-Mud) Sediment: Enhanced Modelling of Sedimentation in Estuarine and Coastal Waters**
    Yi Yuan, Alan Cuthbertson, Tom Eaves
    School of Science and Engineering, University of Dundee

38. **Active fluid-induced dynamics of passive polymers**
    Zahra Valei Tyler N. Shendruk
    School of Physics and Astronomy, The University of Edinburgh

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

The 37th Scottish Fluid Mechanics Meeting will be held in the new Nucleus Building in the King's Buildings (KB), the main campus of the College of Science and Engineering of The University of Edinburgh.

Edinburgh Airport

Edinburgh Airport is situated to the west of Edinburgh, around 12 miles from the School of Engineering.

- Edinburgh airport details
  www.edinburghairport.com

Edinburgh Train Stations

Edinburgh Waverley (2.3 miles) and Haymarket (3.1 miles) train stations are the closest to the School of Engineering

- National Rail Edinburgh train station details
  https://www.nationalrail.co.uk/stations/edinburgh/

- ScotRail Edinburgh train station and travel information
  https://www.scotrail.co.uk/plan-your-journey/stations-and-facilities/edb

Edinburgh Bus, Tram and Coach Travel

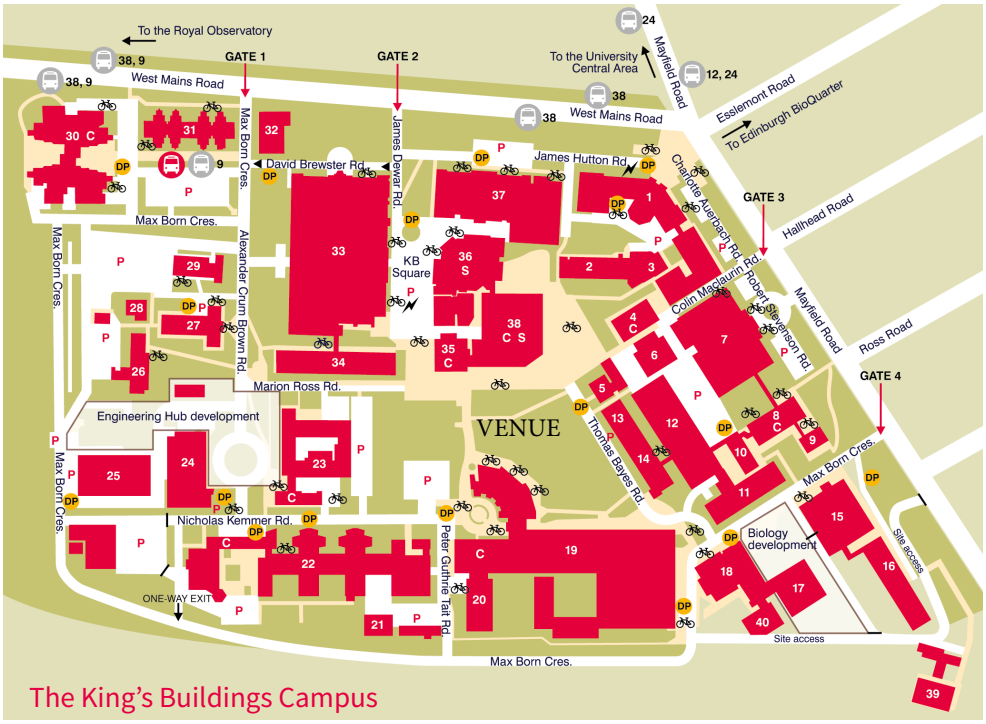
Edinburgh has an excellent bus network, operated by Lothian Buses and Edinburgh Trams.

- Edinburgh Council Public Transport
  https://www.edinburgh.gov.uk/public-transport

- Lothian Buses
  https://www.lothianbuses.com

- Edinburgh Trams
  https://edinburghtrams.com

Map of the King's Buildings campus



14 Alexander Graham Bell Building
11 Alrick Building
32 Arcadia Nursery
1 Ashworth Building
34 Christina Miller Building
29 Crew Annex
27 Crew Building
26 Crew Laboratory
15 Daniel Rutherford Building
17 Darwin Building
6 Engineering Structures Lab
20 Erskine Williamson Building
21 Estates Hub
10 Faraday Building
12 Fleeming Jenkin Building
25 FloWave Ocean Energy Research Facility
39 Glasshouse
37 Grant Institute
8 Hudson Beare Building
9 Hudson Beare Lecture Theatre
19 James Clerk Maxwell Building; Learning and Teaching Cluster
5 John Muir Building
2 John Murray Building
33 Joseph Black Building
36 KB House: EUSA
3 March Building
4 Mary Brück Building
40 MEP Building
30 Murchison House
38 The Nucleus
35 The Noreen and Kenneth Murray Library
22 Peter Wilson Building (SRUC)
23 Roger Land Building
7 Sanderson Building
24 Scottish Microelectronics Centre
31 Student accommodation
18 Swann Building
28 UK Biochar Research Centre
16 Waddington Building
13 William Rankine Building

Bike racks
C Cafe
DP Disabled permit parking
Electric car charging point
Pedestrian area
P Permit parking
Public bus
S Shop
Shuttle bus to the Central Area
Traffic barrier

The timetable for the shuttle bus between the Central Area and the King's Buildings can be viewed at: **www.ed.ac.uk/shuttle-bus**

Nucleus Building, King's Buildings campus

Abstracts

37^{th} Scottish Fluid Mechanics Meeting

Flow and Jamming of Granular Material in TPMS structures

Aashish Kumar Gupta, Sina Haeri
School of Engineering (IMP), University of Edinburgh
King's Buildings, Edinburgh, EH9 3FB, United Kingdom

27^{th} May 2024

Abstract

Triply Periodic Minimal Surface (TPMS) scaffolds are being explored for revolutionary applications in bone-tissue regeneration, lightweight impact-resistant structures, and compact heat exchangers for miniaturized electronics. Due to the complex topology and intricate internal pores, conventional cutting or milling methods (aka subtractive manufacturing) cannot be applied to fabricate TPMS structures, and additive manufacturing (AM) is necessary. However, evacuating the powder trapped inside the cavities of the final build is very challenging after the selective melting stage is completed in a metal-powder-based AM process. The Discrete Element Method (DEM) [1] simulations can offer valuable insights into the flowing-jamming transition in these structures allowing us to tailor effective de-powdering strategies. In this study, the natural flow (under the influence of gravity) of trapped powders inside a unit cell of the commonly used TPMS structures is simulated using DEM. A computationally efficient protocol to create a granular packing akin to the actual AM process is first developed by utilizing the framework of Lees-Edwards shear.[2] A subset of the particles is then frozen, per the mathematical equation representing the desired TPMS structure, to replicate the embedded shell. The kinematics and the force-transmission are then studied as the structure empties. A drastic slowing down of particles about halfway through the evacuation is observed, indicating a transient arching-like phenomenon posited earlier for granular discharge through relatively narrow orifices. The corresponding signatures are also present in the underlying force-chains. The hypothesis of arching is further corroborated by the statistics of the particle-level contact-force at different stages in the simulation.

References

[1] Cundall, P. A. and Strack, O. D., *geotechnique*, Vol. 29, No. 1, 1979, pp. 47–65.

[2] Berry, N., Zhang, Y., and Haeri, S., *Powder Technology*, Vol. 389, 2021, pp. 292–308.

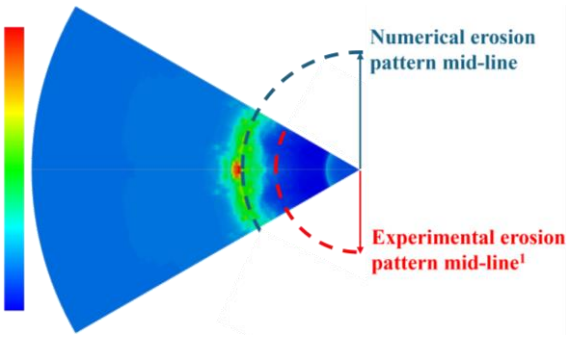
37th Scottish Fluid Mechanics Meeting
Development and validation of a cavitation erosion model

Adriano Evangelisti[1,2], Giuliano Agati[1], Domenico Borello[1], Paolo Venturini[1], Paolo Capobianchi[2]
[1]Sapienza University of Rome, Italy
[2]University of Strathclyde, Glasgow, UK

27th May 2024

Abstract

Cavitation erosion is a major issue in the working life of hydraulic turbomachines. Vapour formation and bubbles' implosions can lead to the damage of the solid boundaries, provoking performance losses and fatigue failure over time. This paper aims at validating a new approach of cavitation erosion prediction, reproducing a circular sector of the stainless-steel nozzle used in the experiment of J. P. Franc[1]. The Singhal et al. Full Cavitation Model[2], based on the bubble dynamics treatment, is implemented in conjunction with the *k-ω* SST turbulence model within Ansys Fluent. Bubbles detached from the cavitation cloud collapse close to the solid surfaces, whenever the static pressure of the target plate is higher than the vapour pressure. In the ultimate life stage of an imploding bubble, a microjet is formed; subsequently the jet may impact on the solid boundaries of a working machine. If the microjet velocity overcomes the material critical velocity, a function of the material yield strength, a permanent deformation finally occurs. The developed cavitation erosion model accounts for the number of implosions in the unit volume, giving different importance to the erosive mechanisms, according to the amount of vapour collapsed in the target plate cells. Time averaged results of the numerical simulations are compared against the nozzle experimental results, displaying the cavitation erosion mid-line annular locations.



Ongoing works are dedicated to test mesh sensitivity and new cavitation models.

References

[1] J.-P. Franc, "Incubation Time and Cavitation Erosion Rate of Work-Hardening Materials," *J Fluids Eng*, vol. 131, No. 2, 2009, pp. 021303 1-14.
[2] A. K. Singhal, M. M. Athavale, H. Li, and Y. Jiang, "Mathematical Basis and Validation of the Full Cavitation Model," *J Fluids Eng*, vol. 124, No. 3, 2002, pp. 617-624.

37^{th} Scottish Fluid Mechanics Meeting

The Role of Activity Patterning in Active Nematic Flow Transitions

Alexander J.H. Houston, Nigel J. Mottram
School of Mathematics and Statistics, University of Glasgow
University Place, Glasgow, G12 8QQ

27^{th} May 2024

Abstract

Active nematics model a wide range of living systems, including cells layers and bacteria.[1] However, real biological systems will not have uniform activity, either due to population variance or due to the presence of distinct species or cell types. As well as arising naturally, it has been recently demonstrated that the structure of activity in a material can be controlled through modulating light intensity.[2] This provides motivation to understand the effects of activity patterning, both to gain insight into the in vivo behaviour of biological systems and to open up routes to engineer desired dynamics in active matter. One of the hallmarks of active nematics is that, when confined to a channel, they will transition to a flowing state once above a threshold activity,[3] and it is in this context that we study the role of activity patterning. The system is described by an operator that is non-Hermitian and non-local, both features that complicate the analysis. We provide a solution for an arbitrary piecewise-constant activity profile. From this we show that patterning allows a significant reduction in the net activity required for a flow transition and also makes it possible to control the structure of the flowing state.

References

[1] Doostmohammadi, A., Ignés-Mullol, J., Yeomans, J. M., and Sagués, F., "Active nematics," *Nature communications*, Vol. 9, No. 1, 2018, pp. 3246.

[2] Zhang, R., Redford, S. A., Ruijgrok, P. V., Kumar, N., Mozaffari, A., Zemsky, S., Dinner, A. R., Vitelli, V., Bryant, Z., Gardel, M. L., et al., "Spatiotemporal control of liquid crystal structure and dynamics through activity patterning," *Nature materials*, Vol. 20, No. 6, 2021, pp. 875–882.

[3] Voituriez, R., Joanny, J.-F., and Prost, J., "Spontaneous flow transition in active polar gels," *Europhysics Letters*, Vol. 70, No. 3, 2005, pp. 404.

37^{th} Scottish Fluid Mechanics Meeting

Investigation of three-dimensional effects of an isolated barnacle on a tidal turbine blade section

Alexander McIntosh, Lada Murdoch, Richard Green, Angela Busse
James Watt School of Engineering, University of Glasgow
University Avenue, Glasgow, G12 8QQ

29^{th} May 2024

Abstract

Biofouling is an issue known to degrade the performance of marine current turbines. Due to their operating environment, turbine blades are susceptible to large scale macrofouling, with barnacles identified as the main culprit in UK waters.[1] In the present study, the effect of barnacle-biofouling on the performance on a NACA63-619 turbine blade section is investigated using Reynolds-Averaged Navier-Stokes (RANS) simulations using the software package STAR-CCM+. The standard approach in RANS investigations of flow past aerofoils or hydrofoils is to simulate the problem in two dimensions. However, roughness caused by barnacle-biofouling is inherently three-dimensional. To investigate the three-dimensional effects of barnacle biofouling, flow past blade sections of increasing spanwise width, each fouled by a single barnacle, is simulated. Both steady conditions, where the angle of attack of the blade is fixed, and unsteady conditions, where the blade section undergoes a pitching motion, are investigated. The results are compared the experiments conducted by Walker et al[2] for validation. Following this, the investigation will be extended to conditions where a larger number of barnacles has settled on the blade section.

References

[1] Vance, T., Ellis, R., and Fileman, T., "ETI MA1001 - Reliable Data Acquisition Platform for Tidal (ReDAPT) project," Tech. Rep. ME 8.5, Plymouth Marine Laboratory, 2014.

[2] Walker, J. S., Green, R. B., Gillies, E. A., and Philips, C., "The effect of a barnacle-shaped excrescence on the hydrodynamic performance of a tidal turbine blade section," *Ocean Engineering*, Vol. 217, 2020, pp. 107849.

37th Scottish Fluid Mechanics Meeting

Dropwise Condensation on Silicone Oil Grafted Surfaces

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

[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 39161, Pakistan

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

22th April 2024

Abstract

Condensation plays a crucial role in numerous industrial applications. It has been proved in literature that dropwise condensation can achieve heat transfer rates significantly higher (6-8 times) than filmwise condensation[1], making it particularly advantageous. Several surface modification methods empowering dropwise condensation have been investigated. One of these methods involve applying low surface energy uniform coatings with minimum contact angle hysteresis (CAH)[2, 3], hence low pinning; which facilitates the shedding of smaller sized droplets. We present here a simple procedure that involves grafting of commercially available silicone oil to impart a hydrophobic coating on an otherwise hydrophilic silicon substrate. By promoting dropwise condensation, the heat transfer rate can also be improved. We make use of different fabrication parameters during the grafting of the silicone oil (oil viscosity and/or volume/method, temperature, etc.) onto smooth silicon substrates henceforth referred to as grafted surfaces.[2] On one hand, we observe dropwise condensation on all grafted surfaces irrespective of the viscosity and/or number of layers of silicone oil grafted. While on the other hand, the radii of droplet shedding from the surface and the time of departure are influenced greatly by the viscosity and/or the number of layers of the oil grafted. The departure of smaller droplets and the higher rate of droplet shedding are noticeable on surfaces grafted with high viscosity oil (100 cSt) as compared to its counterpart grafted with low viscosity oil (5 cSt). This correlation corresponds to the lower contact angle hysteresis (CAH) on high viscosity oil grafted surfaces, leading to reduced droplet pinning and maximized droplet shedding. This research provides insights for selecting grafting parameters to prepare low energy surfaces that facilitate dropwise condensation. Moreover, the manipulation of the fabrication parameters may enable control over the droplet size distribution and their growth.

References

1. Weisensee, P.B., Y. Wang, H. Qian, D. Schultz, W.P. King, and N. Miljkovic, Condensate droplet size distribution on lubricant-infused surfaces, *International Journal of Heat and Mass Transfer*, 109, 2017, pp. 187-199.
2. Abbas, A., G.G. Wells, G. McHale, K. Sefiane, and D. Orejon, Silicone Oil-Grafted Low-Hysteresis Water-Repellent Surfaces, *ACS Applied Materials & Interfaces*, 15(8), 2023, pp. 11281-11295.
3. Armstrong, S., G. McHale, R. Ledesma-Aguilar, and G.G. Wells, Pinning-Free Evaporation of Sessile Droplets of Water from Solid Surfaces, *Langmuir*, 35(8), 2019, pp. 2989-2996.

Numerical Modelling of Nanofiltration Membranes Using Polyethylene Glycol Solutions

A. Puigdefabregas Nogueras[1], E. H. Papaioannou[2], A. Kazakidi[1]

[1]University of Strathclyde, UK. [2]Lancaster University, UK
16 Richmond St, Glasgow, G1 1XQ

Abstract

Polyethylene Glycols (PEG) solutions have found applications in many industries (pharmaceutical, chemical, etc.) [1]. Pressure-driven membrane processes, such as nanofiltration (NF), are prone to concentration polarization (CP). CP occurs when the concentration of solute molecules near the membrane surface increases as the solvent is forced through to the permeate side. This increased concentration results in higher osmotic pressures ($\Delta\pi$) on the retentate side, which causes an additional resistance to the applied transmembrane pressure (TMP) resulting in a flux decline. Computational fluid dynamics (CFD) is an inexpensive tool that can provide insight into the effects of CP. The aim of the current work is to develop a CFD model capable of predicting the performance of a commercially available membrane during the NF of a dilute (1% w/v) PEG aqueous solution.

A flat-sheet NF membrane was evaluated experimentally at a semi-pilot system with crossflow rates in the range of 500-1500ml/min and TMP of 2-7 bar. Commercially available PEG with a molecular weight of ~400 Da (PEG400) was used. A 2-D CFD model was developed using the software COMSOL Multiphysics (Version 6.2.0.290, COMSOL inc.). The dimensions of the model were created following the membrane specifications (area, 0.00225 m^2, channel height, 2.5 mm, membrane thickness 0.25mm). The membrane was modelled using the Brinkman equations for porous media flow while the main channel flow was governed by the Navier-Stokes equations. The transport of diluted species was governed by Fick's Law. The osmotic pressure was approximated as a quadratic polynomial regression fit obtained from the literature [1].

The results from the evaluation in the semi-pilot scale, under the applied TMPs (2 to 6 bar), showed a flux decrease of ~32% of the PEG solution with respect to pure water flux permeability (PWFP), which was mainly attributed to CP. After the PEG400 experiments, a decrease of the initial PWPF (<1%) confirmed the absence of irreversible fouling. The CFD model was able to capture the formation of a CP layer near the membrane surface and showed good agreement with experiments for 2 to 4 bar TMP, whereas it overestimated the flux decline for 5 and 6 bar TMP. This difference can be attributed to the simplicity of the polynomial fit to predict the osmotic pressure contribution at higher PEG concentrations near the membrane surface. Refinement of the osmotic pressure computation is expected to accurately predict flux for all ranges of TMP.

Acknowledgements

The authors thank BBSRC NorthwestBio DTP for the financial support. Dr Papaioannou would like to thank in addition EPSRC (EP/X018660/1) for supporting the experimental implementation.

References

[1] N. P. Money, "Osmotic Pressure of Aqueous Polyethylene Glycols' Relationship between Molecular Weight and Vapor Pressure Deficit," 1989.

37^{th} Scottish Fluid Mechanics Meeting

Reconstructing ocean's density stratification from surface data

Anirban Guha
School of Science and Engineering, University of Dundee
Dundee, DD1 4HN

27^{th} May 2024

Abstract

Oceans are by and large stably stratified, that is, the density of ocean water monotonically increases with depth. Depending upon the strength of stratification, the vertical structure of ocean's density is divided into three major layers: (i) *top* - weakly stratified surface mixed layer, (ii) *middle* - strongly stratified pycnocline, and (iii) *bottom* - weakly stratified abyss. Accurate knowledge of the ocean's density field is crucial for ocean and climate modelling. Ocean's density is a function of both temperature and salinity, both of which are measured using CTD (Conductivity, Temperature and Depth) sensors. These sensors, while descending (or ascending) through the ocean water, collect the necessary information. The vertical profiles of temperature and salinity thus obtained are then substituted into the equation of state to yield ocean's density profile at a given latitude–longitude. However, there are no indirect or non-invasive techniques that can estimate oceanic density profile.

In this talk, I will discuss our proposed inverse technique[1] that can provide a reasonably accurate estimate of the ocean's density stratification profile *only* by analyzing the ocean free surface. To achieve this, we scrutinize one of the most important consequences of stable density stratification – the internal gravity waves (IGWs), and specifically, the internal tides. Internal tides are a variety of IGWs, nearly omnipresent in global oceans, which are produced when the stably stratified ocean water is driven back and forth over submarine topography by tidal currents. The pivotal point of our research is the realization that the ocean surface signature of internal tides carries the information of the ocean's density stratification, and can in principle be inverted to reconstruct the latter.

Presentation type: Oral.

References

[1] Kar, S. and Guha, A., "An inverse technique for reconstructing ocean's density stratification from surface data," *Ocean Model.*, Vol. 147, 2020, pp. 101561.

37^{th} Scottish Fluid Mechanics Meeting

Numerical investigation of pulsating flow around a circular cylinder

Brains Vasco, Oleksandr Zhdanov, Angela Busse
James Watt School of Engineering, University of Glasgow
University Avenue, Glasgow, G12 8QQ

29^{th} May 2024

Abstract

This study presents a two-dimensional numerical investigation of a circular cylinder exposed to a pulsating freestream composed of a finite mean velocity with a superimposed sinusoidal fluctuating component $U_\infty(t) = \overline{U}_\infty + \tilde{U}_\infty \sin(2\pi f_s t)$. The flow is in the laminar regime at a Reynolds number of 300 based on the mean freestream velocity $\overline{U}_\infty$ and the cylinder diameter. The focus of this research is to explore the influence of the pulsation frequency (f_s) and amplitude ($\tilde{U}_\infty$) on the wake behaviour and fluid dynamic forces exerted on the cylinder. The systematic variation of these parameters allows for a detailed investigation into how pulsation characteristics impact the vortex shedding patterns[1] and the resultant lift and drag coefficients. In the next stage, this investigation will be extended to three dimensions and the Reynolds number dependency will be considered.

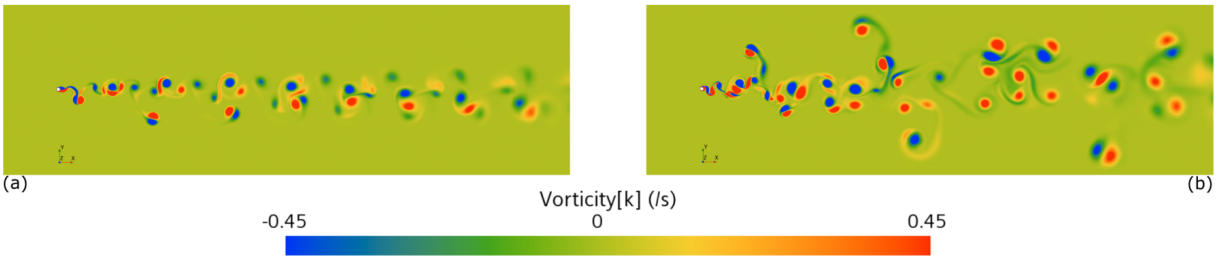


Figure 1: Instantaneous vorticity field generated by a pulsation frequency equal to half the shedding frequency that would emerge for uniform freestream condition ($f_s = 0.5 f_{sh}$), for two different pulsation amplitudes: (a) $\tilde{U}_\infty/\overline{U}_\infty = 0.5$; (b) $\tilde{U}_\infty/\overline{U}_\infty = 1.5$

References

[1] Detemple-Laake, E. and Eckelmann, H., "Phenomenology of Kármán vortex streets in oscillatory flow," *Experiments in Fluids*, Vol. 7, No. 4, 1989, pp. 217–227.

37th Scottish Fluid Mechanics Meeting

CFD Simulation of Fast Fluidized Bed Carbonator for CO_2 Capture Process using the Eulerian-Eulerian Model.

Darlington Amadi, Aditya Karnik, Mamdud Hossain

School of Engineering, Robert Gordon University, Aberdeen, United Kingdom.

27th May 2024

Abstract

Global warming is a major issue affecting the environment by raising the average temperature of the earth's surface. CO_2 is a greenhouse gas released by human activity and the industrial burning of fossil fuels. Many technologies have been employed to mitigate the amount of CO_2 emitted into the atmosphere. One of the effective methods is the use of a fluidized bed reactor (carbonator) system with a dry granular sorbent to remove CO_2 from industrial flue gases. In this work, the performance of Potassium Carbonate (K_2CO_3) dry sorbent throughout the CO_2 capture process is evaluated using a 2D CFD simulation of the carbonator. A chemical reaction of the flue gases and Potassium Carbonate (K_2CO_3) fluidized bed is incorporated into the Eulerian-Eulerian model to account for chemical kinetics via the Arrhenius equation in the species transport. Fluidized bed particle motions were modelled using the kinetic theory of granular flow (KTGF). The K-epsilon turbulence model is applied to enhance the interaction between the flue gases and Potassium Carbonate (K_2CO_3) in phase interactions and chemical reactions between the species. The effects of clusters in the carbonator, the non-uniform distribution of the solid-phase volume fractions and the effects of flue gas velocity on the CO_2 removal will be monitored in the carbonator by the commercial software ANSYS FLUENT. The outcome of this work will help inform future carbonator designs.

References

[1]. Yi, C.K., Jo, S.H., Seo, Y., Lee, J.B. and Ryu, C.K., 2007. Continuous operation of the potassium-based dry sorbent CO2 capture process with two fluidized-bed reactors. *International Journal of Greenhouse Gas Control*, *1*(1), pp.31-36.

[2]. Yu, G., Dai, B., Chen, J., Liu, D., Yu, X., Shao, J. and Zhao, L., 2014. CFD Modeling of a Carbonation Reactor with a K-Based Dry Sorbent for CO2 Capture. *Chemical Engineering & Technology*, *37*(10), pp.1705-1712.

[3]. Nouri, M., Rahpaima, G., Nejad, M.M. and Imani, M., 2018. Computational simulation of CO2 capture process in a fluidized-bed reactor. *Computers & Chemical Engineering*, *108*, pp.1-10.

[4]. Benjaprakairat, T., Piumsomboon, P. and Chalermsinsuwan, B., 2020. Development of computational fluid dynamics model for two initial CO2 concentration in circulating fluidized bed reactor. *Energy Reports*, *6*, pp.137-145.

37^{th} Scottish Fluid Mechanics Meeting

Evaporation of droplets on porous substrates

David Craig[1], Stephen K. Wilson[1], Alexander W. Wray[1], Khellil Sefiane[2]
[1] Department of Mathematics and Statistics , University of Strathclyde,
26 Richmond St, Glasgow
[2] School of Engineering , University of Edinburgh
Robert Stevenson Rd, Edinburgh

27^{th} May 2024

Abstract

The vast majority of the previous work on the evaporation of sessile droplets has focused on the situation in which the substrate is solid. However, in many applications, such as printing onto paper and fabric textiles, the substrate is porous, and the imbibition of liquid into the substrate may play a significant role in the evolution and lifetime of and the deposition from a particle-laden droplet.[1] We extend recent work on droplet evaporation on a solid substrate[2] and develop analytical models, based on lubrication theory, for the evolution of and deposition from a thin particle-laden droplet situated on an initially dry porous substrate. The evaporation from the droplet is driven by the diffusion of liquid molecules into the passive atmosphere.[2] In contrast, the imbibition from the base of the droplet is driven by capillary pressures within the porous substrate. We use Darcy's law to model the flow of liquid within the substrate. We analyse the evolution, and hence the lifetime, of such droplets in a variety of modes of evaporation, specifically the constant angle, constant radius, stick–slide, and stick–jump modes. While, as expected, the addition of imbibition leads to a decrease in the lifetime of the droplet, the dynamics of a droplet undergoing evaporation and imbibition are qualitatively different from those of a droplet undergoing pure evaporation. Furthermore, we demonstrate that in the regime in which diffusion is faster than axial advection, but slower than radial advection, the coffee-ring effect is enhanced by imbibition.

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.

[2] D'Ambrosio, H.-M., Wilson, S. K., Wray, A. W., and Duffy, B. R., "The effect of the spatial variation of the evaporative flux on the deposition from a thin sessile droplet," *Journal of Fluid Mechanics*, Vol. 970, 2023, pp. A1.

37^{th} Scottish Fluid Mechanics Meeting

Evaporation-induced translation of Multiple Binary Droplets

Debarshi Debnath[1], Anna Malachtari[2], Geroge Karapetsas[2], Daniel Orejon[1], Khellil Sefiane[1], Alidad Amirfazli[3], and Prashant Valluri[1]
[1] School of Engineering, University of Edinburgh, Edinburgh EH9 3FB, UK
[2] Department of Chemical Engineering, Aristotle University of Thessaloniki
[3] Department of Mechanical Engineering, York University, Toronto

27^{th} May 2024

Abstract

The evaporation of multiple sessile droplets is a complex physical scenario due to their dynamic interaction through the vapour phase.[1] In situations when such evaporating droplets comprise a binary mixture, their interaction becomes more complex due to the inclusion of solutal Marangoni.[2] The physical understanding of such instances is extremely important for many crucial applications ranging from inkjet printing and spray cooling to disease diagnosis and DNA chip manufacturing. We propose a lubrication theory-based finite element model for two evaporating sessile drops comprising a binary mixture on a heated surface. We consider a diffusion-limited evaporation regime for thin droplets in the presence of a precursor film. The liquid components are considered to be ideally mixed. The net surface tension is taken as a linear function of both temperature and concentration. Further, we also solve the vapour concentration for both the volatile components in the presence of air in 2D using the standard diffusion equation. Our findings demonstrate that thermal and solutal Marangoni can cause attraction, coalescence (1b), or repulsion (1a) between adjacent droplets with identical binary mixture.

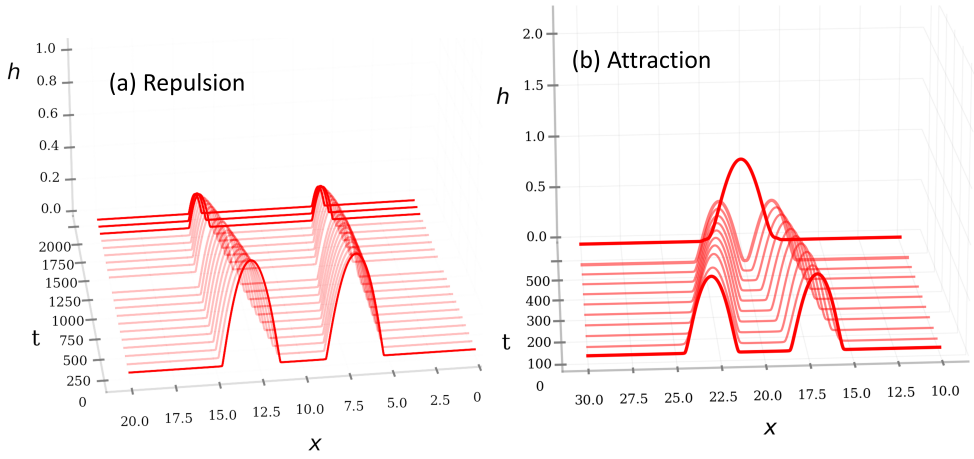


Figure 1: Space-time plots of (a) repulsion and (b) attraction of two binary drops.

References

[1] 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.

[2] Williams, A. G. L., Karapetsas, G., Mamalis, D., Sefiane, K., Matar, O. K., and Valluri, P., "Spreading and retraction dynamics of sessile evaporating droplets comprising volatile binary mixtures," *Journal of Fluid Mechanics*, Vol. 907, Jan. 2021, pp. A22, Publisher: Cambridge University Press.

37th Scottish Fluid Mechanics Meeting

Evidence of seabed currents enhancements in abyssal tropical ocean driven by the surface mesoscale eddies

Dmitry Aleynik*, Andrew Dale

Scottish Association for Marine Science (SAMS),
Oban, Argyll, PA37 1QA, United Kingdom

29th May 2024

Abstract

Recent measurements in abyssal planes between Clarion and Clipperton transform faults in the Eastern Tropical Pacific reveal the presence of aperiodic enhancement of currents affecting natural and expected artificial plumes of suspended sediments. Weakly stratified low-energetic deep-sea occasionally become more turbulent, while the suppressing turbulence vertical density gradient and buoyancy frequencies remain weak (N^2~10^{-7} rad s^{-1}). Records from two- and one-year-long Acoustic Doppler profilers deployed in a water column 4.2 km below the surface at 16 and 8, 92, 379, 667 and 954 m above seabed in two separate experiments (in 2013-15[1] and 2023-24, SMARTEX project) demonstrate that currents at all depths become an order of magnitude more energetic during a few weeks in a response to the westward passage of significant (0.4 - 0.8 m) Sea-level Anomalies identified as mesoscale eddies (Fig. 1). Satellite altimetry, Argo floats and Vessel Mounted ADCPs enable evaluation of eddies 3D parameter scales: mixed layer deepened to 100 m, the horizontal radius was ~100 miles, maximum orbital velocity exceeded 0.5 m·s^{-1} and average westward transport was 0.11-0.14 m·s^{-1}. Analysis of the time scales, magnitudes and consistent D-shape veering of seabed currents revealed a statistically significant coherence with the amplitude and direction of strong surface currents during eddies propagation over the mooring sites. The identified mostly clockwise rotating eddies (anticyclonic) were traced back to their origin at the meanders of the poleward Costa Rica Coastal Current under the Central American Gap Winds. Eddies became vertically tilted due to bottom friction either on a narrow and steep continental slope or during their drag over the East Pacific Rise and chain of Mathematical Seamounts (110°W, 12-14°N). After eddy separation from CCRC and stirring by strong curling wind stress beneath certain hurricanes, eddies became capable of transporting huge volumes of trapped warmer coastal waters westward above tall obstacles (Guyots) and even trespassing over high sea mountains if their slope Froud number F_{ss} exceeds 0.5 [2].

The filled gap in scarce *in-situ* observations increases the ability to predict the time, location, and scales of aperiodic excitation of abyssal currents 3-4 times above the background tidal flow (4 cm·s^{-1}), and thus numerically evaluate both the positive (wide larvae spreading speeding up

References

[1] Aleynik, D., Inall, M.E., Dale, A. and A. Vink. Impact of remotely generated eddies on plume dispersion at abyssal mining sites in the Pacific. *Scientific Reports*, 7, 16959. 2017. https://www.nature.com/articles/s41598-017-16912-2

[2] Here $F_s=U\cdot(L\cdot N\cdot\sin\alpha)^{-1}$, where U, and L are currents and topographic scales, N is buoyancy, and α is the tilt or slope angle.

post-impact recovery) and negative (wider blanketing of vulnerable habitats) burden of suspended sediment plumes. Pending UN International Seabed Authority regulations required such a knowledge-based development of the effective mitigation measures that aimed to mediate or offset the negative impacts, i.e., by slowing down or halting disturbance or harvesting operations near the seabed during the passage of mesoscale eddies, that could be detected in advance.

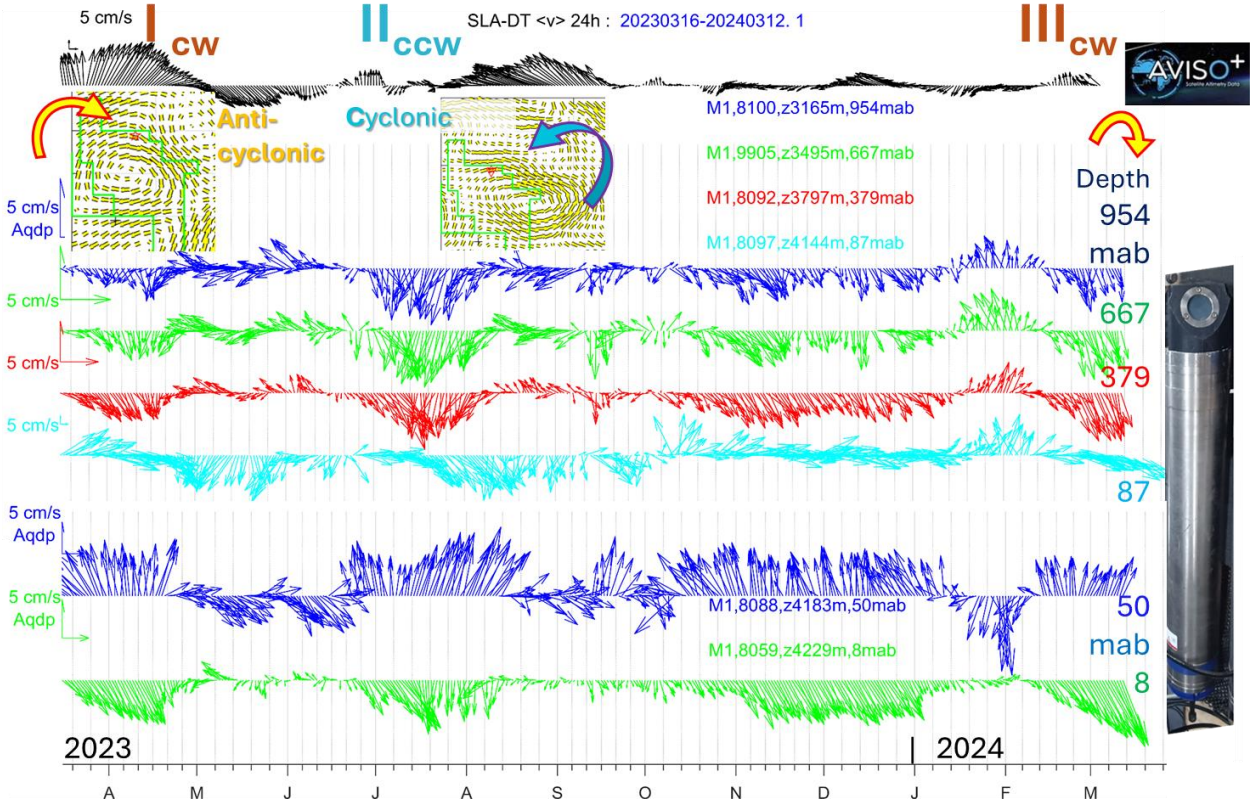


Figure 1. Currents at the sea surface (top, derived from Sea Level Anomalies, AVISO) and measured by AquaDopp 2mHz de-tided currents in the water column at depths from 8 to 954 m above the seabed (4.2 km) and recovered from the mooring 1 at UK1 CCZ in JC257 NERC-SMARTEX research cruise, note that scales vary at each depth.

37^{th} Scottish Fluid Mechanics Meeting

On the flow around 3D porous disks at incidence

Doudou Huang[1], Chandan Bose[2], and Ignazio Maria Viola[1]

[1]School of Engineering, Institute for Energy Systems, University of Edinburgh.

[2]Aerospace Engineering, College of Engineering and Physical Sciences, University of Birmingham.

29^{th} May 2024

Abstract

We numerically investigate the flow around the three-dimensional (3D) porous disks at different incidence angles. Computational Fluid Dynamics (CFD) simulations are performed to study the effects of inclination on the aerodynamics of the porous disks. The simulations are conducted using the open-source CFD package OpenFOAM, with Reynolds number $Re = 30$ and 100 and non-dimensional permeability Darcy number (Da) from 10^{-6} to 10^{-3}. The incompressible Navier-Stokes equations are calculated for the clear fluid, while the Darcy-Brinkman equations are solved for the fluid within the porous media, using a self-developed numerical porousIcoFoam solver. The Arbitrary Mesh Interface is employed to achieve a high quality mesh at every incidence angle. The results revealed that both the permeability and the angle of inclination have a great impact on the wake topology: at $Da = 10^{-3}$, no recirculation regions are observed in the wake, while the number of recirculation regions vary from 0 to 2 with increasing inclination angle α when $Da \leq 10^{-4}$. Subsequently, forces acting on the porous disks are evaluated. For both Re, the drag shows a monotonic increasing trend as α increases. Also, $Da = 10^{-3}$ exhibits a lower value of drag than all other permeabilities for all α, whereas $Da = 10^{-4}$ experiences the highest value of drag when $\alpha > 60°$. These results can inform the design and control of bio-inspired micro-fliers in terms of achieving minimal terminal velocity and maximal floating time.

37th Scottish Fluid Mechanics Meeting

Advanced Thermophysical Properties Determination within CCUS Operations: Thermodynamic Modelling for Phase Equilibrium in CO_2-rich mixtures

Elahe Rostaminikoo[1], Edris Joonaki[2], Hamid Reza Nasriani[1]

[1] *School of Engineering and Computing, University of Central Lancashire, Preston, PR1 2HE, United Kingdom*

[2] *TÜV SÜD UK National Engineering Laboratory (UK NEL), East Kilbride, Glasgow, G75 0QF, United Kingdom*

27th May 2024

Abstract

In the dynamic field of environmental engineering, namely in the domains of Carbon Capture, Utilisation, and Storage (CCUS), accurate modelling of phase equilibrium has become essential for the planning and implementation stages of these systems. It is crucial to have a predictive understanding of phase behaviour and the thermophysical properties of mixtures with high levels of CO_2 in the domain of CCS transportation. These mixtures are distinguished by the presence of minor amounts of contaminants, such as sulphur dioxide, nitrogen, nitric oxide, hydrogen, oxygen, various alkanes, hydrogen sulphide, argon, ammonia, amines, and water. To achieve precise and reliable forecasts for these mixtures, it is necessary to employ optimised Equation of State (EoS) models.

This academic study assesses the effectiveness of various thermodynamic models in accurately predicting the phase behaviour and thermophysical parameters of complex blends containing high levels of CO_2. The models being examined are the Peng-Robinson (PR), Soave-Redlich-Kwong (SRK), Cubic Plus Association (CPA), Perturbed Chain-Statistical Associating Fluid Theory (PC-SAFT), and Valderrama-Patel-Teja (VPT) equations of state. The performance of these models has been thoroughly evaluated by comparing them to a complete experimental dataset obtained from literature. The tuning of Binary Interaction Parameters (BIPs) is essential in enhancing the precision of model predictions.

The results of this study demonstrate a strong agreement between the predictions made by the model and the experimental data. Specifically, it emphasizes the reliability of the VPT Equation of State in accurately forecasting the vapor-liquid equilibria (VLE), as well as the density and viscosity of mixtures containing a high concentration of CO_2 under conditions commonly encountered during pipeline transportation. The pressure ranges for VLE assessments vary from 8 to 39 barg, while the temperature intervals span from -5 to 45°C. For density and viscosity assessments, the pressure ranges are from 5 to 39 barg, while the temperature intervals are from 5 to 45°C. The mixes analysed have contaminant amounts ranging generally from 5 to 6.5 mole percent.

This investigation validates the accuracy of the VPT Equation of State and enhances the scientific comprehension of phase equilibria in CO_2-rich mixtures. It makes a significant contribution to the field of environmental engineering and sustainability.

37th Scottish Fluid Mechanics Meeting

Stress dependent permeability of granular polymer material as a temporary plug in subsurface reservoirs

Feng Zhao[1,2], Yukie Tanino[2], Jianchun Guo [1] and Amer Syed[3]
[1]School of Oil & Natural Gas Engineering, Southwest Petroleum University, Chengdu, China.
[2]School of Engineering, University of Aberdeen, Aberdeen AB24 3UE.
[3]School of Engineering, The University of Edinburgh, Edinburgh EH9 3DW.

27th May 2024

Abstract

Flow of fluids in geological reservoirs depends on the nature of the porosity and permeability of the reservoir. Mineral composition, sedimentation, tectonic stresses as well as engineering operations such as placement of wells and hydraulic fracturing, results in localisation of porosity and permeability of the reservoir strata. It is often necessary to control the permeability either temporarily or permanently for sustained operations. This work investigates the use of granular polymer compacted as an *in-situ* plug to control local permeability within the formation.

The theoretical framework is based on the premise that the permeability k of the material is a function of stress as $k = f(\sigma)$. For uniaxial fractured reservoirs such as coalbeds, the relationship is of the form $k = e^{C(\sigma-\sigma_o)}$, where C is the compressibility.[1] Permeability of compressible granular media is also related to its porosity ϕ with pore pressure P as a controlling parameter and can be defined as [2]

$$k(P) = \frac{m^2}{s_o}\phi_o^{2\beta-1}e^{(2\beta-1)(P-P_o)}, \tag{1}$$

where m and s_o are parameters related to pore structure,[3] ϕ and ϕ_o are porosity at pressures P and P_o and β is a formation specific constant. The interpretation of the results will be drawn on the basis of the theoretical framework described above. The pore structure of the media will be discussed based on the data acquired through x-ray computed tomography and will be related to the measured permeability.

References

[1] Seidle, J. P., Jeansonne, M. W., and Erickson, D. J., "Application of Matchstick Geometry To Stress Dependent Permeability in Coals," Vol. All Days of *SPE Rocky Mountain Petroleum Technology Conference / Low Permeability Reservoirs Symposium*, 05 1992, pp. SPE–24361–MS.

[2] Barry, D., Lockington, D., Jeng, D.-S., Parlange, J.-Y., Li, L., and Stagnitti, F., "Analytical approximations for flow in compressible, saturated, one-dimensional porous media," *Advances in Water Resources*, Vol. 30, No. 4, 2007, pp. 927–936.

[3] Brace, W. F., "Permeability from resistivity and pore shape," *Journal of Geophysical Research (1896-1977)*, Vol. 82, No. 23, 1977, pp. 3343–3349.

37th Scottish Fluid Mechanics Meeting
Ferrohydrodynamics of Ferrofluid-gaps for Direct Drive Wind Turbine Generators

Mr. Fergus Hall, Prof. Alasdair McDonald, Prof. Markus Mueller
School of Engineering, University of Edinburgh
Scotland, EH9 3JG

27th May 2024

Abstract

In a direct drive (DD) wind turbine, there is no gearbox that steps up the rotational speed and steps down the torque produced from the wind by the blades before the mechanical power enters the electrical generator. A single 15 MW DD wind turbine, such as the largest commercial models available today, may require approximately 25 tons of high-grade permanent magnet material (PMM) which is both expensive and environmentally damaging to procure [1]. In this PhD project we investigate the concept of filling the air-gap region between the rotor and stator of DD generators with magnetically permeable ferrofluid. A 'ferrofluid-gap' would have a greater magnetic permeability than a conventional air-gap, therefore less PMM would be required to achieve the same generator performance [2]. A clear drawback of the ferrofluid-gap concept is an increased mechanical drag within the generator which would result in a decreased efficiency and therefore energy yield of the wind turbine. The dynamic magnetic field within the generator influences the flow of the ferrofluid, accurate prediction of ferrofluid-gap drag therefore requires consideration of 'ferrohydrodynamic' behavior. Simulations of ferrofluid-gap flow have been developed using COMSOL Multiphysics and a scaled ferrofluid-gap test generator has been developed to validate the results. In this talk we will present progress to date and discuss interesting preliminary findings.

Fig. 1. Custom ferrofluid-gap test generator rotor (left) and stator (right).

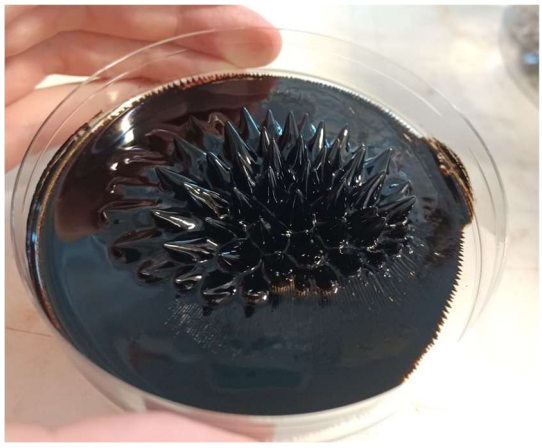


Fig. 2. A ferrofluid sample with a magnet positioned beneath.

References

[1] E. Gaertner, J. Rinker, L. Sethuraman, B. Anderson, F. Zahle, and G. Barter, "IEA Wind TCP Task 37: Definition of the IEA 15 MW Offshore Reference Wind Turbine," pp. 1–44, 2020.

[2] F. P. Hall, A. S. McDonald, and M. A. Mueller, "A Magnetic Analysis of Ferrofluid-Gaps for Direct-Drive Wind Turbine Generators — an Unconventional Approach for Reducing Air-Gap Reluctance," in *IET Conference Proceedings*, Institution of Engineering and Technology, 2023, pp. 225–232.

37^{th} Scottish Fluid Mechanics Meeting

Simulating collective bacterial swarming in sparse systems

François de Tournemire, Kristian Thijssen, Gavin Melaugh, Tyler Shendruk
School of Physics and Astronomy, University of Edinburgh

April 26, 2024

Abstract

Collective swarming of bacterial swimmers is a complex behaviour, often-seen in experiments. It consists of organisms moving rapidly while creating flow structures, such as vortices or jets, on a scale much larger than individual cells. While computational models have been successful at modelling dilute suspensions of swimming bacteria,[1] they have struggled to reproduce the complexity of swarming. In particular, swarming is known to occur in sparser systems than predicted by current models, due to a near-field interaction between flagella and cell bodies. To tackle this complexity, we introduce a new minimal model to efficiently simulate bacteria, in which cell bodies are treated as dumbbells subject to a propulsive force and flagellar bundles are modelled as phantom forces applied directly to the surrounding fluid. The fluid is modelled using Multi-Particle Collision Dynamics (MPCD),[2] a coarse-grained simulation technique that solves the fluctuating Navier-Stokes equations. We validate this new 'Active Dumbbell' model with a multipole expansion[3] and single swimmer simulations. We demonstrate that this numerical approach allows for swarming to occur at a sparser packing fraction than predicted for suspensions of swimmers. Our results are in line with experiments,[4] and the relatively low computational costs of this approach will allow future studies to probe more complex systems than previously considered, such as swarming in the vicinity of large obstacles or in porous media.

References

[1] V. Škultety, C. Nardini, J. Stenhammar, D. Marenduzzo, and A. Morozov, *Swimming supresses correlations in dilute suspensions of pusher microorganisms*, Physical Review X **10**, 031059 (2020)

[2] M. P. Howard, A. Nikoubashman, and J. C. Palmer, *Modeling hydrodynamic interactions in soft materials with Multi-Particle Collision Dynamics*, Current Opinion in Chemical Engineering **23** (2019)

[3] J. Yeomans, D. O. Pushkin, and H. Shum, *An introduction to the hydrodynamics of swimming microorganisms*, The European Physical Journal Special Topics **223** (2014)

[4] A. Be'er and G. Ariel, *A statistical physics view of swarming bacteria*, Movement Ecology **7** (2019)

37th Scottish Fluid Mechanics Meeting
Numerical experiments to assess the dynamics of sediment particle entrainment

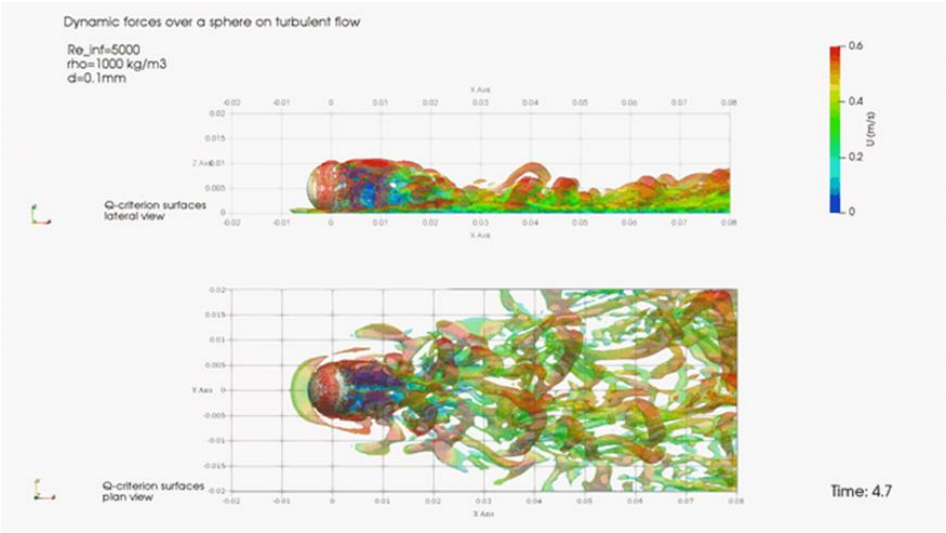
Gaston Latessa, Angela Busse, Maggie Creed and Jin Sun

James Watt School of Engineering, University of Glasgow
James Watt South Building, G12 8QQ

29th May 2024

Abstract

Fluid flow and particle dynamics are closely related in sediment transport processes along natural water courses. These phenomena interact through the transfer of momentum from the fluid flow over the sediment particles (either on riverbeds or during suspension) and through the mass transfer and forces from the particle into the surrounding fluid. Collision between particles is yet another driver to particle entrainment and suspension.
This work focuses on the dynamic behaviour of fluid flow acting upon a sediment particle using numerical experiments by the means of Computational Fluid Dynamics simulations. The objective is to further develop a recent approach where the importance of intermittent effects of forces – and their durations – over sediment particles entrainment is considered, as an alternative to conventional average bulk flow descriptors.
Numerical experiments are run and compared against flume experiments results, such as particle entrainment frequency. The opportunities laid from such numerical approach through the postprocessing of flow velocity, pressure and turbulence fields are also explored in detail. Forces and torques are interpolated across a fixed spherical particle under different conditions: sitting in free flow, lying on a flat bed surface, and finally sitting on a rough bed surface.
The actual forces acting on streamwise and crosswise directions and the torques around a centre of rotation are assessed statistically to obtain an overall summary of these actions over the particle and to confirm the dynamic behaviour and the potential advantages of such an approach compared to classical bulk flow methods. Drag correlations for these geometrical conditions are also investigated and compared against established approaches.



37th Scottish Fluid Mechanics Meeting
Generation of quasi-2D isolated spanwise vortex gusts

G. Pavar, T. Bruce, B. Peterson, A. McDonald
School of Engineering, University of Edinburgh
The King's Buildings, Edinburgh EH9 3FB, UK

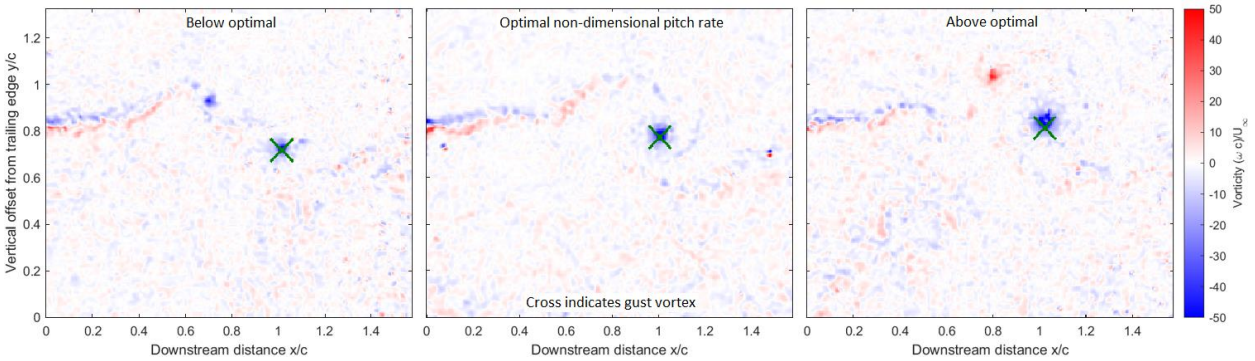
27th May 2024

Abstract

Fluid-structure interactions in steady flow have been extensively studied and are well modelled by potential flow theories. In contrast, unsteady interactions are less well understood, being more complex due to significant viscous interactions that are absent in potential flow. The effects are particularly significant during interactions involving massive flow separation, during which the fluid boundary layer attached to the solid surface becomes detached and sheds into the wake. This separation can be triggered by flow disturbances, commonly known as gusts.

So far, the complexity of these interactions has prevented successful development of a low-order, physics-based model that can predict the fluid forces on the structure. To facilitate development of these models, we need to distinguish the unsteady forces and force production mechanisms from those of the steady flow. This necessitates experimentally generating canonical interactions that match the complexity of current models. Thus, there is a need to generate isolated gusts. However, generating isolated vortex gusts is challenging due to the inherent creation of opposite-sign vorticity, to satisfy the conservation of angular momentum.

Here we show how to generate isolated vortex gusts of variable strength in a water channel, using a pitching foil with a chord-based Reynolds number of 50,000. The opposite-sign vorticity is held in the foil boundary layer, resulting in an isolated gust. As expected from related work, the vortex shedding characteristics are shown to depend on the reduced frequency and pitching angle. These can be combined to yield a "non-dimensional pitch rate". At the optimal value, the vorticity shed during the pitching motion rolls up into a single coherent vortex, as desired. Below or above this value generates either two same sign or two opposite sign vortices, respectively.



Our results provide insight into how to find suitable kinematics to produce isolated vortex gusts of variable strength. Hence, our findings facilitate the experimental generation of a wider range of vortex gusts and thereby more robust validation of current and future low-order models as well as a better understanding of the flow physics of vortex-structure interactions.

37^{th} Scottish Fluid Mechanics Meeting

Machine learning based PAH modelling for LES of turbulent non-premixed flames

Geveen Arumapperuma, Oliver Bladek, Antonio Attili
School of Engineering, Institute for Multiscale Thermofluid, The University of Edinburgh
Edinburgh EH8 3JL, UK

29^{th} May 2024

Abstract

A convolutional neural network (CNN) based on a U-Net architecture[1] is employed to model the PAH mass fraction (naphthalene) in a turbulent non-premixed sooting flame. The CNN models were trained using data from a series of high-fidelity direct numerical simulations (DNS) of temporally evolving planer jet flames.[2,3] The CNN models are trained to take only the mixture fraction (Z) field as the input and predict the corresponding PAH field as the output. Figure 1 (left), shows the joint probability density function (JPDF) between the target DNS output and the CNN prediction while Figure 1 (right) shows the results obtained from a flamelet model, which is a popular approach typically used in this context. The CNN model shows much better performance compared to the flamelet model. This indicates that the CNN model is able to capture the spatial structure of the turbulent Z field. Moreover, this shows that information about the PAH field is embedded within the Z field and CNN models can be trained solely based on the Z field to predict PAH with a much better accuracy than a flamelet-based model.

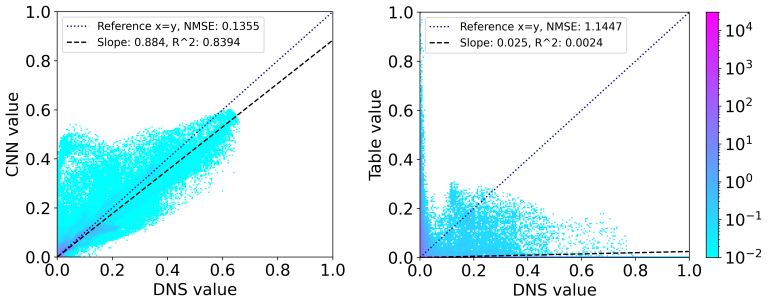


Figure 1: Left: JPDF between the DNS and CNN prediction. Right: JPDF between the DNS and flamelet table. Both models are predicting the PAH (naphthalene) mass fraction.

References

[1] Lapeyre, C. et al., *Combustion and Flame*, Vol. 203, 2019, pp. 255–264.

[2] Attili, A. et al., *Combustion and flame*, Vol. 161, No. 7, 2014, pp. 1849–1865.

[3] Attili, A. et al., *Proceedings of the Combustion Institute*, Vol. 35, No. 2, 2015, pp. 1215–1223.

37^{th} Scottish Fluid Mechanics Meeting

Effect of substrate wettability on the heat transfer coefficient in pool boiling

Giada Minozzi[1], Alessio D. Lavino[2], Edward R. Smith[3], Tassos Karayiannis[3], Khellil Sefiane[1], Omar K. Matar[2], David Scott[1], Timm Krüger[1] and Prashant Valluri[1]
[1] School of Engineering, University of Edinburgh, Edinburgh, UK
[2] Department of Chemical Engineering, Imperial College London, London, UK
[3] Department of Aerospace and Mechanical Engineering, Brunel University London, London, UK

27^{th} May 2024

Abstract

Boiling phenomena is one of the most effective heat transfer mechanisms. This phenomena is common in many industrial applications, including power generation plants and thermal management of micro-devices distinguished by the high heat power density and high dissipation rate which requires a high-level thermal management. Complexity in nucleate boiling arises due to non-equilibrium thermodynamics intertwined with heat, mass and momentum transport and surface processes. Wettability of surfaces plays an important role in the nucleate boiling heat transfer coefficient. We develop a direct numerical simulation model framework using our in-house TPLS Solver,[1] thorough the diffuse interface method[2] to investigate the role of surface wettability on nucleate boiling heat transfer coefficient (NBHTC), bubble growth and departure. This method removes stress singularity at the contact line, thereby allowing imposition of a contact angle boundary condition to prescribe surface wettability.[3] We analyse the effects of going from small-scale i.e. $O(1)$ nucleation sites to large-scale i.e. $O(1000)$ nucleation sites, as shown in Fig 1. We validate our simulations against our nucleate boiling experiments using FC-72 on silicon surfaces.[4] Our simulations show that hydrophilic substrates enhance the NBHTC. Conversely, the hydrophobic cases retain a larger residual vapour on the heated surface limiting the heat transfer.

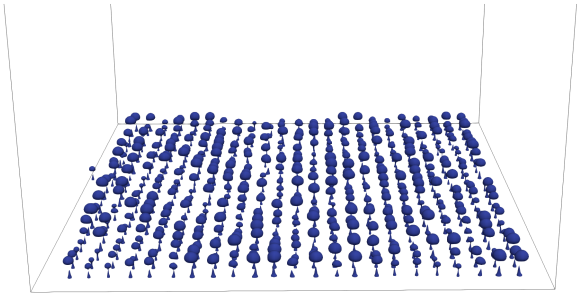


Figure 1: 3D simulation in multiple bubble system with random generation of bubble seeds.

References

[1] L. Ó Náraigh, P. Valluri, D. Scott, T. Collis, I. Bethune, and P. Spelt, "TPLS," 2013.

[2] Sáenz, 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, 2015, pp. 705–739.

[3] Ding, H. and Spelt, P. D., "Wetting condition in diffuse interface simulations of contact line motion," *Physical Review E - Statistical, Nonlinear, and Soft Matter Physics*, Vol. 75, No. 4, 2007, pp. 1–8.

[4] Liu, J., Orejon, D., Zhang, N., Terry, J. G., Walton, A. J., and Sefiane, K., "Bubble Coalescence During Pool Boiling with Different Surface Characteristics," *Heat Transfer Engineering*, 2023.

37^{th} Scottish Fluid Mechanics Meeting

Scattering of surface waves by ocean currents

Han Wang, Ana B. Villas Bôas, Jacques Vanneste and William R. Young

School of Mathematics and Maxwell Institute for Mathematical Sciences,
University of Edinburgh, EH9 3FD, UK;
Department of Geophysics,
Colorado School of Mines, Golden CO 80401, USA;
Scripps Institution of Oceanography,
University of California at San Diego, La Jolla CA 92093-0213, USA.

27^{th} May 2024

Abstract

Ocean turbulence at meso- and submesocales affects the propagation of surface waves through refraction and scattering, inducing spatial modulations in significant wave height (SWH), with profound impacts for air-sea tracer transports, hazard predictions, and satellite data retrievals. We develop a theoretical framework[1,2] that relates these modulations to the current that induces them. We exploit the smallness of the ratio of typical current speed to wave group speed to derive the relations between surface current velocity and SWH anomaly under different asymtptic regimes, and test the results against WAVEWATCH III numerical simulations for both idealised and realistic current configurations. Our framework also provides analytical insights on the relations between power spectra of currents and SWH, and shed light on the different roles of rotational and divergent currents on SWH.

References

[1] Wang, H., Bôas, A. B. V., Vanneste, J., and Young, W. R., "Scattering of surface waves by ocean currents: the U2H map," *arXiv preprint arXiv:2402.05652*, 2024.

[2] Wang, H., Bôas, A. B. V., Young, W. R., and Vanneste, J., "Scattering of swell by currents," *Journal of Fluid Mechanics*, Vol. 975, 2023, pp. A1.

37^{th} Scottish Fluid Mechanics Meeting

The effect of contact line motion on the deposition of particles from an evaporating sessile droplet

Hannah-May D'Ambrosio, Stephen K. Wilson, and Alexander W. Wray
Department of Mathematics and Statistics, University of Strathclyde
26 Richmond Street, Glasgow, G1 1XH

27^{th} May 2024

Abstract

The evaporation of sessile droplets occurs in numerous physical contexts, with applications in nature, industry, and biology. As a consequence of the wide variety of everyday applications, the evolution of, and deposition from, an evaporating droplet has been subject to extensive investigation in recent years.[1] Particular attention has been paid to droplet lifetimes and the ring-like deposit (the "coffee-ring") that often forms near the contact line of a pinned evaporating droplet. Previous work has shown that the mode in which a droplet evaporates is a key factor in determining the lifetime of a droplet undergoing diffusion-limited evaporation. However, few studies have investigated the effect of the mode of evaporation on the deposition of particles from an evaporating droplet, and those that do often use a spatially-uniform evaporative flux to approximate the diffusion-limited model. In this talk we investigate the effect of contact line motion and the local evaporative flux on the deposition of particles from an evaporating droplet. For a thin axisymmetric droplet, we determine the resulting flow due to evaporation, the evolution of the concentration of particles within the droplet, and the evolution of the mass of deposit on the substrate for a droplet undergoing diffusion-limited evaporation in four different modes of evaporation, and compare the results with those determined previously for spatially-uniform evaporation when the contact line is pinned and receding. We find qualitatively different deposit types depending upon the mode in which the droplet is evaporating, as well as on the local evaporative flux. In particular, we show that spatially-uniform evaporation is not an accurate approximation to the diffusion-limited model for the flow within, and deposition from, an evaporating droplet when the contact line is receding.

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.

37th Scottish Fluid Mechanics Meeting

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

Haolin Yang[1], Nigel Mottram[1], Katarzyna Kowal[1]
[1]School of Mathematics and Statistics, University of Glasgow
University Place, Glasgow, G12 8QQ

27th May 2024

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.

37^{th} Scottish Fluid Mechanics Meeting

Evolution of an Evaporating Sessile Droplet According to the One-Sided Model

Henry T. Sharp, Stephen K. Wilson, Alexander W. Wray
Department of Mathematics and Statistics, University of Strathclyde
Livingstone Tower, 26 Richmond Street, Glasgow G1 1XH, United Kingdom

27^{th} May 2024

Abstract

The evaporation of a sessile droplet is a phenomenon of significant interest due to its many appearances in nature and in a wide range of applications, including fuel synthesis and the cooling of microelectronics.[1,2] In the case of an evaporating sessile droplet where the surrounding atmosphere is composed entirely of vapour, or where vapour moves away very quickly from the surface of the droplet, the liquid and gas phases are decoupled, and the influence of the vapour can be neglected when examining the dynamics of the droplet.[3] The associated divergence from thermodynamic equilibrium is what drives evaporation and can be described by the so-called "one-sided" model of evaporation.[4] In the present work, it is shown how the degree of divergence from thermodynamic equilibrium affects the dynamics of a sessile droplet evaporating according to the one-sided model in the so-called stick-slide (SS) and stick-jump (SJ) modes of evaporation. More specifically, it is shown that decreasing the degree of thermodynamic equilibrium decreases the lifetime of the droplet, and that a greater divergence from thermodynamic equilibrium changes the extent to which the slide phase in the SS mode and the jump phase in the SJ mode affect the dynamics of the droplet.

References

[1] Wang, F., Yao, J., Yang, S., Liu, R., and Jin, J., "A new stationary droplet evaporation model and its validation," *Chinese Journal of Aeronautics*, Vol. 30, No. 4, 2017, pp. 1407–1416.

[2] Kabakuş, A., Yakut, K., Özakın, A. N., and Yakut, R., "Experimental determination of cooling performance on heat sinks with cone-jet electrospray mode," *Engineering Science and Technology, an International Journal*, Vol. 24, No. 3, 2021, pp. 665–670.

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

[4] Knudsen, M., "Die maximale verdampfungsgeschwindigkeit des quecksilbers," *Annals of Physics*, Vol. 352, No. 13, 1915, pp. 697–708.

37^{th} Scottish Fluid Mechanics Meeting

A single-phase Level-Set method for unsteady free surface flows: geometric-based reinitialisation and physically-consistent dynamic boundary conditions

Jorge Sandoval, Cristián Escauriaza, David Ingram
School of Engineering, Institute for Energy Systems, University of Edinburgh
Kings Buildings Campus, Edinburgh EH3 5BQ, UK

23^{rd} April 2024

Abstract

In the present work, we introduce a novel numerical solver which integrates an Artificial-Compressibility 3D Navier-Stokes solver with a Level-Set method, and a geometric-based reinitialisation approach that ensures accurate global volume conservation. The governing equations are reforumlated in generalised curvilinear coordinates, which allows the solver to effectivley handle complex geometries.

Dynamic boundary conditions at the free surface, both in normal and tangential directions, are consistently formulated and utilized to extrapolate velocity and pressure values to adjacent nodes using the ghost-fluid method. Furthermore, a geometric-based technique is employed to construct weighted averages of the extrapolated variables near the free surface, thereby incorporating the spatial distribution of the free surface into the calculations.

The results show good agreement with benchmark cases, demonstrating the solver's ability to represent complex unsteady flow and free-surface patterns accurately.

37^{th} Scottish Fluid Mechanics Meeting

Drug inhalation simulations in patient airways with two-way and four-way coupling

Josh Williams[1,3], Uwe Wolfram[2,3], Ali Ozel[3]

[1] Hartree Centre, STFC Daresbury Laboratory, [2] Institute for Material Science and Engineering, TU Clausthal, [3] School of Engineering and Physical Sciences, Heriot-Watt University

Abstract

Inhalers transport drugs as aerosolised solid particles or liquid droplets (henceforth called 'particles'), by spraying a large number of particles ($N_p > 10^8$ for a 100 µg dose) into the airways. We expect this spray to create a significant momentum transfer between particle and gas phases. Due to the large computational resources required to simulate 10^8 particles for realistic dosages of $50 - 100$ µg, the effect of fluid-particle interaction modifying fluid transport (two-way coupling) or particle-particle interaction (four-way coupling) on deposition has not been studied previously. In this study, we aimed to evaluate the influence of a realistic number of particles with two-way and four-way coupling on deposition.

We used the multiphase particle-in-cell[1] OpenFOAM solver MPPICFoam to track fluid and particle motion in patient-specific airways from a healthy adult male patient.[2] Particle motion is computed by Newton's equations of motion including drag force and gravity. Four-way coupled simulations also include contributions of the particle-phase granular pressure,[3] computed on the fluid grid and interpolated to the particles. We studied particle diameters $d_p = 4 - 20$ µm at doses $10, 50$ and 100 µg. For the case of $d_p = 10$ µm, $N_p = 16 \times 10^6, 80 \times 10^6, 160 \times 10^6$ for a dosage of $10, 50, 100$ µg, respectively.

We observed a significant increase in deposition fraction for all particle sizes. Compared to one-way coupled simulations, deposition with $d_p = 10$ µm and 100 µg was 2.6 times larger. Four-way coupling effects became important at 100 µg, causing a decrease in deposition. These findings show that future deposition simulations must include four-way coupling to accurately predict drug transport and deposition that will be used for tailoring of clinical treatments.

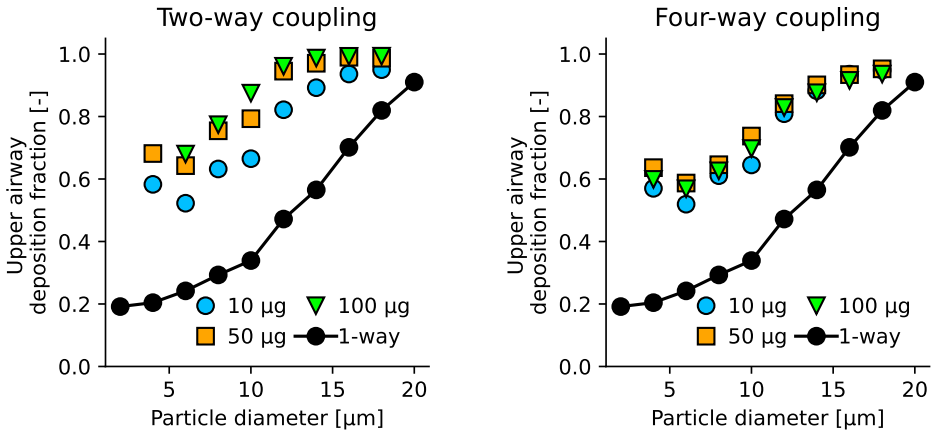


Figure 1: Effect of drug mass on deposition with two-way and four-way coupling.

References

[1] Snider, D. M., *Journal of computational physics*, Vol. 170, No. 2, 2001, pp. 523–549.

[2] Williams, J. et al., *International Journal of Pharmaceutics*, Vol. 612, 2022, pp. 121321.

[3] Harris, S. and Crighton, D., *Journal of Fluid Mechanics*, Vol. 266, 1994, pp. 243–276.

37^{th} Scottish Fluid Mechanics Meeting

Instabilities of thin-film flow over a spinning disk

Laura Milne[1], Alexander W Wray[1], Omar K Matar[2], Marc Pradas[3], and Stephen K Wilson[1]

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

[2] Department of Chemical Engineering, Imperial College London
South Kensington Campus, London, SW7 2AZ

[3] School of Mathematics and Statistics, The Open University
Walton Hall, Milton Keynes, MK7 6AA

27^{th} May 2024

Abstract

We study the dynamics of a thin, axisymmetric film of Newtonian fluid on a uniformly rotating, topographically patterned disk. The system is modelled via a thin-film approximation together with the method of weighted residuals.[1] The resulting model is a closed initial-value problem for the film thickness and the radial and azimuthal fluxes, which includes the effects of inertia, viscosity, centrifugation and capillarity. We determine simplified models in the far field to investigate the spatial and temporal stability and find there exist three distinct regions that exhibit different behaviors. We also study a family of substrate shapes with parameters controlling the asymmetry, smoothness, amplitude and frequency of the topography. The effect of topography on the flow is quantified using an integral measure of the interfacial waviness[2] amongst other measures. In particular, we find that the presence of topography can cause additional interfacial waves that increase the surface area of the film.

References

[1] Ruyer-Quil, C. and Manneville, P., "Improved modeling of flows down inclined planes," *Eur. Phys. J. B*, Vol. 15, 2000, pp. 357–369.

[2] Matar, O. K., Sisoev, G. M., and Lawrence, C. J., "Thin film flow over spinning discs: The effect of surface topography and flow rate modulation," *Chem. Eng. Sci.*, Vol. 63, No. 8, 2008, pp. 2225–2232.

37th Scottish Fluid Mechanics Meeting

Experimental investigation of flow boiling performance of microchannel with different hydraulic diameter and aspect ratio

Luwen Qin[a], Arif Widyatama[a], Julio Cesar Passos[b], Khellil Sefiane[a], Daniel Orejon[a]

a. School of Engineering, University of Edinburgh

b. Department of Mechanical Engineering, Federal University of Santa Catarina, Florianopolis, SC, Brazil

Institute for Multiscale Thermofluids, School of Engineering, University of Edinburgh, The King's Buildings, Edinburgh, EH9 3FD, UK

27th May 2024

Abstract

In light of the progressive trend towards miniaturization and compactness in micro-electromechanical components, the efficient and rapid dissipation of accumulated heat flux from their surfaces has emerged as a prominent focus of research within thermal management[1]. During flow boiling, the latent heat and sensible heat carried away during phase change can dissipate heat more quickly and effectively compared to the sensible heat dissipation in single-phase flow, resulting in more efficient cooling of electronic components. However, as the size of the electronic components decreases so do the necessary paths for the working fluid, the better understanding of the phase-change mechanisms taking place as the length scales of the paths are reduced is of need. Hence, this research focuses on rectangular microchannels of varying hydraulic diameters (D_h = 727 μm, 762 μm and 909 μm) and width to height aspect ratios (AR = 10 and 20) and investigates the flow boiling heat transfer characteristics and bubble dynamics under different Reynolds number (Re = 22, 45 and 68) employing Novec-7000 as the working fluid. As the Re increases, the onset of boiling (ONB) gradually occurs at higher effective input heat, leading to a progressive enhancement in the boiling heat transfer performance of the working fluid. When looking into the different channels, the D_h has a more significant impact on the boiling heat transfer performance than the AR. As the hydraulic diameter increases, under the same effective heat flux, the wall temperature in flow boiling is higher, leading to a gradual decrease in the heat transfer performance. Before the bubble undergoes elongated growth, the growth rates oof transverse L_B and longitudinal W_B tend to be consistent, and the maximum longitudinal W_B growth of the bubble is slightly smaller than the width of the microchannel. After reaching the onset of bubble elongation, the bubble continues to grow both transverse L_B and longitudinal W_B, with the transverse L_B growth rate far exceeding the longitudinal W_B growth rate. Findings reported here will find usefulness on the design of thermal management solutions.

References

[1] Arif Widyatam, Mand Vente, Daniel Orejon, Khellil Sefiane, Experimental investigation of bubble dynamics and flow patterns during flow boiling in high aspect ratio microchannels with the effect of flow orientation, International Journal of Thermal Sciences, Vol. 189, 108238, 2023.

37^{th} Scottish Fluid Mechanics Meeting

Simulation of the transient dissolution of a spherical particle using CFD

Mahsa Hassanpour[1], Sina Haeri[1], and * Stacie Tibos[2]

[1]School of Engineering, University of Edinburgh, Robert Stevens Road, The King's Buildings, Edinburgh, EH9 3FB.
[2]PepsiCo International's R&D center, Beaumont Park, Leicester, LE4 1ET.

27^{th} May 2024

Abstract

One of the strategies to reduce the consumption of salt is the physical modification of the salt crystal structure to increase the dissolution rate. Computational Fluid Dynamics (CFD) can be used to investigate the dissolution rate of different particle shapes and find the optimum structure that will dissolve faster. In this study, we conduct an unsteady simulation of the dissolution of a spherical particle using the Volume of Solid (VOS) method in OpenFOAM. The objective is to model the transient behaviour of the concentration of the solid within the fluid as the particle dissolves over time. The novel solver that we developed is based on the GeoChemFoam toolbox offered by Maes et al.[1]

In order to accurately investigate the dissolution phenomenon in this study, particle fluid interface tracking is done using the VOS method by solving for the volume fraction of each phase (fluid and dissolved material) at each mesh cell. This method enables the simulation of complex interface behaviour over time. Instead of deforming or updating the mesh to accommodate changing boundaries, the solver dynamically computes and updates the interface position within the fixed mesh framework.[2]

The dissolution rate of a particle strongly depends on fluid and solid properties, the saturation concentration of the solid in the fluid, and the surface-to-volume ratio of the particle. These parameters are implemented in the developed code with a new kinetic reaction model accounting for the dissolution process. This solver performed well in convergence tests and is able to explain the complex interaction between diffusion, fluid flow, and particle dissolution with a reduced computational cost. The findings from these simulations can be valuable in industries such as food processing, as well as other sectors interested in understanding and optimizing solid dissolution rates.

References

[1] Maes J, Soulaine, C. and Menke, H. P., "Improved volume-of-solid formulations for micro-continuum simulation of mineral dissolution at the pore-scale," .

[2] Haeri, S. and Shrimpton, J. S., "On the application of immersed boundary, fictitious domain and body-conformal mesh methods to many particle multiphase flows," *International Journal of Multiphase Flow*, Vol. 40, 2012, pp. 38–55.

*Stacie Tibos is an employee of PepsiCo Inc. The views expressed in this abstract are those of the author and do not necessarily reflect the position or policy of PepsiCo Inc.

37^{th} Scottish Fluid Mechanics Meeting

Thermodiffusive Instabilities in Ammonia/Hydrogen Laminar Flames

Markos-Gerasimos Viagkinis, Sofiane Al-Kassar, Antonio Attili
School of Engineering, Institute for Multiscale Thermofluid,
The University of Edinburgh, Edinburgh EH8 3JL, UK

29^{th} May 2024

Abstract

Ammonia and hydrogen burn without carbon dioxide emissions so they have recently gained significant interest due to their possible use to store renewable energy. However, their implementation in technical devices present several challenges, related to the high reactivity, propensity of flashback, and high flame speed of hydrogen and the low reactivity and low flame speed of ammonia. In addition, thermo-diffusive instabilities have a strong impact in hydrogen flames, leading to complex flame patterns and increased flame speeds.[1] These instabilities have also been observed in ammonia/hydrogen blends.[2] The stability of ammonia/hydrogen flames is investigated with Direct Numerical Simulations (DNS), for both the linear and non-linear phases of the instability. In the linear phase, the growth rate ω of harmonic perturbations for different wavelengths $\lambda = 2\pi/k$ is assessed.[3] In the non-linear phase, the flame patterns and enhanced flame speed are investigated. Figure 1 shows the dispersion relations computed with DNS for different ammonia/hydrogen flames. It is observed that the dependency of the instability intensity on the share of ammonia/hydrogen is non-linear. In addition, it is evident that even a very small amount of hydrogen can trigger a significant enhancement of the instability. Similar results are also observed for the non-linear regime. These results can help the understanding of combustion of these new fuels, with the final goal of improving the efficiency and safety of new devices.

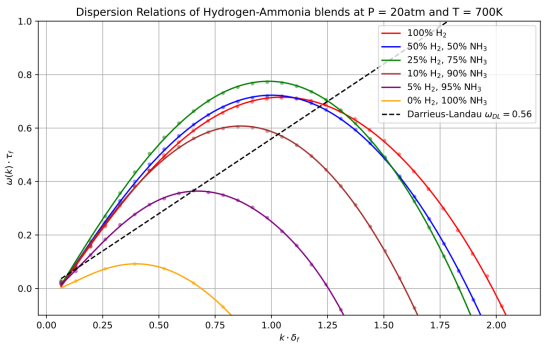


Figure 1: Dispersion relation for different ammonia/hydrogen blends.

References

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

[2] D'Alessio, F., Matteucci, C., Lapenna, P., and Creta, F., *Fuel Communications*, Vol. 19, 2024, pp. 100110.

[3] Al-Kassar, S., Berger, L., Lapenna, P., Creta, F., Pitsch, H., and Attili, A., *arXiv preprint arXiv:2306.10901*, 2023.

37^{th} Scottish Fluid Mechanics Meeting

Ocean wave-induced microplastic transport in the presence of the Basset-Boussinesq history force

Mary Eby[*, a] and Cathal Cummins[†, a, b]

[a] Maxwell Institute for Mathematical Sciences, Department of Mathematics
Heriot-Watt University, Edinburgh, United Kingdom EH14 4AS

[b] Institute for Sustainable Built Environment
School of Energy, Geoscience, Infrastructure and Society
Heriot-Watt University, Edinburgh, United Kingdom EH14 4AS

27^{th} May 2024

Abstract

Over the course of the "plastic age," plastic pollution in the global oceans has been a growing issue. Microplastics (plastic particles smaller than 5mm in diameter) are of special concern, as these particles negatively affect marine ecosystems, and millions of tons of microplastics are added to the global oceans each year. There is an observable size-dependent preferential loss of microplastic particles from the ocean surface to the water column,[1] but the mechanism behind this process is not fully understood. The movement of rigid, spherical, inertial particles in linear waves can be modelled using the Maxey-Riley equation, however, present models of microplastic transport rely on neglecting the Basset-Boussinesq history term and the assumption of infinitely deep water.[2–5] We extend the applicability of these models by including the Basset-Boussinesq history term using a multi-step integration scheme,[6] and employing a velocity field that does not rely on the deep water assumption.

In this talk, we discuss our implementation of the Maxey-Riley equation, including the treatment of the Basset-Boussinesq history term, and the subsequent error analysis performed with our model. Further, we present analyses of the effects of the Basset-Boussinesq history term when other parameters, such as the Stokes number, particle density, and water depth, are varied. Finally, we discuss the significance of this contribution as it relates to microplastic transport.

[*] me2017@hw.ac.uk
[†] c.cummins@hw.ac.uk

References

[1] Kooi, M., Van Nes, E. H., Scheffer, M., and Koelmans, A. A., "Ups and Downs in the Ocean: Effects of Biofouling on Vertical Transport of Microplastics," *Environmental Science and Technology*, Vol. 51, No. 14, 2017, pp. 7963–7971.

[2] Santamaria, F., Boffetta, G., Martins Afonso, M., Mazzino, A., Onorato, M., and Pugliese, D., "Stokes drift for inertial particles transported by water waves," *EPL*, Vol. 102, 2013, pp. 14003.

[3] Haller, G. and Sapsis, T., "Where do inertial particles go in fluid flows?" *Physica D: Nonlinear Phenomena*, Vol. 237, No. 5, 5 2008, pp. 573–583.

[4] Prasath, S. G., Vasan, V., and Govindarajan, R., "Accurate solution method for the Maxey–Riley equation, and the effects of Basset history," *Journal of Fluid Mechanics*, Vol. 868, 6 2019, pp. 428–460.

[5] Dibenedetto, M. H., Clark, L. K., Pujara, N., Dibenedetto, M. H., Clark, L. K., and Pujara, N., "Enhanced settling and dispersion of inertial particles in surface waves," *J. Fluid Mech*, Vol. 936, 2022, pp. 38.

[6] Daitche, A., "Advection of inertial particles in the presence of the history force: Higher order numerical schemes," *Journal of Computational Physics*, Vol. 254, 12 2013, pp. 93–106.

37^{th} Scottish Fluid Mechanics Meeting

The energetics of pilot-wave hydrodynamics

Matthew Durey* and John W. M. Bush
School of Mathematics and Statistics, University of Glasgow
University Place, Glasgow, G12 8QQ

27^{th} May 2024

Abstract

A millimetric droplet may bounce and self-propel across the surface of a vertically vibrating liquid bath, guided by the slope of its accompanying Faraday wave field.[1] The 'walker', consisting of a droplet dressed in a quasi-monochromatic wave form, is a spatially extended object that exhibits many phenomena previously thought exclusive to the quantum realm.[2] Although the walker dynamics can be remarkably complex, steady and periodic states arise in which the energy added by the bath vibration necessarily balances that dissipated by viscous effects. The system energetics may then be characterised in terms of the exchange between the bouncing droplet and its guiding or 'pilot' wave.

We here characterise this energy exchange by means of a theoretical investigation into the dynamics of the pilot-wave system when time-averaged over one bouncing period. Specifically, we derive simple formulae characterising the dependence of the droplet's gravitational potential energy and wave energy on the droplet speed. Doing so makes clear the partitioning between the gravitational, wave and kinetic energies of walking droplets in a number of steady, periodic and statistically steady dynamical states. We demonstrate that this partitioning depends exclusively on the ratio of the droplet speed to its speed limit, which yields a beguiling connection to the Lorentz factor in relativistic mechanics.

References

[1] Couder, Y., Protière, S., Fort, E., and Boudaoud, A., "Walking and orbiting droplets," *Nature*, Vol. 437, 2005, pp. 208.

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

37^{th} Scottish Fluid Mechanics Meeting

A unifying heat transport model for magnetoconvection

Matthew McCormack[1], Andrei Teimurazov[2], Olga Shishkina[2] and Moritz Linkmann[1]

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

[2]Max Planck Institute for Dynamics and Self-Organization, 37077 Göttingen, Germany

27^{th} May 2024

Abstract

A long-standing question in thermal convection is how important response parameters of the system, such as the convective heat transport, depend on the control parameters of the governing equations *e.g.* the dimensionless thermal driving (Rayleigh number Ra). At high values of Ra, the heat transport is known to be well described by simple power laws. In this talk, I will present an analytic model derived from physical assumptions which extends this framework to magnetoconvection, where an electrically conducting fluid is influenced by a vertical magnetic field in addition to the thermal driving. The model predicts scaling exponents in regimes dominated by the magnetic field from knowledge only of convection without a magnetic field. Furthermore, the model suggests a collapse of intermediate regimes onto a master curve, characterising the heat transport over a wide range of parameter values. We will demonstrate that this collapse can be utilised to construct simple analytic models which explicitly describe the heat transport as a function of the control parameters. The theory is supported by our direct numerical simulations and data from the literature.

37^{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

27^{th} May 2024

Abstract

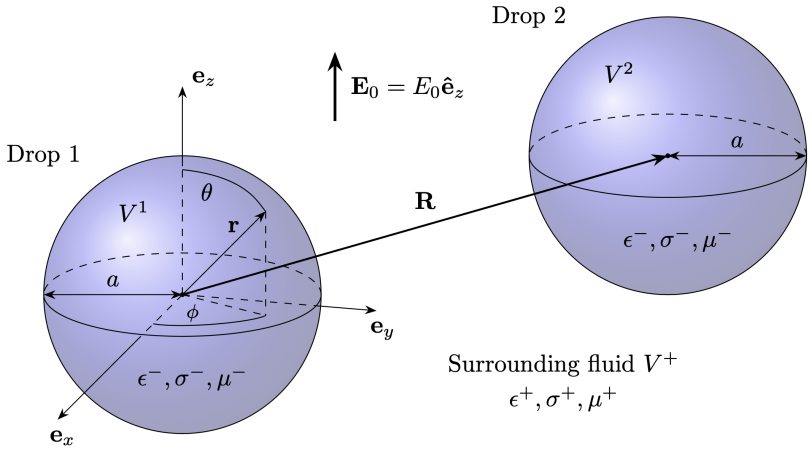


Figure 1: A pair of identical leaky dielectric drops in a uniform electric field

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] The present work predicts that interactions can either destabilise or stabilise each drop, promoting or resisting drop rotation, depending on the relative positions of the drops and the initial perturbations to the surface charge density.

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.

37th Scottish Fluid Mechanics Meeting

The Influence of Inlet, Headland, and Tidal Current Relative Positioning on the Formations of Ebb Tidal Jet and Eddy at Tidal Inlets

Munawir Pratama & Vengatesan Venugopal
School of Engineering, the University of Edinburgh
The King's Buildings, Edinburgh, EH9 3FB

29th May 2024

Abstract

As the water elevation at a tidal inlet rises/falls, the inlet generates inflow/outflow that regulates the movement of matters at and adjacent to the inlet. While studying tidal hydrodynamics at the Montrose Tidal Inlet System on the East Coast of Scotland, UK, a distinctive outflow formation is observed, resulting in an ebb jet, which is followed by only one eddy [1]. Meanwhile, at other tidal inlets, the flow field saw a different ebb jet curvature accompanied by two eddies, for instance, Tauranga Inlet System, New Zealand [2]. From these two inlets, the emerging hypothesis is that ebb jet and eddy formation could be characterized based on its headland and tidal current relative positioning, which will be examined and visualized in this study.

A set of numerical modelling experiments on schematized tidal inlets is developed, with geometry, bathymetry, and tidal conditions adapted from the case of the Montrose Tidal Inlet System. Depth-averaged tidal hydrodynamics modelling is constructed in D-Flow FM of Delft3D-Flexible Mesh, an open-source software system that is based on the unsteady shallow water equations. The schematized model underwent sensitivity analysis, and the evaluation result is that the modelled flow patterns show a very good agreement and maturity with the real case model.

Further, we examined the variability of flood and ebb flow formations under differing headland and tidal current scenarios. Various demonstrations are presented, comparing flow fields at straight or headland-disturbed coast, and being under along-shore or parallel-shore currents, and also their combinations. When comparing the ebb flow formations, two distinctive ebb jet and eddy dynamics are seen, depending on whether the along-shore current approaches from the upwind or downwind side relative to the system. These dynamics explain the ebb jet–eddy systems observed at Montrose Inlet System, UK and Tauranga Inlet System, New Zealand. The introduction of a passive tracker to the model shows a good qualitative similarity between the simulated and observed turbidity patterns in Sentinel-2 satellite imagery at Montrose and Tauranga.

References

[1] Pratama, M., Venugopal, V., and Poleykett, J. “Wave and Tidal Hydrodynamics Characterization of Montrose Bay: A Severely-Eroding Coast in East Scotland.” *Proceedings of the 40th IAHR World Congress, Vienna, Austria*, 21-25 August 2023, pp. 546–553.

[2] Spiers, K. C., Healyf, T. R., and Winter, C. “Ebb-Jet Dynamics and Transient Eddy Formation at Tauranga Harbour: Implications for Entrance Channel Shoaling.” *Journal of Coastal Research*, Vol. 25, No. 1, 2009, pp. 234–247.

37^{th} Scottish Fluid Mechanics Meeting

Numerical turbulent viscosity estimation

Nguyen Q. Chien
School of Engineering, University of Edinburgh

27^{th} May 2024

Abstract

For homogeneous turbulence the decline in total kinetic energy is known[1] to relate with the viscosity as: $-\frac{\mathrm{d}E_{\mathrm{kin}}}{\mathrm{d}t} = -\frac{\mathrm{d}}{\mathrm{d}t}\int_V \frac{1}{2}|\mathbf{u}|^2 \mathrm{d}V = \nu \int_V |\vec{\omega}|^2 \mathrm{d}V$. The formula provides a means to estimate ν based on the total kinetic energy and vorticity of the system, which can be numerically integrated in a convenient way for solvers such as Gerris.[2]

Simulation shows that the computed viscosity (ν_{comp}) fluctuates rapidly at first but then converges toward a mean value. For $\nu_{\mathrm{comp}} = \nu_{\mathrm{num}} + \nu_{\mathrm{init}}$, the target is to obtain $\nu_{\mathrm{comp}} \approx \nu_{\mathrm{goal}}$, the kinematic viscosity of water.

ν_{init} can be found iteratively. For the first iteration, choose $\nu_{\mathrm{init}} = \nu_{\mathrm{goal}}$. After a certain simulation time until the system reaches stability, we compare ν_{comp} to ν_{goal}. If $\nu_{\mathrm{comp}} \approx \nu_{\mathrm{goal}}$ then this ν_{init} is chosen; otherwise $\nu_{\mathrm{init}} + (\nu_{\mathrm{goal}} - \nu_{\mathrm{comp}})$ is chosen as the input for the next simulation. ν_{init} can be negative because it will be added up with ν_{num} to represent a physical quantity.

Test cases are performed with several cases for an axisymetric (submerged) jet, with $nu_{\mathrm{init}} \sim O(-10^{-5})$ we obtain $\nu_{\mathrm{num}} \sim O(+10^{-5})$.

Estimating the numerical viscosity can help in certain cases, e.g. to approximate turbulent (eddy) viscosity, which serves for modeling the sub-grid scales (SGS) stress in large-eddy simulations (LES).

References

[1] Davidson, P., Turbulence: An Introduction for Scientists and Engineers, Oxford University Press, 2004.

[2] Popinet, S., "Gerris: a tree-based adaptive solver for the incompressible Euler equations in complex geometries," J. Comput. Phys., Vol. **190**, No. 2, 2003, pp. 572–600.

37^{th} Scottish Fluid Mechanics Meeting

Experimental modelling of rock dumping through vertical and inclined pipes

Otto Neshamar, Alan Cuthbertson, Øyvind Thiem, Peter Davies
School of Science and Engineering, University of Dundee
DD1 4HN, Dundee, UK

29^{th} May 2024

Abstract

Recent plans for tunnel construction along Sørfjorden, Norway, are projected to generate O(10^7) m^3 of blasted rock, most of which would require direct disposal within the fjord waters. This necessitates an efficient disposal strategy which minimises the negative environmental impact as well as the overall carbon footprint of the disposal process. Two deposition strategies are proposed: mass dumping via (1) vertical pipelines and (2) inclined shaft tunnels. In both cases, rocks are released at a significant depth, with the expectation that fines will deposit to the floor of the fjord. Both strategies were investigated in laboratory experiments at the University of Dundee. Experiments involve releasing particle mixtures through vertical and inclined pipes under a range of conditions at 1:50 scale, with dynamic similarity achieved through Froude scaling. Measurements were conducted by imaging from 5 colour CMOS cameras, with particles colour-coded based on size.

Figures show two example snapshots from a single camera during a vertical-pipe experiment. The example shown is for a 600 mm long 46 mm diameter pipe, and the two images are approx. 5 seconds apart. A range of image processing techniques are employed, including colour thresholding to classify particles and optical flow methods to obtain velocities from image sequences. Results show how dumping large volumes of particles into narrow pipes produces a 'traffic jam' where particles linger within the pipe for much longer than otherwise. Pipe inclination is seen to produce preferential flow paths, which results in particles clearing out of the pipe more quickly. Accumulation of fines in the pipe is found to be insignificant; results indicate that, in relatively still waters, wide dispersal of fine particles is minimal.

37th Scottish Fluid Mechanics Meeting
Modelling ferrofluid emulsions using a bulk stress approach

Paolo Capobianchi[1], Ghulam Sultan[1], Marcello Lappa[1], Fernando T. Pinho[2], Mónica S.N. Oliveira[1]
[1]University of Strathclyde
75 Montrose Street, G1 1XJ, Glasgow,
[2]University of Porto
Rua Dr Roberto Frias, 4200-465 Porto

27th May 2024

Abstract

The rheology of a dilute emulsion made of ferrofluid droplets dispersed in a non-magnetisable immiscible liquid has been modelled through the bulk stress model of Batchelor[1] and simulated numerically using a Volume of Fluid approach considering, uniform magnetic fields and different viscosity ratios. The results revealed that, even for magnetic fields of relatively modest intensity, the rheological properties of the emulsion are significantly altered by the magnetic force which affects the rheology in two ways: directly, by introducing additional stresses[2], and indirectly, by altering the drop conformation[1]. In particular, we observed an increase of the effective viscosity of the emulsion, and a reversal of the sign of the two normal stress differences for relatively high magnetic field intensities and low shearing forces. Comparisons between the results predicted by our rheological model and the numerical experiments have provided evidence of the reliability of the model in predicting the effective viscosity and normal stress differences of the ferrofluid suspension in the presence uniform magnetic fields when modest droplet deformations are contemplated.

References

[1] G.K. Batchelor, “The stress system in a suspension of force-free particles,” *Journal of Fluid Mechanics,* Vol. 1, No. 41, 1970, pp. 545-570.
[2] P. Capobianchi P., M. Lappa, M.S.N Oliveira, F.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.

37th Scottish Fluid Mechanics Meeting
Surface Vibration Effects on Nanoscale Ice Nucleation

Pengxu Chen, Rohit Pillai, and Saikat Datta
School of Engineering, University of Edinburgh
The King's Buildings, Edinburgh, EH9 3FB
29th May 2024

Abstract

Controlling ice nucleation plays an essential role in various engineering applications, such as food freezing[1], cryopreservation[2] and developing anti-icing coatings[3]. In recent years, the use of high-frequency surface vibrations has emerged as a promising anti-icing/de-icing technology[4]. However, the effects of such vibrations on nanoscale ice nucleation (i.e. formation of critical ice clusters that precede bulk ice formation) remain poorly-understood. Using molecular dynamics (MD) simulation, this study explores how surface vibrations affect nanoscale ice nucleation from supercooled water. We demonstrate that ice nucleation can be either hindered or enhanced by surface vibrations, depending on setup and vibrational parameters. We observe that, in water nano-films, GHz-order surface vibrations inhibit the formation of large pre-critical ice-like clusters, reducing the ice nucleation probability (Fig. 1a). Conversely, in confined nano-pores, MHz-order surface vibrations are shown to trigger regions of negative pressure, which enhances the probability of ice nucleation (Fig. 1b). This study advances our understanding of the interplay between surface vibration dynamics and ice nucleation, which can enable future anti-icing and freezing technologies.

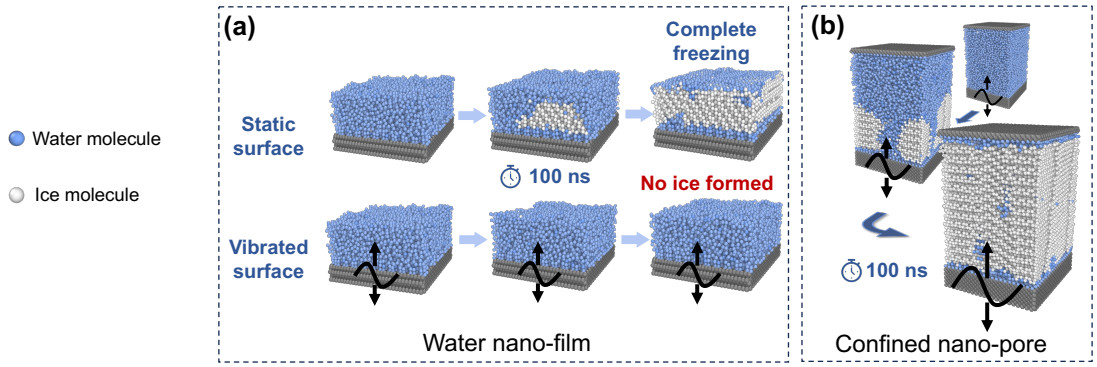


Fig. 1 Ice nucleation probability is (a) reduced by GHz-order surface vibrations on water nano-films, and (b) enhanced by MHz-order surface vibrations in nano-pores.

References

[1] Jia, Guoliang, et al. "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, Kathryn A., and Matthew I. Gibson. "Chemical approaches to cryopreservation." *Nature Reviews Chemistry,* Vol. 6, No. 8, 2022, pp. 579-593.
[3] Kreder, Michael J., et al. "Design of anti-icing surfaces: smooth, textured or slippery?." *Nature Reviews Materials* Vol. 1, No. 1, 2016, pp. 1-15.
[4] Del Moral, Jaime, et al. "A Holistic Solution to Icing by Acoustic Waves: De-Icing, Active Anti-Icing, Sensing with Piezoelectric Crystals, and Synergy with Thin Film Passive Anti-Icing Solutions." *Advanced Functional Materials*, Vol.33, No. 15, 2023, pp. 2209421.

37th Scottish Fluid Mechanics Meeting

Post-impact gas-cushioning in liquid-solid impacts

Peter Hicks
School of Engineering, University of Aberdeen
Kings' College, Aberdeen, AB24 3UE.

29th May 2024

Abstract

Gas cushioning in droplet impacts has been investigated using a distinguished limit between an inviscid liquid and a thin lubricating gas layer (between the liquid and impact site), for over twenty years.[1] However, despite notable successes in determining the shape and volume of the gas bubble trapped during impact, this framework breaks down at the instant of initial touchdown when the gas layer separating the droplet from the impact site ruptures. In this talk, preliminary work will be presented showing how pre-impact cushioning of liquid-solid impacts can be extended into the post-touchdown regime by incorporating disjoining van der Waals forces and surface tension.

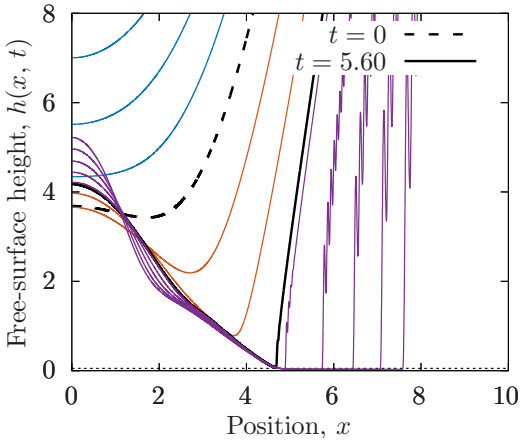


Figure 1: One half of a droplet free-surface as it is initially decelerated by a gas pressure build-up, touches down (solid black line), and then evolves post-impact (purple).

At touchdown, the rupture of the gas cushion is regularised using a thin pre-cursor gas layer, which allows the post-impact free surface to be calculated (see figure 1). As the surface tension is increased, the free-surface evolution will be shown to deviate ever more from potential flow models for liquid impacts that neglect gas cushioning and surface tension. We conclude by discussing how models of post-impact gas cushioning might be extended to liquid-liquid impacts and other impact configurations.

References

[1] Smith, F. T., Li, L., and Wu, G. X., "Air cushioning with a lubrication/inviscid balance," *J. Fluid Mech.*, Vol. 482, 2003, pp. 291–318.

37th Scottish Fluid Mechanics Meeting

Experimental Studies on the Hydraulic Behaviour of Nigerian Soils for Efficient Water Management

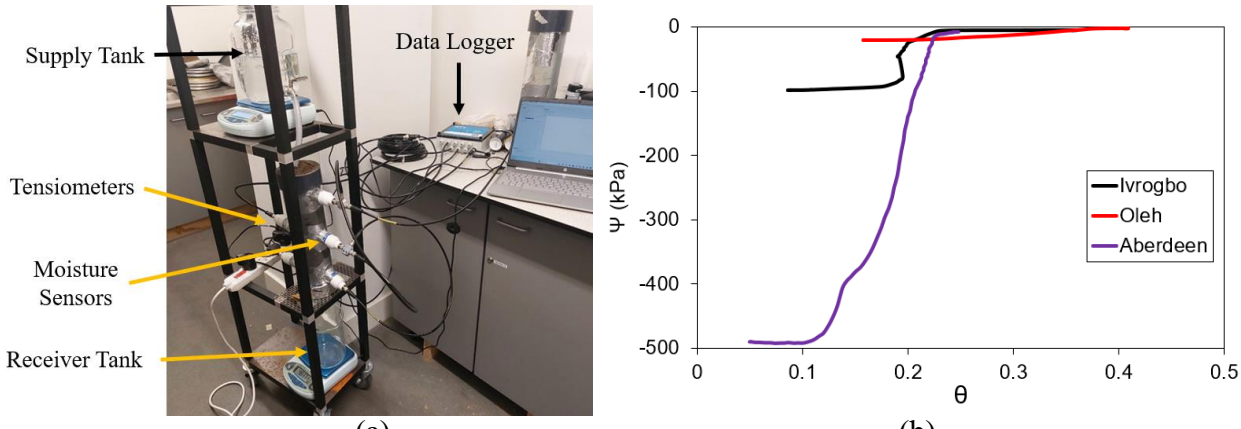
Peter Uloho Osame, Taimoor Asim, Sheikh Zahidul Islam, Dallia Ali
School of Engineering, Robert Gordon University
Garthdee Road, Aberdeen, AB10 7GJ

27th May 2024

Abstract

Nigeria severely lacks in clean water with 28% of her population having no access to drinking water. The agriculture sector of Nigeria consumes more than 1/3rd of its available clean water. More efficient water management is required in the agriculture sector of Nigeria as the water scarcity is expected to rise to dangerous levels by 2030, potentially leading to mass pooulation displacement. Farms mostly rely on experience rather than scientific evidence when it comes to irrigation, with flood irrigation being the most common and popular method. Efforts are being currently made to shift towards more water-efficient irrigation methos like drip irrigation etc. However, it is also important to better understand the irrigation requirements based on different soil types and the comfort zone of different crops. Thus, in this study, efforts have been made to provide insights into the hydrodynamic bahaviour of soils from two different regions of Nigeria, Ivrogbo and Oleh, in order to provide scientific evidence for developing water-efficient irrigation strategies.

A bespoke soil column test-rig has been developed, shown in figure 1(a), integrated with soil moisture sensors and equitensiometers, to carry out infiltration experiments on the monolith soil samples, mimicking drip irrigation. Soil moisture content and water matric potential have been recorded at different depths of the sample every 1 minute during the infiltration experiments. For comparative purposes, a soil sample for Aberdeen has also been sourced. Soil water characteristic curves have been obtained for all the soil samples [1-2], as shown in figure 1(b). All soil samples were left to dry for atleast 4 weeks. Soil sample from Oleh region demonstrates highest matric potential (excellent water retention capability) requiring less amount of water to reach field capacity (i.e. EU standard for crop comfort zone). Soil sample from Ivrogbo region has significantly lower matric potential at the start of the infiltration experiment, requiring significantly higher amount of water to reach field capacity. According to these findings, drip-irrigation can be easily adopted by the farmers in the Oleh region. As for Aberdeen soil sample, its water retention capability is quite poor compared to other soil samples considered.



(a) (b)

Figure 1. (a) Test setup (b) characteristic curves of different soils

References

[1] Zhou W, Yuen K, Tan F "Estimation of soil–water characteristic curve and relative permeability for granular soils with different initial dry densities", *Engineering Geology,* Vol. 179, 2014, pp. 1-9.

[2] Yang C, Wu J, Li P, Wang Y, Yang N "Evaluation of Soil-Water Characteristic Curves for Different Textural Soils Using Fractal Analysis", *Water,* Vol. 15, No. 4, 2023, pp. 772.

37th Scottish Fluid Mechanics Meeting

Possible ubiquitous nature of solitary waves in the stable atmospheric boundary layer

Philip S. Anderson
Scottish Association for Marine Science
Dunstaffnage, Argyll and Bute

27th May 2024

Abstract

Reanalysis of measurements of stratified atmospheric boundary layers (SABLs) hints that coherent events may be ubiquitous in such flows. These wave-like structures may explain unexpected variation in turbulent Prandtl Number (*Pr*) at super-critical Richardson Number ($Ri > 0.25$)[1]. If so, these phenomena may solve a long standing puzzle of the observed complex behaviour of the evolution of SODAR echo-grammes SABLs over very flat terrain. SABLSs occasionally exhibit unexpected behaviour, such as coherent structures within turbulence, or apparent laminar (non-turbulent) flow which retains turbulent-like viscosity. Attempts to model these flows in climate or mesoscale models results in divergence from observation: either turbulent fluxes remain too large or the flow evolves to ever larger *Ri* and thence laminar flow. A partial solution is to modify the *Pr* as a function of *Ri*, but this is problematic because no plausible "missing physics" has been detected which explains the proposed deviation of turbulent *Pr* from unity.

This paper will present new analysis of coherent signatures in data collected on the Brunt Ice Shelf in 2003. The novel approach implies that solitary-like wave events, which were previously noted on rare occasions at this site, might in fact be ubiquitous. Further work is necessary from the community to assess whether trains of solitons might transport momentum in the vertical without an associated heat flux.

References

[1] Anderson, P. S., "Measurement of Prandtl Number as a Function of Richardson Number Avoiding Self-Correlation" *Boundary-Layer Meteorol 131, 345–362 (2009)*

37^{th} Scottish Fluid Mechanics Meeting

Numerical Simulation of Inertial Particle Motion in Heterogeneous Suspensions

Qi Zhou, Benjamin Owen and Timm Krüger
School of Engineering, Institute for Multiscale Thermofluids
The University of Edinburgh, EH9 3FD, UK

29^{th} May 2024

Abstract

Inertial microfluidics (Reynolds number Re $\sim \mathcal{O}(10-100)$) emerged in the late 2000s and has contributed to wide applications in biomedical engineering [1]. One key application is label-free, high-throughput separation of target particles from heterogeneous suspensions (e.g. blood) for diagnosis purposes. Such separation relies on the migration of different particle types to their own focusing positions determined by the balance of inertial and drag forces within microchannel cross-sections. However, current inertial microfluidic devices are mostly limited to handling diluted samples (concentration <5%), due to a lack of understanding of the interplay of inertial particle migration and particle-particle interactions in dense suspensions, which are both experimentally and numerically challenging.

Using an *in house* 3D lattice-Boltzmann code with incorporated immersed-boundary and finite-element methods [2], we perform numerical simulations to elucidate the mechanisms for inertial particle motion in heterogeneous suspensions. A range of effects including flow inertia, sample concentration, mixture fraction and particle size/softness heterogeneity are investigated. We demonstrate that in homogeneous suspensions with increasing concentration, the hydrodynamic interactions between particles introduce substantial fluctuations to their motion and result in weakened focusing behaviour. For heterogeneous suspensions, the interparticle interactions are more complex, affecting the lift velocities and collective motion of particles distinctly based on their absolute and relative size. Subject to appropriate particle confinement and mixture fraction, satisfactory focusing or enrichment of target particles can be achieved in dense suspensions (e.g. 20%) under high inertia conditions (Re >100). This work furthers our understanding of the inertial lift of particles in concentrated samples and provide evidence for the design and optimisation of microfluidic devices aimed at particle separation from dense heterogeneous suspensions.

References

[1] J.M. Martel & M. Toner, Inertial Focusing in Microfluidics. *Annual Review of Biomedical Engineering*, 16(1):371-396, 2014.

[2] B. Owen *et al.*, Lattice-Boltzmann modelling for inertial particle microfluidics applications - a tutorial review. *Advances in Physics: X*, 8(1):2246704, 2023.

37th Scottish Fluid Mechanics Meeting
Nucleate pool boiling of carbon dioxide on a vertical flat copper surface in confined spaces

Rafael B. A. Passarella, Arthur K. A. Araújo, Adonis Menezes, Júlio C. Passos

Department of Mechanical Engineering, Federal University of Santa Catarina
Florianópolis/SC, 88040-900, Brazil

27th May 2024

Abstract

The concern about greenhouse gases and their impacts on global warming is one of the driving forces behind research on more environmentally friendly fluids. In this context, carbon dioxide (CO_2) has re-emerged as an alternative to synthetic refrigerants, due to its zero Ozone Depletion Potential (ODP) and Global Warming Potential (GWP) equal to unit. CO_2 – when treated as a refrigerant it's called R-744 – is a natural, abundant, cheap, non-toxic, and non-flammable fluid. The nucleate boiling has, as the main characteristic, the transfer of high heat fluxes to lower temperature differences between the heating surface and the refrigerant[1], which allows designing more compact and smaller heat exchangers. The pool boiling process has been used in refrigeration systems in various engineering applications involving thermal energy dissipation, such as high-power electronics, heat exchangers, flooded evaporators, and nuclear reactors. Confined nucleate boiling in a narrow space has heat transfer characteristics that may be different from those of conventional unconfined boiling. However, experimental data for CO_2 nucleate boiling are scarce in the open literature. Thus, to understand the behavior of carbon dioxide during nucleate boiling with confinement, this paper presents experimental results on a vertical flat copper surface, with and without confinement. Test conditions include a saturation pressure of 2.8 MPa and heat fluxes ranging from 56 to 309 kW/m², without confinement and with confinement of 0.8, 0.5, and 0.3 mm, which yields Bond numbers of 0.97, 0.61, and 0.36, respectively. The dimensions of the vertical flat surface of the copper are 10 mm in width and 30 mm in height. The experimental results shown a tendency to accelerate the partial dryout on the surface with the decreasing confinement distance. The experimental heat transfer coefficient results were compared with Rohsenow's[2] correlation due to its wide range of applicability and with the Gorenflo's[3] and Liu's[4] correlations, both developed for CO_2. For the case without confinement, the experimental heat transfer coefficients showed good agreement with the proposed correlations for CO_2.

References

[1] CAREY, Van P. Liquid-vapor phase-change phenomena: an introduction to the thermophysics of vaporization and condensation processes in heat transfer equipment. CRC Press, 2020.

[2] ROHSENOW, Warren M. *et al.* A method of correlating heat-transfer data for surface boiling of liquids. Transactions of the American Society of Mechanical Engineers, v. 74, n. 6, 1952, p. 969-975.

[3] GORENFLO, D.; KENNING, D. Pool Boiling. VDI Heat Atlas, Second edition, Chapter H2, Springer-Verlag Berlin Heidelberg, Berlin, Heidelberg, 2010, p.755-792.

[4] LIU, Shengchun *et al.* A new correlation for carbon dioxide boiling heat transfer coefficient outside evaporating tubes. Journal of Cleaner Production, v. 276, 2020, p. 123050.

37^{th} Scottish Fluid Mechanics Meeting

A 2D model for the drainage of melt ponds on Arctic sea ice

Russell Campbell, David Rees Jones, David Dritschel,
School of Mathematics and Statistics, University of St Andrews,
Mathematical Institute, North Haugh, St Andrews KY16 9SS.

27^{th} May 2024

Abstract

In the summer months, large pools of water known as melt ponds form on the surface of Arctic sea ice. The evolution of melt pond coverage is heavily dependent on the availability of drainage pathways in the ice. Brine channels are a common drainage feature which initially develop during ice growth as a means of desalination. As melt water starts to drain, two possible regimes occur. Either these brine channels freeze shut, causing pond growth, or they expand, accelerating pond drainage.[1,2] Which regime dominates depends on the initial channel radius and its relation to a critical value.[1]

Here, we present a 2D model for brine–channel drainage, which we use to improve our understanding of what controls the critical value. Through numerical simulation of the 2D mushy-layer equations, we demonstrate the two regimes of channel evolution and their dependence on the various non-dimensional parameters of the physical system. Our 2D calculations motivate a simplified, analytical, quasi-steady-state 1D model which allows us to reliably estimate the critical radius in the 2D system.

References

[1] Polashenski, C., Perovich, D., and Courville, Z., "The mechanisms of sea ice melt pond formation and evolution," *J. Geophys. Res.*, Vol. 117, No. C1, 2012.

[2] Polashenski, C., Golden, K. M., Perovich, D. K., Skyllingstad, E., Arnsten, A., Stwertka, C., and Wright, N., "Percolation blockage: A process that enables melt pond formation on first year Arctic sea ice," *J. Geophys. Res.*, Vol. 122, No. 1, 2017, pp. 413–440.

37th Scottish Fluid Mechanics Meeting

Multiphase Flow Measurement of Wet Gas Flow Using Machine Learning Modelling Algorithms

Seyedahmad Hosseini [a], Gabriele Chinello [b], Gordon Lindsay [c], Sheila Smith [c], Don McGlinchey [a]

[a] Department of Mechanical Engineering, Glasgow Caledonian University, Glasgow, UK
[b] TÜV SÜD National Engineering Laboratory, Glasgow, UK
[c] Department of Applied Science, Glasgow Caledonian University, Glasgow, UK

27th May 2024

Abstract

This paper aims to deal with the big problems of measuring wet gas flow, especially when there is liquid mixed in with the gas. This can cause a higher pressure drop, which needs to be fixed, often called overreading. This challenge comes from the need to figure out how much liquid there is using a parameter called Lockhart-Martinelli. While this parameter. To tackle this challenge, this study introduces a new way to measure gas and liquid flow rates using machine learning (ML) modelling. It helps overcome the limitations of existing methods. However, it would be beneficial to be able to predict gas-liquid flow rates directly without many correlation side tracks from liquid content to overreading and flowrate measurement.

The research method uses advanced machine learning tools like Deep Neural Networks (DNN), Long Short-Term Memory (LSTM) networks, and Random Forest (RF), along with combinations like LSTM-DNN and RF-DNN. These models help predict how much gas and liquid will flow based on sensor data. The dataset includes important details like pressure, temperature, throat, and recovery differential pressure, recorded at a frequency of 1 Hz.

The individual and ensemble models are carefully trained to accurately forecast gas and liquid flow rates, forming the vital core of our comprehensive approach. The study shows that LSTM-DNN and RF-DNN ensemble models work better than traditional methods for predicting gas and liquid flow rates in wet gas conditions. The effectiveness of the proposed approach is rigorously confirmed through evaluation metrics such as MSE, RMSE, MAE, and error plots. The study shows that using a combination of different models, especially RF-DNN, makes predictions of gas and liquid flow rates more accurate.

This new technology can help engineers measure wet gas flow more accurately and reliably. It will make industrial processes run smoother and more efficiently.

37th Scottish Fluid Mechanics Meeting
Molecular kinetic modelling of non-equilibrium evaporative flows

Shaokang Li[1], Livio Gibelli,[1] and Yonghao Zhang[2]

[1]*School of Engineering, The University of Edinburgh, Edinburgh EH9 3FB, UK*
[2]*Centre for Interdisciplinary Research in Fluids, Institute of Mechanics, Chinese Academy of Sciences, Beijing, 100190, PR China*

Non-equilibrium evaporation is an ubiquitous phenomenon in nature and has numerous applications in nanoscale devices, such as chip cooling and nano-assemblies [1]. In the general case, the bulk of the vapour and liquid phases are well described by the hydrodynamic equations at the macroscopic scale, while the interface has a complex structure that still requires experimental and theoretical investigation. This interface consists of two regions: the vapour-liquid interface, whose size is of the order of the molecular diameter, and a Knudsen layer, whose size is of the order of the mean free path. The macroscopic variables are subject to strong variations in these two regions, which manifest themselves as "jumps" on the macroscopic scale. The traditional kinetic treatment of evaporation processes is limited to the Knudsen layer described by the Boltzmann equation. The structure of the liquid-vapour interface is not resolved and the molecular exchange process with the liquid phase is described by a phenomenological boundary condition [2]. A more thorough description of the evaporation processes is provided by the Enskog-Vlasov (EV) equation, which is capable of describing both the liquid and vapour phases, including the interfacial region. However, this kinetic equation is computationally demanding, making it impractical for widespread use in real-world scenarios [3].

In this work, we report a simplified kinetic model that has a similar level of accuracy as the EV equation, but is significantly more efficient in terms of computational solution. The robustness of the kinetic model is first assessed by studying the equilibrium state. At a given temperature, the number densities of the liquid and vapour phases calculated by our model are in agreement with the theoretical values obtained from the liquid-vapour coexistence curve. In addition, our model predicts interfacial thicknesses close to those obtained from the EV equation, with the differences arising from the way the collision term is treated in each model. We then simulate evaporation into a vacuum to determine the evaporation coefficient, a key parameter in the kinetic boundary conditions at the liquid-vapour interface. As shown in Figure 1(b), when the liquid-vapour interface temperature is set to the separation value rather than that of the liquid bulk, the evaporation coefficient remains nearly constant, in qualitative agreement with the previous results [4]. Finally, evaporation into vapour was studied to demonstrate the ability of our model to capture the non-equilibrium Knudsen layer that forms near the evaporating surfaces. The predicted jump relations for number density and temperature are compared in Fig. 1(c) with those from the direct simulation Monte Carlo method and the analytical solution, showing good overall agreement.

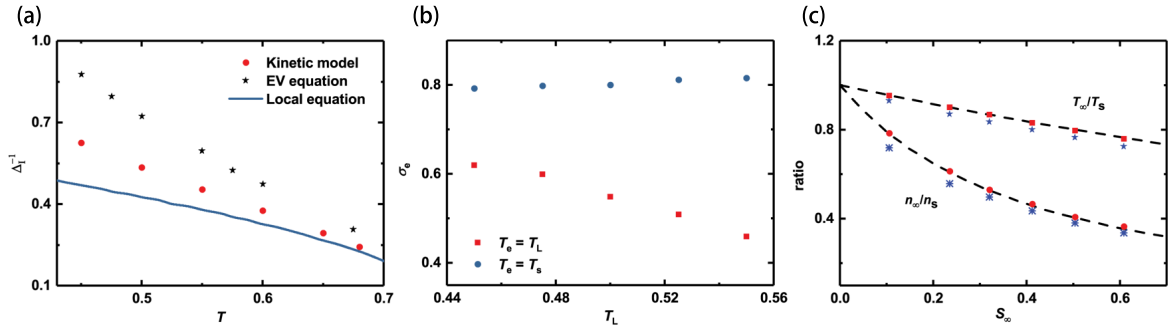


Fig. 1. (a) Comparison of vapour-liquid thickness. (b) Evaporation coefficient σ_e versus temperature. The subscripts L and s denote the liquid bulk and the location where the parallel and normal temperatures separate, close to the centre of the liquid-vapour interface. (c) Jump relations along the Knudsen layer for various far-field evaporation velocities evaporative velocities S_∞. The dashed line shows the analytical solution; the blue symbols are the results from our simplified kinetic model; the red symbols are the results from DSMC simulations.

REFERENCES

[1] D. Zang, S. Tarafdar, Y. Y. Tarasevich, M. D. Choudhury, T. Dutta, *Physics Reports* **804**, 1–56 (2019).
[2] A. Frezzotti, *Physics of Fluids* **23**, 030609 (2011).
[3] A. Frezzotti, P. Barbante, L. Gibelli, *Physics of Fluids* **31**, 062103 (2019).
[4] A. Frezzotti, L. Gibelli, S. Lorenzani, *Physics of Fluids* **17**, 012102 (2005).

37^{th} Scottish Fluid Mechanics Meeting

Dynamics of flow around wind-dispersed dandelion-inspired polyimide flyers

Soumarup Bhattacharyya[1], Bappa Mitra[1,2], Marc Desmulliez[2],
Ignazio Maria Viola[1]
[1]School of Engineering, Institute for Energy Systems,
University of Edinburgh, EH9 3BF.
[2]School of Engineering & Physical Sciences, Heriot-Watt University, EH14 4AS.

27^{th} May 2024

Abstract

The aerodynamics of the dandelion seed has been a topic of interest for the past decade because of the seed's efficient dispersal mechanism. The porous pappus of the seed facilitates flight by the formation of separated vortex rings.[1,2] To understand the changes in fluid dynamics due to the porosity of the pappus several polyimide replicas with various porosity were fabricated. These replicas were made to hover in a vertical wind tunnel constructed at the University of Edinburgh. Flow visualization shows that the porosity of the flyers affects the separated vortex ring. Particle image velocimetry revealed that the terminal velocity of the flyers with the same mass density decreases with a decrease in the filament length. This experimental study would further aid in the manufacturing of efficient dandelion-inspired drones for environment monitoring.

References

[1] Cummins, C., Seale, M., Macente, A., Certini, D., Mastropaolo, E., Viola, I. M., and Nakayama, N., "A separated vortex ring underlies the flight of the dandelion," *Nature*, Vol. 562, No. 7727, 2018, pp. 414–418.

[2] Seale, M., Zhdanov, O., Soons, M. B., Cummins, C., Kroll, E., Blatt, M. R., Zare-Behtash, H., Busse, A., Mastropaolo, E., Bullock, J. M., Viola, I. M., and Nakayama, N., "Environmental morphing enables informed dispersal of the dandelion diaspore," *eLife*, Vol. 11, nov 2022, pp. e81962.

37th Scottish Fluid Mechanics Meeting
Two-phase blood flow downstream of a 3D bifurcation.

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

29th May 2024

Abstract

A mechanistic link between haemorheology and disease pathogenesis such as ischemia or vascular diseases is poorly understood due to the small size of microvessels and their intrinsic complexity[1]. Studies of microcirculation have reported that a formation of a cell-free layer (CFL) due to the movement of red blood cells (RBCs) affects the velocity profiles and wall shear stress (WSS) distribution. This study investigates computationally the three-dimensional (3D) haemodynamics and the CFL development in a symmetric Y-shaped bifurcation where both RBC and CFL phases were taken into account. The bifurcation with a parent diameter of 20 μm was constructed and meshed with polyhedral and prismatic elements in Star-ccm+ (Siemens). A two-phase model consisting of a particulate RBC phase dispersed in a CFL phase was developed utilizing the Eulerian Multiphase (EMP) method. Each phase had its own rheological properties relating to the parent vessel diameter and was governed by the Navier-Stokes equations. The model was simulated under steady and time-dependent flow conditions with an implicit unsteady solver. The data were analysed at a physical time of 1.1 seconds. The results showed that the CFL phase surrounded the RBC phase in a concentric annulus shape in the parent vessel. At the entrance of the bifurcation, the RBC phase was shifted towards the inner wall (IW) of the bifurcation, while the CFL phase was larger near the outer wall (OW). The CFL fluid returned to a concentric annular shape at a long distance downstream of the junction. The development of the symmetric CFL recovery occurred at variable lengths downstream of the junction, depending on the shear rate and branch flow split ratios between the daughter vessels. Moreover, the velocity profiles depended on the asymmetric CFL with the maximum velocity being skewed towards the enlarged CFL at the OW. This flow behaviour has been previously observed in animal arterioles[2]. The CFL eccentricity had also an effect on the haematocrit and WSS variation. Further studies are required to confirm these results in realistic vessels and expand for asymmetric bifurcations into more complex microvascular networks.

References

[1] Alexy et al, “Physical properties of blood and their relationship to clinical conditions”, *Front Physiol,* 13:906768, 2022.
[2] Ng et al, “Symmetry recovery of cell-free layer after bifurcations of small arterioles in reduced flow conditions: effect of rBC aggregation”, *American Journal of Physiology-Heart and Circulatory Physiology*, 311(2), 2016.

37^{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

27^{th} May 2024

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.

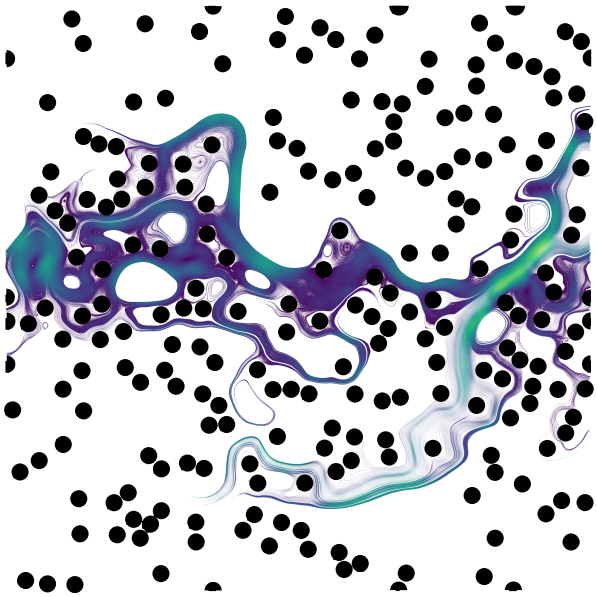
37^{th} Scottish Fluid Mechanics Meeting
Active Darcy's Law

Ryan R. Keogh, Timofey Kozhukhov, Kristian Thijssen and Tyler N. Shendruk
School of Physics and Astronomy, University of Edinburgh
James Clerk Maxwell Building, Peter Guthrie Tait Road, EH9 3FD

27^{th} May 2024

Abstract

Bacterial swarms exhibit active turbulence in vacant spaces; however, many naturally occurring examples of collective bacterial motion arise in heterogeneous and crowded environments, such as in porous soils. On one hand, confinement favours coherent motion; on the other, heterogeneity disorders flow. We study the cooperative effect of disorderly active flows on pressure-driven transport through porous materials of fixed obstacles.[1] While purely active flows average to zero flux, we numerically show hybrid active/pressure-driven flows display greater drift than purely pressure-driven fluids. We find that activity positively enhances global drift, even in the active turbulence limit. This effect endures as the number density of obstacles is increased, resulting in undiminished kinetic energy despite increased dissipative drag—suggesting active flows autonomously fill the available porous length scales. However, we find this enhancement is non-monotonic with activity, leading to an optimal activity to maximise flow rate. We conclude that the flux of active fluids through porous media are described by a modified Darcy's law, which we understand through incorporating the active contribution into an active Darcy's law, which may serve to help understand anomalous transport of swarming in porous media.



References

[1] Keogh, R., Kozhukhov, T., Thijssen, K., and Shendruk, T., "Active Darcy's Law," *Physical Review Letters*, 2024, in press.

37th Scottish Fluid Mechanics Meeting

3D roughness effects on the aerodynamics of the outboard section of a wind turbine blade

Wasina Preamsakul, Oleksandr Zhdanov, Angela Busse
James Watt School of Engineering, University of Glasgow
Glasgow G12 8QQ, UK

29th May 2024

Abstract

In 2020, wind energy generation accounted for 24% of the total electricity generation in the United Kingdom, significantly contributing to reducing greenhouse gas emissions.[1] Surface roughness can degrade the efficiency of wind turbines, for example, leading edge erosion is widely known to affect the aerodynamic performance of wind turbine blades. Other forms of roughness, such as fouling and contamination, can also significantly affect the blades' aerodynamic characteristics.

The current study investigates the influence of roughness wavelength and amplitude on the roughness-induced performance degradation of a typical aerofoil used for the outboard section of a wind turbine blade (DU96-W-180). The roughness pattern is represented by the so-called egg-carton roughness which is generated by the combination of sine waves in the chordwise and spanwise direction of the aerofoil. The roughness is applied towards the leading edge on the suction side of the aerofoil from 0.037 to 0.132 chord (c), occupying 10% of the upper surface area.

The effects of the roughness pattern wavelength and amplitude on the aerodynamic performance are investigated using 3D Reynolds-averaged Navier-Stokes simulations using the CFD software STAR-CCM+. Simulations are conducted for a range of angles of attack at Reynolds number $Re = 1.5 \times 10^6$. The change in the aerodynamic performance is found to be dependent on roughness amplitude but can also be influenced by the roughness wavelength. There are also significant differences on the aerofoil's sensitivity to roughness amplitude and wavelength between the current 3D simulations and earlier 2D results. In the next stage, the influence of anisotropic patterns will be investigated by varying the ratio of the chordwise to the spanwise wavelength for the same blade section.

References

[1] Thomas, G.N., Wind energy in the UK: June 2021, Wind energy in the UK - Office for National Statistics. (2021) Available at: https://www.ons.gov.uk/economy/environmentalaccounts/articles/windenergyintheuk/june2021 (Accessed 26.4.2024).

37^{th} Scottish Fluid Mechanics Meeting

Turbulent Hydrogen Flames at High Pressure

William Lauder, Sofiane Al-Kassar, Geveen Arumapperuma, Antonio Attili
School of Engineering, Institute for Multiscale Thermofluid,
The University of Edinburgh, Edinburgh EH8 3JL, UK

29^{th} May 2024

Abstract

Premixed hydrogen flames are susceptible to thermodiffusive instabilities, which manifest themselves with superadiabatic temperatures and alternating regions of enhanced reactivity and extinction in highly curved regions of the flame surface. They lead to complex fractal patterns and up to a five-fold increase of flame speed in laminar conditions;[1] in the turbulent regime, they interact synergistically with the flow field and cause even larger effects.[2]

In this work, the effect of elevated pressure and temperature on thermodiffusive instabilities in turbulent hydrogen flames is investigated using large-scale Direct Numerical Simulations (DNS). Figure 1 shows a 2D slice of the temperature field obtained in the 3D DNS at high pressure, normalised by the maximum temperature in a 1D laminar freely-propagating flame at the same conditions of the 3D DNS. To investigate the effect of pressure, the results are compared with available data for a low-pressure case.[2] It is found that thermodiffusive instabilities are enhanced at elevated pressures. For example, the relative increase of the average mixture fraction in the turbulent flames compared to the mixture fraction in a laminar 1D flame, which is a reliable measure of thermodiffusive effects, was found to be 56% in the high-pressure case in comparison to 39% at low pressure. These observations suggest that the synergistic interactions between thermodiffusive instabilities and turbulence is enhanced at high-pressure conditions.

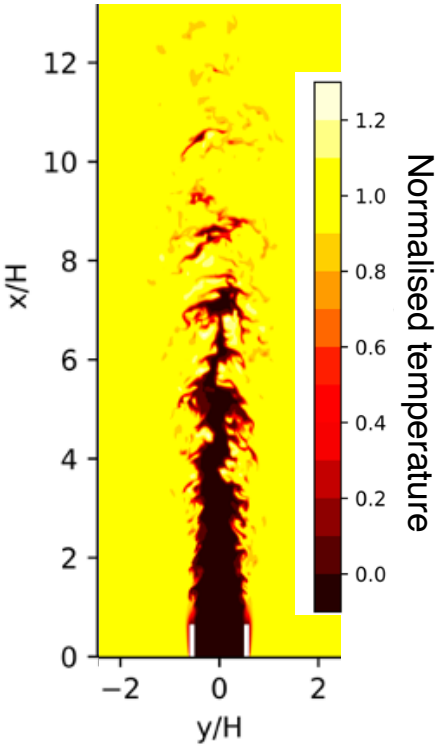


Figure 1: Temperature field in a 2D slice of the 3D DNS.

References

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

[2] Berger, L., Attili, A., and Pitsch, H., *Comb. Flame*, Vol. 244, 2022, pp. 112254.

37th Scottish Fluid Mechanics Meeting

The interaction of waves with suspended seaweeds: wave attenuation and in-canopy hydrodynamics

Xinyi Zhang, Dominic van der A, Tom O'Donoghue
School of Engineering, University of Aberdeen
King's College, Aberdeen, AB24 3FX

27th May 2024

Abstract

Cultivated seaweeds play an important role in the aquaculture industry and have the potential to serve as a nature-based coastal protection measure. With the global expansion of seaweed farms and the need for sustainable coastal defences, researchers are interested in exploring their wave attenuation capabilities. Being suspended in the water column, seaweed farms have the potential to attenuate wave energy more effectively than benthic vegetation.

Understanding the mechanics of wave attenuation across seaweed farms requires knowledge of the in-canopy hydrodynamics (mean and turbulent velocities), which are also critical in the survival and productivity of cultivated seaweeds. While much attention has been paid to wave attenuation of benthic vegetation, few studies have focused on suspended seaweed.[1] Therefore, this study aims to investigate the wave attenuation potential of suspended seaweeds and their in-canopy hydrodynamics in detail.

Laboratory experiments will be conducted at the University of Aberdeen in a 20 m long, 0.45 m wide, 0.7 m deep wave flume. Regular and irregular waves will propagate over a model seaweed canopy, and the wave decay will be measured with wave gauges positioned along the section. Additionally, Laser Doppler Anemometry (LDA) will be used to measure in-canopy velocities. Based on the obtained data, the wave attenuation effect of farm parameters (size, seaweed density, suspension height) and seaweed properties (morphology, flexibility) will be identified for a wide range of incident wave conditions. The in-canopy mean and turbulent velocities will be revealed. Furthermore, we aim to develop semi-empirical formulae to predict wave attenuation of seaweed farms in realistic scenarios to help design new seaweed farms.

The poster will present the project's experimental design, preliminary results, and a plan for further work.

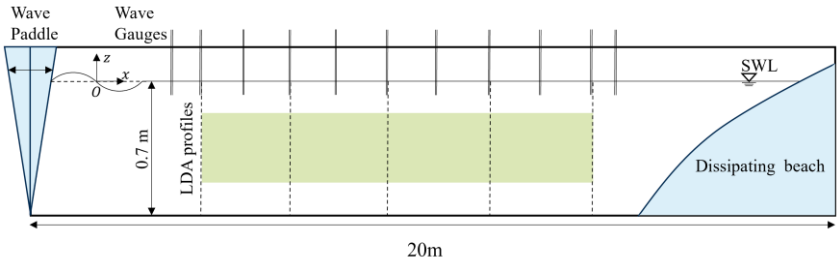


Figure 1. Schematic of the experimental setup, the dashed vertical lines are LDA profiles, the solid vertical lines denote wave gauges, and the green shaded area is the seaweed canopy.

References

[1] Zhu et al. "Wave attenuation by suspended canopies with cultivated kelp (Saccharina latissima)," *Coastal Engineering,* Vol. 168, 2024, 103947.

37th Scottish Fluid Mechanics Meeting
Decoupling Heat Transfer Mechanisms of Micrometre Droplets during Steady State Phase-Change

Xuecong Wang, Nenad Miljkovic, Daniel Orejon
School of Engineering, University of Edinburgh
Old College, South Bridge, Edinburgh EH8 9YL

27th May 2024

Abstract

Wetting and evaporation of micrometre droplets are common phenomena in daily life and influence a wide range of industrial applications, including the printing of complex fluids, spray cooling and DNA analysis.[1] While a plethora of fundamental studies have been carried out on addressing the heat and mass transfer mechanisms during wetting and evaporation phase-change at the droplet scale the complete coupling of the effect of substrate on which the droplets are deposited, ambient around the droplets, and properties of the fluid, is still not fully addressed. In particular, very little is known and quantified in terms of temperature distributions within the solid substrate, the droplet and the ambient, for micrometre sized droplets. While the temperature distribution around micrometre evaporating droplets has been recently uncovered experimentally for droplets evaporating in steady state on insulating and non-insulating surfaces[2], details on how the heat is transferred or removed from the surface are still lacking. This study makes use of a micro goniometer, a zoom lens and a thermocouple, to produce, visualise and quantify the heat removed from the thermocouple, respectively, for droplets ranging from 10 to 500 microns evaporating in steady state. By comprehensively considering the heat of conduction within the droplet, the substrate, and the ambient, results and analysis anticipates a better understanding of the energy flow and heat transfer mechanism for micrometre sized droplets.

References

[1] Brutin, D. and Starov, V., "Recent advances in droplet wetting and evaporation," *Chemical Society Reviews,* Vol. 47, No. 2, 2018, pp. 558-585.
[2] Mousa, M. H., Günay, A.A., Orejon, D. Khodakarami, S., Nawaz, K., Miljkovic, N., "Gas-Phase Temperature Mapping of Evaporating Microdroplets," *ACS Appl. Mater. Interfaces*, Vol. 13, No. 13, 2021, pp. 15925–15938.

37th Scottish Fluid Mechanics Meeting

Exploring Critical size and temperature of size effect: Insights into Adsorption and Diffusion in Nanometer-Scale Confinement

Xueling Zhang[1,2], Qiang Ye[1], Yingfang Zhou[2]

[1.]*School of Energy and Power Engineering, Zhengzhou University of Light Industry, Zhengzhou 450002, China;*

[2.]*School of Engineering, University of Aberdeen， Aberdeen, AB24 3UE.*

27th May 2024

Abstract :

In the realm of confined domains, factors such as size and temperature play a crucial role in influencing the adsorption and diffusion characteristics of fluids. However, there is currently a lack of analysis regarding the combined impact of size effects and temperature. Specifically, the elusive concept of whether a critical size and temperature exists for size effects remains unexplored. In this study, molecular dynamics simulations were employed to investigate the n-heptane adsorption and diffusion characteristics of hydroxylated quartz in nano-confinement, the analysis elucidated the effects of size and temperature on the adsorption and diffusion of fluids, clarifying the critical temperature at which size effects manifest.

The results revealed that the adsorption of n-heptane in quartz nanoscale pores exhibits a transition at 6 nm. When the pore size is ⩾6 nm, shale oil coexists in both free and adsorbed states, with the peak of the first adsorption layer remaining unchanged with pore size, while the peak and amount of adsorption gradually decrease with increasing temperature. When the pore size is <6 nm, n-heptane exists entirely in the adsorbed state, with the interaction energy between n-heptane and quartz surface showing a decreasing trend as the pore size decreases due to the overlapping effect of both walls, leading to a gradual reduction in adsorption. Below 303K, the peak of the first adsorption layer increases with decreasing pore size, reflecting the size overlap effect. Above 323K, due to the decrease in adsorption energy, the size effect weakens, and the density of the first adsorption layer no longer exhibits a significant decreasing trend.

Regarding diffusion coefficients, when the pore size is >6 nm, both total diffusion coefficients and those perpendicular to the pore wall (Z direction) increase with increasing pore size and temperature, while those in the X-Y plane only increase with temperature, with no clear trend with pore size. When the pore size is <6 nm, the Z-direction diffusion coefficient rapidly increases from zero with increasing scale, with little effect from temperature; both total diffusion coefficients and those parallel to the pore wall (X-Y plane) decrease rapidly with decreasing pore size at low temperatures (<303K), indicating a significant size effect, while the trend with pore size becomes less pronounced above 323K due to the influence of molecular thermal motion, diminishing the size effect.

This study, from the perspective of molecular dynamics, reveals a new viewpoint that both size and temperature effects have important influences on adsorption and diffusion, the 6 nm is the critical size of the size effect, under low temperatures (<303K), the size on the adsorption or diffusion is more prominent, while at

temperatures above 323K, the thermal motion greatly weakens the size effect, providing a theoretical basis for the research of nanoconfined fluids.

References

[1] HUGHES J D. A reality check on the shale revolution [J]. Nature, 2013, 494(7437): 307-8.

[2] LIANG S, WANG J-M, LIU Y-K, et al. Oil Occurrence States in Shale Mixed Inorganic Matter Nanopores [J]. Frontiers in Earth Science, 2022, 9: 1456.

[3] PEREZ F, DEVEGOWDA D. Spatial distribution of reservoir fluids in mature kerogen using molecular simulations [J]. Fuel, 2019, 235: 448-59.

[4] VISHAL V, CHANDRA D, BAHADUR J, et al. Interpreting Pore Dimensions in Gas Shales Using a Combination of SEM Imaging, Small-Angle Neutron Scattering, and Low-Pressure Gas Adsorption [J]. Energy & Fuels, 2019, 33(6): 4835-48.

[5] XU H W. Probing nanopore structure and confined fluid behavior in shale matrix: A review on small-angle neutron scattering studies [J]. International Journal of Coal Geology, 2020, 217: 103325.

[6] HUANG L, ZHOU W, XU H, et al. Dynamic fluid states in organic-inorganic nanocomposite: Implications for shale gas recovery and CO2 sequestration [J]. Chemical Engineering Journal, 2021, 411: 128423.

[7] XU J L, WANG R T, ZAN L. Shale oil occurrence and slit medium coupling based on a molecular dynamics simulation [J]. Geoenergy Sci Eng, 2023, 220: 15.

[8] CHEN C, HU W F, SUN J Y, et al. CH4 adsorption and diffusion in shale pores from molecular simulation and a model for CH4 adsorption in shale matrix [J]. Int J Heat Mass Transf, 2019, 141: 367-78.

[9] FANG Y J, LI Z X, YANG E R, et al. Molecular Dynamics Simulation Study on the Occurrence of Shale Oil in Hybrid Nanopores [J]. Molecules, 2024, 29(2): 14.

[10] SUN L H, JIA N H, FENG C, et al. Exploration of Oil/Water/Gas Occurrence State in Shale Reservoir by Molecular Dynamics Simulation [J]. Energies, 2023, 16(21): 14.

[11] FEI J S, WANG M, LI J B, et al. Molecular Dynamics Simulation of Adsorption and Absorption Behavior of Shale Oil in Realistic Kerogen Slits [J]. Energy Fuels, 2023: 18.

[12] Li X, Wang P, Yan Z, et al. The miscible behaviors of C_3H_8/C8H18 (C7H17N) system in nanoslits: Effects of pore size and rock surface wettability[J]. Chemical Engineering Journal, 2022, 431: 133988.

[13] ZHAN S, SU Y, JIN Z, et al. Study of liquid-liquid two-phase flow in hydrophilic nanochannels by molecular simulations and theoretical modeling [J]. Chemical Engineering Journal, 2020, 395: 125053.

[14] CAO Z, JIANG H, ZENG J, et al. Nanoscale liquid hydrocarbon adsorption on clay minerals: A molecular dynamics simulation of shale oils [J]. Chemical Engineering Journal, 2021, 420: 127578.

37th Scottish Fluid Mechanics Meeting

Fluid-Solid Flow Transition in Mixed (Sand-Mud) Sediment: Enhanced Modelling of Sedimentation in Estuarine and Coastal Waters

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

27th May 2024

Abstract

Sedimentation processes in natural estuarine or coastal sediment systems are normally divided into two regimes: (1) the hindered settling regime, where volumetric concentrations of polydisperse settling particles generate return flows and interactions through the faster-settling particles (e.g. sands) affecting the settling rates of other slower-settling particles (e.g. clays, muds)[1]; and (2) the consolidation regime, where the deposited sediment bed layers are compressed by the overburden at the top of the settling unit due to gravity[1]. Understanding of sedimentation processes has significant implications for improving predictions of mixed (i.e. sand-mud) sediment transport dynamics, as particularly crucial to maintaining navigation channels, ports and harbours, understanding morphological evolution in affected coastlines, and preserving and enhancing sensitive habitats (e.g. mudflats).

In the simplest case, we can consider a well-mixed monodispersed suspension (i.e. cohesive clay suspension) settling within a vertical column under the assumption that (1) column side wall effects on sedimentation are negligible; (2) the two-phase (water-clay) suspension settles and compacts vertically in one-dimension; and (3) the initial clay concentration is lower than its gelling point ($\phi_0 < \phi_g$) at which a bed deposit layer structure begins to form. The full sedimentation process for a monodisperse suspension is shown schematically in *Figure 1*. At the initial time, the clay particle concentration is distributed evenly within the column [*Figure 1* (i)]. Throughout the sedimentation process [*Figure 1*: (ii)], the column region in which the hindered settling regime occurs progressively reduces in thickness as the interface separating the supernatant (clear water) layer from the settling suspension reduces in elevation, while the interface separating the hindered settling and consolidation regimes gradually increases in elevation from the bottom of the column. This increasing interfacial height, due to the deposition of clay particles from the hindered settling regime, is to some extent counteracted by compaction processes in the consolidation regime. Once the sedimentation processes has finished, the consolidated bed layer structure has a single interface separating from the upper clear water layer and the compacted clay deposit at the bottom [*Figure 1* (iii)]. The spatial and temporal transitions between these hindered settling and consolidation regimes are not currently well-understood or modelled, particularly when considering the sedimentation behaviour of bi-disperse and poly-disperse mixtures of cohesive (i.e. mud) and non-cohesive (i.e. sand) sediments. The aim of this project is therefore to develop a new integrated hindered settling – compaction model for multifractional sediments, accounting for polydisperse hindered settling and flow-driven compaction. This model development is informed and supported by laboratory experiments, conducted in a bespoke settling column, with the aim of improving the

physical understanding and representation of hindered settling and compaction, and particularly the crucial fluid-solid regime transitions, for natural sediment mixtures during the sedimentation process.

Preliminary comparison of the monodispersed compaction model predictions and experimental measurements are presented in *Figure 2* for selected parametric runs. It is clearly shown that the model predictions correlate well with experimental measurement of the upper fluid-suspension interface for runs with lower initial settling heights ($h_0 \leq 0.165m$). However, the model slightly underpredicts the deposit height variations during the consolidation regime, including the final consolidated height, for runs with higher initial settling heights ($h_0 \geq 0.255m$). The main reason for this discrepancy is that characteristic physical parameters [i.e. gel point ϕ_g and permeability $k(\phi)$] of the clay suspension are calibrated based on the measurements with relatively low initial heights, meaning that the gravitational loading on the evolving layer structure is less than for runs with higher initial settling height. This may suggest that the calibrated linear behaviour between yield stress $\sigma_y'(\phi)$ and concentration ϕ near the gel point is in fact only valis in the low-ϕ limit, with a non-linear relationship [$\sigma_y'(\phi) \sim \phi^2$] valid for higher solid fractional concentrations ϕ, the relation for which can be referenced from previous studies[2].

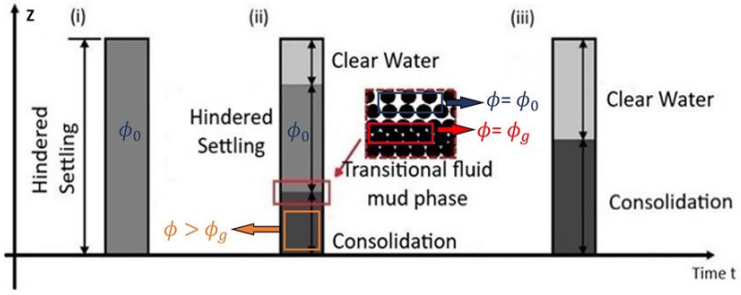


Figure 1 Geometry arrangements of well-mixed monodisperse suspension (low ϕ_0) settling with time lapses driven by gravity: (i) beginning of the settling (ii) intermediate state of the sedimentation process (iii) final deposit structure.

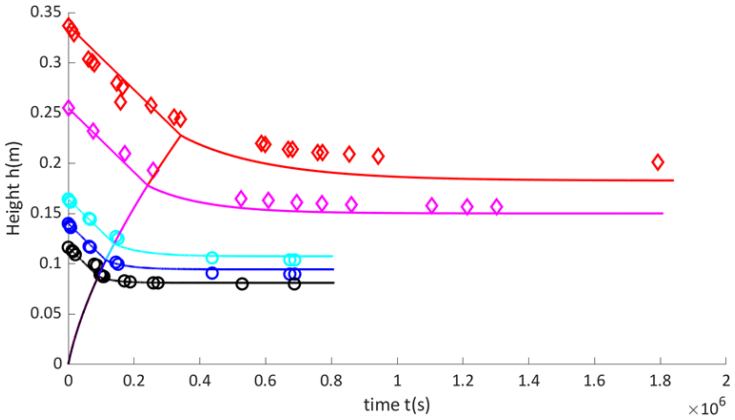


Figure 2 Model predictions (solid line) and experimental measurements (hollow points) of the fluid-sediment interface height over time using a suspended mixture of clays settling at different initial heights ($\phi_0 = 0.094$)

References

[1] TE SLAA, S., VAN MAREN, D. S., HE, Q. & WINTERWERP, J. C. (2015). Hindered Settling of Silt, *Journal of Hydraulic Engineering,* Vol. 141, No. 9.

[2] PATERSON, D. T., EAVES, T. S., HEWITT, D. R., BALMFORTH, N. J. & MARTINEZ, D. M. (2019). Flow-driven compaction of a fibrous porous medium," *Physical Review Fluids,* 4, Article 074306.

37th Scottish Fluid Mechanics Meeting
Active fluid-induced dynamics of passive polymers

Zahra Valei [1], Tyler N. Shendruk[1]
[1]School of Physics and Astronomy, University of Edinburgh
Edinburgh, EH9 3FD, United Kingdom
29th May 2024

Abstract
Anisotropic and rheologically complex materials are ubiquitous in living systems, in which biochemically fuelled activity can give rise to intriguing nonequilibrium effects that act on small solutes, whole organelles, and biomolecular polymers. Of these, passive polymers embedded in active liquid crystalline solvents are particularly interesting because of the interplay between anisotropy, activity and internal degrees of freedom. While previous studies have either focused on active polymers embedded in passive fluids [1] or passive polymers in nonequilibrium thermal baths [2], this talk will report the first study of a passive polymer in active nematic flows. This work reports the first study of a passive polymer in active nematic flows. To study the conformation and dynamics of individual flexible polymers coupled to the velocity field of 2D extensile active turbulence, we employ a hybrid Multi-Particle Collision Dynamics [3] and Molecular Dynamics [4] simulation. We find the diffusivity of polymers enhances considerably with activity. It increases with activity up to the onset of turbulence, at which point it saturates. This is due to the competition between active forcing and conformational changes. We show how the characteristic length scale of the fluid dictates the conformation of the polymers. Polymers longer than active length scale are stretched, while short polymers are compressed. This demonstrates how activity represents a pathway by which biological systems can control polymer properties.

References
[1] Martín-Gómez, Aitor, Thomas Eisenstecken, Gerhard Gompper, and Roland G. Winkler. "Active Brownian filaments with hydrodynamic interactions: conformations and dynamics." Soft Matter 15, no. 19 (2019).
[2] Harder, Joseph, Chantal Valeriani, and Angelo Cacciuto. "Activity-induced collapse and re-expansion of rigid polymers." Physical Review E 90, no. 6 (2014).
[3] Kozhukhov, Timofey, and Tyler N. Shendruk. "Mesoscopic simulations of active nematics." Science Advances 8, no. 34 (2022).
[4] Boldon, Lauren, Fallon Laliberte, and Li Liu. "Review of the fundamental theories behind small angle X-ray scattering, molecular dynamics simulations, and relevant integrated application." Nano Reviews 6, no. 1 (2015).