University of Dundee

School of Science and Engineering

32nd Scottish Fluid Mechanics Meeting

Book of Abstracts

University of Dundee

Thursday May 30th, 2019

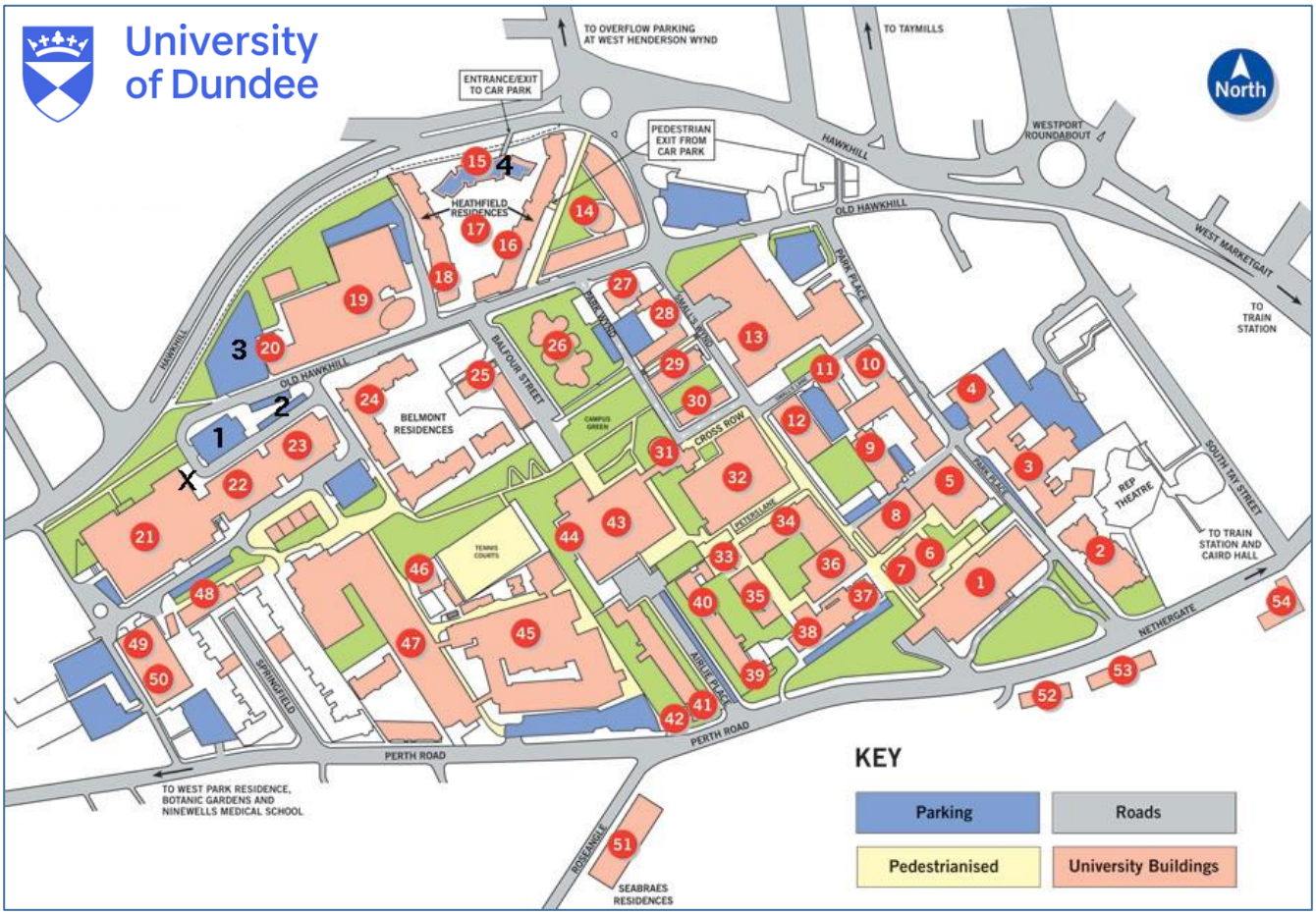
DANTEC DYNAMICS

Contents

Introduction and Welcome

Welcome to the University of Dundee and the 32nd Scottish Fluid Mechanics Meeting. We thank you for your participation in this year's meeting, which promises as always to showcase the strength and depth of Fluid Mechanics research in Scotland. We were delighted to receive a total of 58 abstracts for the meeting and have tried to offer everyone a chance to present their research either as an oral or poster presentation. As such, we have a wide and varied range of talks and posters from researchers across Scotland (and further afield) and we hope you enjoy the programme we have put together.

The meeting will take place in the **Dalhousie Building** on the University of Dundee's City Campus. This building is listed as **number 14** on the campus map below.



Sponsorship of this year's meeting by **Dantec Dynamics** is gratefully acknowledged. Graham Hassall will be on hand throughout the day to answer any of your questions regarding Dantec measurement systems.

During the meeting, if you have any questions, please approach a member of the local organising committee: **Alan Cuthbertson, Peter Davies, Masoud Hayatdavoodi, Yong Sung Park, David Pontin, Mohamed Salim.**

Programme Timetable

Time	Presenter	Title
0900 – 0945	**Registration and coffee**	
0945 – 1000	**Introductions and welcome (Peter Davies)**	
Session 1	Chair: Yong Sung Park	
1000 – 1015	Stuart Cameron	Drag forces on sediment particles in open-channel flows: effects of very large scale motions and implications for particle entrainment
1015 – 1030	Masoud Hayatdavoodi	Hydroelastic response of very large floating bodies to nonlinear waves
1030 – 1045	Francesco Neglia	Computational fluid dynamics simulations of experimental and natural granular flows: first insights on the flow-wall interaction dynamics
1045 – 1100	Jiaqi Liu	Bore impact on coastal structures
1100 – 1115	Jean Reinaud	Quasi-geostrophic vortex arrays and Jupiter's polar atmosphere
1115 – 1145	**Coffee break and poster session**	
Session 2	Chair: Stephen Wilson	
1145 – 1200	Hannah D'Ambrosio	Evaporation of a sessile droplet in a shallow well
1200 – 1215	Vasileios-Martin Nikiforidis	Modeling ice formation on surfaces using molecular dynamics simulations
1215 – 1230	Feargus Schofield	A 2D model for the evaporation of a pair of shallow liquid ridges
1230 – 1245	Ewan Ryecroft	The Effects of Complex Rheology on the Swimming Velocities of a Flagellated Alga
1245 – 1300	Tom O'Donoghue	Jason Reese
1300 – 1415	**Lunch, poster session and group photograph**	
Session 3	Chair: Vladimir Nikora	
1415 – 1430	Shuji Otomo	On the Lift and Vortices of an Asymmetrically Pitching Foil
1430 – 1445	James Cairns	Optimisation of a Ship Wind-Assisted Propulsion System
1445 – 1500	David Pickles	Experimental simulation of the Vortex Ring State
1500 – 1515	Mamdud Hossain	Comparison of Turbulence modelling in Hydrocyclone Simulations
1515 – 1530	Angel Zarev	Experimental investigation of a two-bladed propeller at yaw
1530 – 1600	**Coffee break and poster session**	
Session 4	Chair: David Pontin	
1600 – 1615	Simon Candelaresi	Relaxation of Vortex Braids
1615 – 1630	Luke Fulford	Modelling Direct Brain Cooling for Ischemic Stroke
1630 – 1645	Yatin Suri	Effect of Fracture Roughness on proppant transport in hydraulic fractures using CFD-DEM model
1645 – 1700	Rohan Vernekar	Red blood cell 'choking' of microfluidic pillar array geometries
1700 – 1715	Peter Szabo	Thermomagnetic convection in thermally heated baroclinic annulus and spherical shell
1715 – 1730	**Concluding remarks and goodbyes (Alan Cuthbertson)**	

Oral Presentation Abstracts (in order of presentation)

Drag forces on sediment particles in open-channel flows: effects of very large scale motions and implications for particle entrainment

S. M. Cameron (s.cameron@abdn.ac.uk) and V. I. Nikora

School of Engineering, University of Aberdeen

The estimation of drag forces acting on aquatic surfaces is of interest in many areas of hydraulic and eco-hydraulic engineering. There remains, however, a shortage of data to evaluate mechanisms responsible for drag force generation and to test and refine models coupling velocity and drag force fluctuations. The aim of this study is to address this shortage with comprehensive measurements and analysis of the instantaneous drag forces acting on roughness elements combined with synchronous measurements of the surrounding velocity field. Mechanisms responsible for the entrainment of sediment particles are also explored by computing the ensemble average velocity field at the instant of entrainment of spherical particles.

Stereoscopic particle image velocimetry was used in a transverse-vertical plane passing through the centre of a 'target' particle. The protrusion of the target particle, which was equipped with a drag force sensor, was systematically varied between 0 and 0.5 particle diameters relative to the surrounding hexagonally packed spheres. Long duration measurements (up to 90 minutes) were conducted in order to provide highly resolved spectra covering the full range of possible scales in the flow. Additional experiments with lightweight mobile particles captured the velocity field at the instant of particle motion for 25 repeated entrainment events across a range of particle protrusions and flow depths.

The results showed that the pre-multiplied drag force spectra have a bimodal shape characterised by a low frequency peak and a high frequency peak. With increasing particle protrusion, the drag force variance becomes increasingly dominated by the contributions from the low frequency process. The low frequency spectral peak is associated with high coherence between the streamwise velocity component and the drag force and likely results from the action of very large scale motions (VLSMs) which extend up to 50 flow depths in the streamwise direction. High frequency drag force fluctuations, previously thought to be related to wake turbulence, however, do not show appreciable coherence with point velocity measurements around the particle. Instead, we propose that the high frequency region of the drag force spectra is dominated by the action of pressure spatial gradients in the overlying turbulent flow. Ensemble averaged velocity fields at the instant of particle entrainment confirm that VLSMs contribute significantly to the entrainment and transport of sediments.

Hydroelastic response of very large floating bodies to nonlinear waves

Masoud Hayatdavoodi (m.hayatdavoodi@dundee.ac.uk)

School of Science and Engineering, University of Dundee

Design and analysis of Very Large Floating Structures (VLFSs) subject to large waves require a hydroelastic assessment of the wave-structure interaction problem. Similarly, hydroelastic response of large floating ice sheets must be studied to determine the wave- ice dynamics at marginal ice zones. These problems have remained significant challenges to engineers and researchers due to the very large size of the body and difficulties in scaling simultaneously the hydrodynamic and structural prosperities. Efficient and robust nonlinear hydroelastic models are required to predict accurately the interaction of large waves with very large floating bodies. However, most of the theoretical methods on the hydroelasticity of very large floating bodies are developed within the linear wave theory framework. Theoretical approaches that can consider the global hydroelastic response of a VLFS to nonlinear waves are very limited due to the complexities of the problem.

This presentation is concerned with the interaction of nonlinear waves of solitary and cnoidal types with deformable floating bodies on water surface. The hydroelastic problem is studied by directly coupling the structure with the fluid, by use of the Level I Green-Naghdi theory for the fluid motion and the Kirchhoff thin plate theory for the body, whether VLFS or large floating ice sheets. Equations of the coupled fluid-structure system are solved by use of the finite-difference method in two-dimensions. In general, there are N number of the deformable bodies with arbitrary sizes and properties. Comparisons show close agreement between the numerical model and existing laboratory experiments. Results include wave diffraction, wave-induced loads, and deformation of the bodies due to solitary and cnoidal waves.

Computational fluid dynamics simulations of experimental and natural granular flows: first insights on the flow-wall interaction dynamics

Francesco Neglia[1,2,3], Fabio Dioguardi[2] (fabiod@bgs.ac.uk), Roberto Sulpizio[1] and Raffaella Ocone[3]

[1] University of Bari, Dipartimento di Scienze della Geoambientali, Bari, Italy
[2] British Geological Survey, The Lyell Centre, Edinburgh
[3] Heriot-Watt University, Institute of Chemical Science, Edinburgh

In the last decade, granular flows modelling has become one of the main tools used to replicate past events in order to investigate potential hazard linked to future phenomena and to better understand the dynamics of volcanic granular flows. The research on behaviour of volcanic granular flows is one of the main topics in present day geophysics and volcanology due to the hazard they pose on areas located in the vicinity of the volcanoes and that can be potentially affected by the passage of these flows. Volcanic granular flows (like many pyroclastic density currents, debris avalanches, etc.) can be defined as gravity-driven currents of solid particles in which the particle-particle interaction dominates the motion, in the sense that they are poorly-to-non influenced by effects on the interstitial fluid and cohesion. The interaction with the topography is another major process controlling the dynamics and the evolution of these flows.

The aim of this research is to investigate on the dynamics of dense geophysical flows generated during explosive volcanic eruptions by combining field study on past volcanic flow deposits, large-scale experiments with computational fluid dynamics simulations. We used FLO2D and TITAN2D, two of the most widely used models for modelling natural granular and debris flows for hazard assessment purposes. Both these models solve for the depth-averaged conservation equations of mass and momentum; the multiphase mixture is treated as a homogeneous fluid. In this study, we applied both models to replicate past volcanic flows at Colima volcano (Mexico) and Sarno area (Campania, Italy) and large-scale experiments on granular flows carried out over the last few years in San Luis Potosí (Mexico). Preliminary results show that the model predictions are significantly affected by the employed rheological model and the way in which the interaction between the flow and the local topography is modelled.

Bore impact on coastal structures

Jiaqi Liu (j.o.liu@dundee.ac.uk) and Masoud Hayatdavoodi

School of Science and Engineering, University of Dundee

Bores generated by (i) breaking of a solitary wave, (ii) dam-break, and (iii) initial mound of water and their propagation over horizontal and inclined surfaces are studied by use of theoretical approaches. Calculations are carried out in two and three dimensions, and effect of viscosity is studied by considering various turbulence models. Particular attention is given to the bore impact on horizontal and vertical structures. Three methods are used in this study, namely the Reynolds-Averaged Navier-Stokes equations (RANS), the Green-Naghdi (GN) equations and Saint Venant equations (SV). The governing equations subject to appropriate boundary conditions are solved with various numerical techniques. Results of these models are compared with each other, and with laboratory experiments when available. Discussion is given on the applicability of these methods to the modelling of bore generation, propagation and impact on horizontal and vertical structures, considering assumptions and limitations of each approach. Particular attention is given to the physics of the flow field for some applied and engineering applications. It is found that results of the GN equations are in close agreement with the RANS equations, while the SV equations substantially simplify the solution. Parametric study show that bore behaviour and its impact on costal structures change significantly with the generation mechanism and the downstream water depth.

Quasi-geostrophic vortex arrays and Jupiter's polar atmosphere

Jean Reinaud (jean@mcs.st-and.ac.uk) and David Dritschel

Mathematical Institute, University of St Andrews

Pictures of Jupiter taken by the Juno spacecraft have revealed the presence of regular vortex arrays over the giant gas planet's poles. There is an array of eight vortices surrounding a central vortex over the North Pole, while there is an array of five vortices around a central vortex over the South Pole, see Adriani *et al.* (2018). The dynamics of Jupiter's weather layer is strongly influenced by the stable stratification of the fluid and by the rapid rotation of the planet. Flows developing in such environments are accurately modelled within the quasi-geostrophic theory.

To explain the observations, we study arrays of quasi-geostrophic vortices in mutual equilibrium in a continuously stratified, rapidly rotating fluid. We also address their linear stability. We first model the vortex arrays using point vortices. In the absence of a central vortex, we show that only vortex arrays of $N \leq 5$ point vortices are stable (in contrast with $N \leq 7$ in a two-dimensional flow). Vortex arrays of $N > 5$ vortices are unstable. Vortex arrays of $N \leq 8$ may however be stabilised by the addition of the central, like-signed, vortex of moderate strength. We next determine numerically equilibrium states for arrays of finite-volume vortices. In this case, owing to the ability of finite-volume vortices to deform when subjected to intense strain, the equilibria are unstable if the vortices are close together. There are nonetheless large regions of the parameter space where stable equilibria for the finite-volume vortex arrays exist.

We finally propose the first plausible scenario for the formation of the vortex arrays by studying the stability and the nonlinear evolution of tori of potential vorticity. The tori are unstable and self-organise into arrays of vortices. We show that the destabilisation of a potential vorticity torus can generate robust, persistent 5 and 8-vortex arrays with a central vortex.

References

Adriani, A., Clusters of cyclones encircling Jupiter's poles, *Nature* **555**, 216-219 (2018)

Reinaud, J.N., Three-dimensional quasi-geostrophic vortex equilibria with m-fold symmetry, *J. Fluid Mech.* **863**, 32-59 (2019)

Reinaud, J.N., Dritschel, D.G, The stability and the nonlinear evolution of quasi-geostrophic toroidal vortices, *J. Fluid Mech.* **863**, 60-78 (2019)

Evaporation of a sessile droplet in a shallow well

H.-M. D'Ambrosio[1] (hannahmay.dambrosio@strath.ac.uk), T. Colosimo[2], S. K. Wilson[1], B. R. Duffy[1], C. D. Bain[2], and D. E. Walker[3]

[1] University of Strathclyde
[2] Durham University
[3] Merck Chemicals Ltd, Southampton

The evaporation of sessile droplets occurs in a wide variety of physical contexts, with numerous applications in nature, industry and biology. Examples of practical applications include blood splatter in forensic science, DNA mapping and gene analysis in the medical industry, the spreading of pesticides on leaves, and coating technologies. In particular, the evaporation of droplets plays a key role in the manufacture of Organic Light Emitting Diode (OLED) displays. In this talk I will formulate and analyse a mathematical model for the evolution of a thin droplet undergoing diffusion-limited evaporation on a non-uniform substrate, specifically in an axisymmetric well similar to those encountered in OLED manufacturing applications. I will use the model to describe the evolution of the height profile, contact radius, and hence the volume of the droplet, from its initial configuration until total evaporation. In particular, I will show how the evolution of the droplet is qualitatively dependent on the cross-sectional profile of the well. I will then compare the theoretical predictions of the model with detailed experimental results for the evaporation of droplets in cylindrical wells of various sizes, and show that theory and experiment (and, in particular, the critical times at which the droplet first touches down and at which total evaporation occurs) are in excellent agreement for fluids that conform to the model assumptions.

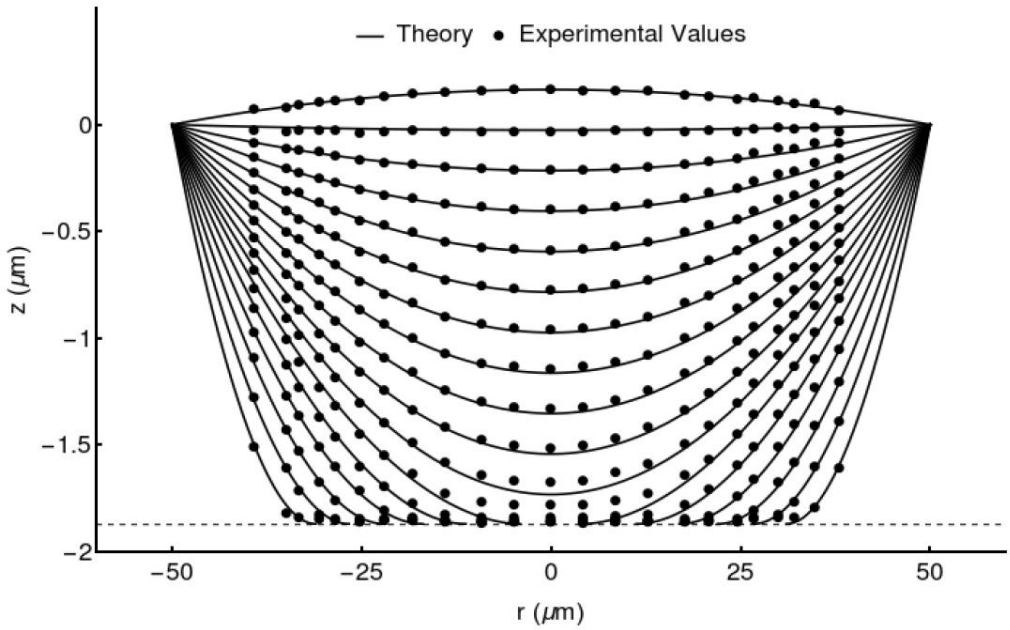


Figure: Comparison of the height profile predicted by the theoretical model with experimental data for a methyl benzoate droplet evaporating in a cylindrical well of depth 1.87 μm and radius 50 μm at equally-spaced times.

Acknowledgements: H.-M.D'A acknowledges the financial support of the United Kingdom Engineering and Physical Sciences Research Council (EPSRC) via EPSRC Doctoral Training Partnership grant EP/R512205/1, the University of Strathclyde, and Merck Chemicals Ltd.

Modeling ice formation on surfaces using molecular dynamics simulations

Vasileios-Martin Nikiforidis (m.nikiforidis@ed.ac.uk), Rohit Pillai and Matthew K. Borg

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

Ice formation is an everyday phenomenon that can negatively impact a variety of industrial applications, like deteriorating aircraft efficiency and safety, and diminishing power production from wind turbines. To mitigate these issues, research in this area has focused on creating coatings that inhibit the formation of ice. To develop icephobic coatings a better understanding is needed of the factors and characteristics of a surface that can inhibit ice formation. Ice formation is a result of two processes, nucleation and crystal growth. Nucleation is a first-order phase transition of water molecules, in liquid phase, that starts to form crystalline clusters. When one of these clusters exceeds a critical size, a new thermodynamic phase starts and leads to the crystal growth of the ice nuclei which were initially formed. Brute-force molecular dynamics (MD) enables us to study the ice formation process at the level of individual molecules, and can help shed light on the parameters that inhibit ice nucleation. However, in reality, brute-force MD simulations of all-atom water are unable to simulate ice nucleation of water molecules on top of surfaces that do not the share the same or similar lattice structure with ice, known as `lattice mismatch'. This constraint limits our ability to study the nucleation process for the majority of metallic and mineral monoatomic surfaces. Typically, this limitation has been overcome by the use of coarse-grained water models, which neglect the Coulombic interactions between water molecules, or using `enhanced sampling' techniques, which artificially inflate the probability of ice nucleation. In this work we present a novel approach that enables us to perform brute-force MD simulations of all-atom water models using smooth metallic surfaces of FCC and BCC crystal structures despite their lattice mismatch. Using this approach, we are able to simulate and compare ice nucleation and crystal growth on a larger variety of surfaces and provide new insights into the formation and growth of ice at the nanoscale.

A 2D model for the evaporation of a pair of shallow liquid ridges

F. G. H. Schofield[1] (feargus.schofield@strath.ac.uk), D. Pritchard[1], A. W. Wray[1], S. K. Wilson[1] and K. Sefiane[2]

[1] Department of Mathematics and Statistics, University of Strathclyde
[2] School of Engineering, University of Edinburgh

Understanding the dynamics of sessile liquid droplet evaporation on solid substrates is critical for many industrial processes, such as ink-jet printing, coating, and spray cooling, as well as for drug delivery systems and chemical spill containment. Consequently, in recent years there has been a rapid growth of scientific interest in all aspects of droplet evaporation, including determining the lifetimes of evaporating droplets. Much of the previous theoretical work has focused on the evaporation of a single droplet. However, there is very little research that captures the physics of the neighbouring evaporating droplets which are present in almost all industrial processes.

In the present work, to provide insight into the 3D problem for a pair of neighbouring droplets, we investigate the equivalent 2D problem for a pair of neighbouring liquid ridges. We use conformal mapping techniques to investigate the 2D problem of an evaporating shallow ridge of liquid. We then extend the single-ridge model to include an identical neighbouring ridge of liquid and obtain analytical expressions for the evaporative flux from the ridges. From this analysis we observe the so-called “shielding effect", in which the atmosphere between the ridges becomes more concentrated with vapour thus dampening the evaporative flux into this region. From the evaporative flux we obtain analytical expressions for the evaporation rates, and hence the lifetimes, of the ridges in various modes of evaporation. In particular, we quantify how decreasing the distance between the neighbouring ridges will increase the magnitude of the shielding effect, leading to longer lifetimes.

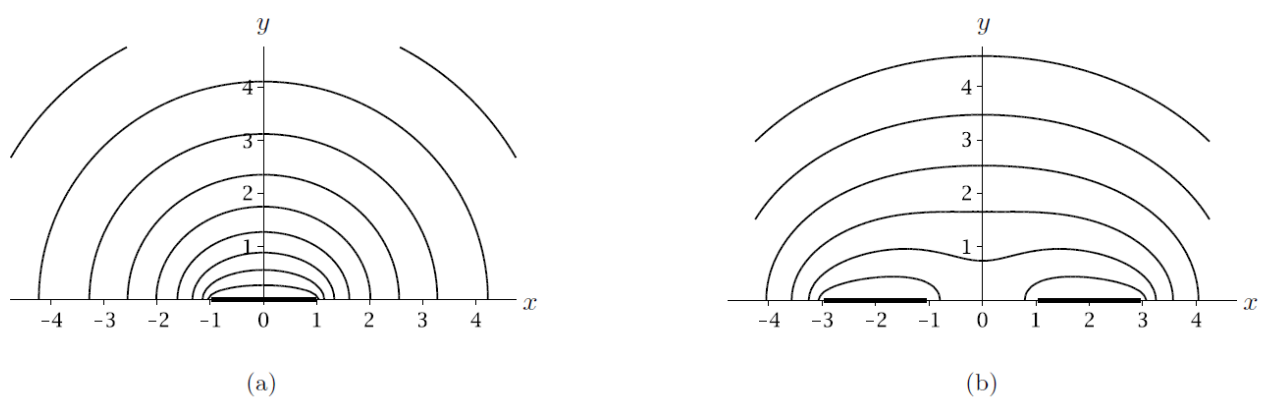


Figure 1: Contour plots of the atmospheric vapour concentration for (a) a single isolated shallow ridge and (b) two identical neighbouring shallow ridges. In both figures the contour lines are in equal increments, with a maximum value on the flat surfaces of the ridges.

Acknowledgements: FGHS acknowledges the financial support of the United Kingdom Engineering and Physical Sciences Research Council (EPSRC) via Doctoral Training Partnership grant EP/N509760/1, the University of Strathclyde, and the University of Edinburgh.

The Effects of Complex Rheology on the Swimming Velocities of a Flagellated Alga

Ewan Rycroft[1] (ewan.rycroft@strath.ac.uk), Mark Haw[2] and Mónica S. N. Oliveira[1]

[1] Department of Mechanical and Aerospace Engineering, University of Strathclyde
[2] Department of Chemical and Process Engineering, University of Strathclyde

The motility and the swimming behaviour of microorganisms is greatly influenced by the interactions with the medium in which they live. An understanding of their swimming dynamics in complex media is key for the enhancement of applications such as artificial swimming, micro-robotics and targeted drug delivery. In this work, we examine the effects of rheological characteristics, such as viscosity, viscoelasticity and shear thinning characteristics, of the medium on the swimming dynamics of a eukaryotic bi-flagellated puller alga, *Dunaliella salina*. We make use of dilute algae suspensions, in which alga-alga interactions are not important. To visualise the algae swimming behaviour, we employ microfluidic chambers of rectangular cross-section and rely on microscopy and image analysis to quantify the swimming kinematics. We observe analogous algae swimming velocities, orientations and beating kinematics in Newtonian (*Ficoll PM400*), viscoelastic (*Polyacrylamide*) and shear thinning (*Xanthan Gum*) solutions across a substantial range of fluid viscosities. In all cases, swimming velocities decrease as the viscosity is increased up to a point where velocities plateau, exhibiting an approximately constant velocity for high viscosity solutions. Observations in the shear thinning fluid allows us to further understand the shear responses experienced by this particular swimmer and approximate the characteristic shear rate. Further, we examine the algae behaviour in a second Newtonian fluid (*Glycerol*). In this case the behaviour is significantly altered, with a more pronounced decrease in swimming velocities as the viscosity is increased and a defined point of non-motility, which was not observed in other cases. We attribute this to cell interaction with the glycerol, changing cell biology and effecting swimming efficiencies.

Jason Reese

Tom O'Donoghue and Ian McEwan

School of Engineering, University of Aberdeen

On the Lift and Vortices of an Asymmetrically Pitching Foil

Shūji Ōtomo[1] (S.Otomo@ed.ac.uk), Karen Mulleners[2], Kiran Ramesh[3] and Ignazio Maria Viola[1]

[1] School of Engineering, Institute for Energy Systems, University of Edinburgh
[2] Ecole Polytechnique Fédérale de Lausanne (EPFL), Switzerland
[3] Aerospace Sciences Division, School of Engineering, University of Glasgow

Research on unsteady aerodynamics has significantly grown in recent years due to its relevance to micro-air vehicles, mechanical swimmers, and flapping-foil energy harvesters. In these applications, a leading-edge vortex (LEV) might occur, resulting in a strongly non-linear relationship between forces and kinematics. We present experimental (force measurement and particle image velocimetry) and theoretical (Theodorsen's theory) studies on unsteady lift generated by a pitching foil and the corresponding dynamics of the LEV. We experimentally investigate the lift generation mechanism of a pitching NACA 0018 aerofoil section. The foil having a chord length of $c = 0.15$ m is exposed in a water flume at free stream velocity of $U_\infty = 0.215$ m/s resulting in a Reynolds number of 3.2×10^4. Time-resolved direct force measurement and two-dimensional particle image velocimetry are performed. We employ smoothed triangular pitching kinematics to have a constant pitch rate for most of the period of oscillation, minimising the effect of pitching acceleration (i.e. non-circulatory forces). The reduced frequency is varied between $0.22 \leq k \leq 0.88$ and pitching amplitude between $4° \leq \theta_0 \leq 64°$. We examine the effect of changing the amount of asymmetry in pitching kinematics. We find that the asymmetric kinematics allows to augment net lift. For the reduced frequencies $k_\xi \gtrsim 0.4$ (k_ξ is the reduced frequency based on the faster part of asymmetric pitching kinematics), the wing experiences its maximum lift at the beginning of the deceleration phase before reaching the maximum angle of attack. This occurs for every pitching amplitude and any asymmetric configuration. These experimental results and those of Theodorsen's theory are in qualitative agreement. The connection between the LEV formation and the lift coefficient is investigated. FIG. 1 presents the phase-averaged lift coefficient C_L and the instantaneous non-dimensional vorticity field $\omega c/U_\infty$ of corresponding points (a-e) as defined in C_L plot.

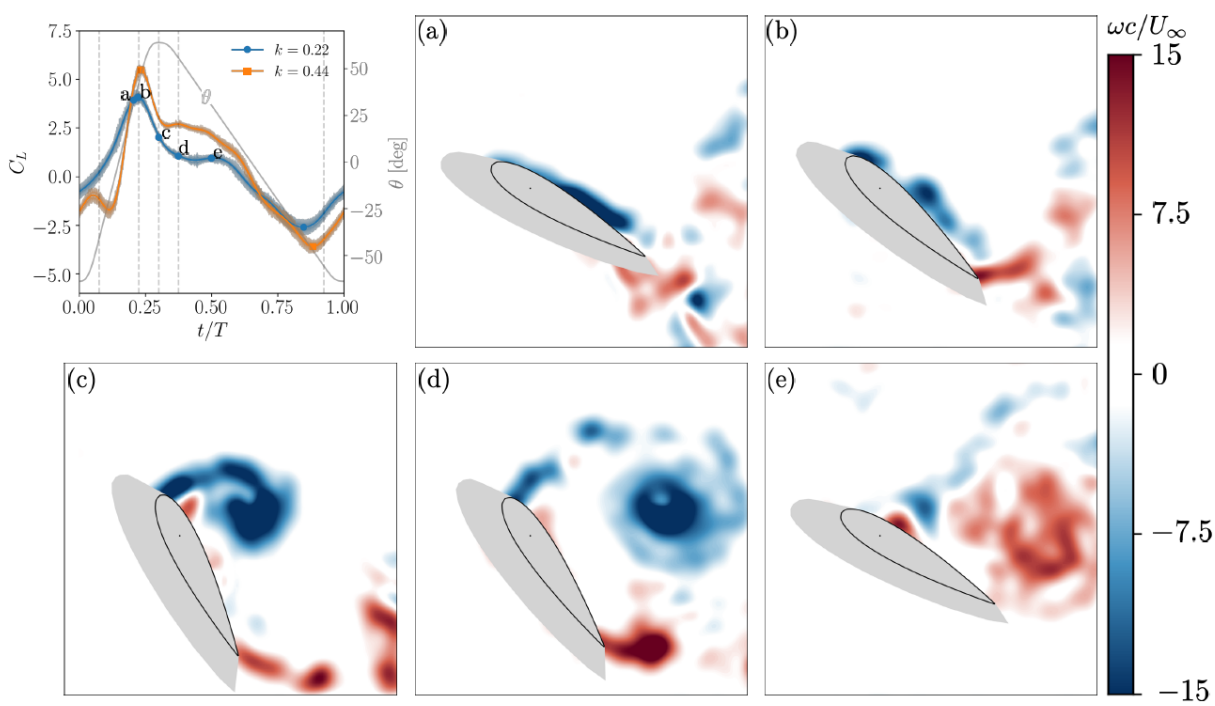


FIG. 1. (Top left) Phase-averaged lift coefficients of asymmetrically pitching foil, where the angle of attack θ varies with time t with an an amplitude $\theta_0 = 64°$ and period T; (a-e) snapshots of the non-dimensional vorticity field ($k = 0.22$, $\theta_0 = 64°$) for five consecutive instants.

Optimisation of a Ship Wind-Assisted Propulsion System

James Cairns (j.cairns.1@research.gla.ac.uk)

Department of Aerospace Sciences, University of Glasgow

Significant legislative changes are set to revolutionise the commercial shipping industry. Upcoming emissions restrictions will force operators to look at technologies that can improve the efficiency of their vessels, reducing fuel costs and emissions. This project investigates the optimisation of a ship wind-assisted propulsion system (SWAP), an actively controlled aerofoil mounted vertically on the deck of a ship in order to generate a forward propulsive force from the wind. The device functions in a similar manner to a sail on a yacht, whereby the aerodynamic forces generated by the sail reach an equilibrium with the hydrodynamic forces of the hull and a resulting forward velocity develops. The current version of the proposed circulation control system employs blowing and suction from the leading-edge and trailing-edge of the aerofoil respectively, utilising the Coanda effect to energise the boundary layer, with the aim of generating high lift and low drag. Computational Fluid Dynamics (CFD) focussed optimisation will consider both the aerofoil aerodynamic performance associated with the circulation system and configuration aspects on board a ship deck. The National Wind Tunnel Facility (NWTF) located at the University of Glasgow provides experimental validation and has been used for two testing campaigns aimed at investigating lift and drag characteristics under different attitude and circulation conditions. This is compared with numerical simulations to make a model which accurately captures the relevant flow phenomena and can predict lift and drag values over a large design space of geometric and circulation parameters. It is intended that the high fidelity data gained from this project will be used to create lower order, fast response models which can be integrated with a ship Velocity Prediction Program (VPP).

Acknowledgement: PhD project industrially linked with SMAR Azure Ltd.

Experimental simulation of the Vortex Ring State

D. Pickles (d.pickles.1@research.gla.ac.uk), R. Green and A. Busse

University of Glasgow

The fluid dynamics of a helicopter rotor present many challenging phenomena. The flow is dominated by a system of intertwined helical vortices that trail from the rotor blades that persist and remain in the vicinity of the rotor for a long time to dominate the wake, and subsequently cause significant interactional, vibration and aeroelastic effects. Of interest here is the vortex ring state (VRS), typically associated with the descent of a rotor into its own wake, where the trailed vortex system collapses from its usual helical structure to form a toroidal vortex ring of the same scale as the rotor diameter. The vortex ring is highly unsteady, and sheds off and reforms, leading to large thrust oscillations. It is often supposed that the phenomenon is the result of the behaviour of the trailed vortices, and subsequently computation and experimentation have revolved around modelling the rotor blade system. Instead, this presentation will describe results of an experiment where a specially designed ventilated open core annular jet is used to simulate the mean flow below a rotor, and the subsequent evolution of this flow field into a state analogous to the VRS is observed. Laser Doppler Anemometry (LDA) of the jet inlet and outlet planes highlights the formation of a conical region of reverse flow through the centre of the jet, which increases in size as the notional descent velocity increased. Planar Particle Image Velocimetry (PIV) and Smoke Flow Visualisation conducted in the University of Glasgow, de-Havilland wind tunnel was used to investigate the development and subsequent shedding of a large toroidal vortex notionally similar to that produced by a rotor operating in the VRS. The experiment will be described in detail and preliminary results comparing the VRS of a rotor and that of the jet will be presented.

Comparison of Turbulence modelling in Hydrocyclone Simulations

Bola Adewoye, **Mamdud Hossain** (m.hossain@rgu.ac.uk), Aditya Karnik and Sheikh Zahidul Islam

School of Engineering, Robert Gordon University, Aberdeen

Many researchers have investigated the use of LES, RSM and $k-\epsilon$ models in analysing the turbulence in hydrocyclone and results of these studies have shown that LES and RSM are best in capturing the anisotropy of turbulence in hydrocyclone. Refinements have however been done on the $k-\epsilon$ models over the past few years therefore there is expected to be a change in the way the refined $k-\epsilon$ model captures the anisotropy nature of turbulence in hydrocyclone.

The current study, therefore, focuses on the modelling of turbulence in hydrocyclone using $k-\epsilon$, $k-\epsilon$ RNG and $k-\epsilon$ realisable model and comparing the result with that of RSM model. The study also focuses on evaluating the accuracy of the results using, turbulence intensity, separation efficiency, velocity distribution and forces acting in the cyclone. The result shows that the efficiency decreased by 20-50% when $k-\epsilon$ model were used as compared with the RSM. It was also observed that the drag force increases for all the models when compared with the RSM model, leading to more particles flowing to the overflow of the cyclone. The velocity profiles were also completely different with maximum tangential velocity was observed to be towards the wall of the cyclone when the $k-\epsilon$ model was used, however, the use of RSM model shows that the maximum velocity moves towards the core of the cyclone indicating poor spatial resolution and near wall representation in $k-\epsilon$ model when compared with RSM model.

In conclusion the result of the simulations show better separation efficiency, improved spatial resolution and better wall presentation with the use of RSM.

Experimental investigation of a two-bladed propeller at yaw

Angel Zarev (a.zarev.1@research.gla.ac.uk), Ross Higgins and Richard Green

University of Glasgow

After decades in the shadow of turbofans and turbojets, the focus of a large section of the aeronautical sector is slowly shifting back to propeller based propulsion. One of the aspects of propeller propulsion left largely unexplored is their behaviour when exposed to an incident airflow. There are no recent or accurate studies on the topic and most of the widely used mathematical models still work under the assumption of uniform induced flow distribution across the disc, laid out by de Young [3] 53 years ago, or the 'steady-state' assumption, laid out by Crigler [2] 66 years ago. Studying the inflow of propellers at incidence is an opportunity to provide a solid foundation for validation of numerical methods, while also highlighting any gaps in the studies performed so far.

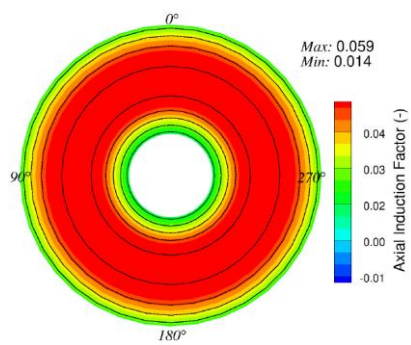
To this end, an experimental investigation has been performed in the 2.06 x 2.66m De Havilland National Wind Tunnel Facility to measure inflow into a two-bladed propeller over a range of five advance ratios and five yaw angles. The measurements were conducted using a three-component Laser Doppler Anemometry (LDA) one chord away from the propeller plane over a coordinate grid of 360 points. The results have been converted into propeller axial, tangential, and radial induction factors and compared against widely used engineering level mathematical models of propellers at incidence [1, 2]. Preliminary investigation of the results demonstrates that the aforementioned methods fail to correctly predict inflow distribution over the propeller plane. This puts in question the ability of said methods to determine forces over the propeller plane itself.

The presentation will encompass the motivation behind the research, experimental methodology, notable data trends of propellers at incidence, comparison between experimental results, mathematical models (including CFD), and future research aims.

References:

[1] P. D. Chappell. In-plane forces and moments on installed inclined propellers at low forward speed. ESDU, 1989.
[2] John L Crigler and Jean Gilman Jr. Calculation of aerodynamic forces on a propeller in pitch or yaw. Technical report, National Aeronautics and Space Administration Washington DC, 1952.
[3] John De Young. Propeller at high incidence. Journal of Aircraft, 2(3):241-250, 1965.

Acknowledgements:



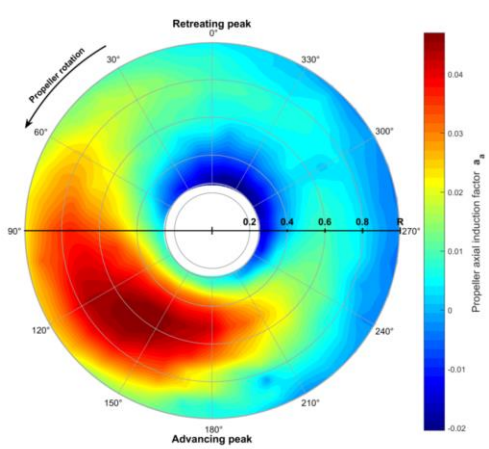


Figure 1: Propeller induced axial induction factor
$J = 1.57$, $V_\infty = 30\ m/s$, $\theta = 20\ °$, $\beta = 49\ °$, $RPM = 2342$

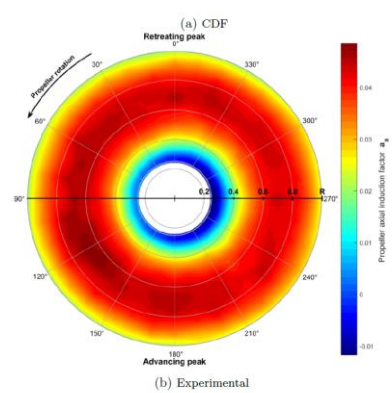


Figure 2: Initial CFD results comparison
$J = 0.89$, $V_\infty = 20\ m/s$, $\theta = 0\ °$, $\beta = 28\ °$, $RPM = 2756$

Relaxation of Vortex Braids

Simon Candelaresi (s.candelaresi@dundee.ac.uk), Gunnar Hornig, Benjamin Podgera and David I. Pontin

Division of Mathematics, University of Dundee

We study the relaxation behaviour of a topologically non-trivial "vortex braid", i.e. a flow in which the vorticity field lines are braided. In particular we measure effects from reconnection of vorticity field lines. The braid is placed in a cylindrical wedge domain which eliminates spurious boundary effects. The model flow has zero net kinetic helicity - a topological quantity important for the flow evolution. Analogous to the magnetic braid case, we find that as the fluid evolves the vortex structure unbraids into two oppositely twisted vortex tubes. The field-line helicity distribution simplifies, while the integral of their magnitude is well conserved. This suggests that vortex braids unwind and simplify just as magnetic braids while conserving information about their topologically non-trivial initial state in the form of the kinetic field-line helicity.

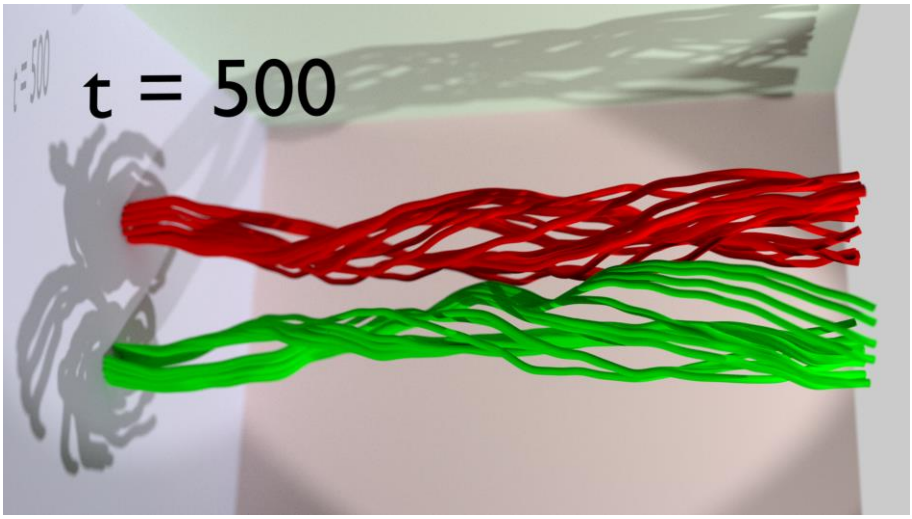


Figure 1: Unbraided vortex braid under viscous Navier-Stokes evolution.

Modelling Direct Brain Cooling for Ischemic Stroke

Luke Fulford (luke.fulford@ed.ac.uk), Ian Marshall, Peter Andrews and Prashant Valluri

University of Edinburgh

Ischemic stroke is a major cause of death in the world, where the disruption of normal blood flow creates an ischemic region preventing normal metabolic function and ultimately leading to cell death. Metabolism is a temperature dependant process, with lower temperatures able to delay the onset of necrosis. Cerebral temperature can be successfully lowered via whole-body hypothermia but implementation is impractical for use in first-response settings. By targeting cooling where needed, direct brain cooling via the scalp could offer the prospect of increasing the treatment window, if an easy to use and effective method can be found. The vast majority of previous computational studies of direct brain cooling for stroke have shown little cooling in the ischemic region, however these have made use of models that fail to account for blood flow. This study makes use of the VaPor model [1], developed at the University of Edinburgh, and extends that to specifically encompass the modelling of temperature distributions in stroke-affected brains. VaPor seeks solutions to the mass, momentum, and energy equations in the brain considering a 1-dimensional vasculature (arterial and venous) embedded in a 3-dimensional porous tissue. As demonstrated in recently published work, the VaPor model produces 3-dimensional blood perfusion and temperature distributions in line with clinical data. In this work we simulate a stroke by placing a flow obstruction anywhere within the 1-dimensional arterial tree. This means that varying degrees of stroke severity can be simulated, with reduced flow rates in the affected 1-dimensional vessels and 3-dimensional porous media. The resolution of flow and temperature profiles within the brain allows for a level of detail not currently possible with in-vivo tests. Generated temperature profiles indicate the vital role blood flow normally plays in thermoregulation within the brain, and demonstrate that the ability of scalp cooling to provide a useful degree of cooling in the ischemic region is highly dependent upon the position of that region within the brain. Initial Transient simulations show cooling can be rapidly achieved, further demonstrating the potential of direct brain cooling via the scalp as a first-response action.

Effect of Fracture Roughness on Proppant Transport in Hydraulic Fractures using CFD-DEM model

Yatin Suri (y.suri@rgu.ac.uk), Sheikh Zahidul Islam and Mamdud Hossain

School of Engineering, Robert Gordon University, Aberdeen

For hydraulic fracturing design in unconventional reservoirs, it is important to accurately predict proppant distribution in a fracture, as the distribution of proppant affects fracture conductivity and fractured well productivity. In this paper, a three-dimensional multiphase modelling approach has been applied, and the equations defining the fluid-proppant and inter-proppant interaction have been solved using the finite volume technique that uses hybrid Computational Fluid Dynamics model (Euler model coupled with Discrete Element Method) to simulate proppant distribution in hydraulic fractures in the unconventional reservoir. The predicted simulation results were validated against the experimental study by Tong and Mohanty [1]. Based on the previous literature [2-3], the existing studies used planar fracture geometry to study proppant transport neglecting the fluid leak off from the fracture wall or fracture-matrix interface and fracture tip screen out. In this work, a realistic fracture geometry is developed with fluid leak off rate defined along the fracture length to mimic the fluid leak-off from the fracture into the surrounding porous reservoir. Additionally, the effect of fracture roughness using Joint Roughness Coefficient (JRC) on proppant transport in hydraulic fractures is investigated in detail. The hydrodynamic and mechanical behaviour of proppant transport was found to greatly dependent on the fracture roughness and flow regime. When particles transport between smooth fractures, the planar walls exert extra hydrodynamic retardation, which causes particle transport velocity to decrease with the decrease in the fracture aperture. In contrast, when particles transport between rough fractures (Figure 1), due to frequent interaction between particles, the mechanical interaction-induced retardation becomes dominant and further decreases proppant transport velocity. Particle longitudinal migration is frequent because of inter-particle interaction, which hinders its transverse transport and even causes particle agglomeration in a fracture during horizontal transport. In addition, the mechanical retardation is significantly dependent of particle transport regimes, and its effect gradually increases and changes to be dominant at high particle Reynold number regime.

References:

[1] Tong, S. and Mohanty, K.K., 2016. Proppant transport study in fractures with intersections. Fuel, 181, pp.463-477.

[2] Zhang, G., Gutierrez, M., and Li, M., 2017. A coupled CFD-DEM approach to model particle-fluid mixture transport between two parallel plates to improve understanding of proppant micromechanics in hydraulic fractures. Powder Technology, 308, pp.235-248.

[3] Zeng, J., Li, H. and Zhang, D., 2016. Numerical simulation of proppant transport in hydraulic fracture with the upscaling CFD-DEM method. Journal of Natural Gas Science and Engineering, 33, pp.264-277.

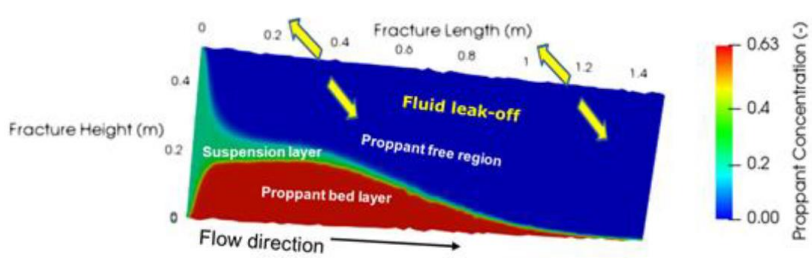


Figure 1: Proppant concentration inside hydraulic fracture at 10s

Red blood cell 'choking' of microfluidic pillar array geometries

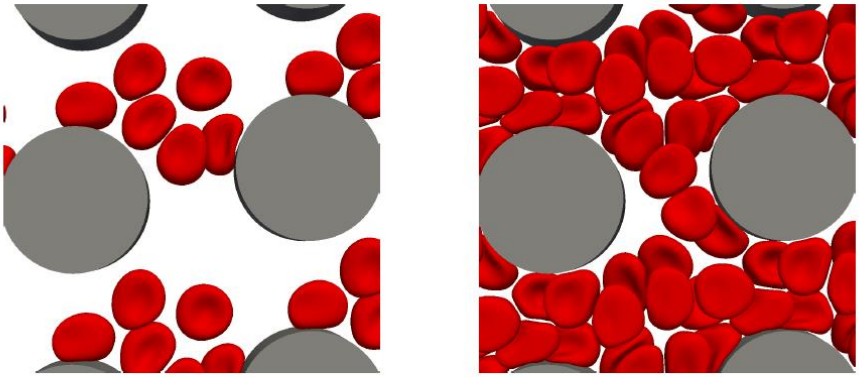
Rohan Vernekar (R.Vernekar@ed.ac.uk) and Timm Krüger

School of Engineering, University of Edinburgh

Microfluidic systems have great potential towards achieving rapid and cost-effective particle and bio-cellular separations, with possible applications ranging from chemical synthesis to bio-medical diagnostics. A microfluidic particle separation technique that has gained much traction in recent years is the *deterministic lateral displacement* (DLD) [1]. The DLD consists of an array of micro-pillars in a rectangular flow channel. It achieves size-based bi-modal particle separation, by *laterally displacing* large-sized particles on off-flow trajectories through the device. The DLD's main advantage lies in its high size-resolution for particle separation combined with simple operation.

DLD devices have been used for separating cellular blood components and also detecting red blood cell (RBC) deformability differences [2]; a bio-marker for multiple diseases (e.g. malaria). The device operation relies solely on flow advected micro-particles (such as RBCs) interacting with individual pillars of the device. Since RBCs occupy > 40% of blood volume (haematocrit, *Ht*) in humans, processing blood through the DLD has required significant amount of dilution. This is a drawback for quick, portable processing of blood samples.

We numerically investigate and statistically quantify the effect of using undiluted RBC suspensions through the DLD (fig. 1) [3]. RBC membrane deformability is varied to mimic the effect of increasing viscous stresses at higher flowrates. We discover that increasing RBC haematocrit causes a breakdown of the large particle trajectory mode in the DLD. Surprisingly the trajectory mode for the small particle remains relatively robust. Our results suggest that it would be difficult to separate a few smaller particles from a dense suspension of larger particles (e.g. platelets from blood), without dilution. However, separating a few larger particles from a dense small particle suspension (e.g. white blood cells from am RBC background) should remain viable.



(a) $Ht = 16.1\%$, diluted $\sim 1/3$ (b) $Ht = 45.6\%$, undiluted

Figure 1: Computer simulations of increasing red blood cell haematocrit (Ht) in a DLD.

References:

[1] L. R. Huang, E. C. Cox, R. H. Austin, J. C. Sturm, Science 304, 987–990, issn: 0036-8075, 1095-9203 (May 14, 2004).

[2] J. A. Davis et al., Proceedings of the National Academy of Sciences 103, 14779–14784, issn: 0027-8424, 1091-6490 (Oct. 3, 2006).

[3] R. Vernekar, T. Krüger, Medical Engineering & Physics 37, 845–854, issn: 1350-4533 (Sept. 2015).

Thermomagnetic convection in thermally heated baroclinic annulus and spherical shell

P. S. B. Szabo (P.Szabo@hw.ac.uk) and W.-G. Früh

School of Engineering and Physical Sciences, Heriot Watt University, Edinburgh

The study of large-scale convection on stellar or planetary interiors is a fundamental feature of geophysical flows. However, experiments are difficult to reproduce in laboratory scale as such would be restricted to buoyancy by terrestrial gravity. One way to produce the desired central force field is the utilisation of the di-electric force[1] or the Lorenz force[2] but are still restricted to buoyancy if experiments are conducted under terrestrial conditions. To study such flows experiments are conducted at microgravity conditions e.g. via parabola flights, the Texus rocket program or even on the international space station.

Here, we present the extend of the novel approach studied by Früh[3] to simulate convection in a central force field for laboratory investigations which exploits the Kelvin body force. The approach considers a central force field generated by a magnetic material placed in the centre of the domain. When a fluid sensitive to temperature and magnetic fields is used a body force in the radial direction is present and can induce thermomagnetic convection as investigated by Früh's numerical simulations that provided a 2D section through a non-rotating spherical shell. This study was further extended by Szabo and Früh[4] who rotated the system about its vertical axis (Ω).

The current study presents further numerical simulations of a magnetic fluid in a differentially heated (ΔT) baroclinic annulus and a spherical shell ($r_i = 35$ mm, $r_o = 100$ mm). Both convections can be characterised via two main non-dimensional parameters the Taylor number - a ratio between Coriolis and viscous force and the magnetic Rayleigh number - a ratio of Kelvin body force over viscous and thermal dissipation:

$$\mathrm{Ta} = \left(\frac{2\Omega r_o^2}{\nu}\right)^2 \qquad\qquad \mathrm{Ra_M} = \frac{\mu_0 K |\nabla H| (r_o - r_i)^3 \ \Delta T}{\rho \nu \kappa}$$

with μ_0 the permeability of free space, $K = (\partial M/\partial T)_H$ the pyromagnetic coefficient, ∇H the magnetic field gradient, ρ the density, ν the kinematic viscosity and κ thermal diffusivity. The parametric numerical study covers a range of Ta and $\mathrm{Ra_m}$ up to $\approx 10^8$. Results present columnar convection cells along the tangent cylinder. To characterise the observed baroclinic wave solutions, a further non-dimensional parameter (Θ) had to be developed that quantified the ratio of magnetic forces to rotational forces and is the equivalent to the thermal Rossby number. A regime diagram was developed for the baroclinic annulus and spherical shell that represented all observed flow regimes shown in Figure 1 (a) and (b) respectively.

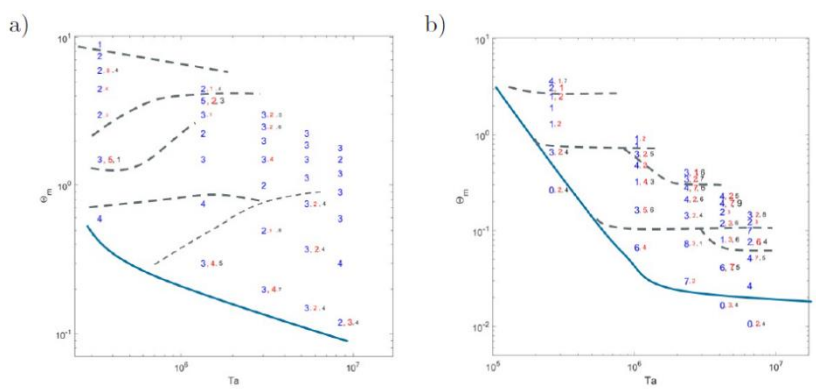


Figure 1: Regime diagram plots for baroclinic annulus (a) and spherical shell (b).

[1]Futterer et al., *Acta Astronomica* **66**, 193 (2010).
[2]Olson et al., *Phy. Earth. Plan. Sci.* **92**, 109 (1995).
[3]Früh, *Nonlin. Prog. Geophys.* **12**, 877 (2005).
[4]Szabo et al., *PAMM* **18**, 1 (2018).

Poster Presentation Abstracts (in alphabetical order)

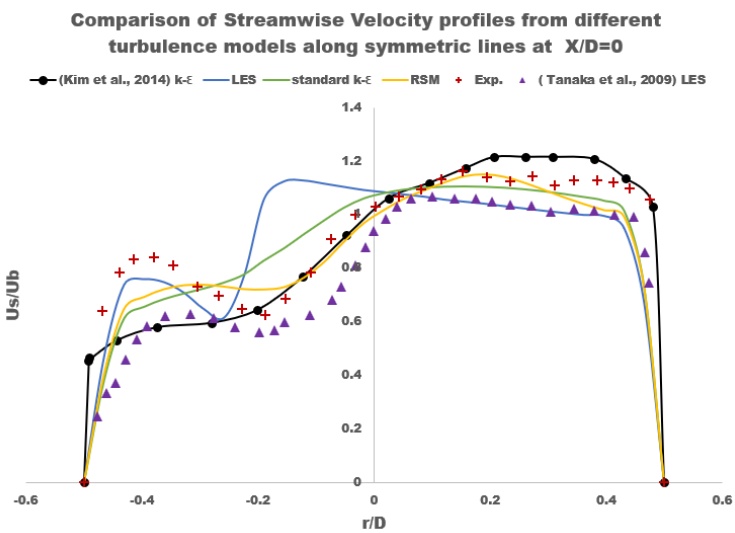
Numerical Simulation of Turbulent Pipe Flow with Elbow Bend: Comparison between RANS and LES

A. Abuhatira, S.M. Salim (m.s.salim@dundee.ac.uk) and J.B. Vorstius

School of Science and Engineering, University of Dundee

Primary and secondary flow in 90-degree pipe elbow with curvature radius ratio of $R_c/D = 2$ and Reynolds Number = 60,000 is investigated using CFD to evaluate relative performance between RANS (standard *k-ε* and RSM) and LES turbulence approaches. The present numerical results are compared against both experimental data (Sudo et al., 1998) and numerical simulations (Kim et al., 2014).

In previous publications, RANS and LES have been investigated separately for different flow regimes (based on different Reynolds numbers and curvature radius ratios). The current work aims to make direct comparison between the two different turbulence approaches against validation data and preliminary results are presented in the figure below.



The results indicate that RSM predicts the flow regime sufficiently accurate and is less computationally expensive than LES. The computed flow field variables will be coupled to ANSYS Mechanical to model the structural response of the pipe and subsequently the induced vibration.

References:

KIM, J., et al. (2014) Characteristics of Secondary Flow Induced by 90-Degree Elbow in Turbulent Pipe Flow. *Engineering Applications of Computational Fluid Mechanics,* **8** (2), 229-239.

SUDO, K., et al. (1998) Experimental investigation on turbulent flow in a circular-sectioned 90-degree bend. *Experimental Methods and their Applications to Fluid Flow,* **25** (1), 42-49.

Using "Smart Sphere" for Studying Incipient Motion

Khaldoon Al-Obaidi (2372435A@student.gla.ac.uk), Athanasios Alexakis and Manousos Valyrakis

School of Engineering, University of Glasgow

Sediment transport in rivers and estuaries environments represents one of the major challenges to engineers and researchers in the field of earth surface dynamics. Specifically, of interest for this study is to identify the flow events causing the entrainment of a coarse particle at low mobility conditions. In this work, the quadrant analysis technique is linked to the impulse criterion, a dynamic criterion in literature for defining incipient motion [1]. The goal of this work is to use the "smart sphere", developed by Valyrakis et al [2] and the 3D Acoustic Doppler Velocimeter (ADV) to assess the impulse criterion using different types of sensors for monitoring the hydrodynamic forces and relating the results to flow structures via linking particle and flow dynamics [3-4].

The experiments took place at a tilting and water recirculating research flume in the Water Engineering Laboratory at the University of Glasgow. A rectangular flume with 7m length and 0.9m width that can provide water at a capacity of 50l/s and carry flows up to 0.41m deep was used. The bed was covered with three layers of uniformly sized fine sand with a d50 of 0.6mm that were glued with a high-performance varnish throughout the entire bed and a layer of fairly uniform sized gravel (d50 = 15mm) to achieve an adequate hydraulic roughness (see Figure 1). To monitor the particle's movement, a high speed camera and a smart sphere (7cm in diameter) were used (Figure 2). Flow velocities were recorded using a 3D Acoustic Doppler Velocimeter (Nortek Vectrino-I ADV) at 20 vertical positions (with an average distance of 7mm between measurements) and 2 longitudinal positions at 1cm and 7cm upstream the particle's location (see Figure 3 and 4).

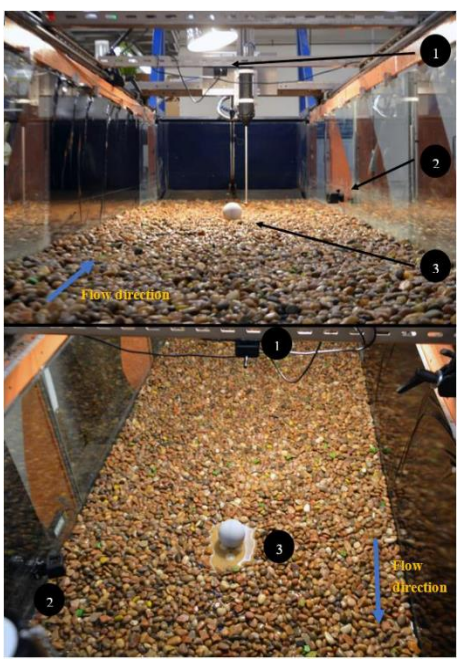


Fig. 1: Experimental set-up

Fig. 2: "Smart Sphere"

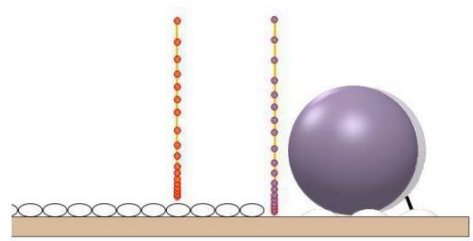


Fig. 3: Velocity measurements

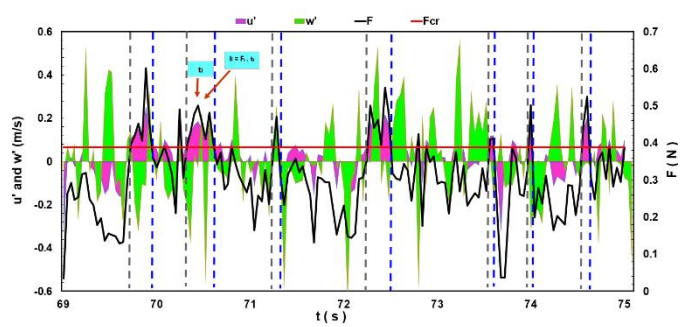


Fig. 4: Force, u' and w' vs. time with F_{cr}= 0.39 N

References: [1] Valyrakis et al, Role of instantaneous force magnitude and duration on particle entrainment, J. Geophys. Res. 115 (2010). [2] Valyrakis et al, "Smart pebble" designs for sediment transport monitoring, 17 (2015). [3] Valyrakis et al, Entrainment of coarse grains in turbulent flows: An extreme value theory approach, Water Resour. Res. 47 (2011) 1. [4] Valyrakis et al, Entrainment of coarse particles in turbulent flows: An energy approach, J. Geophys. Res. Earth Surf. 118 (2013) 42.

Applying frequency domain unsteady lifting-line theory to time domain problems

Hugh Bird (h.bird.1@research.gla.ac.uk) and Kiran Ramesh

Aerospace Sciences, School of Engineering, University of Glasgow

Frequency domain Unsteady Lifting-Line Theory (ULLT) allows the oscillation of finite wings to be studied at low computational cost with wing tip effects. Small amplitude frequency domain ULLTs are well established, with methods available valid for all oscillation frequencies, and swept and curved wings. In contrast, time domain ULLT is still in its infancy. Methods that have lower computational costs assume a pseudo-steady wake, limiting their validity to very long wake wavelengths. Meanwhile, more expensive numerical time-marching methods remain frequency limited and applicable only to straight wings. For this reason, ULLT is only applicable to a small range of kinematics if the validity of current theory is observed, limiting its applicability to real world problems.

Here, a method is devised by which a frequency domain ULLT can be applied to the time domain. Consequently, the generality of a frequency domain solution is inherited by the time domain solution. Arbitrary small amplitude kinematics can therefore be investigated for a low computational cost. The method is separated into three steps: firstly, an approximation of the frequency domain three-dimensional induced downwash function; secondly, an inverse Fourier transform to obtain a step response for a finite wing, and finally, the application of the step response using the Duhamel integral. Such a procedure is like that applied to Theodorsen's function to obtain approximations of Wagner's function in classical literature, albeit with modifications.

Whilst the method is not yet as flexible as the most general frequency domain methods, no significant obstacles are anticipated its further development.

Blood flow in the pulmonary bifurcation under healthy and diseased conditions

M. Boumpouli[1] (maria.boumpouli@strath.ac.uk), M. Danton[2], T. Gourlay[1] and A. Kazakidi[1]

[1] Department of Biomedical Engineering, University of Strathclyde
[2] Scottish Adult Congenital Service, Golden Jubilee National Hospital, Clydebank

Adult patients with congenital heart disease are at risks of chronic complications including pulmonary regurgitation [1] and stenosis in the left pulmonary branch [2]. Long-term pulmonary stenosis is also associated with abnormal lung development and elevated pulmonary vascular resistance [3]. In this study, the haemodynamic environment of the pulmonary bifurcation is investigated, assuming different physiological and pathological cases based on various geometries and boundary conditions.

Within the finite volume method framework of OpenFOAM®, blood flow simulations were performed in simplified two- and three-dimensional models of the pulmonary bifurcation. Newtonian and non-Newtonian blood rheology was considered for incompressible fluid flow governed by the Navier-Stokes equations.

The simulation results demonstrated that the pulmonary arterial flow can be significantly affected by different geometrical characteristics and boundary conditions. Flow separation varied in models with different branch angle, origin, and obstruction. Local stenosis in the left pulmonary artery had a notable effect in the axial velocities and shear stresses developed on the vessel wall. Peripheral stenosis and pressure difference in the branch outlets resulted in variations in the branch flow splits, representative of pulmonary hypertension conditions. The branch pressure ratio was further analysed, based on disease severity, as an indicator of flow discrepancies between the different cases. The obtained results were comparable to flow simulations in a three-dimensional model, for both steady and unsteady flow. Future work will involve simulations in more complex patient-specific geometries from patients with congenital heart diseases. Peripheral resistances and patient-specific blood flow inlet waveform will be also considered.

Acknowledgments:
This work is supported in part by the University of Strathclyde Research Studentship Scheme (SRSS) Research Excellence Awards (REA), Project No 1208 and the European Union's Horizon 2020 research and innovation programme, under the Marie Skłodowska-Curie grant agreement No 749185.

References:

1. Kogon B.E., et al (2015) Seminars in Thoracic and Cardiovascular Surgery. 27 p57-64
2. *McElhinney D.B., et al (1998) The Annals of Thoracic Surgery. 65 p1120-1126*
3. Harris M.A., et al (2011) Cardiovascular Imaging. 4 p506-513

Modelling of Multiple Normal Shock Wave Boundary Layer Interactions

Kiril Boychev (k.boychev.1@research.gla.ac.uk), G. N. Barakos and R. Steijl

School of Engineering, University of Glasgow

The interaction of a shock wave with a boundary layer (SWBLI) occurs in many applications such as supersonic wind tunnel diffusers, and supersonic (high-speed) intakes. Under specific operating conditions multiple shock wave/boundary layer interactions (MSWBLI) can form. In the present work, Reynolds Averaged Navier Stokes (RANS) simulations are used with the HMB3 CFD solver of Glasgow University to investigate the flow physics, and the sensitivity of MSWBLI to modelling assumptions for a rectangular duct (M = 1.61, Re_δ = 162000). Several eddy-viscosity models and an explicit algebraic Reynolds stress model are considered. A methodology for matching the experimental conditions before and after the interaction was first established. This was followed by a grid resolution investigation and a series of three-dimensional simulations were performed to quantify the spanwise confinement effects. Using the same methodology, additional test cases were computed and compared to experiments. Across the test cases, the explicit algebraic Reynolds stress model was found to give the best agreement with the experiments and displayed consistency in MSWBLI predictions. The ability of the non-linear model to better resolve the corner flows resulted in narrower corner separations than linear eddy-viscosity models. A detailed account of the flow physics of MSWBLI will be presented at the event.

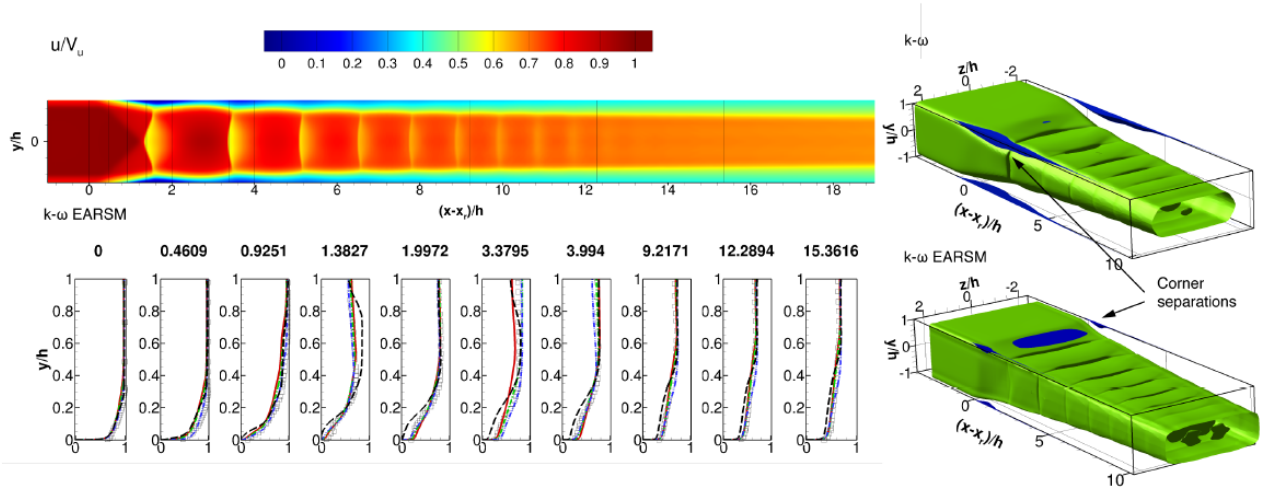


Figure 1: Streamwise velocity contours (top), profiles (bottom), and ($M = 1, u/V_u = -1 \times 10^{-3}$) isosurfaces (right) for a MSWBLI ($M = 1.61$, $Re_{\delta_u} = 162000$).

On the aerodynamics of the gliding seeds of Javan cucumber

Daniele Certini[1] (D.Certini@ed.ac.uk), C. Cummins[1,2,3], E. Mastropaolo[1], N. Nakayama[2,3,4] and I. M. Viola[1]

[1] School of Engineering, University of Edinburgh
[2] School of Biological Sciences, Institute of Molecular Plant Sciences, University of Edinburgh
[3]SynthSys Centre for Systems and Synthetic Biology, University of Edinburgh
[4]Centre for Science at Extreme Conditions, University of Edinburgh

Wind dispersed seeds inspired flight pioneers. Igo Etrich studied the Javan cucumber (*Alsomitra macrocarpa*) because it is an inherently stable glider with one of the lowest terminal velocities ($0.4\,\mathrm{m\,s^{-1}}$). Unlike seeds from pioneer trees such as maples, Javan cucumber does not rely on wind, gusts and updrafts to cover distances up to hundreds of meters. It outperforms autorotating seeds like the maple seed and equals pappose seeds like the dandelion. This amazing flight is not only due to the flight mechanics but also to the aerodynamics provided by the shape. Our research aims to uncover the flow structures around the Javan cucumber, gliding at a Reynolds number around 4000. Here, we describe the morphology of real seeds and the airfoil with 3 D scans and image processing.

We argue that the flight mechanics of Javan cucumber cannot be understood using a two-dimensional wing. Different authors stated that it glides at different angles of attack: $5\,^\circ$ and $12\,^\circ$. However, we found that a rectangular flat plate with the same surface and wing-span of Javan cucumber provides the correct lift and drag at $5\,^\circ$ but, differently from the real seed, it is unstable in pitching when flying at this angle of attack. A triangular flat plate with the same surface and wing-span, as the rectangular one glides stably at $5\,^\circ$, but shows a lift which is almost half of that of the Javan cucumber. Conversely, a flat plate with a planar geometry resembling the Javan cucumber is stable when it flies at an angle of attack of $12\,^\circ$, but it provides a lift that is more than double that the lift of the real seeds. Finally, if the angle of attack was lowered to $5\,^\circ$, it would provide the right lift but it would again be unstable in pitching. More research is ongoing to demonstrate that the correct forces and stability can only be achieved with a three-dimensional wing. This work aims to inspire the design of more efficient drones, as in the past the flight mechanics of Javan cucumber has inspired the design of aircraft.

Validation Study of Large Eddy Simulation Modelling of Turbulent Blood Flow through the FDA Nozzle

Tze Liang Chee[1] (s1443131@sms.ed.ac.uk), Rudolf Hellmuth[2] and Dong-hyuk Shin[1]

[1] School of Engineering, University of Edinburgh
[2] Vascular Flow Technologies, Dundee

This study conducted Large Eddy Simulation (LES) for turbulent and transitional flows in the 'FDA nozzle' to explore its merits as a potentially more accurate predictor than RANS models. The US Food and Drug Administration (FDA) has been more open to computational models in regulatory applications of medical devices. In 2012, the FDA has published the results of an inter-laboratory challenge, dubbed the "Critical Path Project", to evaluate the success of computational fluid dynamics (CFD) in predicting flow features in a 'Nozzle' benchmark, a geometry with features commonly found in blood passing medical devices [1]. Among many groups, Vascular Flow Technologies (VFT) also had investigated how an axisymmetric Reynolds-averaged Navier–Stokes (RANS) modelling can reproduce the experimental measurements [2].

This study furthers the VFT studies, exploring 3D-RANS and LES models. As part of the comparative analysis, centreline velocity, axial velocity profiles, wall shear stress and wall pressure were used in the comparison against the PIV results. Among the simulations compared, LES overall had similar or superior performance in the quantities of interest investigated, particularly in reproducing jet behaviour. As a result of this LES investigation, a workflow for LES in vascular device applications was developed; however, further optimisations can be made to improve the simulation accuracy whilst maintaining a reasonable simulation time through mesh optimisation, numerical solver control and runtime control. With ever increasing computational power, LES may potentially see mainstream use in regulation and prototyping of medical devices.

References:

[1] S. Stewart, E. Paterson, G. Burgreen, P. Hariharan, M. Giarra, v. Reddy, S. Day, k. Manning, S. Deutsch, M. Berman, m. Myers and R. Malinauskas, "Assessment of CFD Performance in Simulations of an Idealized Medical Device: Results of FDA's First Computational Interlaboratory Study," Cardiovascular Engineering and Technology, vol. 3, no. 2, pp. 139-160, 2012.

[2] J. Williams, "FDA Nozzle: CFD Validation," Vascular Flow Technologies, Dundee, 2018.

Using Fluid-Structure Interaction to Determine Optimal Application for Microbial Induced Calcite Precipitation in Soil

Harry Eggo (1605472@abertay.ac.uk), Ehsan Jorat, Carl Schaschke and Ruth Falconer

Abertay University, Dundee

With an expanding human population and reduced area for development, ground improvement is required to support existing and future infrastructure. Conventional techniques, such as grouting or compaction, are associated with environmental disruption in the form of toxic additives to the soil and high levels of embodied CO_2. The possibility of improving the mechanical properties of soil, undisturbed in situ, through the process of Microbial Induced Calcite Precipitation (MICP) has been an area of interest due to its potential as an environmentally sustainable alternative.

One challenge in practical implementation involves achieving a predictive model which considers the chemistry, fluid-dynamics and complex interacting processes that occur in a heterogeneous soil matrix. A specific problem involves the method used to induce MICP which in a real case scenario, will result in different fluid-dynamic configurations.

Finite element modelling was applied to fluid–structure interaction to replicate varying fluid-dynamic configurations. An arbitrary structure was imposed to replicate the soil matrix and characteristics - pore space and shear strength, defined by Mohr-Coloumb theory. Data obtained from 2-D radial flow experiments was used to calibrate the behaviour of the soil matrix.

The model produced was demonstrated to run with computational efficiency and achieved validation for clean sands but there was ambiguity for natural soils, unless data was available to indicate more specific pore space distribution. A parametric analysis, based on clean sands, considered the optimal fluid-dynamic conditions and indicated which configurations result in stable distribution of calcite in clean sands.

Low-order Prediction and Modelling of Intermittent Flow Separation and Reattachment in Unsteady 2D Flows

Desanga Fernando (Desanga.Fernando@glasgow.ac.uk) and Kiran Ramesh

Aerospace Sciences Division, School of Engineering, University of Glasgow

Discrete-vortex numerical methods can improve the design process of aerofoils when compared to traditional Computational Fluid Dynamics (CFD) techniques. Recently, these methods have been successfully applied to high-amplitude, high-frequency unsteady manoeuvres where the aerodynamics is dominated by leading-edge vortex shedding. Here, the criterion of shedding the vortices from the leading edge is based on Leading-Edge-Suction-Parameter (LESP) ["Discrete-vortex method with novel shedding criterion for unsteady aerofoil flows with intermittent leading-edge vortex shedding". J. Fluid Mech.751 pp. 500-538 (2014)]. However, the concept of LESP is only applicable to the cases where the flow detachment only occurs at the leading edge of the aerofoil. The generalisation of this method to the external aerodynamics problems with arbitrary flow separation/ reattachment on the surface can further broaden the application areas of LEPS.

The current study presents a general approach that can automatically determine the time and location of flow separation on arbitrary unsteady flows by using an integral boundary layer method. The formation of “Van Domellen singularities” in the boundary layer solution indicates the onset of unsteady flow separation and this viscous flow solution is coupled to the inviscid discrete-vortex solution using a physics-based coupling schema. Separated shear layers are represented by shedding discrete-vortex particles from the dynamically determined separation point. This novel method is computationally efficient and mimics the arbitrarily separated/ reattached flow fields over aerodynamic bodies, and the framework can be easily extended to three-dimensional flow fields.

The presentation will be elaborated the initial validations of this computational model for flows past a cylinder under a range of Reynolds numbers Re = 10,000 – 1000,000, and the outcomes of the method will be compared with similar CFD simulation results.

Applying the Goldilocks Principle to predict coral habitat engineering

Konstantinos Georgoulas (Konstantinos.Georgoulas@ed.ac.uk)

University of Edinburgh

The ecosystem services provided by coral reefs are worth over $100 billion annually and include coast line protection, tourism, food and medical derivatives. However, the health of the constituent corals can be significantly impacted by climate change. The occurrence and proliferation of reef-forming cold-water corals is reliant upon optimal current conditions, where provision of organic material is at a velocity suitable for prey capture by the coral. The occurrence of a significant proportion of dead skeletal framework on reefs highlights that when flow is sub-optimal, prey capture and ingestion rates are likely inadequate to facilitate survival.

The reef forming coral Lophelia pertusa has an optimal range of flow velocities in which they can capture food efficiently. This 'Goldilocks Zone', where the flow is neither too fast nor too slow, will promote coral growth compared to zones of sub-optimal flow velocity. Disruption of flow by the corals also creates sub-optimal velocity regions behind it, contributing to mortality of downstream corals.

A Computational Fluid Dynamics (CFD) model has been created in order to understand coral growth in 'optimal conditions', simulating current and possible future environments. Smoothed Particle Hydrodynamics (SPH) – a mesh-free Lagrangian method- is being used as its advantages over traditional grid methods are exploited in the coral growth model. The model is written in the C++ programming language and will be parallelized with the Open Multi-Processing (OpenMP) application programming interface to allow for time-effective high resolution simulations. Our data and models of how corals modify their own flow environment, provides an explanation to the cold-water coral paradox of living in fast velocity environments, but requiring slow velocities for prey capture.

Mach Effects on Particle-Wall Interactions: A Parametric Study

Jack A. Hanson (jack.hanson@strath.ac.uk) and Dr. Sina Haeri

Dept. of Mechanical and Aerospace Engineering, University of Strathclyde

This paper focuses on a parametric study of the effects of Reynolds, Mach and dimensionless gap height on particle drag/lift as it approaches a wall. Accurate modeling of drag and lift coefficients in this regime is needed where particle driven erosion is a concern. A 4th order accurate finite difference scheme is used to discretize the compressible NS equations on a globally orthogonal mesh. Experiments are performed over Re (100 – 1000), M (0.3 – 0.5) for a set of dimensionless gap heights - h(4, 2, 1, 0.75). These gap heights and Reynolds numbers are well documented in the incompressible regime, providing benchmarking for the methods used. Results for a free stream flow agree well with the values of St and C_D available in the literature for a range of Mach numbers. Preliminary, coarse grid results for a particle near a wall show an increase in the Strouhal number as predicted in the literature, the further effects of increasing Mach number will be reported in detail.

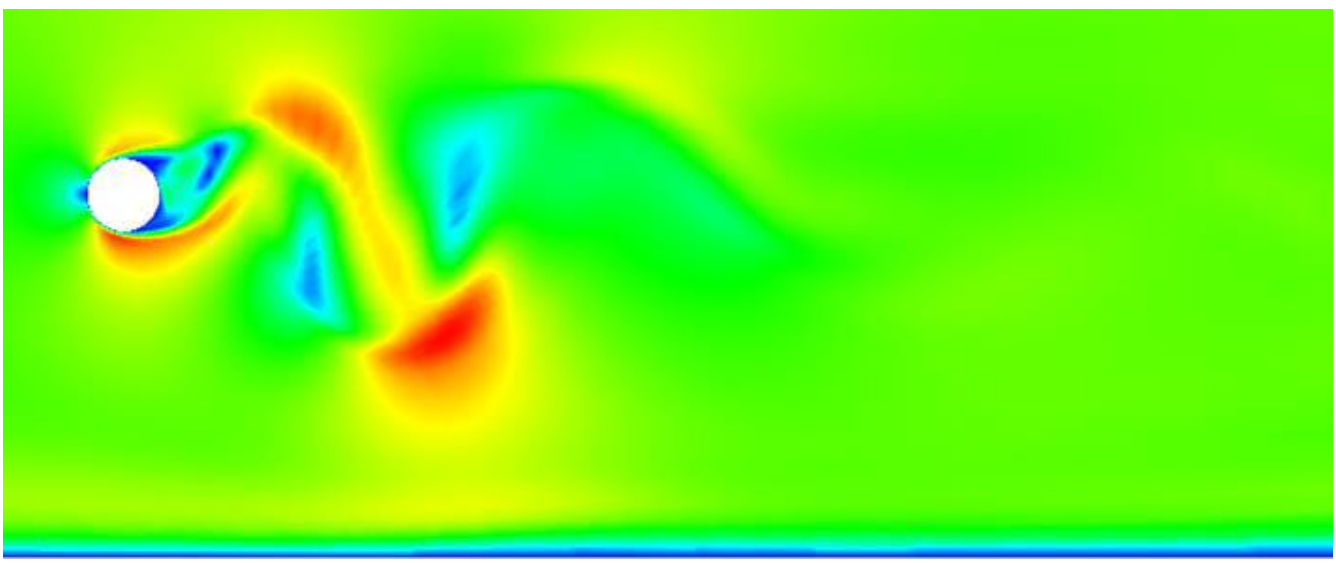


Figure 1 – Ma = 0.3 at Re = 500

Implementation of Explicit Windkessel Boundary Condition

Rudolf Hellmuth (Rudolf.Hellmuth@vascular-flow.com)

Vascular Flow Technologies, Dundee, UK

Blood flow and pressure waves emanate from the heart and travel throughout a tree-like network of compliant vessels, where they are damped, dispersed and reflected due to changes in vessel caliber, branching, and tissue properties. Computational fluid dynamics (CFD) simulations of blood flow are represent just a small region of interest of the entire cardiovascular system. Therefore, if a simulation requires to take into account any sort of dynamic effect that occur outside the CFD domain, then models that couple pressure and flow must be applied as boundary conditions (BCs). Such models can be as simple as a polynomial function, to an ordinary differential equation (ODE), or even a multinodal circuit of 1-D elements representing a large network of blood vessels. Two big challenges arise from implementing such special boundary conditions. First, it decreases computational stability, if the if the pressure and velocity fields are segregated in the CFD solver, because the BCs of those fields dynamically update each other [1]. Second, eventual physiologic backflows at outlets may artificially alter the local flow dynamics, if not specially treated [2].

The classical Windkessel model, which combines lumped effects of downstream resistance and compliance, was implemented as a BC using OpenFOAM. The pressure Dirichlet BC is coupled to the Neumann velocity BC by the implicit backward integration of the Windkessel ODE. The patch internal field is used to compute the flow rate, in order to take advantage of relaxation factors of the SIMPLE algorithm. The BC of individual cell surfaces operate as Neumann for positive fluxes, but they switch to Dirichlet BC with the values obtained from the patch-face normal component of the internal-cell value, for negative fluxes. This strategy allows having both positive and negative fluxes on the same outlet patch.

Verification was performed using a mesh consisting of an array of 10 cells, and compared with the Windkessel ODE integrated with Runge-Kutta. Validation was performed using both an idealised iliac bifurcation with 75% stenosis in one iliac artery, as well as a left coronary artery tree. As clinically observed, a 75% stenosis is asymptomatic with the Windkessel model, because the resistance of the BC is much higher than the resistance caused by the reduced cross-sectional area. The flow inside the left coronary tree is much more homogeneously distributed with the Windkessel model, than with constant outlet pressure BCs.

References:

[1] I. E. Vignon-Clementel, C. A. Figueroa, K. E. Jansen, and C. A. Taylor, “Outflow boundary conditions for three-dimensional finite element modelling of blood flow and pressure in arteries", Computer Methods in Applied Mechanics and Engineering, vol. 195, pp. 3776-3796, 2006.

[2] M. E. Moghadam, I. E. Vignon-Clementel, Figliola, R. and A. L. Marsden, “A modular numerical method for implicit 0D/3D coupling in cardiovascular finite element simulations". Journal of Computational Physics, vol. 244, pp. 63-79, 2013.

CFD investigation of the effect of pipe diameter on multiphase flow induced vibration

Nkemjika Mirian Chinenye-Kanu, **Mamdud Hossain** (m.hossain@rgu.ac.uk), Ghazi Mohamad Droubi and Sheikh Zahidul Islam

School of Engineering, Robert Gordon University, Aberdeen

This study investigates effect of pipe diameter in multiphase flow induced vibration (FIV) in 90^0 pipe bends. Such geometries commonly form important sections of flow lines and process systems in offshore and onshore hydrocarbon production and process systems. Due to inherent slug and churn multiphase flow patterns, the pipe bends are usually excitation sources for FIV. The vibrations potentially result in resonance and increase the likelihood of fatigue failures or production interruptions in the systems. Most practical pipeworks have large diameters but experimental studies that investigate multiphase FIV are mostly based on small pipe diameters. Therefore in this study, a validated numerical modelling approach used for a 52.5mm I.D. pipe geometry has been applied to a 203.1mm I.D. pipe geometry. The multiphase flow patterns and turbulence were modelled using the volume of fluid (VOF) method and the $k-\epsilon$ Reynolds-Averaged Navier-Stokes (RANS) equations respectively. The small and large pipes were each used to simulate 7 slug and churn flow case studies. The PDF of the void fraction fluctuations in the large pipe compared well with that of the small pipe showing that the slug and churn flows were well modelled in the large pipe. Also, the frequency of the slugs, velocity and pressure signals were compared between the two pipe geometries. The results showed that slug frequency values obtained in the small pipe are higher than the values obtained for the large pipe.

Sand Erosion Prediction in Complex Multiphase Flows in Double Bend geometries

Oluwademilade Ogunsesan and **Mamdud Hossain** (m.hossain@rgu.ac.uk)

School of Engineering, Robert Gordon University, Aberdeen

Piping geometry and flow pattern remain important parameters influencing surface material removal. Most previous studies have focused on sand erosion in simplified geometries with upstream length before the elbow long enough for flow development, however exploration of fossil in extreme conditions has called for the use of more complex piping systems with little or no allowance between the elbows. This study provides more in-depth understanding of sand erosion in elbows mounted in series.

Computational Fluid Dynamics (CFD) modelling techniques have been employed to investigate the complex flow interactions between two elbows mounted in series. Eulerian Multifluid-VOF Model was employed to simulate the air-water two phase flow and sand particles were tracked using the Lagrangian Discrete Phase Model. The two-phase flow was also resolved and analysed as a single phase. The distance (L/D) and angle between the elbows were varied, and the effects of these and flow pattern on erosion rate and location were studied.

Results generated showed satisfactory agreement with experimental and other published data. In the representative single phase condition, average erosion rate in the second elbow was less than the first elbow in most cases while the opposite is observed in the actual multiphase flow. Erosion pattern in the second elbow was also observed to be different from that in the first elbow. Additionally, the interaction between erosion in the two elbows, effects of change in L/D and flow pattern on erosion magnitude are presented.

Modelling granular media with dynamical density functional theory

B. G. Goddard[1], **T. D. Hurst[1]** (s1111092@sms.ed.ac.uk), and R. Ocone[2]

[1] School of Mathematics and the Maxwell Institute for Mathematical Sciences, University of Edinburgh
[2] School of Engineering and Physical Sciences, Heriot-Watt University, Edinburgh

Systems of granular media play several important roles in industry and the natural world. Their dynamics are very complicated, as microscopic interactions between particles can have a large effect on the entire system. This also presents a difficulty when trying to model a system of granular media: even state of the art computational power cannot handle a large enough number of particles to fully simulate a system of industrial scale, and existing continuum models can neglect important particle interactions that take place, which can make their flow predictions inaccurate.

Recently, dynamical density functional theory (DDFT) has proved successful in modelling colloidal fluids, such as paint or milk. DDFT incorporates interparticle interactions and volume exclusion at a mesoscopic level, using the well-studied Helmholtz free energy functional. They can be fine-tuned by deriving empirical forms of parameters from particle simulations with small numbers of particles.

We present a new DDFT that is suitable for granular media. A model is constructed which includes volume exclusion effects and other interparticle interactions via the free energy functional, but also includes effects from inelastic collisions using a collision operator. We simulate small systems of inelastic hard particles using event driven particle dynamics (EDPD), and use statistics from these simulations to construct parameters that are not analytically tractable, for example the radial correlation function. These parameters are then used in example DDFT simulations. The results are promising, and several important and fundamental questions relating to granular media arise during the derivation.

Fluid-structure interaction simulation of the brachial artery undergoing flow-mediated dilation

G. Hyde-Linaker (g.hyde-linaker@strath.ac.uk), R. Black and A. Kazakidi

Department of Biomedical Engineering, University of Strathclyde

Flow-mediated dilation (FMD) permits a non-invasive clinical assessment of endothelial dysfunction, a key indication of early atherosclerosis and cardiovascular diseases. This has significant implications with paediatric patients. FMD necessitates the measurement of brachial artery dilation from transient hyperaemia following a period of temporary ischemic occlusion. In addition to arterial diameter changes, the wall shear stress, blood pressure, and wall stiffness vary transiently in FMD, making it a complex fluid-structure interaction (FSI) problem. This work seeks to model the haemodynamic mechanisms associated with FMD utilising the open source OpenFOAM-extend library[1]. Prior studies have demonstrated the suitability of this library for cardiovascular simulations[2]. Two FSI solvers, based on strong and weak coupling, were implemented for comparison. Both solvers utilise a partitioned approach, where the fluid and structure are solved separately and the information in each domain is exchanged at the FSI interface for each time step. This is achieved using a dynamic mesh solver based on a discretisation of Laplace's equation. The fluid flow solution is based on the finite volume method (FVM) and the displacement of the solid domain is solved by a Lagrangian FVM solver. The artery wall was modelled as a straight tube with physiological values for the internal diameter, density, wall thickness, Young's modulus, and Poisson's ratio[3]. A Newtonian incompressible fluid was assumed with physiological density and viscosity[4]. The inlet velocity for the fluid domain is specified from an in-vivo hyperaemic condition[5]. The simulation results demonstrate an important variation in the diameter of the arterial vessel during FMD, while haemodynamic wall shear stress and pressure values are also ascertained. These preliminary results are useful for comparing the implementation of strong and weak FSI solvers and for correlating arterial wall displacement with the prescribed in-vivo inlet velocity. Future work will focus on FMD in idealised and patient-specific bifurcation models where ischemic occlusion will be prescribed for the distal branching arteries.

Acknowledgments:
This work is supported in part from the University of Strathclyde International Strategic Partner Research Studentships, and the EU's Horizon 2020 research and innovation programme under the Marie Sklodowska Curie grant agreement No749185.

References:

1. Extend-Project (2018) *The foam-extend*. https://sourceforge.net/projects/foam-extend/
2. Tukovic, Zeljko, Karaˇc, Aleksandar, Cardiff, Philip, Jasak, Hrvoje and Ivankovic, Alojz. (2018). *OpenFOAM Finite Volume Solver for Fluid-Solid Interaction*. Transactions of FA- MENA. 42. 1-31. 10.21278/TOF.42301.
3. Takashima, K. Kitou, T. Mori, K. Ikeuchi, K. (2006) *Simulation and experimental observation of contact conditions between stents and artery models.* Medical Engineering & Physics 29 (2007) 326–335
4. Ku, D. (1997) *Blood flow in Arteries*. Annu. Rev. Fluid Mech. 1997. 29:399–434
5. van Bussel, FCG et al. *A control systems approach to quantify wall shear stress normalization by flow mediated dilation in the brachial artery*. PloS one (2015) 10:e0115977

Use of the Padé Approximant in Solution to a Model of Vortex Shedding

D. Johnston (daniel.johnston.2015@uni.strath.ac.uk) and M.Z. Afsar

Department of Mechanical and Aerospace Engineering, University of Strathclyde

Solutions to nonlinear differential equations are often determined by numerical methods such as a Runge-Kutta scheme. The Padé Approximant, however, is a method to approximate such solutions as the ratio of two absolutely convergent power series. In this paper we apply the Padé Approximant theory to a previously developed model of vortex shedding (proposed by Facchinetti et al [1]) that takes the form of the diffusive Van der Pol Oscillator. The Padé Approximations obtained and presented in this paper are determined through appropriate asymptotic expansions and compared to their corresponding Taylor Series and the 4th Order Runge-Kutta solution found using the MATLAB's "ode45" numerical solver. Our main findings are that the Padé

Approximations consistently show closer agreement to the numerical solution over their corresponding Taylor Series especially at O(1) values of the small parameter in VDP model for an extended interval in the independent variable domain (see Fig. 1). We discuss how the Padé Approximation of the VDP solution in this model offers the advantage of being amenable to further mathematical manipulation as well as reducing the need to store large numerical solutions.

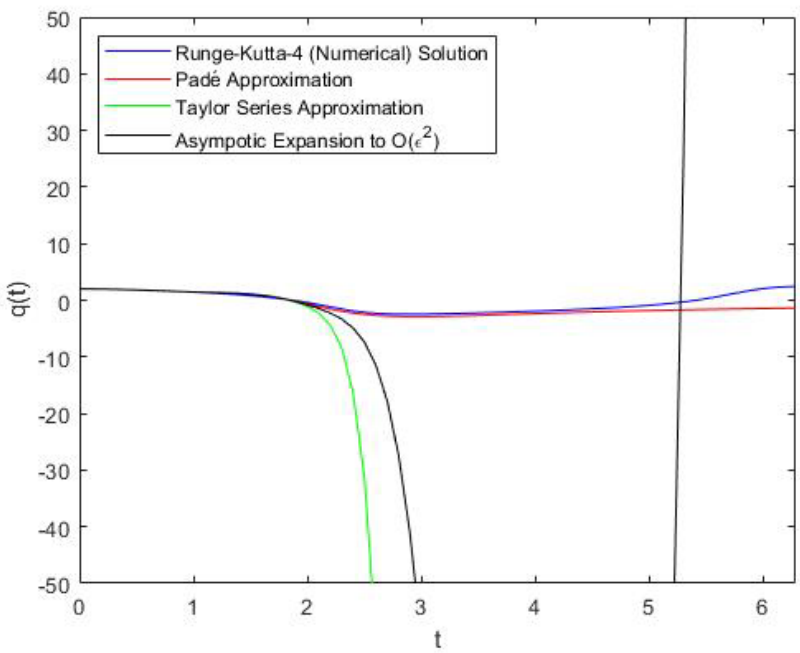


Figure 1: Approximations of the diffusive Van der Pol equation for a Padé Approximation of order L = 5 and M = 6; and with ε = 1.

Reference:

[1] Facchinetti M.L. et al., CR Mechanique 330 (2002) 451-456.

Blood Flow Simulations in the Human Aortic Arch in Relation to Obesity

L. Johnston (lauren.johnston@strath.ac.uk) and A. Kazakidi

Department of Biomedical Engineering, University of Strathclyde

The global obesity epidemic is worsening with 10% of the world's population now classified as obese [1]. In 2015, obesity contributed to 4 million deaths globally, 41% of which were due to cardiovascular disease. The healthy human aorta has a complex anatomy often associated with disturbed flow dynamics, while in obese individuals structural and functional changes to the cardiovascular system lead to abnormal aortic function [2]. Such changes are also associated with coronary artery disease, hypertension, and diabetes; disorders which themselves are thought to be accelerated by obesity. In this study, we utilised computational fluid dynamic (CFD) methods to examine various haemodynamic parameters, namely blood flow velocity, blood pressure, and wall shear stress (WSS), in the aortic arch and proximal branches. Two idealised three-dimensional geometries of the human aortic arch, based on anatomically-correct data from two different patient groups [3], were created using the ANSA pre-processor (BETA CAE Systems). A mesh independence study was completed to determine the optimum number of elements for the geometries. CFD simulations were performed in the open-source library OpenFOAM®, with the material properties of the working fluid, blood, in accordance with current literature [4]. Preliminary results considered both steady and time-dependent (pulsatile) flow for the solution of the incompressible Newtonian Navier-Stokes equations. The initial focus of the flow analysis was on the distribution of wall shear stress. Flow patterns observed for both models showed regions of considerable flow disturbance. The results demonstrate low shear stresses at locations on the aortic wall which are known to be susceptible to the development of atherosclerotic plaques. Blood flow parameters are significantly affected by the local anatomy of the aortic arch, highlighting important differences between the two models. The future direction of this work is to improve the accuracy of the simulations by implementing more complex boundary conditions. The investigation will then be extended to patient-specific aortic models to confirm the results of this work.

Acknowledgments: This work is supported in part from the University of Strathclyde Research Studentship Scheme (SRSS) Student Excellence Awards (SEA) Project No1619, and the EU's Horizon 2020 research and innovation programme under the Marie Sklodowska-Curie grant agreement No749185.

References:

1. Afshin A et al. *N Engl J Med* 2017; 377:13-27.
2. Robinson R et al. *J Cardiovasc Magn Reson* 2008; 10(1):10.
3. Boufi M. et al. *Eur J Vasc Endovasc Surg* 2017; 53: 663-670.
4. Benim A.C. et al. *Applied Mathematical Modelling* 2011; 35: 3175-3188.

Conjugate heat transfer between a solid and a boiling fluid

Robin Kamenicky[1] (robin.kamenicky@strath.ac.uk), Michael Frank[1], Dimitris Drikakis[2] and Konstantinos Ritos[1]

[1] University of Strathclyde
[2] University of Nicosia, Cyprus

Conjugate heat transfer between boiling fluid and solid occur in a number of industrial applications such as manufacturing processes (quenching), heat exchangers, nuclear reactors and high power electronics. In our research, we primarily focus on the nucleate boiling heat transfer regime. We give considerable attention to the essential parts, the partitioning model and the closure models, which determine nucleate site density, bubble detachment diameter and bubble detachment frequency. We compare various partitioning models in order to determine which one of those can accurately describe the nucleate boiling as close as possible to the critical heat flux point. In our comparison, we include the widely used RPI (Rensselaer Polytechnic Institute) mechanistic model [1], as well as more recent ones [2 – 4]. The later ones take into account additional physical phenomena such as the sliding bubble heat transfer or the microlayer impact on the surface evaporation.

We will present the development and validation of our multiphysics solver implemented within the OpenFOAM framework. The solver is capable to numerically estimate the behaviour of two or more regions, which can be either solid or fluid. The solid region is described using the energy equation. On the other hand, the fluid part is allowed to boil and therefore two phases, liquid and gas can be present. Each of these phases is governed by Navier-Stokes equations, which are interconnected using interfacial terms. Our results are compared to previous studies, such as the backward-facing step by Ramsak [5], as well as to more complex problems simulating conjugate heat transfer with nucleate boiling.

References:

[1] Kurul, N., Podowski, M.Z., On the modeling of multidimensional effects in boiling channels, ANS. Proc. National Heat Transfer Con. Minneapolis, Minnesota, USA, pp. 30–40, (1991).

[2] Gilman, L., Baglietto, E., A self-consistent, physics-based boiling heat transfer modeling framework for use in computational fluid dynamics, International Journal of Multiphase Flow, pp 35–53, (2017).

[3] Basu, N., Gopinath, W., Dhir, V.K., Wall Heat Flux Partitioning During Subcooled Flow Boiling: Part 1—Model Development, Journal of Heat Transfer-transactions of The Asme - J HEAT TRANSFER (2005).

[4] Yeoh, G. H., Vahaji, S., Cheung, S. C. P., Tu, J. Y. Modeling subcooled flow boiling in vertical channels at low pressures – Part 2: Evaluation of mechanistic approach, International Journal of Heat and Mass Transfer, pp 754–768, (2014).

[5] Ramsak, M., Conjugate heat transfer of backward-facing step flow: A benchmark problem revisited, International Journal of Heat and Mass Transfer, 84, pp. 791–799, (2015).

Analysis of Thin Leaky-Dielectric Layers Subject to an Electric Field

Matthew Keith (matthew.keith@strath.ac.uk), Stephen Wilson and Alexander Wray

University of Strathclyde

The application of an electric field can have a significant impact on the behaviour of fluids. For example, electrohydrodynamic (EHD) instabilities can lead to the breakup of fluids into droplets or the formation of patterned structures. In particular, these instabilities play a key role in an abundance of industrial situations such as inkjet printing and the production of micro-electronic devices. The recent review by Papageorgiou [1] gives an overview of the recent work on electrohydrodynamic instabilities. Our present work investigates the two-dimensional problem of an electric field applied across a bilayer of a leaky-dielectric liquid and gas contained between two solid walls. This work builds on that of Wray et al. [2] who considered the enhancement and suppression of EHD instabilities, investigating the axisymmetric problem of a fluid layer on the outside of a solid cylinder. Using a long-wave approximation, we explore the linear stability of the system. An investigation of the nonlinear regime highlights three characteristic behaviours of the system, namely, asymptotic thinning, the return of the interface to its flat state, and singular touchdown. Numerically calculated plots of appropriate parameter planes are obtained. Of particular interest are the critical conditions for the transitions between these characteristic states, which we investigate both analytically and numerically. In addition, we explore the self-similar dynamics of the liquid-gas interface near touchdown.

References:

[1] Papageorgiou, D.T., 2019. Film flows in the presence of electric fields. Annual Review of Fluid Mechanics, 51, pp.155-187.

[2] Wray, A.W., Papageorgiou, D.T. and Matar, O.K., 2013. Electrified coating flows on vertical fibres: enhancement or suppression of interfacial dynamics. Journal of Fluid Mechanics, 735, pp.427-456.

Design and Analysis of Floating Offshore Structures with Multiple Wind Turbines

Arun Kumar (a.s.kumar@dundee.ac.uk), Azin Lamei, Shuijin Li and Masoud Hayatdavoodi

School of Science and Engineering, University of Dundee

The offshore wind resources have received significant attention in recent years due to strong and consistent wind fields. Most of the existing floating offshore wind turbines (FOWT), whether built or in concept, host a single turbine. With the aim of reducing the overall cost of the energy production, concepts of floating structures with multiple turbines are introduced in recent years. The main conceptual challenge of placing multiple wind turbines on a single floating platform is the blockage effect of the leading turbines; under variable wind directions, the leading turbines may block the wind against the trailing turbines, and reduce the efficiency of the system. In this work, concept design of a wind-tracing floating structure accommodating three wind turbines is presented. The floating structure uses a single-point mooring system which allows for the entire structure to rotate in response to the change of the wind direction. Because of the particular configuration of the floating structure, it is essential to consider simultaneously the wind, wave and current loads, along with the responses of the structure. In addition, due to the large size of the structure, hydroelastic and aeroelastic responses must be considered. In this study, motion and elastic response of FOWTs to combined environmental loads from different directions are studied by use of the constant panel approach of the Green function method for the hydrodynamic loads and the Blade Element Method for the aerodynamic loads, directly coupled with a finite element method for the structural analysis. The equations are solved simultaneously in frequency domain, and results are compared with laboratory measurements. The model is used to study the motion and elastic response of the triangular platform with three turbines to waves and wind from different directions. We also use FAST, CFD and laboratory experiments to study this problem. These approaches are under development.

A multi-compartment lumped-parameter model for assessing the role of haematocrit in foetal circulation

L. S. Marinou[1] (lydmarin@gmail.com), V. Vavourakis[2,3] and A. Kazakidi[1]

[1] Department of Biomedical Engineering, University of Strathclyde
[2] Department Mechanical and Manufacturing Engineering, University of Cyprus
[3] Department of Medical Physics & Biomedical Engineering, University College London

Foetal circulation, being different from neonatal and adult circulation, is an intricate system. Current knowledge of its haemodynamics is limited[1], while the role of haematocrit at different gestational ages has not yet been examined extensively. This work aims to investigate the effect of haematocrit variations using a multi-compartment lumped parameter model (LPM) of the foetal circulation. The LPM model is developed in Simulink® and includes 19 elastic arterial segments and 12 peripheral vascular beds, represented, respectively, by electrical circuits and a 3-element Windkessel model[2,3]. Previous data[1,2] and allometric laws[4] were used to calculate the inflow and boundary conditions for a 33-week gestational age and foetus weight. Two validation studies were completed, one comparing results with adult flow waveforms and another examining the foetal Isthmic Flow Index. Different values of haematocrit (Hct), ranging from 10% to 80% Hct, were investigated, representing a range of anaemic, healthy, and polycythaemic conditions. Results from the validation studies were in good agreement with literature. The foetal LPM enabled calculations of blood flow waveforms at various arterial positions. Computations with 10%, 45%, and 80% Hct were further performed to demonstrate the effect of haematocrit on the foetal arterial flow. A clear difference between the 45% and 80% Hct models at the position of the ascending aorta was evident, whereas no apparent difference was detected between the models for 10% and 45% Hct. Similarly, this effect was manifested at the positions of the aortic isthmus, the thoracic aorta, and the umbilical artery. However, at the position of the ductus arteriosus there was no difference between the three models. Finally, the calculations revealed an almost exponential relationship between mean resistance and Hematocrit. Investigating haematocrit variations revealed an important effect on the foetal circulation, resulting in significant changes in vascular resistances and the pulsatility indices of the flow rate waveforms. Further investigation is required aiming at the improvement of the accuracy of the inflow and boundary conditions.

Acknowledgements:
This work is supported in part by the EU's Horizon 2020 research and innovation programme, Marie Skłodowska-Curie grant agreement No749185.

References:

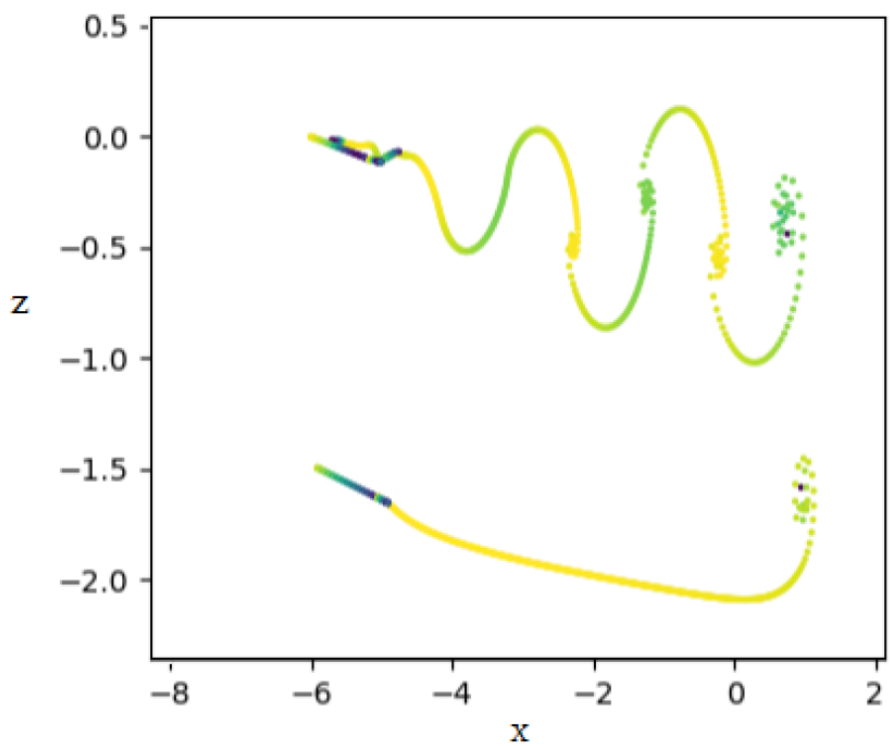
1. van den Wijngaard J. et al, Am J Physiol Regul Integr Comp Physiol 2006; 291: R1222-R1233.
2. Garcia-Canadilla P. et al., *PLoS Computational Biology* 2014; 10:e1003667.
3. Milisic V, Quarteroni A E. SAIM: Mathematical Modelling and Numerical Analysis 2004; 38(4):613-632
4. Pennati G., Fumero, R. *Annals of Biomedical Engineering* 2000; 28(4):442-45s

Deformable bodies for leading-edge vortices control

Alfonso Martínez (a.martinez-carmena.1@research.gla.ac.uk) and Kiran Ramesh

Aerospace Sciences Division, School of Engineering, University of Glasgow

Motivated by the growing popularity of MAVs within modern aerospace engineering, the phenomenon of the leading-edge vortex has been on the heart of numerous studies aiming to polish the knowledge about the conditions where such vortices exist, to allow them achieve designs increasingly closer to what has been a source of inspiration for aeronautics since its inception: natural fliers. In this work a previously developed theoretical model to determine the onset, growth and shedding of LEVs is extended to deformable bodies. Based on unsteady thin aerofoil theory and the leading-edge suction parameter, this low-order model is able to trigger or delay the formation of these vortices as per aerodynamic requirements. Temporal camber deformations are accounted for by a harmonically deflected trailing-edge flap. It is shown that slight variations on the design parameters (amplitude, initial deflection, reduced frequency and relative hinge position) can modify the cycle in which LEVs appear. Thus, pre-determined the critical LESP value, suitable alterations of the motion kinematics drive the occurrence of the vortex. The results obtained from this first approach, although restricted to small deflections, are encouraging to test more complex flap movements which will, perhaps, led to a greater dominance over the vortex.



LEV shedding after flap deflection (upper)
Absence of vortex without deflection (lower)

Dynamics of a single gas/vapour microbubble under acoustic forcing of very low frequency

Davide Masiello[1] (D.Masiello@ed.ac.uk), Ignacio T. Montes[1], Rama Govindarajan[2] and Prashant Valluri[1]

[1] School of Engineering, University of Edinburgh
[2] International Centre for Theoretical Sciences, Tata Institute of Fundamental Research, Bengaluru, India

Gas bubbles in liquids undergoing acoustic forcing of sufficient amplitude can expand to many times their initial volume, followed by a rapid collapse to a very small size. Due to the violent compression, the mixture of gas and vapour in the bubble reach very high temperature and pressure. This in turn causes chemical dissociation. The dynamics of the bubble's radial motion are of fundamental importance for the basic understanding of this phenomenon and its applications. Acoustic forcing with driving frequency higher than 20 kHz have been widely studied and experimentally observed while lower frequency ranges are still unexplored. In the present work, the dynamics of an acoustically forced gas/vapour single micro-bubble in water have been simulated for driving frequencies in the audible range. In order to do so, we developed a reduced-order model accounting for all the critical thermo-mechanical contributions to the bubble's dynamics. The model has been validated on previous simulations and experimental data and then used to perform a large frequency-parametric study, checking that the fundamental assumptions are not broken in the novel ranges. Results obtained at high amplitudes (roughly > 1.1 atm) suggest that water phase-change and vapour segregation play a key role in slowing down the collapse at lower frequencies, yielding a different kind of dynamics where the first collapse is not necessarily the strongest one (which is the case for higher frequencies). However, at moderate acoustic pressure (between 1.0 and 1.1 atm), low frequency forcing yields bubble dynamics comparable to high frequency high amplitude ones in terms of dynamics, peak temperature and pressure, and generation of highly reactive chemicals. A direct consequence is the implication that the optimal frequency for applications based on cavitation could lie in very low frequency ranges, which therefore would be beneficial to include in the design process.

Reducing the Wake Drag of Bluff Bodies Using Dielectric Barrier Discharge Plasma Actuation

Craig Nolan (2082725n@student.gla.ac.uk), Angela Busse, Konstantinos Kontis

Aerospace Sciences, School of Engineering, University of Glasgow

As modern transport moves away from fossil fuels, the need for more efficient road vehicles is only increasing to reduce fossil fuel consumption and to improve the endurance of electric vehicles. For large road vehicles, such as trucks, vans and SUVs, the drag induced by recirculating flow in the wake contributes about 40% of the total drag of the vehicle (Chainani & Perera, 2008).

In the experiments of Roy et al. (2016), plasma actuators were used to reduce wake drag by disrupting the wake shear layer in flow over of a truck geometry at Reynolds numbers up to 100,000. This technique yielded up to 14% reduction of the wake drag proving the effectiveness of this method. However, the flow characteristics at a Reynolds number of 100,000 differ significantly from the full-scale case where operational Reynolds numbers exceed 3,000,000.

In the current study, we plan to extend the concept described above to higher Reynolds numbers by first conducting experiments at intermediate Reynolds numbers up to 250,000. The aim is to provide a platform for applying plasma actuators to simple road-vehicle geometry to reduce the wake drag at higher Reynolds numbers than previously attempted. A wind tunnel investigation is currently being conducted to investigate the wake characteristics of a Windsor-type body equipped with dielectric barrier discharge (dbd) plasma actuators using Particle Image Velocimetry (PIV) and hot-wire based frequency analysis. This will be accompanied by a detailed PIV-based investigation of the frequency-dependence of the flow induced by the dbd plasma actuators.

References:

Chainani, A., Perera, N. (2008). CFD Investigation of airflow on a model radio control race car. *London, World Congress on Engineering 2008.*

Roy, S., Zhao, P., DasGupta, A., & Soni, J. (2016). Dielectric barrier discharge actuator for vehicle drag reduction at highway speeds. *AIP Advances, 6*, 025322.

Adaptive Reduced Basis Methods for reconstruction of Unsteady Aerodynamics flows

Gaetano Pascarella[1] (gaetano.pascarella@strath.ac.uk), Marco Fossati[1] and Gabriel Barrenechea[2]

[1] Aerospace Centre of Excellence, University of Strathclyde
[2] Department of Mathematics and Statistics, University of Strathclyde

Obtaining accurate CFD solutions of unsteady flows during the design process or sensitivity studies of an aircraft can be a time-consuming task. To address this problem, a common practice is the use of Reduced Basis Methods (RBM) [1], in the attempt to reduce the number of degrees of freedom required to describe the physics of the fluid system in a cost-effective manner without severely reducing the accuracy.

The Proper Orthogonal Decomposition (POD) introduced by Lumley has been widely used in the literature for the specific case of unsteady flows. To overcome the limits of the classical POD, which lacks of any temporal connection between snapshots when applied to unsteady regimes [2], alternatives to POD have been proposed.

The aim of this work is to propose a method which aims at automatically and adaptively selecting the most accurate reduction technique among the classical snapshot POD and more recent techniques such as SPOD and Recursive DMD. The method will be paired with an equation-based approach to evaluate the accuracy in reconstruction. Problems of high relevance to the aerodynamics field will be considered such as the impulsive start of 2D and 3D high-lift configurations.

References:

[1] Taira, Kunihiko, et al. "Modal analysis of fluid flows: An overview." AIAA Journal (2017): 4013-4041.
[2] Noack, Bernd R. "From snapshots to modal expansions bridging low residuals and pure frequencies." Journal of Fluid Mechanics 802 (2016): 1-4.

Re-casted Navier-Stokes: application to shock structure description

M. H. Lakshminarayana Reddy (l.mh@hw.ac.uk) and S. Kokou Dadzie

School of Engineering and Physical Sciences, Heriot-Watt University, Edinburgh

Classical Navier-Stokes equations are known to be inadequate in describing some flows in both the compressible and incompressible configurations. A ubiquitous simple example of Navier-Stokes failure in the case of compressible flow configuration, is shock wave structure predictions [1,2,3,4]. Here, we derive a new class of continuum hydrodynamics equations that are systematically thermo-mechanically consistent mass diffusion type of fluid flow equations. We call these the "re-casted Navier-Stokes equations". The re-casting methodology is based on applying a transformation technique which involves transforming the fluid flow velocity variable to an appropriately selected change of variable within the standard fluid flow equations. We then analyse the stationary shock wave problem by solving the re-casted Navier-Stokes equations numerically using a finite difference global scheme (FDGS) of Reese et al. [2]. The results on the shock structures are presented for different Mach numbers ranging from supersonic to hypersonic and compared with the experimental results of Alsmeyer [1] and also with that of classical Navier-Stokes predictions. The re-casted NS model appear to be better suited for predicting actual shock profiles.

Acknowledgments:
Authors would like to gratefully acknowledge the funding from Engineering and Physical Sciences Research Council (EPSRC), UK, through Grant No. EP/R008027/1.

References:

[1] H. Alsmeyer, J. Fluid Mech. 74, 497–513 (1976).
[2] J. M. Reese, L. C. Woods, F. J. P. Thivet, and S. M. Candel, J. Comput. Phys. 117, 240–250 (1995).
[3] C. J. Greenshields and J. M. Reese, J. Fluid Mech. 580, 407–429 (2007).
[4] M. H. L. Reddy and M. Alam, J. Fluid Mech. 779, p. R2 (2015).

On the versatile forms of classical Navier-Stokes

M. H. Lakshminarayana Reddy (l.mh@hw.ac.uk) and S. Kokou Dadzie

School of Engineering and Physical Sciences, Heriot-Watt University, Edinburgh

Classical Navier-Stokes equations fails to describe some flows in both the compressible and incompressible configurations. In the last decade, the equations have been subject to a number of modifications/extensions [1,2,3,4,5]. In the current work, starting with the classical Navier-Stokes, a new form of Navier-Stokes equations are derived by a recasting methodology which is based on transforming the fluid velocity vector field within it. We name these new class of hydrodynamic models the re-casted Navier-Stokes. These new models are systematically thermo-mechanically consistent mass diffusion type of fluid flow equations and they are more complete than those previously proposed substitutions to the original [1, 2, 3, 4, 5]. The re-casted Navier-Stokes equations termed as "mass-diffusion", "pressure-diffusion" and "thermal-diffusion" Navier-Stokes equations appear better suited for compressible flows, flows involving thermal stresses and other transport processes. The plane wave analysis of these new models confirmed that these are stable in both space and time like the classical. They are also found to better interpret the experimental data of Rayleigh-Brillouin light scattering in gas.

Acknowledgments:
Authors would like to gratefully acknowledge the funding from Engineering and Physical Sciences Research Council (EPSRC), UK, through Grant No. EP/R008027/1.

References:

[1] M. Carrassi and A. Morro, Il Nuovo Cimento B 9, 321–343 (1972).
[2] H. Brenner, Physica A 349, 11–59 (2005).
[3] H. Brenner, Physica A 349, 60–132 (2005).
[4] F. Drust, J. Gomes, and R. Sambasivam, "Thermofluiddynamics: Do we solve the right kind of equations?" in Proceeding of the International Symposium on Turbulence, Heat and Mass Transfer, Vol. 5, edited by K. Hanjalic, Y. Nagano, and S. Jakirlic (2006), pp. 3–18.
[5] S. K. Dadzie, J. Fluid Mech. 716, p. R6 (2013).

Premixed flame kinematics subject to an oscillating flame holder

Dong-hyuk Shin[1] (D.Shin@ed.ac.uk), Andy Aspden[2], Sukruth Somappa[3], Vishal Acharya[3] and Tim Lieuwen[3]

1. School of Engineering, University of Edinburgh
2. School of Engineering, Newcastle University
3. School of Aerospace Engineering, Georgia Institute of Technology, USA

This study investigates the kinematics of premixed flames stabilized on an oscillating flame holder, using direct numerical simulation (DNS) with a detailed chemistry. The investigation is motivated by the onset of combustion instability in modern gas turbines for power generation [1]. Combustion instability is the phenomenon where the pressure oscillation increases significantly when coupled with the unsteady heat release. Once the instability happens, pollutant emissions can increase, and internal components, such as turbine blades and liners, can be damaged.

For the simulations, PeleLM (a DNS solver developed by the Center for Computational Sciences and Engineering at the Lawrence Berkeley National Laboratory) was used employing adaptive mesh refinement (AMR), which greatly saves computational time and storage space [2]. The simulations emulate the experiments done by Humphrey et al. [3] where methane and air are premixed near stoichiometry in the upstream and the flame is stabilized by a thin heated wire which harmonically oscillates in the cross-stream direction as shown in Figure 1. The forcing frequencies are 750 and 1250 Hz running over 20-40 forcing periods to get ensemble averages. The computational time was awarded by Extreme Science and Engineering Discovery Environment (Xsede), funded by National Science Foundation (NSF) in the USA.

The ensemble-averaged flame kinematics illustrates that (i) turbulent flow fluctuations cause flame front wrinkles at the forcing frequency are damped, and (ii) the turbulent flame speed exhibits linear dependencies on the ensemble-averaged flame curvatures.

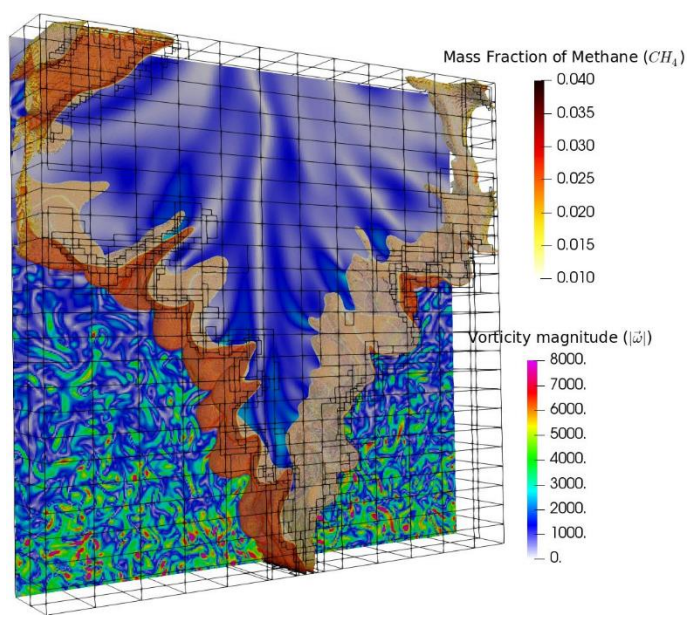


Figure 1. 3D visualization of the simulated data, highlighting (i) iso-contour of temperature coloured by mass fraction of methane - an indication of flame front, (ii) magnitude of vorticity coloured by a rainbow colour map, (iii) boxes of adaptively refined meshes in black solid lines.

References:

[1] T. C. Lieuwen, "Unsteady combustor physics," Cambridge University Press, 2012.
[2] M. S. Day and J. B. Bell, "Numerical simulation of laminar reacting flows with complex chemistry," Combustion Theory Modelling, 2000.
[3] L. J. Humphrey, B. Emerson, and T. C. Lieuwen, "Premixed turbulent flame speed in an oscillating disturbance field," Journal of Fluid Mechanics, 2018.

Towards an Improved Understanding of Induced Drag

K. Singh[1] (kevin.singh@strath.ac.uk), K. Ritos[1], M. Frank[1], I. Kokkinakis[1], D. Drikakis[2]

[1]Department of Mechanical and Aerospace Engineering, University of Strathclyde
[2] University of Nicosia, Cyprus

This poster will focus on induced drag, a component of the total drag caused by the production of lift. This component of the drag force acting on an aircraft can be as large as 90% during take-off and 40% during cruise flight [1]. With an ever-increasing requirement of designing more fuel-efficient aircraft, it is becoming important to properly understand and quantify the fundamental physical processes governing induced drag, in order to optimise aircraft design. Up until recently, most methods of quantifying induced drag have made assumptions which are generally violated at the cruise conditions of commercial aircraft.

We will review the most prevalent theoretical methods of estimating the induced drag acting on an aircraft and delve into pros and cons of each. These are compared against Computational Fluid Dynamics (CFD) simulations for the flow over a NACA 0012 wing, over a range of flow regimes, ranging from a truly incompressible flow to a near-transonic flow. Using the data collected from these simulations, we will assess the relative accuracies of these different theoretical approaches will be evaluated for a range of Mach numbers.

Reference:

[1] I.M. Kroo. “Drag due to Lift: Concepts for Prediction and Reduction”. Annual Review of Fluid Mechanics, (2000), pp. 587-617, doi:10.1146/annurevfluid.33.1.587

4D flow MRI-derived CFD for investigating haemodynamics in large arteries with severe stenotic disease

S. Skopalik[1] (s.skopalik.1@research.gla.ac.uk), P. Hall Barrientos[3], J. Matthews[4], A. Radjenovic[2], P. Mark[2], G. Roditi[2] and M. C. Paul[1]

[1] School of Engineering, University of Glasgow, [2] Institute of Cardiovascular & Medical Sciences, University of Glasgow, [3] Queen Elizabeth University Hospital, Glasgow, and [4] Canon Medical Research Europe Ltd, Edinburgh

Cardiovascular disease is one of the primary causes of death worldwide. The narrowing of the blood vessels, known as stenosis, can lead to the reduced blood supply to organs. The severity of these stenoses is currently being assessed by invasive pressure measurement using a catheter in the intervention room. Fractional flow reserve (FFR), which is the ratio of the mean distal arterial pressure (P_d) over the mean proximal pressure (P_p) during maximum hyperaemia [1], is the current "gold standard" for determining stenosis severity in the coronary arteries [2]. Computational Fluid Dynamics (CFD) combined with time-resolved phase-contrast MRI (PC-MRI/4D Flow) is a potential non-invasive method for deriving the pressure field across diseased arteries. CFD derived FFR (vFFR) is available commercially for coronary artery assessment, however, this concept has not as yet been tested in other common areas such as the iliac and femoral arteries. 4D Flow MRI measurements were conducted on a patient with a focal 78% area reduction (and >50% diameter reduction) left iliac artery stenosis as part of a comprehensive non-invasive assessment. The morphological 3D geometry of the abdominal aorta and both iliac arteries were also reconstructed from the non-invasive imaging data using the open source Vascular Modelling Toolkit (www.vmtk.org). The vessel wall was assumed rigid and the domain was discretized using tetrahedral elements with 1 mm base and 15 boundary layers near the inlet and outlets with a local mesh refinement around the stenosis with element base size reduced to 0.3 mm and 20 boundary layers. Constant mass flow rate was applied at the inlet for two flow rate scenarios (mean and systolic) at rate of 20.2 mL/s and 56.3 ml/s corresponding to diameter based Reynolds number at the inlet of Re≈415 and Re≈1150 respectively. Pressure outlet boundary conditions were used for both outlets, with values based on the catheter-derived blood pressures for resting and hyperaemic scenarios [3]. Blood was modelled as a Newtonian fluid with density of 1060 kg/m^3 and dynamic viscosity of 4 mPa·s [4]. The standard two-equation k-Omega SST turbulence model was used for all scenarios [5]. Results, presented in Figure 1, showed good agreement with in-vivo measurements presented in literature [3], thus the study concludes that vFFR has good potential to characterise iliac artery stenotic disease.

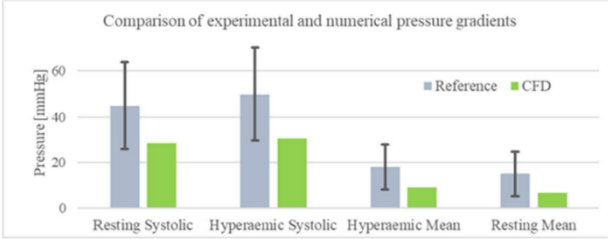


Figure 1: Comparison of pressure drop values from CFD with reference values for study group with 76-90% stenosis [3].

Acknowledgement:

This work is kindly funded by Medical Research Scotland (grant PhD-888-2015) and Canon Medical Research Europe Ltd.

References:

1. Lotfi, A., et al., Expert consensus statement on the use of fractional flow reserve, intravascular ultrasound, and optical coherence tomography: a consensus statement of the Society of Cardiovascular Angiography and Interventions. Catheter Cardiovasc Interv, 2014. 83(4): p. 509-18.
2. Morris, P.D., et al., "Virtual" (Computed) Fractional Flow Reserve: Current Challenges and Limitations. JACC: Cardiovascular Interventions, 2015. 8(8): p. 1009-1017.
3. Archie, J.P., Analysis and Comparison of Pressure Gradients and Ratios for Predicting Iliac Stenosis. Annals of Vascular Surgery, 1994. 8(3): p. 271-280.
4. Linge, F., M.A. Hye, and M.C. Paul, Pulsatile spiral blood flow through arterial stenosis. Computer Methods in Biomechanics and Biomedical Engineering, 2014. 17(15): p. 1727-1737.
5. Paul, M.C. and A. Larman, Investigation of spiral blood flow in a model of arterial stenosis. Med Eng Phys, 2009. 31(9): p. 1195-203.

Investigating the enhanced mass flow rates in pressure-driven water flow through nanopores

Alexandros V.A. Stamatiou (alexandros.stamatiou@hw.ac.uk), Kokou S. Dadzie, Lakshminarayana Reddy R.M and Alwin M. Tomy

Heriot-Watt University, Edinburgh

Over the past two decades, several researchers have presented experimental data from pressure-driven water flow through nanotubes. They quote flow velocities which are four to five orders of magnitude higher than those predicted by the Navier-Stokes equations. Thus far, attempts to explain these enhanced mass flow rates at the nanoscale have focused mainly on the introduction of wall-slip boundary conditions for the velocity field. In this paper, we present a different theory. A change of variable on the velocity field within the classical Navier-Stokes is adopted to transform the equations into mathematically and physically different equations. The resulting equations termed re-casted Navier-Stokes equations contain additional diffusion terms whose expressions depend upon the driving mechanism. The new equations are then solved for the pressure driven flow configuration in the case of water flow in a long nano-channel. Analogous to previous studies of gas flow in micro- and nano-channels, a perturbation expansion in the aspect ratio allows for the construction of a 2D analytical solution for the stream-wise component of the new velocity. In contrast to the slip-flow models, this solution is specified by a no-slip boundary condition at the channel walls. The mass flow rate can be calculated explicitly and compared to available data.

Structure of high frequency Green's function in non-axi-symmetric (chevron-type) transversely sheared mean flows using a Ray tracing solver within the generalized acoustic analogy formulation

S. Stirrat (stirrats@virginmedia.com) and M.Z. Afsar

University of Strathclyde

The chevron nozzle continues to remain a popular approach to reducing jet noise, which works by breaking up the turbulence structures at all scales. In this paper we investigate the effect of chevron type mean flow in an acoustic analogy model in which the wave propagation reduces to the solution of the Rayleigh equation and is calculated using a ray theory model for a jet represented as a transversely sheared mean flow. Since the generalised acoustic analogy (GAA) shows that the acoustic pressure is given by the convolution product of a rank 2 tensor propagator and the fluctuating Reynolds Stress inasmuch as $p(\boldsymbol{x},\omega) = \int_{V_\infty(\boldsymbol{y})} \hat{G}_{ij}(\boldsymbol{y}|\boldsymbol{x};\omega)/\hat{T}_{ij}(\boldsymbol{y},\omega)d^3\,y$ we determine $\hat{G}_{ij}$ (that is related to the vector adjoint Green's function of the linearised Euler operator) using the high frequency Ray theory developed in Goldstein (1982) for an isotropic model of the fluctuating Reynolds stress.

Our calculation of the Rayleigh equation at high frequencies for a series of chevron mean flow patterns with multiple lobes represents a novel application of the ray tracing method. We show how the chevron jet introduces a richer structure in the Green's function (i.e. more non-periodic modulation) with a local decrease in the jet region. The high frequency noise component will then reduce if the turbulence is also small in the jet and regions where the Green's function possesses a minima.

References:

Goldstein, M. (1982). High frequency source emission from moving point multi-pole sources embedded in arbitrary transversely sheared mean flows. *Journal of Sound and Vibration*, 80(4), pp.499–522.

FDA Nozzle Validation Study

Josh Williams[1] (josh.williams96@live.co.uk) and Rudolf Hellmuth[2]

[1] Heriot-Watt University, Edinburgh
[2] Vascular Flow Technologies, Dundee

The validity of studies using CFD is shrewdly monitored in industry, particularly in medical devices. In an attempt to review the suitability of CFD as a regulated device development technique, the Food and Drug Association of America (FDA) commissioned an inter-laboratory study with five sets of independent particle image velocimetry (PIV) experiments to replicate blood flow through an idealised medical device domain [1]. This inhibited complex flow patterns such as increased velocity in a converged throat, leading to recirculation at a sudden expansion. Before releasing the PIV data, 28 groups from around the world submitted simulation results for five flow rates, spanning laminar, transitional and turbulent flow [2]. The simulations showed considerable variation from each other and from the experiments.

In our study, flow was simulated in steady-state on an axisymmetric domain using various RANS turbulence models. Quantities such as velocity, pressure, Reynolds stress were compared to the experimental data at axial and radial cuts of the domain. Wall pressure and wall-shear stress were also investigated. Mesh convergence was tested against experimental data at key points (directly before and after the expansion) using the Richardson extrapolation for fully turbulent flow (Re_{throat} = 6500) and transitional flow (Re_{throat} = 2000).

Models performance differed across varying Reynolds numbers, regions of the domain and in measuring different properties. The extrapolated results showed that Spalart-Allmaras gave optimal overall performance. Particularly so, when measuring the physiologically relevant quantities wall- shear stress and Reynolds stress—which affect blood vessel remodelling and blood cell damage respectively. Although the models' performance was mixed, the results observed were similar or superior to the best computational results submitted to the FDA study, showing that good practices were followed throughout.

References:

[1] P. Hariharan, M. Giarra, V. Reddy, S. W. Day, K. B. Manning, S. Deutsch, S. F. Stewart, M. R. Myers, M. R. Berman, G. W. Burgreen, et al. Multilaboratory particle image velocimetry analysis of the FDA benchmark nozzle model to support validation of computational fluid dynamics simulations. Journal of biomechanical engineering, 133(4):041002, 2011.

[2] S. F. Stewart, E. G. Paterson, G. W. Burgreen, P. Hariharan, M. Giarra, V. Reddy, S. W. Day, K. B. Manning, S. Deutsch, M. R. Berman, et al. Assessment of CFD performance in simulations of an idealized medical device: results of FDA's first computational interlaboratory study. Cardiovascular Engineering and Technology, 3(2):139–160, 2012.

Estimation of turbulent flow features in the vicinity of a circular pier

Yi Xu[1] (2421380X@student.gla.ac.uk), Manish Pandey[2] and Manousos Valyrakis[1]

[1] Water Engineering Lab, University of Glasgow
[2] Indian Institute of Technology, Roorkee, India

Local scour near piers is still a main reason for bridge failures. Maximum local scour depth estimation is an important aspect of river engineering. While many research studies focus on scour depth estimation have been completed the high-resolution flow turbulence data near cylindrical structures, such as bridge piers, and the linkage of these to scour processes have yet to be comprehensively studied. Thus, the goal of the present study is to undertake a properly designed experimental measurement campaign towards comprehensively estimating derivative turbulent flow quantities near circular piers under clear water scour conditions, using acoustic Doppler velocimetry (ADV). A range of turbulent flow features (such as three-dimensional mean flow velocity components, size of primary horseshoe vortex, Reynolds shear stresses, turbulence intensities and turbulence kinetic energy), are estimated for different pier diameters under equilibrium scour conditions. The primary horseshoe vortex was found to be the major cause for scour processes. Horseshoe vortex size increases with pier diameter resulting at a maximum equilibrium scour depth which also scales with the shed vortices.

Characteristics of Wakes in Branching Blood Vessels under Re = 500

Yidan Xue[1] (s1782793@sms.ed.ac.uk), Rudolf Hellmuth[2] and Dong-hyuk Shin[1]

[1] School of Engineering, University of Edinburgh, Edinburgh, EH9 3DW, UK
[2] Vascular Flow Technologies, Dundee, DD2 1TY, UK

Atherosclerosis, the leading causes of death in the developed world, preferentially occurs near the junction of branching vessel, where blood recirculation tends to occur. Studies have shown that the endothelial cells covering the inside of blood vessels develop an atherogenic phenotype when exposed to low wall shear stress in recirculation zones (Malek, Alper and Izumo, 1999). In 1980, Karino and Goldsmith observed the flow patterns of eddies forming in glass models of circular T-junctions having various branching angles, diameter ratios and Reynolds numbers from 0 up to 350. These models served as idealized models of branching vessels in mammalian circulatory systems. Flow parameters for the presence of eddies in main and side branches were mapped.

Validation is a requirement to guarantee credibility and fidelity of computational models in the medical device industry (ASME, 2018). Hence the simulations are conducted by automatic parametric study scripts over the ranges of inflow speeds, ratios of two outflow speeds, branching angles, and diameter ratios. The Reynolds number by the inflow speed varies from 0 to 500, so that the flows remain laminar. The blood is assumed to be Newtonian, which is a valid assumption under moderate shear stresses.

As the inlet flow rate increases, vortices start to form near the junction of branching vessels. The inlet flow rate (equivalently the Reynolds number) at which vortices form are denoted as the critical Reynolds number. For the vortices in the main branch, the simulation was able to reproduce reasonable agreements of the critical Reynolds numbers as the experiments. However, the vortices in narrower side branch are less satisfactory correlated to the measurement, which may have caused by possible subtle difference in geometries in the branching pipe. Furthermore, the sizes of vortices are proportional to the square of inflow speeds. The wall shear stress distributions are also closely related to the recirculation regions.

The experiments of branching vessel flows are replicated by CFD simulation under comprehensive flow conditions. A simulation region with a high level of confidence is also validated. It could potentially benefit the optimisation of medical tools and personalised surgical treatments for cardiovascular diseases.

References:

Karino, T., and Goldsmith, H. L., 1980. "Disturbed flow in models of branching vessels". *Transactions - American Society for Artificial Internal Organs*, **26**, pp. 500–506.

Malek, A. M., Alper, S. L., and Izumo, S., 1999. "Hemodynamic shear stress and its role in atherosclerosis". *Journal of American Medical Association*, **282**(21), pp. 2035–2042.

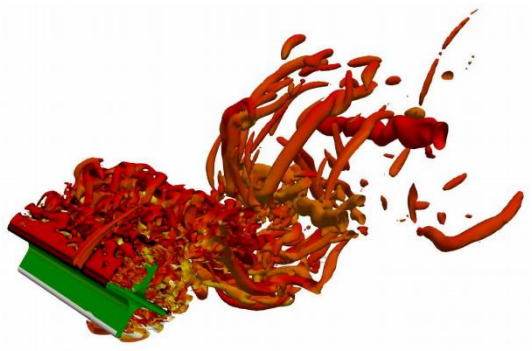
ASME, 2018. "Assessing Credibility of Computational Modeling through Verification and Validation: Application to Medical Devices". pp. 60.

Large-eddy simulations of flow past a cactus-shaped cylinder with a low number of ribs

Oleksandr Zhdanov (o.zhdanov.1@research.gla.ac.uk) and Angela Busse

School of Engineering, University of Glasgow

Despite of their shallow root systems cacti can withstand high winds in their natural environment without being uprooted. Previous studies, inspired by the Saguaro cactus, showed that it experiences less drag and lower aerodynamic force fluctuations compared to the circular cylinder over a range of Reynolds numbers [1]. The Saguaro cactus has 10-30 ribs and can be found in the deserts of the North America. Succulents of the African deserts have independently developed a similar plant structure, including ribs, in the process of convergent evolution. Even though the number of ribs in succulents is lower, they are expected to perform the same function as in cacti. If plant aerodynamics were one of the factors driving the convergent evolution of cacti and succulents, a cactus-shaped cylinder with a low number of ribs would be expected to also display aerodynamic advantages.



Results of large eddy simulations (LES) with wall-adapting local eddy-viscosity (WALE) subgrid-scale model of the flow past cactus-shaped cylinder with four ribs, inspired by a succulent Euphorbia Abyssinica are presented. The simulations were performed using OpenFOAM, at Re=20,000, a Reynolds number that is typical for the natural environment of these succulents. Two orientations were studied, namely when the cavity is facing the flow (zero angle of attack) and when a rib is facing the flow (maximum angle of attack). The results for the aerodynamic coefficients and Strouhal number are compared to experimental and numerical studies for square cylinders at the same angular orientations and to results from URANS simulations [2] for the same shape and Reynolds number.

References:

[1] S. Talley, G. Iaccarino, G. Mungal, N. Mansour (2001), An experimental and computational investigation of flow past cacti, in: Annual Research Briefs 2001, Center for Turbulence Research, NASA Ames/Stanford University, pp. 51–63.

[2] Zhdanov, O. and Busse, A. (2019), Angle of attack dependence of flow past cactus-inspired cylinders with a low number of ribs. *European Journal of Mechanics-B/Fluids,* 75, pp.244-257.

Effect of Changing Ambient Gas Density on the Primary Break-up of Modulated Liquid Jets

Zhen Zhang[1,2] (v1zzhen@exseed.ed.ac.uk) and Dong-hyuk Shin[1]

[1] Institute for Multiscale Thermofulids, School of Engineering, University of Edinburgh
[2] Beijing Institute of Control Engineering, China

The present simulation study investigates the development of modulated liquid jet sprays with gas density oscillation, which were caused by strong harmonic pressure oscillation shown to be the crux of combustion instability. The modulated jet flow was limited to the Reynolds number below 2000 and Weber number around 2000 by setting the velocity oscillation over a certain range and assuming liquid properties as specific values, which were conducive to make the break-up available in the laminar flow.

In order to investigate the continuous effect of changing gas density, a new solver in OpenFOAM was developed, implementing the oscillating ambient gas density determined by ambient pressure oscillation. The calculations showed the good agreements with the simulation results of Srinivasan et al. (2011). The oscillating gas density can stimulate the surface instability of jet flow significantly to generate droplets more easily and earlier, which was related to the mean value and amplitude in the oscillation.

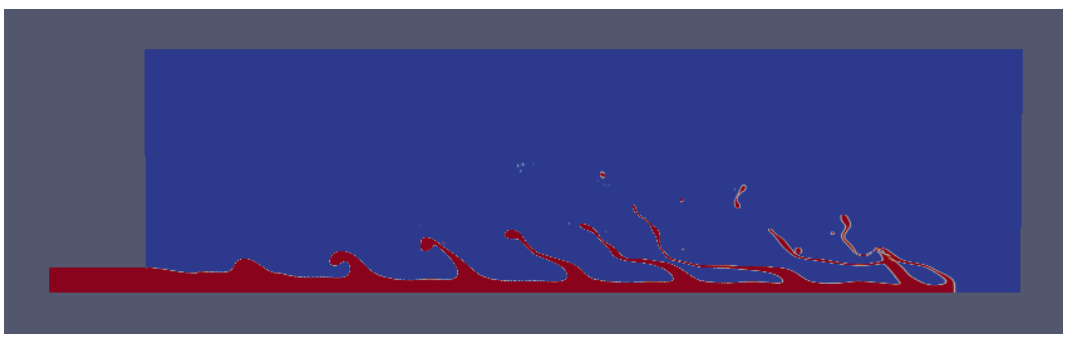


Figure 1. The development of modulated liquid jet under the condition of changing ambient gas density

Reference:

Srinivasan V., Salazar A. and Saito K. Modeling the disintegration of modulated liquid jets using volume-of-fluid (VOF) methodology[J]. Applied Mathematical Modelling, 2011, Vol.35(8), pp.3710-3730.

List of Participants

Ahmed	Abuhatira	University of Dundee
Adebola	Adewoye	Robert Gordon University
Khaldoon	Al Obaidi	University of Glasgow
Hugh	Bird	University of Glasgow
Maria	Bompouli	University of Strathclyde
Kiril	Boychev	University of Glasgow
Angela	Busse	University of Glasgow
James	Cairns	University of Glasgow
Stuart	Cameron	University of Aberdeen
Simon	Candelaresi	University of Dundee
Daniele	Certini	University of Edinburgh
Tze Liang	Chee	University of Edinburgh
Alan	Cuthbertson	University of Dundee
Hannah	D'Ambrosio	University of Strathclyde
Saikat	Datta	University of Edinburgh
Peter	Davies	University of Dundee
Fabio	Dioguardi	British Geological Survey
Duncan	Dockar	University of Edinburgh
Jennifer	Dodoo	University of Edinburgh
Shiliang	Duan	University of Dundee
Harry	Eggo	Abertay University
Desanga	Fernando	University of Glasgow
Michael	Frank	University of Strathclyde
Wolf-Gerrit	Früh	Heriot Watt University
Luke	Fulford	University of Edinburgh
Konstantinos	Georgoulas	University of Edinburgh
Ben	Goddard	University of Edinburgh
Jack	Hanson	University of Strathclyde
Graham	Hassall	Dantec Dynamics
Masoud	Hayatdavoodi	University of Dundee
Rudolf	Hellmuth	Vascular Flow Technologies Ltd
Peter	Hicks	University of Aberdeen
Mamdud	Hossain	Robert Gordon University
Timothy	Hurst	University of Edinburgh
George	Hyde-Linaker	University of Strathclyde
Daniel	Johnston	University of Strathclyde
Lauren	Johnston	University of Strathclyde
Robin	Kamenicky	University of Strathclyde
Asimina	Kazakidi	University of Strathclyde
Matthew	Keith	University of Strathclyde
Arun	Kumar	University of Dundee
Ryan	Kyle	Heriot Watt University
Azin	Lamei	University of Dundee

Shuijin	Li	University of Dundee
Yajin	Lyu	University of Edinburgh
Maxime	Magaud	University of Glasgow
Lydia	Marinou	University of Strathclyde
Alfonso	Martinez	University of Glasgow
Davide	Masiello	University of Edinburgh
Sritay	Mistry	University of Edinburgh
Andrew	Mitchell	University of Strathclyde
Lada	Murdoch	CFD People Ltd
Francesco	Neglia	University of Bari
Vasileios Martin	Nikiforidis	University of Edinburgh
Vladimir	Nikora	University of Aberdeen
Craig	Nolan	University of Glasgow
Tom	O'Donoghue	University of Aberdeen
Mike	O'Sullivan	Vascular Flow Technologies Ltd
Shuji	Otomo	University of Edinburgh
Yong Sung	Park	University of Dundee
Gaetano	Pascarella	University of Strathclyde
Sreehari	Perumanath	University of Edinburgh
David	Pickles	University of Glasgow
Rohit	Pillai	University of Edinburgh
Dubravka	Pokrajac	University of Aberdeen
David	Pontin	University of Dundee
Kiran	Ramesh	University of Glasgow
Lakshminarayana	Reddy	Heriot Watt University
Luke	Reid	University of Dundee
James	Reilly	University of Strathclyde
Jean	Reinaud	University of St Andrews
Konstantinos	Ritos	University of Strathclyde
Ewan	Rycroft	University of Strathclyde
Mohamed	Salim	University of Dundee
Feargus	Schofield	University of Strathclyde
Dong-hyuk	Shin	University of Edinburgh
Kevin	Singh	University of Strathclyde
Simeon	Skopalik	University of Glasgow
Alexandros	Stamatiou	Heriot Watt University
Sarah	Stirrat	University of Strathclyde
Yatin	Suri	Robert Gordon University
Peter	Szabo	Heriot Watt University
Rohan	Vernekar	University of Edinburgh
Jan	Vorstius	University of Dundee
Jonathan	Wilkin	University of Dundee
Josh	Williams	Heriot Watt University
Stephen	Wilson	University of Strathclyde
Yi	Xu	University of Glasgow

Angel Angelov	Zarev	University of Glasgow
Ruiwen	Zhao	University of Edinburgh
Oleksandr	Zhdanov	University of Glasgow
Zhang	Zhen	University of Edinburgh

Key Information

Travelling to the University of Dundee: https://www.dundee.ac.uk/travel/

Taxis

Dundee taxi firm numbers:

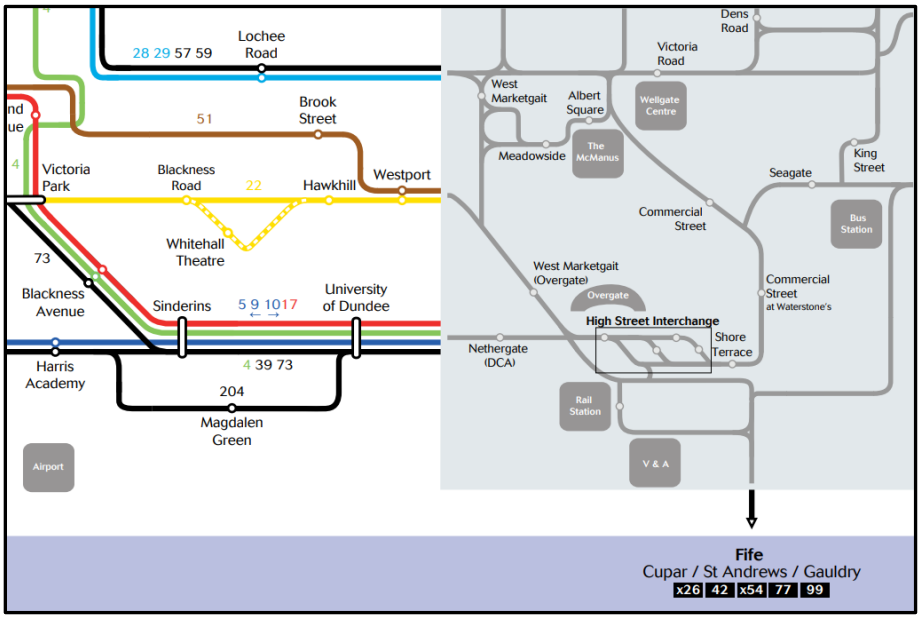
- Dundee City Taxis – 01382 204060
- 505050 Taxis – 01382 505050

The recommended pick-up and drop off location is the main entrance of the Dalhousie Building on Old Hawkhill (Building 14 on the campus map, page 4). This is the building in which the meeting will take place.

Buses

- Citylink and Megabus services run from Edinburgh, Glasgow and Aberdeen to Dundee. These typically arrive at Dundee Bus Station, which is to the east of the city centre and about a 20 minute walk to the University of Dundee.

  See https://www.citylink.co.uk/journeyplanner.php and https://uk.megabus.com/ for details.
- Local bus services (Stagecoach and Xplore) operate on routes run between the City Centre and the University of Dundee (via Perth Road) and the north side of the City Campus (via Hawkhill) – see map below.
- A single fare is about £1.60 and can be purchased from the driver (correct change required). Day tickets can also be bought for £3.50 - £3.70.



http://www.dundeetravelinfo.com/downloads/Dundee_Route_Map_Jan2019.pdf

Train times (Dundee to/from Aberdeen, Edinburgh and Glasgow)

Depart Aberdeen	07:07	07:35	07:52	08:20	**Depart Dundee**	17:01	17:39	18:09	18:51
Arrive Dundee	08:19	08:49	09:05	09:31	**Arrive Aberdeen**	18:17	19:05	19:27	20:05
Depart Edinburgh (Waverley)	06:29	07:00	07:29	07:59	**Depart Dundee**	17:24	17:34	18:24	18:46
Arrive Dundee	08:12	08:22	08:42	09:28	**Arrive Edinburgh (Waverley)**	18:33	19:07	19:44	20:10
Depart Glasgow (Queen Street)	07:41	08:10			**Depart Dundee**	16:46	17:49	18:56	
Arrive Dundee	09:07	09:40			**Arrive Glasgow (Queen Street)**	18:22	19:12	20:26	

Full details of train services to/from Dundee are available at https://www.scotrail.co.uk/tickets.