Fluids in New Zealand 2021

University of Canterbury

27-29 January 2021

Contents

About the Conference

Fluids in New Zealand is a research workshop for discussing fluid mechanics in the broadest sense. FiNZ 2021 will be hosted at the University of Canterbury from Wednesday 27 to Friday 29 January 2021.

FiNZ provides a forum to facilitate the dissemination of ideas across the different branches of fluid mechanics, and to promote interdisciplinary collaborations between New Zealand scientists. The idea is to mix experienced researchers with postgraduate students, structured talks with open discussions, and experimentalists with theorists.

Organising Committee

James N. Hewett
Department of Mechanical Engineering, University of Canterbury, New Zealand

Mathieu Sellier
Department of Mechanical Engineering, University of Canterbury, New Zealand

Phillip L. Wilson
School of Mathematics and Statistics, University of Canterbury, New Zealand

Elizabeth K. McGeorge
School of Mathematics and Statistics, University of Canterbury, New Zealand

Alan Caughley
Callaghan Innovation, Christchurch, New Zealand

Contact

Fluids in New Zealand 2021
Level 5 Civil/Mechanical Building
University of Canterbury
Private Bag 4800
Christchurch 8140
New Zealand

Email: james.hewett@canterbury.ac.nz
Website: https://www.fluids.nz/finz/2021/

Code of Conduct

As a New Zealand conference welcoming members from diverse backgrounds, organisations, and lived experiences, we are committed to the open exchange of ideas, freedom of thought and expression, and respectful debate. These require a community and an environment that recognises the inherent worth of every person and group, that fosters inclusion, dignity, understanding, and mutual respect, and embraces diversity.

Fluids in New Zealand (FiNZ) is dedicated to providing a harassment-free experience for everyone engaging with FiNZ 2021, regardless of gender, sexual orientation, disability, physical appearance, race, ethnicity, political affiliation, nationality, language, or religion – and not limited to these aspects. We are dedicated to cooperation, civility, and respect, and do not tolerate harassment in any form. Sexual or discriminatory language or imagery is not appropriate and will not be tolerated at this conference. Participants violating these rules may be sanctioned or expelled at the discretion of the organisers.

Harassment and hostile behaviour are not welcome at FiNZ. This includes speech or behaviour (including in public presentations and on-line discourse) that intimidates, creates discomfort, or interferes with a person's participation or opportunity for participation in the community or event. FiNZ 2021 aims to be an environment where harassment in any form does not take place, including but not limited to harassment based on race, gender, religion, age, colour, national origin, ancestry, disability, socioeconomic status, sexual orientation, or gender identity.

Harassment includes but is not limited to: inappropriate verbal comments as outlined above; sexual images in public spaces; deliberate intimidation, stalking, or following; harassing photography or recording; sustained disruption of talks or other events; inappropriate physical contact; unwelcome sexual attention; and advocating for or encouraging any of the above behaviour.

Participants asked to stop any harassing behaviour are expected to comply immediately.

Implementation

It is the responsibility of the FiNZ community as a whole to promote an inclusive and positive environment for our scholarly activities. If you are being harassed, notice that someone else is being harassed (active bystander principle), or have any other concerns, please contact a member of the FiNZ 2021 Organising Committee. Your support will help keep our community and our events a safe, welcoming, and friendly space for all fellow participants!

Protocol for conflict resolution

Report

1. If you are being harassed, notice that someone else is being harassed (active bystander principle), or have any other concerns, please immediately contact a member of the FiNZ 2021 Organising Committee.

Escalate

2. Find a member of the FiNZ 2021 Organising Committee. They will assist you. These parties will help participants contact hotel/venue security or local law enforcement, provide escorts, or otherwise assist those experiencing harassment and will help them to feel safe for the duration of the event.

3. If you have been harassed via email or social media, you may send emails or screenshots to phillip.wilson@canterbury.ac.nz.

Resolve

4. The FiNZ 2021 Organising Committee will require anyone engaging in harassing behaviour to cease immediately or face expulsion or other sanctions.

5. Those sanctions can include reporting to the individual's organisation, and supporting the victim in making informal or formal complaints via that organisation's processes.

6. If an incident results in corrective action, then FiNZ will support those harmed by the incident, both publicly (where appropriate) and privately.

7. If any individual encounters problems or issues attempting to help a victim of harassment while following our anti-harassment policy and protocol, they are encouraged to engage with the FiNZ community and FiNZ 2021 Organising Committee for clarifications and/or to seek revisions. We are very open to revising any part of our policy or protocol if needed to ensure a safe and welcoming community in which harassment is not tolerated.

This code of conduct was adapted with permission from that of Te Pūnaha Matatini.

Programme

Day 1: Wednesday 27 January

10:30 Registration

11:00 Welcome

Session 1: Biology & Microfluidics (chair: James Hewett)

11:15 Invited Speaker: Tracie Barber – *The fluid dynamics of arterio-venous fistulae for dialysis*

12:00 Sevgi Onal – *Mechanical cell stimulation and compression in a microfluidic device*

12:15 Yifei Ma and Jeffrey J. Wise – *Microfluidic systems for machine learning*

12:30 Lunch

Session 2: Non-Newtonian & Microfluidics (chair: Richard Clarke)

13:30 Samuel K. Irvine – *Granular flow in a two opening silo: extending the* $\mu(I)$ *model*

13:45 Megan Danczyk – *Numerical modelling of smooth sphere collisions in the pendular regime validated with experimental results*

14:00 Santhosh Kumar Pandian – *Non-Newtonian drop impacts: spread and retraction analysis on micropillars*

14:15 Miguel Moyers-Gonzalez – *To flow or not to flow: the Bingham (fluid) prerogative*

14:30 Sina Safaei – *Instability of slip-asymmetric Janus aggregates in shear flows: a molecular dynamics study*

14:45 Debolina Sarkar – *Can we prevent kauri dieback through the modification of electric fields in natural environment?*

15:00 Afternoon tea

Session 3: Multiphase (chair: Miguel Moyers Gonzalez)

15:30 Ayoub Abdollahi – *Impact of heated dairy droplets on surfaces*

15:45 Emilia Nowak – *Kinetics of polymerisation reaction of alginate gels*

16:00 Kirill Misiiuk – *Development of coating-free super water-repellent micropatterned aluminium for spontaneous droplet motion*

16:15 Andrew J. L. Lange – *Modelling bubble growth in a burning metal droplet*

16:30 Ross G. Shepherd – *Optimal control of spin coating on a spherical substrate*

16:45 Mathieu Sellier – *Thin liquid film dynamics on a spinning ellipsoid*

17:00 Staff club drinks

Day 2: Thursday 28 January

Session 4: Earth Science I (chair: Mathieu Sellier)

09:00 Invited Speaker: Rosalind Archer – *Coupled models in geothermal energy – multiphysics and multiprocess models*

09:45 Elizabeth K. McGeorge – *Recovery of bedrock topography in ice flows: an exploration using optimisation methods*

10:00 Craig D. McConnochie – *Aspect ratio affects iceberg melting*

10:15 Matthew Hayward – *Numerically modelling waves of explosive origin with application on submarine volcanism*

10:30 Morning tea

Session 5: Earth Science II (chair: Sam Lowrey)

11:00 Kunal K. Dayal – *Mesoscale wind resource mapping of the small island developing state (SIDS) of Fiji*

11:15 Michael MacDonald – *DNS of the moist stably stratified surface layer: turbulence and fog formation*

11:30 Pouria Aghajannezhad – *The effects of surface roughness on the flow in discrete fracture networks*

11:45 Sid Becker – *Low-resolution magnetic resonance velocimetry to measure velocity in porous media*

12:00 Poster session

12:30 Lunch

Session 6: Hydrodynamics (chair: Phil Wilson)

13:30 Invited Speaker: Callum Shakespeare – *The world of waves beneath the sea: how internal waves influence the global ocean and climate system*

14:15 Shuen Law – *Large eddy simulation of desalination discharges*

14:30 Michael J. Coe – *Investigation of scaling laws for underwater locomotion and propulsion efficiency*

14:45 Raghu Ande – *Investigation on fluid properties of small, medium and large array size of beams oscillating in fluids*

15:00 Afternoon tea

Session 7: Aerodynamics (chair: John Cater)

15:30 Theresa Bischof – *The effect of vane parameters on buoyancy vortices*

15:45 Warit Chanprasert – *Impact of wake meandering and turbulence on offshore wind turbines under different inflow conditions*

16:00 Duc Nguyen – *Using CFD to examine the sensitivity of incident wind directions on flow dynamics through constructed coastal dune notches*

16:15 Illia Chyrva – *Experimental investigation of upward velocity region between adjacent counter-rotating rotors using SPIV technique*

16:30 Michael J. Kingan – *Noise from large and small propellers*

16:45 James R. Ramsay – *Designing porous skins to reduce drag on bluff bodies*

17:00 Break

18:00 Dinner

Day 3: Friday 29 January

Session 8: Industry & Combustion (chair: Alan Caughley)

09:00 Invited Speaker: Sam Hamilton – *Using CFD and parametric CAD to design a new generation of reverse ducts for waterjet propulsion*

Invited Speaker: Curtis Marsh – *Industrial CFD applications*

09:45 Tao Cai – *Enhancing thermal performance by implementing a bluff-body in a hydrogen-fueled power system*

10:00 Siliang Ni – *Mitigation of NOx emission and energy conversion efficiency improvement study of ammonia powered micro-combustor with ring-shaped ribs in fuel-rich combustion*

10:15 Yuze Sun – *Effect of N_2 dilution on thermal performance and emission characteristics of an ammonia-oxygen micro-thermophotovoltaic system*

10:30 Morning tea

Session 9: Heat Transfer (chair: Sid Becker)

11:00 Gideon J. Gouws – *Evolution of substrate temperature and drop morphology during the evaporation of sessile milk drops*

11:15 Chris Hughes – *3D-printed anisotropic microchannels for improved water management in heat exchangers*

11:30 Diana Kommedal – *Analysis of the effect of infiltration in heat load calculations of a retrofitted office building in Christchurch – New Zealand*

11:45 Salma Radwan – *Empirical heat transfer correlation for expansion pulsed pressure mass transport regime*

12:00 James N. Hewett – *Natural shapes of melting ice blocks*

12:15 Closing

12:20 Annual meeting

Posters

Maryam Abdali – *Designing a PCM-based battery cooling system for an electric racing vehicle*

Ekaterina Lieshout – *Sit ski aerodynamics*

Sam Lowrey – *Experimental investigation of condensation on superhydrophobic surface tension gradients*

Kirill Misiiuk – *Surface wetting phenomena on combined micro-milled and ion-beam processed aluminium*

William Power – *Seiche effects in Lake Tekapo in an Mw8.2 Alpine Fault earthquake*

Oscar Punch – *Collision dynamics of wetted roughened particles*

James Robinson – *The non-local $\phi(I)$ response*

Frederick Steven Wells – *Stretch, splash, spike: ferrofluid drop impacts in non-uniform fields*

Session 1: Biology & Microfluidics

The fluid dynamics of arterio-venous fistulae for dialysis

Invited Speaker: Tracie Barber[1]
[1] *School of Mechanical and Manufacturing Engineering, UNSW Sydney, Australia*

Kidney failure patients require a surgically created arteriovenous fistulae (AVF) to enable vascular access for hemodialysis. Hemodialysis is extracorporeal blood filtration which is needed when the kidneys are no longer able to complete this function. AVF failure is a costly and life-threatening problem for dialysis patients and unfortunately common. Many of the complications are attributed to hemodynamics perturbations including turbulent flow, pathophysiological wall shear stress (WSS) and flow recirculation zones.

My group uses an in-house designed freehand ultrasound scanning system to enable the collection of patient geometry. We work closely with the Dialysis Unit at the Prince of Wales Hospital, Randwick Sydney, attending a clinic each week (pre-COVID!) where we work with vascular surgeons. By segmenting the ultrasound images and using the ECG-synchronised flowrates, we have the information needed to run CFD simulations and also develop benchtop models.

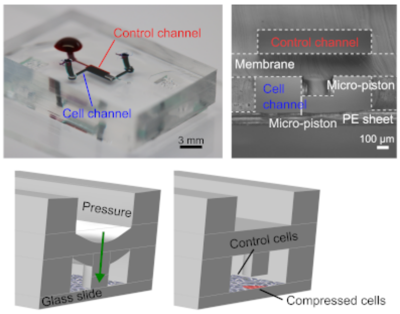
In a longitudinal study where we scanned on patient every week for 15 weeks, we were able to determine the first results for fistula maturation, demonstrating that while flow and cross-sectional area increased over time in the proximal artery and vein, wall shear stress remained fairly constant, while at a higher than baseline level. Our data has also allowed the effects of different surgical configurations to be modelled. Earlier work in our group showed that the traditional AVF geometry, with an acute angle of vein to artery at the juxta-anastomotic region, produced detrimental flow conditions. A "smooth loop" geometry, when modelled computationally, was seen to result in a more beneficial flow circuit. This has now been proven surgically, with patients who have the "smooth loop" geometry – often created via the use of a stent during an intervention – performing better longer term.

In order to create a predictive tool, we have considered a number of flow related parameters to determine if there is any correlation between these and AVF success rate. Hemodynamic impedance through the vascular access was calculated by combining the proximal artery and vein resistances. In one patient case, the first scan featured low flowrates with stenosis noted in two locations of the outflow vein. The second scan was taken after the AVF underwent straight stent implantation in the stenotic region. While greater flowrates results, a new stenosis region formed near the new stent. A further stenting procedure in the juxta-anastomotic region provided a smooth loop geometry and removed all stenoses. Oscillatory shear behaviours were seen at different locations in all scans, localised in regions of low WSS, however impedance was significantly reduced in the third scan, indicating successful restoration of the vascular access.

A comparative study of all scans to date shows excellent agreement between low impedance and AVF success.

Mechanical cell stimulation and compression in a microfluidic device

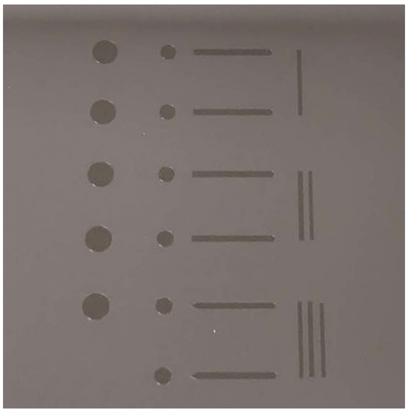
Sevgi Onal[1,2], Maan M. Alkaisi[1,2] and Volker Nock[1,2,3]
[1]*Electrical and Computer Engineering, University of Canterbury, New Zealand*
[2]*MacDiarmid Institute for Advanced Materials and Nanotechnology, New Zealand*
[3]*Biomolecular Interaction Centre, University of Canterbury, New Zealand*



Biomechanics is well-studied for cells in tissues that have explicit biomechanical properties, such as vasculature, muscle, and joints. Less is known about cancer cells which also respond to mechanical forces just like every cell in the body. Among the mechanobiological aspects of cancer research, the influence of compressive forces in particular has been largely neglected until very recently. As such, there is significant demand to investigate the impact of external compressive forces on cancer cells. A novel and comprehensive way to do this, in contrast to bulk compression systems, is to use flexible microdevices to better mimic physiological force values. Here, we present a microfluidic platform with an integrated, actively modulated actuator for the application of compressive forces on cancer cells. The platform can apply compression on cancer cells in a dynamic and controlled manner by modulating applied gas pressure, localization, shape and size of the device compartments. The mechanical actuator was characterized experimentally and computationally, as were cell loading and culture in the device. The actuator was used to perform both, repeated dynamic cell compression at physiological pressure levels, and end-point mechanical cell lysis, demonstrating suitability for mechanical stimulation to study the role of compressive forces in cancer microenvironments.

Microfluidic systems for machine learning

Yifei Ma[1], Jeffrey J. Wise[1], Le Yang[1] and Ciaran P. Moore[1]
[1] *Department of Electrical and Computer Engineering, University of Canterbury, New Zealand*



Droplet microfluidics is an exciting field with known applications in environmental monitoring, medical diagnostics and lab-on-a-chip applications. One relatively unexplored feature of this technology is that the individual nature of the droplets makes them ideal for representing discrete time series data, such as those used in biomimetic spike-based neural networks (SNNs). Like various neurons in the brain, SNNs use series of discrete pulses to communicate information, which is primarily encoded in the temporal frequency of the spikes or pulses. This means that the strength of the signals between neurons is measured in terms of the spike firing rate, which is directly analogous to the droplet flow rate in microfluidic channels.

Here we present an update on our investigations into realising an artificial neural network using microfluidic channels. We present work on droplet generators and combiners, which are necessary for encoding data and mimicking synaptic connections between neurons. We also discuss some of the challenges faced by microfluidic neural networks and evaluate recent advances in artificial neural network architecture that may alleviate these concerns.

Session 2: Non-Newtonian & Microfluidics

Granular flow in a two opening silo: extending the $\mu(I)$ model

Samuel K. Irvine[1], Luke A. Fullard[1], Thomasin A. Lynch[1], Daniel J. Holland[2] and Daniel A. Clarke[3]
[1]*School of Fundamental Sciences, Massey University, New Zealand*
[2]*College of Engineering, University of Canterbury, New Zealand*
[3]*School of Chemical and Physical Sciences, Victoria University of Wellington, New Zealand*

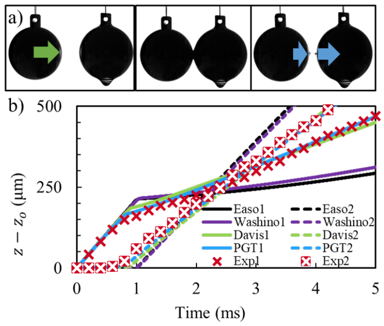
Granular material, such as sand, soil, grain, and powders, is widely used in many kinds of industrial processes. While many models have been developed to attempt to characterise the behaviour of these materials, a comprehensive model for granular flow has not been developed which captures all phenomena which can be observed in granular materials. While the recently developed $\mu(I)$ model provides good predictions for many aspects of granular flow, it falls short in some areas such as flow rate in a silo.

Silos provide a variety of different regimes of flow, with some areas of the silo being stagnant with other areas being in near freefall, providing an excellent testing ground for many different phenomena. In addition to the dynamics displayed by a normal silo, a silo with two openings shows a flow rate "dip" when the openings are close together. Examination of this phenomena could give insight into the dynamics of flowing granular material.

Some models have been developed to capture effects like segregation, dilatancy, and non-locality, however these models have not been applied to silos, and have not been applied together. While the basic $\mu(I)$ model is able to capture some of the dynamics of the two opening silo, extending the model to include segregation, dilatancy, and non-locality and testing this extended model in a silo may provide new insights into the mechanics of these everyday materials.

Numerical modelling of smooth sphere collisions in the pendular regime validated with experimental results

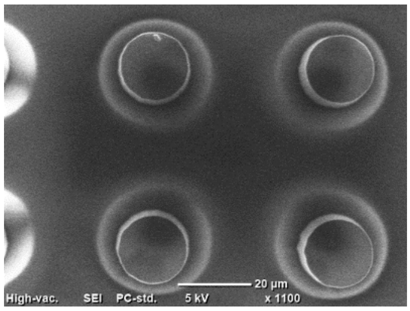
Megan Danczyk[1], Oscar Punch[1], Mathew Hawken[1], Luke A. Fullard[2] and Daniel J. Holland[1]
[1]*Department of Chemical and Process Engineering, University of Canterbury, New Zealand*
[2]*School of Fundamental Sciences, Massey University, New Zealand*



Many industrial and natural applications of granular flow have liquid present that changes the dynamic of the flow interaction. Here, we observe liquid-coated granular collisions in the pendular state, where liquid bridges form at contact points of grains but the pores between particles are empty. We particularly consider cases where the capillary number is high and viscous forces dominate. First, high-speed particle tracking velocimetry was used to study the collisions of two and three smooth spheres with and without liquid coatings. By varying the Stokes number, we find four possible rebound outcomes with three spheres: Newton's cradle, reverse Newton's cradle, full agglomeration, and full separation. We then use these experiments to examine three theoretical models of these wet granular flows. A key limitation of these models is how they address the divergence of the force that is predicted as particles approach each other. Most existing models impose a limit to the viscous force. One recent model instead assumes a hard sphere collision. These fundamentally different approaches produce different rebound outcomes. A fair match between experiments and simulations was obtained when limiting the viscous force. However, an empirically-fitted, and non-physical, maximum force is required. Alternatively, the hard sphere collision model also matches the experimental results well. The hard sphere model still requires an empirically-fitted parameter, the glass transition pressure, however, this parameter appears to be consistent between different experimental conditions and previous work. Thus, it appears that the hard sphere model is better able to describe the collisions between smooth particles studied here.

Non-Newtonian drop impacts: spread and retraction analysis on micropillars

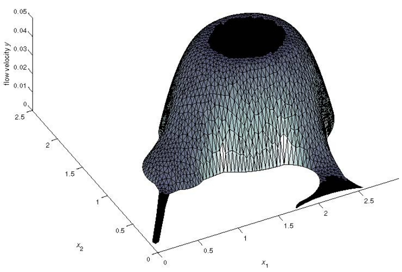
Santhosh Kumar Pandian[1,2], Miguel Balzan[1,2] and Geoff R. Willmott[1,2]
[1]*Department of Physics, The University of Auckland, New Zealand*
[2]*The MacDiarmid Institute for Advanced Materials and Nanotechnology, New Zealand*



Droplet interactions with micro-patterned surfaces are important for numerous microfluidic applications such as surgical endoscopes, biological fluids, MEMS, and applications related to cell growth platforms. Understanding drop dynamics is essential for controlling and optimizing drop deposition on any surface. Drop spreading and retraction on a patterned surface has been studied widely for Newtonian fluids, but impacts of non-Newtonian fluids on patterned surfaces have not been studied in detail. Drop impact of non-Newtonian fluids is of significant interest owing to the common use of such fluids in industrial and biological processes. In this study, the effect of surface patterning on drop hydrodynamics has been studied for both Newtonian and non-Newtonian solutions, and preliminary results will be discussed. We aim to investigate the underlying mechanism between surface patterns and drop dynamics for various solutions. The figure is an SEM image of patterned circular micropillars used in experiments.

To flow or not to flow: the Bingham (fluid) prerogative

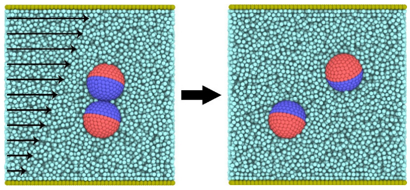
Miguel Moyers-Gonzalez[1] and Ian A. Frigaard[2]
[1]*School of Mathematics and Statistics, University of Canterbury, New Zealand*
[2]*Departments of Mathematics and Mechanical Engineering, University of British Columbia, Canada*



In this talk, I will present the differential form of the conservation equations for a Bingham fluid and its equivalent variational formulation. Through the variational problem, one can develop numerical algorithms for the approximation of velocity, stress and rate of deformation fields. I will briefly explain one of them, the Augmented Lagrangian Algorithm. In addition, in some situations, one can derive exact conditions for which there is no flow for a fixed pressure drop in a given domain. I will go through some examples for which this is possible.

Instability of slip-asymmetric Janus aggregates in shear flows: a molecular dynamics study

Sina Safaei[1], Geoff R. Willmott[1,2] and Shaun Hendy[1,3]
[1]*MacDiarmid Institute for Advanced Materials and Nanotechnology, Department of Physics, University of Auckland, New Zealand*
[2]*School of Chemical Sciences, University of Auckland, New Zealand*
[3]*Te Pūnaha Matatini, Department of Physics, University of Auckland, New Zealand*



The influence of surface wettability in micro- to nanoscale fluid dynamics has been of considerable recent interest, particularly in relation to slip boundary conditions. Amphiphilic Janus nanoparticles have slip-asymmetric boundary conditions due to different wettabilities of their faces. These nanoparticles can self-assemble into complex nanostructures in order to minimise the unfavourable interactions between the solvent and hydrophobic sides. Theoretical studies have shown that slip-asymmetric nanoparticles can also experience a torque in fluid flows. The torque on individual particles destabilises Janus aggregates which promotes their break-up into isolated spheres. In this work, we investigate the effect of the slip length on the thermal and shear break-up of amphiphilic Janus dimers (see figure). We will present our latest results on the possible break-up mechanisms and propose a descriptive equation for calculation of the break-up rate. The figure shows when an amphiphilic Janus dimer is in a shear flow, individual particles of the dimer encounter torque in opposite directions. This promotes break-up of the dimer into isolated spheres.

Can we prevent kauri dieback through the modification of electric fields in natural environment?

Debolina Sarkar[1], Ashley Garrill[1] and Volker Nock[2]
[1]*Department of Biological Sciences, University of Canterbury, New Zealand*
[2]*Department of Electrical and Computer Engineering, University of Canterbury, New Zealand*

The oomycete *Phytophthora* can cause serious threats to native flora, agriculture and food biosecurity. In New Zealand, *Phytophthora agathidicida* and *Phytophthora cinnamomi* are the causative agent of 'kauri-dieback' and is receiving much coverage in the mainstream press, as it can kill a mature kauri tree within few years. These organisms can infect their host plants via motile zoospores. They can swim from site of infection to healthy trees. Zoospores can detect electro-chemical signals released by the plant roots. They can even discriminate between the growing tip and wounded surface on the roots from rest of the root structure, depending on the variance in electric-filed. This helps them to find the specific target area and initiate infection.

My research focuses on mimicking the natural environment around the plant roots, using microfluidic Lab-on-a-Chip (LOC) devices to investigate the electrical parameters combined with the chemical. LOC devices with electrodes have been designed, enabling the study of the swimming behaviour of *Phytophthora* zoospores in presence of electric-fields. Preliminary data indicates that zoospores show a tendency to aggregate around cathode. This device will also allow us to study the signal transmitted from individual zoospore to other zoospores that intensify the infectious effect. In future this research may allow us to design devices that modify electric fields around vulnerable plants and attract zoospores away from roots and thus inhibit their infective capability.

Session 3: Multiphase

Impact of heated dairy droplets on surfaces

Ayoub Abdollahi[1], Frederick S. Wells[1,2] and Geoff R. Willmott[1,2,3]
[1]*The Department of Physics, The University of Auckland, New Zealand*
[2]*The MacDiarmid Institute for Advanced Materials and Nanotechnology, New Zealand*
[3]*School of Chemical Sciences, The University of Auckland, New Zealand*

The impact of heated dairy droplets on stainless steel surfaces is experimentally studied using a free flight single droplet drying technique. A syringe pump was used for reproducible droplet generation, the droplet release height was constant (500 mm), and droplet spreading on the substrates was recorded via high-speed camera. The experiments were carried out under controlled temperature and humidity within an environmental chamber. The chamber temperature was adjusted ranging from 22 °C to 200 °C with a constant humidity. Reconstituted milk was used for consistent liquid preparation. The data obtained from the image analysis of the recorded video was used to determine the effect of solids concentration and temperature variation on droplet spreading and drying.

Kinetics of polymerisation reaction of alginate gels

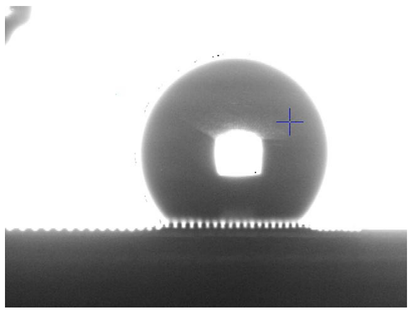
Emilia Nowak[1], Kate Maslova[2] and Geoff R. Willmott[3]
[1] *School of Food and Advanced Technology, Massey University, Manawatu, New Zealand*
[2] *School of Food and Advanced Technology, Massey University, Albany, New Zealand*
[3] *Department of Physics, University of Auckland, Auckland, New Zealand*

Food grade hydrogels are cheap, non-toxic, biocompatible and widely used, e.g. alginate gels are used in food, cosmetic, pharmaceutical and medical industries. Alginate polymerisation reaction is assumed to be instantaneous. However, with the increasing applications and technological innovations, lack of knowledge on the exact kinetics of polymerisation prevents the development of more sophisticated processing and manufacturing conditions.

Recently developed drops coalescence rig (Nowak et al, Colloids and Surfaces A 505 (2016) 124-131) allows probing the rates of considered instantaneous reaction using visual aids. The drop containing monomer (alginate) is being contacted with drop containing polymerising agent (calcium) and the progress of the polymerisation reaction has been successfully visualised with high speed camera. It has been found that the polymerisation reaction is in the same timescale as the bridge expansion of droplets undergoing coalescence. Furthermore, the addition of surfactants proved that the polymerisation reaction is overtaken by Marangoni flows, showing promising encapsulation applications.

Development of coating-free super water-repellent micropatterned aluminium for spontaneous droplet motion

Kirill Misiiuk[1,2], Sam Lowrey[1,2], Richard Blaikie[1,2], Josselin Juras[3] and Andrew Sommers[3]
[1]*Department of Physics, University of Otago, New Zealand*
[2]*MacDiarmid Institute for Advanced Materials and Nanotechnology, New Zealand*
[3]*Department of Mechanical & Manufacturing Engineering, Miami University, Ohio, USA*



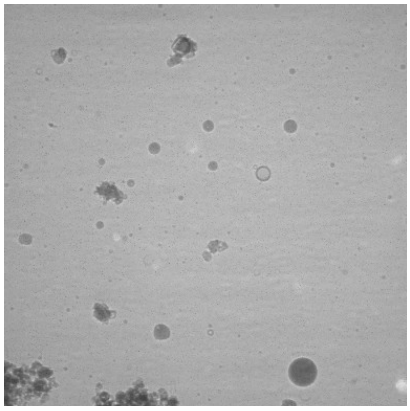
Nature shows various approaches to create superhydrophobicity, such as a lotus leaf, where the superhydrophobic (SHPB) surface arising from its hierarchical surface consists of random microscale bumps with superimposed nanoscale hairs. Some natural systems have even evolved to passively transport water droplets from one part of a surface to another. For example, the hydrophilic spider-silk of some spider's webs can promote passive motion of sub-millilitre water drops through spindle-knot/joint couplings, creating gradients in surface tension and Laplace pressure. We look to combine both ideas and replicate the super-water repellence of the lotus leaf and the surface tension gradient driven motion of spider silk in an all metal, coating-less surface.

SHPB surfaces promote dropwise condensation over filmwise, increasing liquid mobility should a driving force be present. A surface tension gradient can be used to generate in-plane actuation forces to induce spontaneous droplet motion that may allow surfaces to more rapidly remove water droplets, suppressing ice-/frost-formation under extreme weather conditions. Such surfaces could potentially be used for wind turbine blades and refrigeration heat exchanger surfaces, improving energy efficiency.

We present a survey of our microstructures and their resultant wetting properties on fixed-pitch and variable-pitch microstructures. In addition, we report progress on the development of all metal hierarchical superhydrophobic gradient surfaces. The figure shows a water droplet in the Cassie-Baxter micro-wetting state on our laser-etched Al surface.

Modelling bubble growth in a burning metal droplet

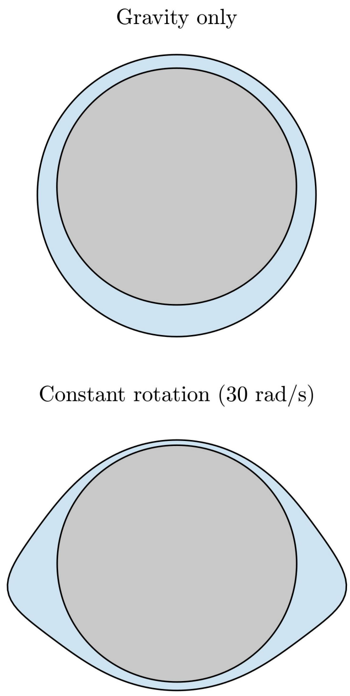
Andrew J. L. Lange[1], Mathieu Sellier[1], James N. Hewett[1], Elliot R. Wainwright[2] and Timothy P. Weihs[2]
[1]*Department of Mechanical Engineering, University of Canterbury, New Zealand*
[2]*Materials Science & Engineering, Johns Hopkins University, Baltimore, US*



Metal powders are a common additive in a number of applications in which high gravimetric and volumetric energy densities from combusting are desirable. For some pure metals, such as Zr and Ti, particles undergoing combustion exhibit small bubbles growing inside the molten droplet particles which is generally attributed to the release of gaseous nitrogen from solution through diffusion and then desorbing across the interface into the bubble. If the growth of this bubble is very large it causes the droplet to split apart in a process termed a "microexplosion". To better understand factors that lead to these microexplosions, the growth of the bubble was modelled and compared to data from experiments done by Wainwright et al. (2019) for an Al:Zr composite powder to find the scheme for molar flow of nitrogen that has the most agreement with the measurements. The model is developed from the Navier-Stokes equations in spherical coordinates while applying assumptions and substituting expressions of mass continuity for simplification to coupled first order ordinary differential equations which may then be solved using MATLAB `ode` functions. Within the limits of uncertainty the model and data agree best when the flow of nitrogen into the bubble is linearly proportional to the bubble radius. These results offer more insights towards the rate at which molecules of gas are adsorbed onto an interface and the mechanisms of mass transfer in high temperature liquid metal solutions.

Optimal control of spin coating on a spherical substrate

Ross G. Shepherd[1], Mathieu Sellier[2] and Edouard Boujo[3]
[1]*Department of Chemical and Process Engineering, University of Canterbury, New Zealand*
[2]*Department of Mechanical Engineering, University of Canterbury, New Zealand*
[3]*Laboratory of Fluid Mechanics and Instabilities, École Polytechnique Fédérale de Lausanne, Switzerland*



Spin coating is a technique commonly used to apply micrometre-scale coatings in electronics manufacturing. Current spin coating methods cannot, however, be used to produce a uniform coating on curved surfaces. We investigate the feasibility of improving spin coating performance by varying the spin speed throughout the coating process. A model for the flow of a solidifying fluid film on the surface of a rotating spherical substrate is derived from lubrication theory subject to surface tension, centrifugal, and gravitational forces. This is then used to determine the effects of different process parameters, including substrate kinematics and fluid properties, on the coating performance. Our model is used to formulate a constrained optimal control problem which is solves with an adjoint equation to show that spin coating cannot be used to improve the coating of the entire surface of a sphere when compared to drainage under gravity alone. We finally suggest alternative conditions under which it may be possible to use spin coating to produce a uniform film on a curved substrate.

Thin liquid film dynamics on a spinning ellipsoid

Selin Duruk[1], Mathieu Sellier[1] and Edouard Boujo[2]
[1]*Department of Mechanical Engineering, University of Canterbury, New Zealand*
[2]*Laboratory of Fluid Mechanics and Instabilities, École Polytechnique Fédérale de Lausanne, Switzerland*

Surfaces are often coated with a decorative, a protective, or a functional layer. Whilst coating a at surface can routinely be achieved with a high degree of accuracy and reliability, coating curved surfaces is much more challenging. In that context, this work explores the impact of a centrifugal force on a thin liquid layer deposited on an ellipsoid rotating around its vertical principal axis is investigated to determine the effects of non-constant curvature together with gravity and capillary forces. A simplified evolution equation is derived in the framework of the lubrication approximation. The validity of the model is confirmed with direct numerical simulations of the full Navier-Stokes equations, and the system's behaviour is examined for a wide range of parameter set. The relative effects of the aspect ratio (characteristic film thickness over characteristic curvature), the ratio of the eccentricities and various forces at play (gravity, surface tension and rotation, characterised by the Bond number and Galilei number) on the interface thickness distribution are reported.

Session 4: Earth Science I

Coupled models in geothermal energy – multiphysics and multiprocess models

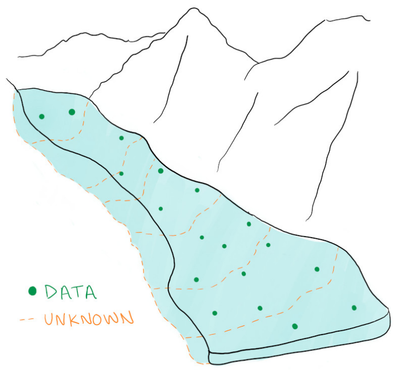
Invited Speaker: Rosalind Archer[1], Musa Aliyu[1], Pengliang Yu[1] and Mohammed Dabbour[1]
[1]*Department of Engineering Science, The University of Auckland, New Zealand*

This talk will outline recent progress on coupled thermo-hydro-mechanical reservoir models (multiphysics) in geothermal energy. Such models are necessary if we wish to consider the development of "engineered/enhanced" geothermal systems where rock permeability is created through engineered processes. Models explore detailed considerations of single fractures in the rock will be considered. The design and performance of wells with multiple fractures will also be presented.

The second part of the talk will present a coupled "multiprocess" modelling approach where we couple heat and mass flow in a reservoir model, with fluid flow modelling through electricity production facilties. We have a specific interest in tracking CO_2 through the system in an effort to understand its impact on phase behaviour and plant performance – and ultimately to support efforts to ensure geothermal energy production is as close to carbon neutral as possible.

Recovery of bedrock topography in ice flows: an exploration using optimisation methods

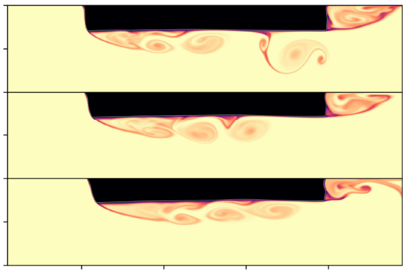
Elizabeth K. McGeorge[1], Miguel Moyers Gonzalez[1], Mathieu Sellier[2] and Phil Wilson[1]
[1]*School of Mathematics and Statistics, University of Canterbury, New Zealand*
[2]*Department of Mechanical Engineering, University of Canterbury, New Zealand*



Non-linear fluid flow problems appear regularly in the physical world. It is therefore useful to have methods of constructing good estimates for fluid parameters based on observation data. Our current work explores doing this (with a specific interest in ice flows) by fitting a model to the surface data. To do this, we consider a cost functional comparing the modelled flow surface and the observed surface. To find the fluid parameters, the functional needs to be minimised subject to PDE constraints. We present some initial results of this optimisation.

Aspect ratio affects iceberg melting

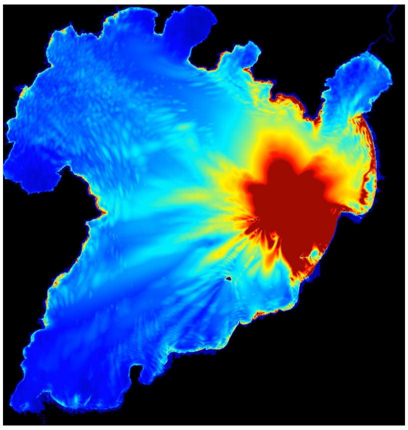
Eric W. Hester[1], Craig D. McConnochie[2], Claudia Cenedese[3], Louis-Alexandre Couston[4,5] and Geoffrey Vassil[1]
[1]*School of Mathematics and Statistics, University of Sydney, Australia*
[2]*Department of Civil and Natural Resources Engineering, University of Canterbury, New Zealand*
[3]*Department of Physical Oceanography, Woods Hole Oceanographic Institution, USA*
[4]*British Antarctic Survey, UK*
[5]*Laboratoire de Physique, ENS de Lyon, France*



Iceberg meltwater is a critical freshwater flux from the cryosphere to the oceans. Global climate simulations therefore require simple and accurate parameterisations of iceberg melting. Iceberg shape is an important but often neglected aspect of iceberg melting. Icebergs have an enormous range of shapes and sizes, and distinct processes dominate bottom and side melting. We show how different iceberg aspect ratios and relative ambient water velocities affect melting using a combined experimental and numerical study. The experimental results show significant variations in melting between different iceberg faces, as well as within each iceberg face. These findings are reproduced and explained with novel multiphysics numerical simulations. At high relative ambient velocities melting is largest on the side facing the flow, and mixing during vortex generation causes local increases in basal melt rates of over 50%. Double-diffusive buoyancy effects become significant when the relative ambient velocity is low. These findings are not reproduced with existing melting parameterisations.

Numerically modelling waves of explosive origin with application on submarine volcanism

Matthew Hayward[1], Colin Whittaker[1] and Emily Lane[2]
[1]*Department of Civil and Environmental Engineering, The University of Auckland, New Zealand*
[2]*NIWA Taihoro Nukurangi, Christchurch, New Zealand*



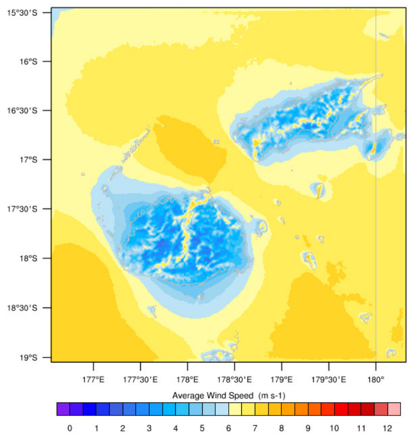
Tsunamis initiated by volcanoes have the ability to greatly extend the hazard footprint of an eruption far beyond the proximity of the eruption itself. Around 26% of direct volcanic fatalities are attributed to these volcanogenic waves. Alongside flank collapse and forcing via pyroclastic flow, an eruptive series of progressively shoaling explosions can generate significant waves, especially in lakes. Due to practical limitations of volcanic observation, the understanding of wave making potential from subaqueous eruptions is poor. Prior studies utilised models of surface waves produced from analogous chemical and nuclear explosions. However, these are derived from dated naval research and require reassessment.

A non-hydrostatic, vertically-Lagrangian multilayer method for free-surface flows is verified against a laboratory flume experiment to assess suitability for modelling waves produced by variable size disturbances. This is then used to evaluate free-surface initial conditions on a U.S. Army submerged explosive series on generating waves in Mono Lake, California. On establishing fitness of the underlying models, these are applied to simulate an analogous hypothetical scenario of a single explosive submarine eruption at Lake Taupō. This approach will help inform and complete hazard maps of local area submarine volcanism.

Session 5: Earth Science II

Mesoscale wind resource mapping of the small island developing state (SIDS) of Fiji

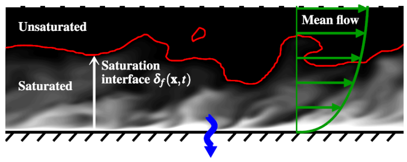
Kunal K. Dayal[1], John E. Cater[2], Michael J. Kingan[1], Gilles D. Bellon[3] and Rajnish N. Sharma[1]
[1] *Department of Mechanical Engineering, University of Auckland, New Zealand*
[2] *Department of Engineering Science, University of Auckland, New Zealand*
[3] *Department of Physics, University of Auckland, New Zealand*



This study presents the mesoscale wind-resource assessment of the SIDS of Fiji using the Weather Research and Forecasting (WRF) model at a grid resolution of 1 km by 1 km using 1deg gridded data from NCEP-FNL. Simulations were performed for 10 years (2009 – 2018) using the dynamical downscaling method employing the tropical suite of the physics parameterization scheme and the two-way nested approach. Our analysis evaluates the wind speed distribution and the Weibull PDFs, the diurnal and annual wind speed patterns, resource maps of annual and seasonal wind speed and the model statistical analysis. The results revealed that the WRF model simulated wind resource parameters are in good agreement with the observations. Higher wind speeds are observed during austral winter than in austral summer. Numerous high wind-resource areas are identified in this study, which were previous unknown. The annual mean wind speed and wind power density varies from 1.5 m/s to 8 m/s and 50 W/m^2 to 300 W/m^2, respectively for onshore land areas at 55 m elevation. This indicates that there is potential for utility-scale wind power generation at selected locations with wind speed and power density greater than 6.4 m/s and 300 W/m^2 (NREL, Wind Power Class 3).

DNS of the moist stably stratified surface layer: turbulence and fog formation

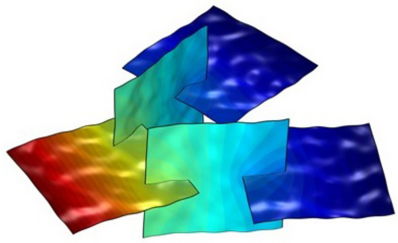
Michael MacDonald[1], Marcin J. Kurowski[2] and João Teixeira[2,3]
[1]*Department of Mechanical Engineering, University of Auckland, New Zealand*
[2]*Jet Propulsion Laboratory, California Institute of Technology, Pasadena CA, USA*
[3]*Joint Institute for Regional Earth System Science and Engineering, University of California, Los Angeles, Los Angeles CA, USA*



We investigate the effects of shear-driven turbulence on fog formation in the stably stratified surface layer, using direct numerical simulations (DNS) of dry and moist open-channel flows. A constant cooling rate is applied at the ground to mimic longwave radiative cooling, leading to either turbulent (weakly stable) or laminar (very stable) flows. Compared to the dry case, the condensation of liquid water in the moist case supports slightly higher cooling rates before leading to turbulence collapse. In the very stable moist cases, runaway cooling leads to substantial condensation of liquid water close to the ground and fog (visibility less than 1km) results over much of the domain. In the weakly stable cases, turbulent mixing narrowly yields visibilities of 1km close to the ground over a similar time period. Despite the idealised nature of the system, the present results suggest that turbulence impedes, although will not necessarily inhibit, fog formation.

The effects of surface roughness on the flow in discrete fracture networks

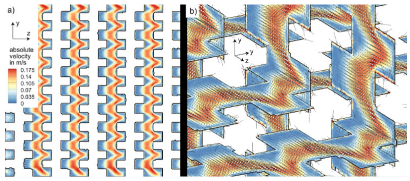
Pouria Aghajannezhad[1] and Mathieu Sellier[1]
[1]*Department of Mechanical Engineering, University of Canterbury, New Zealand*



We present a novel computationally efficient approach for investigating the effect of surface roughness on the fluid flow in discrete fracture networks at low Reynolds number. The effect of parallel and series fracture arrangements on the flow rate and hydraulic resistance was studied numerically by patching Hele-Shaw (HS) cells to represent the network. In this analysis, the impact of surface roughness was studied in different arrangements of the network. For this aim, four models with different sequences of fracture connections were established. The validity of the models was assessed by comparing the results with solutions of the full Navier-Stokes equations (NSE). The approximate hydraulic resistance and flow rate calculated by the HS method were found to be in good agreement with the NSE (less than 7 % deviation). Results suggest a quadratic relationship between the network hydraulic resistance and the joint roughness coefficient (JRC). Notably, an increase in surface roughness caused growth in hydraulic resistance and fall in flow rate. Further insight was provided by drawing an analogy between resistors in electrical circuits and fractures in networks.

Low-resolution magnetic resonance velocimetry to measure velocity in porous media

Sid Becker[1] and Martin Bruschewski[2]
[1]*Department of Mechanical Engineering, University of Canterbury, New Zealand*
[2]*Universitat Rostock, Rostock, Germany*



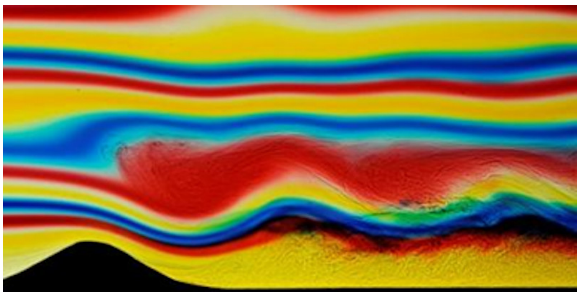
In this study, Magnetic Resonance Velocimetry (MRV) is used to measure the velocity field within porous media. The method presented here relies on the unique properties of MRV that the data contains quantitative velocity information even if the voxel has a mixed signal of fluid and matrix material. Conventional MRV experiments in porous media aim at resolving the flow field at sub-pore scale. A sufficiently high resolution is required which can make the experiment expensive and often impractical. The MRV method presented here does not require that any part of the porous matrix is resolved: the voxel size is purposely chosen larger than all solid structures. As a result, low-resolution MRV measurements provide the internal velocity field without having to resolve the solid structures. Image segmentation is not required since all voxels contain quantitative velocity data. Compared to high-resolution MRV measurements, these low-resolution measurements are fast and have considerably lower velocity uncertainty. The method is verified in a flow experiment with a known porous structure and demonstrate the new capabilities with detailed measurements of the three-dimensional velocity field in an unstructured filter foam. In conclusion, the presented method allows simple fluid mechanics experiments in many porous media applications, where conventional high-resolution MRV measurements are too difficult or simply not possible. The figure shows segmented three-dimensional velocity field of the porous cube experiment obtained with the highest resolution of 0.42 mm: a) selected slice covering most of the porous sample, b) detailed view including velocity vectors.

Session 6: Hydrodynamics

The world of waves beneath the sea: how internal waves influence the global ocean and climate system

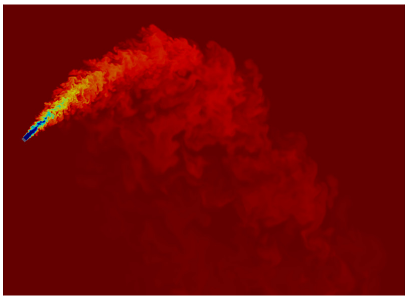
Invited Speaker: Callum Shakespeare[1]
[1]*Research School of Earth Sciences, Australian National University, Australia*

Internal gravity waves are the propagating motions generated when a stratified fluid such as the ocean is pushed away from its equilibrium state. Unlike the more familiar surface gravity waves, internal waves propagate three-dimensionally through the ocean interior. These waves play a number of crucial roles in the global ocean and climate system, some of which are only just beginning to be recognised. Breaking internal waves are responsible for mixing the deep ocean and thereby sustaining the abyssal branch of the oceanic overturning circulation. The forces exerted during their generation also control the amplitude of the ocean tides. Furthermore, the momentum carried by internal waves can enhance currents and eddies in the upper ocean, influencing air-sea exchanges of heat and carbon dioxide. Internal waves are challenging to observe and model owing to their very short lengthscales and fast timescales, and consequently, they remain unresolved in global ocean and climate models. Fundamental understanding of internal wave dynamics is therefore necessary in order to develop parameterisations of their impacts on the resolved flow in such models. In this talk I will review the current state of knowledge of oceanic internal waves and their impacts, and ongoing efforts in parameterisation development. The figure shows breaking internal waves generated by flow over a model hill in the Geophysical Fluid Dynamics Laboratory at the Australian National University.



Large eddy simulation of desalination discharges

Shuen Law[1], Mark J. Davidson[1], Roger I. Nokes[1] and Daniel Lagrava[2]
[1]*Department of Civil and Natural Resources Engineering, University of Canterbury, New Zealand*
[2]*The National Institute of Water and Atmospheric Research (NIWA), New Zealand*

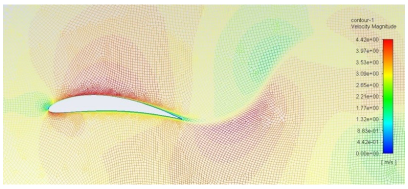


Elevated salinity and contaminant levels in the effluent discharged from large-scale desalination plants are potential threats to the coastal environment. Therefore, many previous researchers have developed theoretical models to predict the behavior of the flow generated from desalination discharges, known as inclined negatively buoyant jets (INBJs). However, substantial discrepancies between the model outputs and experimental data remains. These discrepancies are partly due to the lack of understandings on their physics, because it is difficult to obtain non-intrusive three-dimensional and high spatial-temporal resolution flow measurements. Computational fluid dynamics (CFD) simulations potentially offer the opportunity to analyse these flows in more detail, because they provide the freedom to extract three-dimensional information in various forms.

This study performs large eddy simulations (LES) of INBJs with the adaptive mesh approach using the finite volume method of an open-source CFD library, OpenFOAM. The results are compared against published experimental data. Analyses to date show that the errors in some bulk flow parameter predictions are reduced by up to 90% when compared with other published LES simulations. The simulations are ongoing and will provide more temporal data for further analyses. The outcomes of the simulation provide a framework for improving the existing theoretical models of INBJs.

Investigation of scaling laws for underwater locomotion and propulsion efficiency

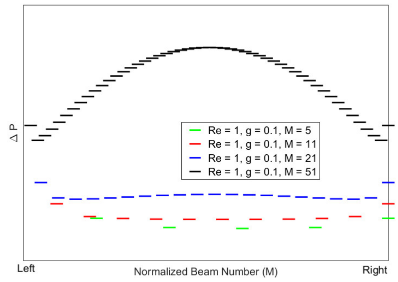
Michael J. Coe[1] and Stefanie Gutschmidt[1]
[1]*Department of Mechanical Engineering, University of Canterbury, New Zealand*



Current research trends in bio-inspired robots have focused on precise modelling and replication of fish locomotion. This type of propulsion has been shown to be up to 1.5 times more efficient than traditional propellers in underwater applications. The current thinking is that outfitting an autonomous underwater vehicle with a flapping propulsors will make it more efficient than traditional propeller vehicles. This research explores the idea that focusing only on propulsion efficiency does not tell the whole story of an efficient underwater vehicle. Using what is called the Cost of Transport, a total system metric of efficiency, various locomotion gaits are studied in terms of energy expended. Whole system scaling laws for flapping and propeller vehicles are formulated in order to better understand the direct comparison between different propulsion mechanisms. This is accomplished with a combination of computational fluid dynamic simulations and experimental robotic and biological data. Our work aims to give insight into where bio-inspired components and traditional components can be utilized to create an overall more efficient underwater vehicle.

Investigation on fluid properties of small, medium and large array size of beams oscillating in fluids

Raghu Ande[1], Stefanie Gutschmidt[1] and Mathieu Sellier[1]
[1] *Department of Mechanical Engineering, University of Canterbury, New Zealand*



An array of beams are finding a wide range of applications in fluid environments such as in, atomic force microscopy, sensors, biology and medicine. Therefore it is important to investigate the effect of fluid properties, when an array of beams oscillate in a fluid environment.

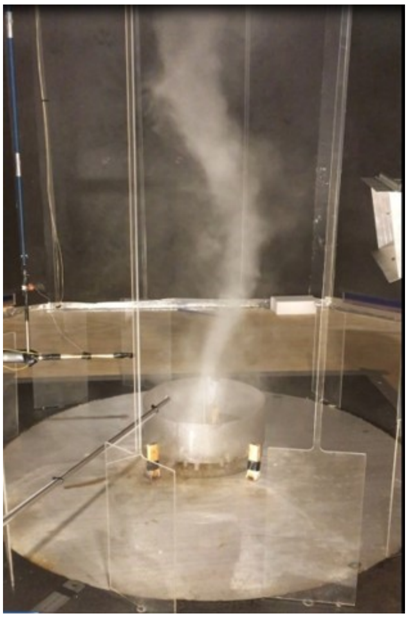
In this work we investigate the overall hydrodynamics of arrays distinguish between sizes of small, medium and large. We are using a well-known closed-form analytical formulation derived from boundary integral method. The analysis is carried out for the velocity configuration of all beams oscillating in an unbounded fluid. We investigate the arrays for different gaps between the beams at different Reynolds numbers.

First results suggest a classification of fluid effects for small to large array sizes (see figure) at critical gaps between the beams for different Reynolds numbers.

Session 7: Aerodynamics

The effect of vane parameters on buoyancy vortices

Theresa Bischof[1], Michael MacDonald[1], John E. Cater[2] and Richard G.J. Flay[1]
[1] *Department of Mechanical Engineering, University of Auckland, New Zealand*
[2] *Department of Engineering Science, University of Auckland, New Zealand*

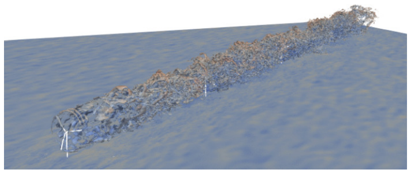


Buoyancy vortices like dust devils are well-formed low-pressure vortices originating from rotating unstable near-surface heated air, and are commonly observed in the turbulent convective boundary layer of the Earth's atmosphere. The dynamics and flow behaviour of buoyancy vortices are complex such that full-scale and laboratory measurements have limitations related to the duration of the vortex, vortex path, and in capturing all aspects of the flow particularly measuring the velocity close to the ground. Therefore, numerical techniques can be extremely beneficial in simulating and investigating the flow patterns and effects of various parameters on these flows in detail. A numerical model was developed using the Unsteady Reynolds-Averaged Navier-Stokes (URANS) method to study the structure, dynamics and flow patterns of buoyancy vortices. Radial vanes and a centrally heated plate produce a buoyancy force for developing a vertical vortex. The effect of radial vane height and angle were studied to examine their effects on the vortex dynamics. It was found that vane height has a minor effect on vortex strength, and the vane angle that produces the strongest vortex, i.e. the largest swirl ratio, is approximately 45°.

The photo of the laboratory vortex was taken by Neil Hawkes, PhD student at the University of Auckland. The vanes can be seen around the periphery of the plate.

Impact of wake meandering and turbulence on offshore wind turbines under different inflow conditions

Warit Chanprasert[1], Rajnish N. Sharma[1], John E. Cater[2] and Stuart Norris[1]
[1]*Department of Mechanical Engineering, University of Auckland, New Zealand*
[2]*Department of Engineering Science, University of Auckland, New Zealand*

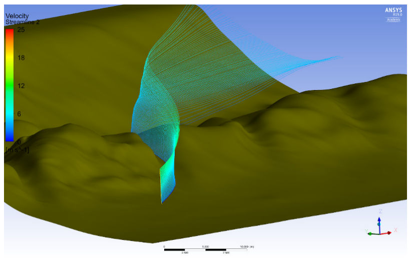


An offshore environment is desirable for siting wind farms due to a lower surface roughness that provides a higher and steadier wind speed with less turbulence compared to that experienced onshore. However, a common challenge for wind farms is the wake interaction effects which reduce the power output of, and increase the structural loads on, downstream turbines. In this study, we investigate the structural dynamics of an array of wind turbines under different seasonal inflow conditions, i.e. winter and summer. The SOWFA-OpenFAST tool is used for the numerical simulations and the metocean conditions were obtained from a buoy anchored off the coast of Nantucket, Massachusetts.

The atmospheric boundary layer is unstable in winter and stable in summer. The ambient stream-wise turbulence intensity in summer is approximately 50% less than that in winter which causes the wakes to propagate farther as illustrated in the figure. Hence, the fluctuating loads on the components of downstream turbines are higher in summer. In addition, wake meandering, the low-frequency flapping of the wake, is more significant in the unstable atmospheric condition in winter. The lateral wake displacement in winter shows a high correlation to the turbine yaw bearing moment about the vertical axis, causing a higher root-mean-square yaw moment of downstream turbines. The figure shows an instantaneous iso-surface of vorticity for the July case.

Using CFD to examine the sensitivity of incident wind directions on flow dynamics through constructed coastal dune notches

Duc Nguyen[1], Sarah Wakes[2] and Mike Hilton[1]
[1]*School of Geography, University of Otago, New Zealand*
[2]*Department of Mathematics and Statistics, University of Otago, New Zealand*



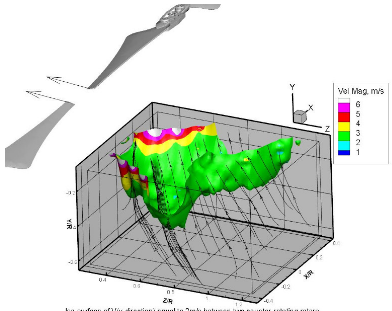
Coastal dunes systems have played a vital role in protection of coastal communities in New Zealand for many decades. Constructing notches on the foredune has been applied to modify the dune morphology to be more resilient, especially from less being severely eroded. Examination of the wind flow dynamics on the notched foredune is important to better understand the sand sedimentation processes and notched foredune evolution.

Our research uses Computational Fluid Dynamics (CFD) to examine the sensitivity of incident wind direction on the flow dynamics through constructed foredune notches. Incident wind is simulated at various directions from shore-normal to shore-parallel at 5 degree increment. The relationship between wind speed within the notch and incident wind direction is presented. The simulated wind results are validated with field data observations at St. Kilda, Dunedin, New Zealand.

The results show that structures of wind within the notch is significantly dependent on incident wind direction. When incident wind direction changes from 190 (notch axis parallel) to 265 degrees (oblique onshore), flow is steered toward notch orientation. Wind speed within the notch decreases when wind direction is more oblique. When incident wind direction is greater than 265 degrees, flow at notch centre is complex. These CFD results are relatively similar to field data observations. Overall, CFD can be used as an effective tool to aid understanding of how notches modify flow structure on the coastal dunes.

Experimental investigation of upward velocity region between adjacent counter-rotating rotors using SPIV technique

Illia Chyrva[1], Mark Jermy[1] and Tara Strand[2]
[1] *Department of Mechanical Engineering, University of Canterbury, New Zealand*
[2] *SCION, New Zealand*



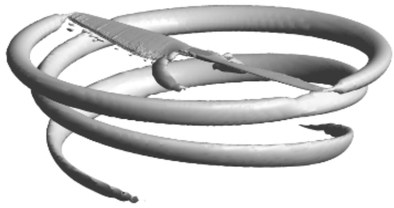
Iso-surface of V(y-direction) equal to 2m/s between two counter-rotating rotors

Multicopter rotors are typically in close proximity to each other and interact aerodynamically. PIV studies revealed the presence of persistent zones of upwash (upwards moving air) between rotors. These influence the distribution of agricultural spray from a multicopter.

For the evaluation of this upward velocity region, two counter-rotating DJI E7000 830mm rotors were used. Stereo particle image velocimetry (SPIV) was used to map the airflow. The magnitude of velocity in the region of upward was evaluated for several relative positions of the rotors. It was observed that:
1) There is a region of upwash between the rotors that exists at every rotor position investigated
2) The strength of the upwash depends on the angular position of the rotor. The strongest upward velocity was observed when rotor tips are in the closest proximity to each other or rotor vortices are in the same angular position
3) The spatial extent of the upward velocity does not depend on the rotational speed (RPM). Neither does the velocity magnitude when normalized by tip speed.

Noise from large and small propellers

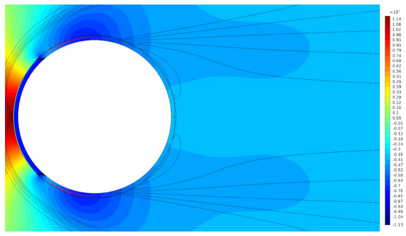
Michael J. Kingan[1], Ryan S. McKay[1], Yan Wu[1], H. Jung[1] and S. T. Go[1]
[1]*Department of Mechanical Engineering, the University of Auckland, New Zealand*



This presentation describes recent work at the University of Auckland investigating the noise produced by propellers of different sizes. These range from large transonic contra-rotating aeronautical propellers down to the small single propellers used on quadcopter unmanned aerial vehicles. The predominant noise generation mechanism for all of these propellers is the unsteady motion of, and loading on, the propeller blades. The cause of this unsteady loading varies depending on the situation. For large contra-rotating propellers, unsteady loading is produced by the propeller blade interacting with the contra-rotating flow-field from the adjacent propeller. For small quadcopter propellers, unsteady loading is produced by the highly turbulent inflow into the propeller and also via the inherent unsteady rotational motion of these propellers. Methods are presented for predicting the noise generated by these mechanisms and these predictions are compared with experimental measurements.

Designing porous skins to reduce drag on bluff bodies

James R. Ramsay[1], Mathieu Sellier[1] and Wei Hua Ho[2]
[1]*Department of Mechanical Engineering, University of Canterbury, New Zealand*
[2]*School of Mechanical, Industrial and Aeronautical Engineering, University of Witwatersrand, Johannesburg, South Africa*



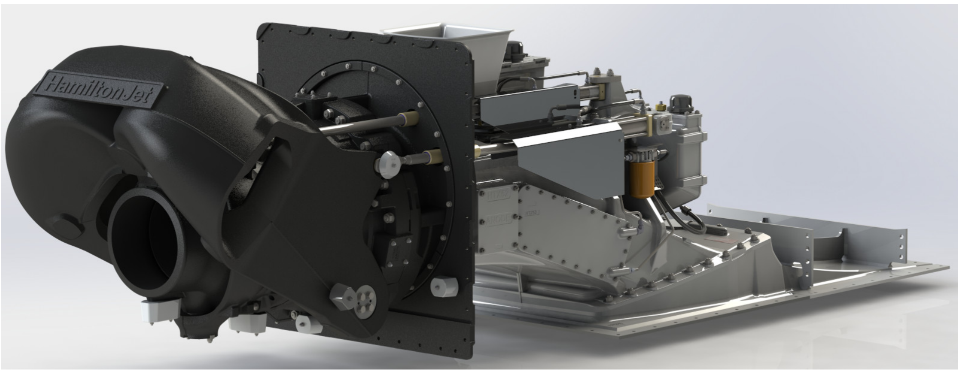
Suction of the boundary layer can have a dramatic effect on flows around bluff bodies. However, the energy required to generate this suction often outweighs the benefits they bring about (reduced drag and stabilised wakes). A similar effect can be achieved by making selected portions of the body surface porous, and connecting them by ducts. The feathers on a bird may operate in a way similar to this. The outer feathers selectively allow air to pass through in certain regions, while the inner feathers act as a porous conductive channel. In our study we try to replicate this passive flow control on the flow around a circular cylinder at Re=120. Two porous skins are layered on top of the cylinder: the inner a uniformly porous and permeable channel to transport removed fluid, and the outer a porous skin with varying permeability to selectively allow fluid to pass through. We perform an optimisation on the material properties of this outer skin to try and achieve a reduction in drag.

Session 8: Industry & Combustion

Using CFD and parametric CAD to design a new generation of reverse ducts for waterjet propulsion

Invited Speaker: Samuel J. M. Hamilton[1]
[1]*Hydrodynamics Research and Development, CWF Hamilton & Co, New Zealand*

HamiltonJet waterjets are renowned for enabling superior vessel manoeuvrability, especially at low speeds. The reverse duct is the component that primarily determines the thrust characteristics of the waterjet at non-straight thrust vectors. The aim of this project was to optimise the design of the reverse duct to achieve better reverse thrust when the duct is fully engaged with a straight nozzle, whilst retaining or exceeding thrust efficiencies at all other direction vectors, as well as reducing actuation loads required. SolidWorks was used to generate parametric geometries that could be easily iterated. These geometries were then solved using ANSYS CFX. A new reverse duct was designed, achieving an increase in full reverse thrust efficiency from 57% to 70% of nominal jet thrust. Significant improvements to complex hydrodynamic components are possible using parametric modelling in conjunction with CFD.



Industrial CFD applications

Invited Speaker: Curtis Marsh[1]
[1] *Engineering CFD, New Zealand*

An overview of several recent projects is presented, including the objectives, design constraints, modelling approaches, validation and some of the challenges that needed to be overcome. The project time scales for many projects are generally counted in weeks, so timely and cost-effective simulations are required, often requiring simplifying assumptions. At the same time, the flow regimes are often complex with multiple phases and transient effects that need to be modelled. Advances in HPC systems and CFD code efficiencies have enabled more complex models to be employed, increasing the value that CFD can add to projects.

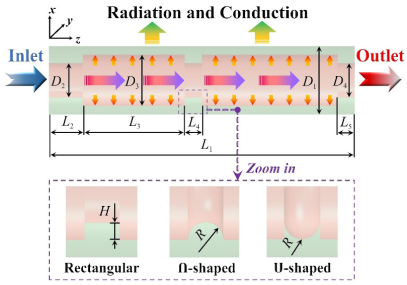
Enhancing thermal performance by implementing a bluff-body in a hydrogen-fueled power system

Tao Cai[1] and Dan Zhao[1]
[1]*Department of Mechanical Engineering, University of Canterbury, New Zealand*

In this work, 3D numerical simulations are conducted to evaluate the bluff-body effect on the thermal performance in hydrogen-fueled meso-combustors. Effects of bluff-body dimensionless axial location l, combustor type and thickness t_1 are considered and assessed. It is shown that the implementation of a bluff-body leads to the fluid flow direction being changed. This change enhances the heat and mass transfer between solid and fluid, thereby resulting in a significant increase on the outer wall temperature (OWT) compared to those in the absence of a bluff-body. In addition, the optimum l is determined and compared for 4 different locations. It is also found that there exists non-monotonic dependence of outer wall temperature on the bluff-body location. The optimum position at which the bluff-body is most effective in enhancing heat transfer is shown to be at $l = 2/5$, under which condition the convective heat transfer coefficient between the fluid and solid is high. Furthermore, Combustor A tends to be associated with a high OWT, while t_1 is found to play a negligible effect on the OWT. The present work sheds lights on how to promote the thermal performance from a hydrogen-fueled power system by reasonably designing the bluff-body parameters.

Mitigation of NOx emission and energy conversion efficiency improvement study of ammonia powered micro-combustor with ring-shaped ribs in fuel-rich combustion

Siliang Ni[1] and Dan Zhao[1]
[1]*Department of Mechanical Engineering, University of Canterbury, New Zealand*



Ammonia, as an alternative option of conventional hydrocarbon energy source, can achieve carbon free emission. However, the production of NOx is responsible for photochemical pollution and human health. In order to reduce NOx emission at the premise of guaranteeing energy conversion efficiency, three micro-combustors with different shaped ribs fuelled by NH_3/O_2 are proposed. Extensive numerical comparisons are made with 3D model built by structured mesh. The effects of 1) inlet velocity, 2) equivalent ratio, 3) rib shape and 4) key parameters of rib are evaluated. In addition, the normalized sensitivity of NO formation and related reaction pathways are analysed. It is found that when the distance between the rib and inlet is 4 mm, comparing the condition of Φ being 1.1 and 1.0, NO emission is reduced by 47.0% while the radiation efficiency is only decreased by 1.76 %. This work has proven that high energy conversion efficiency and low NO emission can be achieved at the same time by properly adjusting the internal structure of the combustor in fuel-rich combustion. It provides an idea for the design of micro-combustors fuelled by NH_3. The figure shows the structure of a micro-combustor with different rib shapes.

Effect of N_2 dilution on thermal performance and emission characteristics of an ammonia-oxygen micro-thermophotovoltaic system

Yuze Sun[1] and Dan Zhao[1]
[1]*Department of Mechanical Engineering, University of Canterbury, New Zealand*

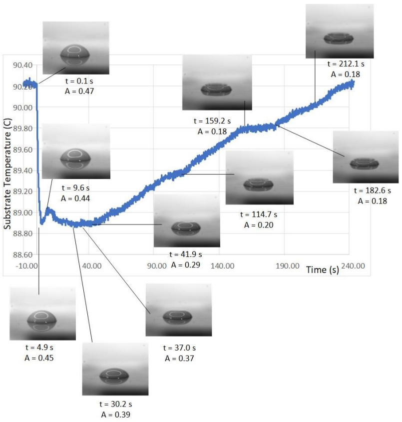
Due to issues of carbon dioxide emissions from carbon-containing fuels, there is growing interest in ammonia as an alternative fuel to reduce greenhouse gas. Oxy-fuel combustion technology is favorable to promote ammonia combustion characteristics. Although it is favorable for stable combustion in power or propulsion systems to use ammonia-oxygen mixture, using diluents is still needed for the purpose of protecting combustor and reducing emission. In the present study, thermal and emission performance of a micro-planar combustor fueled with methane/ammonia mixture are numerically investigated. For this, a low-Mach number reacting flow solver with conjugate heat transfer is developed based on an open source code OpenFOAM. The solver is firstly validated with experimental data available in the literatures. Then the effect of N_2 dilution ratio is examined. It is found that the outer wall mean temperature $\bar{T}_w$ and the non-uniformity $\bar{R}_{Tw}$ are decreased with the increasing N_2 dilution ratio. When inlet velocity $v_i < 0.8\,\mathrm{m/s}$, radiation efficiency η_r varies non-monotonically i.e., increases first then decreases as the N_2 dilution ratio is increased. N_2 dilution is not favorable for promoting η_r when v_i exceeds $0.8\,\mathrm{m/s}$. Moreover, NO emission is found to be reduced by applying N_2 dilution. This work shed light on the thermal and emission performance of an N_2 diluted ammonia-oxygen micro-thermophotovoltaic system.

Session 9: Heat Transfer

Evolution of substrate temperature and drop morphology during the evaporation of sessile milk drops

Gideon J. Gouws[1]

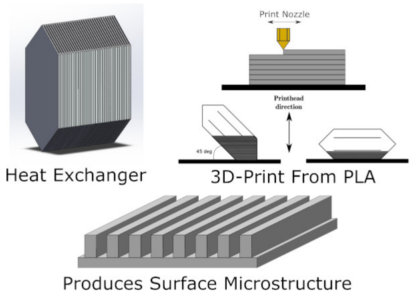
[1]*School of Engineering and Computer Science, Victoria University of Wellington, New Zealand*



Milk is a complex fluid, consisting mainly of water, milkfat, and solids such as proteins and lactose. In dairy processing the composition shows large variations depending on the process step or intended end product. Physical properties such as surface tension, thermal conductivity or heat capacity are strongly influenced by the composition and as a result we expect processes such drop impact and evaporation to change for different milk types and products. This presentation will show results on changes in temperature and drop morphology observed during the impact and evaporation of a sessile milk drop on a hot substrate. The figure below displays the variation in substrate temperature, while also showing the evolution of the shape of the drop during this time. This drop morphology evolves through heating and evaporation of the water, leading to the formation of a permeable crust on the drop. With further evaporation the central dome of the drop will collapse, leading to a characteristic doughnut shape in the deposited solid materials. The differences in this process as observed for a number of milk products at different solid concentrations and substrate conditions will be described.

3D-printed anisotropic microchannels for improved water management in heat exchangers

Chris Hughes[1], Sam Lowrey[1,2] and Zhifa Sun[1,1,2]
[1]*Department of Physics, University of Otago, New Zealand*
[2]*The MacDiarmid Institute for Advanced Materials and Nanotechnology, New Zealand*



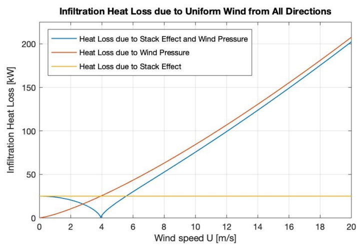
Nature has evolved a mastery of water droplet management over millennia. For example, the surfaces of rice leaves and butterfly wings possess microstructured lines that allow preferential motion of water droplets along the microlines and greater flow resistance perpendicular to lines, leading to excellent water-shedding properties. Incorporating such properties into condensing heat exchanger technology could provide benefits in terms of energy efficiency and as a result, reduced GHG emissions.

Heating, Ventilation, Air-Conditioning and Refrigeration (HVAC&R), represents more than 17% of global electricity consumption. An integral component of most HVAC&R systems is the condensing heat exchanger. Research has shown that a condensing aluminium heat exchanger with micropatterned fins can reduce air-side pressure drop by 36%. This micropatterning has not been trialled for performance improvements in lightweight, less costly polymer heat exchangers.

In the present work, using fused filament fabrication (a 3D-printing technique), we have prepared anisotropic microstructured heat exchanger surfaces and ducting systems with a range of microstructured pitch and various microstructure surface alignment configurations. This work aims to assess whether such micropatterns can improve the drainage properties of a 3D-printed plate-type condensing heat exchanger, compared with smooth control surfaces, via modifying its surface wetting characteristics. The figure shows vertical build-up of print layers results in surface microstructure on 3D-printed heat exchanger.

Analysis of the effect of infiltration in heat load calculations of a retrofitted office building in Christchurch – New Zealand

Diana Kommedal[1], Isabel Andrade[1] and Sid Becker[1]
[1]Department of Mechanical Engineering, University of Canterbury, New Zealand



Winter time air infiltration is a significant contributor to the heating loads of commercial and residential buildings. While the phenomenon is well studied, the number of factors affecting the wind pressure and stack effect add to the complexity in the prediction of the associated heating load. In this study well accepted parametric expressions are used to model the infiltration of air through a medium rise institutional building in order to estimate the total heating loads and the influence of infiltration under different wind conditions. As a test case we analyze the top floor of the Mechanical Engineering Building at the University of Canterbury. This study presents a comparison of the different performance of fenestration choices under different wind conditions on the overall building energy demand.

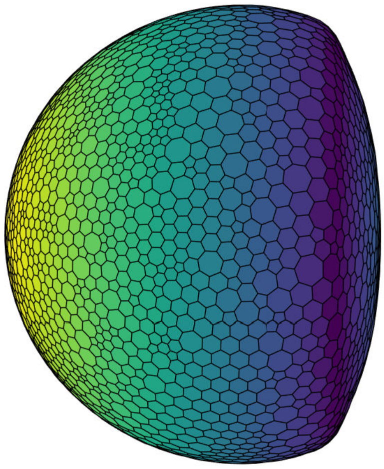
Empirical heat transfer correlation for expansion pulsed pressure mass transport regime

Salma Radwan[1], Daniel Bishop[1], Sid Becker[1], Susan Krumdieck[1] and Natalia Kabaliuk[1]
[1]*Department of Mechanical Engineering, University of Canterbury, New Zealand*

Pulsed pressure metal organic chemical vapour deposition (PPMOCVD) is a method for depositing thin film coatings on a heated substrate in the expansion mass transport regime. Measuring the pulsed heat transfer coefficient is crucial to understanding the fundamental physics of the underlying process and designing industry scale heating systems for coating multiple complex geometry substrates. The theory and non-dimensional parameter space characterising the expansion mass transport regime have been introduced. In this study resistive heating is used to establish the pulsed heat transfer correlations. By adjusting the voltage and current, the power, resistance and temperature are calculated for a Tungsten wire substrate. The resistively heated wire model was implemented in COMSOL and validated against published data. The wire temperature distribution was assessed for various wire geometries to achieve the wire temperature uniformity. The radiation heat transfer and molecular conduction will be experimentally assessed for a range of PPMOCVD process conditions.

Natural shapes of melting ice blocks

James N. Hewett[1] and Mathieu Sellier[1]
[1]*Department of Mechanical Engineering, University of Canterbury, New Zealand*

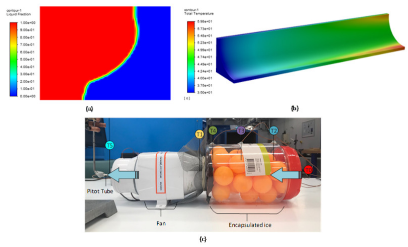


The evolution of melting blocks of ice depends on their surrounding environment, which in turn influences the surrounding medium: causing a coupled moving boundary problem. For example, a hemispherical shape with a flat back in the wake region was developed from a spherical block of ice held in warm flowing water, as shown on the right. Conversely, a pear-shaped profile was observed when melting ice around a heated cylinder; natural vortices were driven by the density variation of the molten ice. We used numerical simulations to model both of these cases and validated against experimental data in the literature. Our node shuffle algorithm was employed for conserving the mesh quality throughout the significant boundary deformations.

Poster Session

Designing a PCM-based battery cooling system for an electric racing vehicle

Maryam Abdali[1], Bill Mohs[1], Daniel Bishop[1], Mathieu Sellier[1] and Natalia Kabaliuk[1]
[1] *Department of Mechanical Engineering, University of Canterbury, New Zealand*



Cooling Li-ion batteries in electric powered race vehicle applications is challenging due to space and power availability limitations. Using a passive cooling method on its own, or in conjunction with an active cooling system, can reduce power consumption and enhance system effectiveness. The use of Phase Change Materials (PCMs) for cooling Li-ion batteries in electric race vehicles is considered. The formulated design solutions include embedding the battery cells in a PCM matrix and PCM-based precooling of the inlet air stream. Analytical, experimental and numerical methods have been used to assess the efficacy of the proposed designs. The airstream temperature required to maintain the batteries surface temperature below the critical 60 °C was estimated to be 15 °C. The use of spherically encapsulated ice for precooling of the inlet air stream up to a period of 30 minutes was evaluated experimentally. The next steps will include numerical modelling of the battery cooling effectiveness of PCM embedding using the enthalpy-porosity method in ANSYS. The figure shows: (a) PCM melting simulation using ANSYS, (b) Air temperature distribution over a Li-ion battery cell, (c) Experimental setup for air precooling using encapsulated ice.

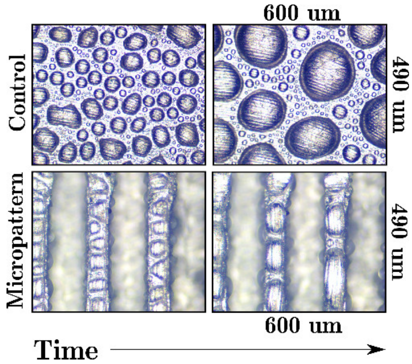
Sit ski aerodynamics

Matt Henderson[1], Ekaterina Lieshout[1], Austin Abbott[2], Cameron Ross[3] and Natalia Kabaliuk[1]
[1]*Department of Mechanical Engineering, University of Canterbury, New Zealand*
[2]*Dynamic Composites, New Zealand*
[3]*High Performance Sport NZ, New Zealand*

Adaptive alpine sit skiing is a competitive discipline in the Paralympic alpine programme. The customization of the sit ski to the athlete is key to performance success in the discipline. Each sit ski is purchased 'off-the-shelf' with minimal customisation and poor aerodynamics. With athletes competing in timed speed events, reaching speeds in excess of 100 km/h, poor aerodynamics of the sit ski is a limiting factor to performance. With elite athletes purchasing a new sit ski every four years, robust and re producible procedures for aerodynamic improvements are required. The goal of this project was to improve the sit ski aerodynamics by reducing drag on the leg coverings. A 3D scan of the athlete in the sit ski was performed to enable computer models to be developed. A number of aerodynamic leg covering design concepts were generated. Each design was assessed based on drag reduction and comparisons made to the original 'non aerodynamic' leg covering. CFD was used to measure drag reduction of each design and a validation step of wind tunnel testing of scaled 3D printed models was performed. The outperforming aerodynamic leg covering design with an estimated time gain of 1.75% was chosen and recommended for manufacture. Planned future aerodynamic research will investigate improving the overall drag aerodynamics of the sit ski.

Experimental investigation of condensation on superhydrophobic surface tension gradients

Sam Lowrey[1,2], Oliver Carter[1], Richard Blaikie[1,2], Andrew Sommers[3] and Geoff R. Willmott[2,4]
[1]*Department of Physics, University of Otago, New Zealand*
[2]*MacDiarmid Institute for Advanced Materials and Nanotechnology, New Zealand*
[3]*Department of Mechanical & Manufacturing Engineering, Miami University, Ohio, US*
[4]*Department of Physics and Chemistry, University of Auckland, New Zealand*



The effect of fixed- and gradient-pitch micropatterned geometry on condensation formation is experimentally investigated on coating-free aluminium surfaces. These metal surfaces are studied because of their importance as a heat exchanger material and are fabricated using laser-etching. Surface tension gradients are designed to reduce the critical droplet radius for spontaneous droplet motion.

During dropwise condensation of water vapour from moist air (condensable plus non-condensable gases) onto a cooled test surface, droplet distributions evolve by nucleation, growth, coalescence and transport. Drop surface coverage dictates the heat transfer characteristics and is a function of the number of drops, and their size. Therefore, drop distribution manipulation is critical to maximizing the condensation heat transfer process.

We measure the time evolution of number density, associated drop size distributions and surface coverage. Preliminary results have been obtained for a control uniform surface and a range of fixed-pitch and surface tension gradient surfaces. These results will be presented along with how the air-side heat transfer coefficient changes between test samples.

The findings of this study will be useful in designing micropatterned metal surfaces operating under condensation conditions to have improved heat and mass transfer characteristics and drainage properties in heat exchanger technology.

Surface wetting phenomena on combined micro-milled and ion-beam processed aluminium

Kirill Misiiuk[1,2], Sam Lowrey[1,2], Richard Blaikie[1,2], Josselin Juras[3], Andrew Sommers[3] and Jérôme Leveneur[4,2]
[1]*Department of Physics, University of Otago, New Zealand*
[2]*MacDiarmid Institute for Advanced Materials and Nanotechnology*
[3]*Department of Mechanical & Manufacturing Engineering, Miami University, Ohio, USA*
[4]*National Isotope Centre, GNS Science - Te Pū Ao, Institute of Geological & Nuclear Sciences, New Zealand*

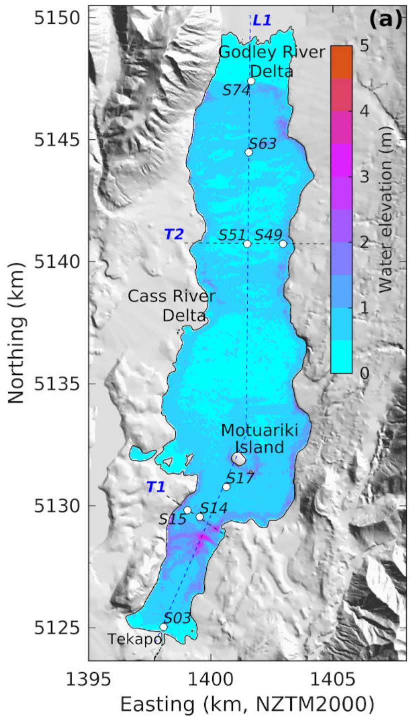
Complex surfaces have evolved in nature that offer unique surface wetting properties. The superhydrophobicity of the lotus is due to its complex surface composed of a combination of microscale bumps with superposed nano-hairs. Furthermore, nature has evolved methods for passive transport of water droplets: a surface tension gradient and a Laplace pressure difference aid the Namib beetle's fog harvesting capability. Its wing-case, consisting of a biphilic hydrophobic/hydrophilic wetting surface, allows increased mass transfer and subsequent droplet rolling motion along the inclined elytra to the mouth of the beetle.

In this work, micro-milling is combined with ion-beam surface modification techniques to attain new surface wetting properties. Ion-beam sputter deposition and implantation enable (A) a superposition of nanoroughness, and/or (B) a modification in the substrate material's Gibbs surface energy, from the periodic microscale lines produced with micro-milling.

A preliminary investigation of the surface wetting properties for these metal surfaces with various roughness - including micro-wetting state, static and dynamic contact angles and in-plane spreading, flat and micro-patterned, and ion-beam processes will be presented. Such combined micro/nanofabrication methods may be useful in engineering unique surface wetting characteristics while remaining coating-free, which can potentially enhance liquid-solid interaction in heat exchanger and wind turbine technologies.

Seiche effects in Lake Tekapo in an Mw8.2 Alpine Fault earthquake

Xiaoming Wang[1], Caroline Holden[1], William Power[1], Yaoru Liu[2] and Joshu Mountjoy[3]
[1]*GNS Science, Lower Hutt, New Zealand*
[2]*State Key Laboratory of Hydroscience and Engineering, Tsinghua University, Beijing, China*
[3]*NIWA, Wellington, New Zealand*



Seiches are standing waves generated in enclosed or partially-enclosed bodies of water and are usually caused by earthquake shaking. They can pose a flooding risk to areas around the edge of the water body, as was observed in Wellington Harbour following the 1855 Wairarapa Fault earthquake. We studied the potential for seiching in Lake Tekapo caused by an Mw8.2 earthquake on the Alpine Fault. Simulations were made of the earthquake ground motion at Lake Tekapo in the scenario earthquake, and these were used as forced boundary conditions in a COMCOT tsunami simulation model of the lake. Seiche wave amplitudes in the 1-4m range were estimated to occur at locations around the lake. While the large amplitude oscillations decayed quickly over 5-10 minutes, long-period oscillations lasted longer, and the oscillations could be linked to theoretical estimates of the modes of oscillation. The size of the modelled ground-motion-triggered seiche waves is sufficient to indicate a potential threat to people and property on the water or onshore close to the lake.

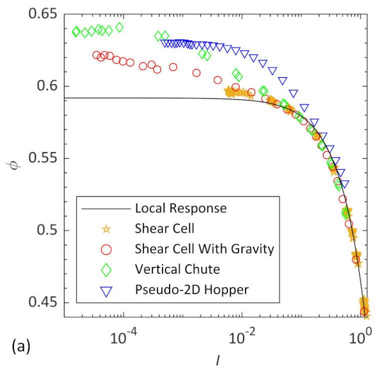
Collision dynamics of wetted roughened particles

Oscar Punch[1], Megan Danczyk[1] and Daniel J. Holland[1]
[1]*Department of Chemical & Process Engineering, University of Canterbury, New Zealand*

Wetted particles are found in many diverse systems both in nature and commercial processing. To characterise and model these systems, the microscopic dynamics between particles must be understood. The Newton's cradle desktop toy is implemented as an apparatus to simulate particle collision where stationary particles are wetted with 3,000 cP silicone oil prior to contact. Three particle roughness cases are investigated in an attempt to resolve the effect of particle roughness on wet particle collision dynamics. The collisions are characterised by the Stokes number, a ratio of inertia forces to viscous force, and the coefficient of restitution, a ratio of the kinetic energy output to kinetic energy input. It has been found that the collision dynamics between wetted roughened particles differs greatly from that of smooth wetted particles. This work is expected to aid simulation of wetted particle systems and provide comprehensive data for wet particle collision numerical model comparison. The figure shows the stages of a reverse Newton's cradle collision outcome: before (left), during (middle), and post-collision (right).

The non-local $\phi(I)$ response

James Robinson[1], Daniel J. Holland[1] and Luke A. Fullard[2]
[1]*Chemical and Process Engineering Department, University of Canterbury, Christchurch, New Zealand*
[2]*School of Fundamental Sciences, Massey University, Palmerston North, New Zealand*

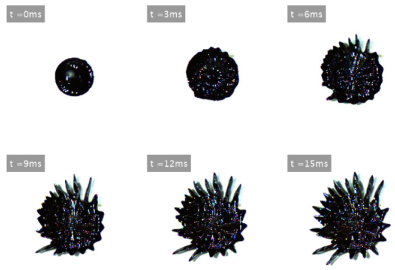


(a)

Granular flows, consisting of numerous individual grains, exhibit many complexities that make a simple continuum description challenging. One major aspect of such flows is their inherent compressibility. Granular flows will expand or contract when under shear, something that is often neglected when modelling dense flows. Prior research has demonstrated that, in simple shear cells, granular packing (ϕ) is a linear function of the dimensionless shear rate (I). However, in more complex systems this relationship breaks down. Here we examine flows in simple shear cells, shear cells with gravity, vertical chutes, and pseudo-2D hoppers. In so doing we demonstrate that ϕ displays a 'local' response, collapsing onto a common curve in the dilute limit and a 'non-local' system dependent response in the dense limit (see figure on right). This relationship is analogous to the non-local response observed in the stress ratio μ. Furthermore, when plotting ϕ against μ we recover a collapse across the full range ϕ range, consistent with recent results, except in the case of pseudo-2D hoppers. These results demonstrate the need to model ϕ as a non-local parameter and further that, in most cases, the non-local ϕ response directly correlates to the non-local stress response.

Stretch, splash, spike: ferrofluid drop impacts in non-uniform fields

Frederick S. Wells[1,2] and Geoff R. Willmott[1,2,3]
[1]*Department of Physics, University of Auckland, New Zealand*
[2]*The MacDiarmid Institute for Advanced Materials and Nanotechnology, New Zealand*
[3]*School of Chemical Sciences, University of Auckland, New Zealand*



Ferrofluids, i.e. fluid suspensions of ferromagnetic particles, can form beautiful and interesting shapes when placed in a magnetic field. Most famously, when a magnet is placed near a ferrofluid drop on a surface, the fluid forms spikes (Rosensweig instabilities) which arise from competition between magnetic forces and surface. When a ferrofluid droplet in air is placed in a uniform magnetic field, it becomes non-spherical.

I will present a video and poster discussing recent experiments with drops falling into a non-uniform magnetic field (simple bar magnet). Using high-speed photography and image analysis methods, droplet shapes are captured. As a drop falls and the magnetic field increases, surface tension becomes less important and the drop elongates, forming a spike at its base. The dynamic evolution of drop shapes is studied by considering energy conservation and forces acting on the drop. The force approach can predict drop shape most effectively, especially if the bulk magnetic force is included.

Author Index

List of Participants

Australian National University

Callum Shakespeare

Callaghan Innovation

Alan Caughley
Lan Le-Ngoc
Ryan McKinlay

Engineering CFD

Curtis Marsh

GNS Science

William Power

HamiltonJet

Sam Hamilton

Massey University

Lucas Corna
Luke Fullard
Sam Irvine
Emilia Nowak
Daniel Uhle

Motovated Design & Analysis

Carlos Alonso
Bevan Hoyt
Tung Ming So

UNSW Sydney

Tracie Barber

University of Auckland

Ayoub Abdollahi
Rosalind Archer
Lily Battershill
Theresa Bischof
John Cater
Warit Chaprasert
Richard Clarke
Kunal Dayal
Matthew Hayward
Michael Kingan
Michael MacDonald
Santhosh Kumar Pandian
Sina Safaei
Steve Wells
Geoff Willmott

University of Canterbury

Maryam Abdali
Pouria Aghajannezhad
Raghu Ande
Isabel Andrade Beltran
Sid Becker
Tao Cai
Jason Chen
Illia Chryva
Michael Coe
Dale Cusack
Megan Danczyk
Stefanie Gutschmidt
Matt Henderson
James Hewett
Daniel Holland
Mark Jermy
Natalia Kabaliuk

Diana Kommedal
Andrew Lange
Shuen Law
Ekaterina Lieshout
Yifei Ma
Craig McConnochie
Elizabeth McGeorge
Bill Mohs
Ciaran Moore
Miguel Moyers-Gonzalez
Siliang Ni
Sevgi Onal
Oscar Punch
Salma Radwan
James Ramsay
Jamie Robinson
Debolina Sarkar
Ali Sefidan
Mathieu Sellier
Refin Sephio
Ross Shepherd
Paul Stephenson
Yuze Sun
Phil Wilson
Jeffrey Wise
Dan Zhao

University of Otago

Chris Hughes
Sam Lowrey
Kirill Misiiuk
Duc Nguyen
Sarah Wakes

Victoria University of Wellington

Gideon Gouws