A NOTE ON THESE PROJECTS

All the projects I offer (except a couple near the end of the list) involve ***one*** (or sometimes more) of

1. Computational Fluid Dynamics (CFD)
2. Some level of programming in either MATLAB ***or*** a proper programming language
3. Use of the Linux operating system

In the past, none of my projects has been too difficult for the students that have taken them, so do ***NOT*** be put off if you don't currently have these skills. However be prepared to have to put some effort into learning them during the early part of the project.

All projects do assume some knowledge of fluid mechanics, a good mathematical ability, a degree of physical intuition and a strong desire to learn and apply new skills.

They are all 12 Credit Point projects and most have 2 titles – the first is the application, the second is the underlying fluid mechanics that are involved.

1) Novel ocean wave damping technology (Flow through a porous plate)

(2 projects, each 12 CP)

On the nano-scale, porous media are used as membranes for reverse-osmosis, on the micro-scale they are used for filtering particles out of suspension and at the macro scale for homogenising flow in turbulent wind tunnels and enhancing heat transfer. A potential new application lies in the partial damping of ocean waves in the place of traditional rock-wall breakwaters.

When flow impacts a porous plate at 90° the pressure drop (and energy required to drive the flow) have been studied previously and can be related to the porosity and Reynolds number of the pore size. However, when the flow impacts the plate at an angle, it exits the other side at a different angle as a result of fluid forces that act tangentially to the plate. Neither the change in angle nor the pressure drop is well characterised, however both are required in order to design applications of porous plates.

There are two possible projects here, both of which will involve the use of unsteady Computational Fluid Dynamics (CFD).

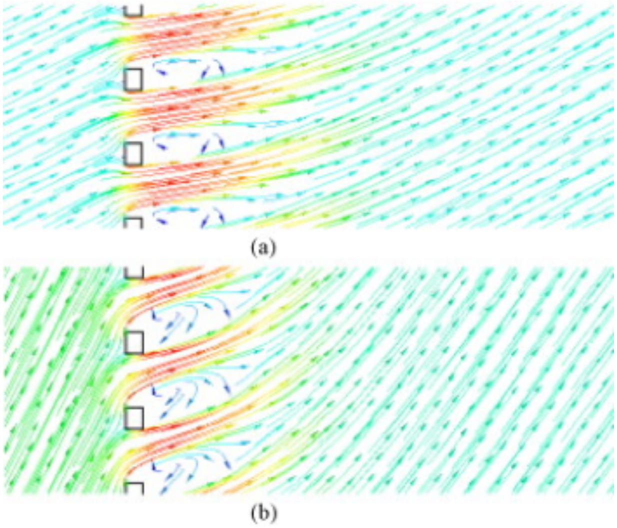

Figure 1 2D simulation of flow through a porous media for different angles of approach.

In Project 1, CFD (in particular Fluent) will be used to first simulate the 3D flow of fluid through a single hole in a solid plate, and then to simulate the flow through multiple holes via different approximations. This project will extend the work started by an FYP student in 2016. This project would best suit a student who has some experience with CFD and mesh generation already (although it is not essential).

In Project 2, a bespoke CFD code based on the Smoothed Particle Hydrodynamics method will be used to investigate the 2D interaction between waves and a porous plate with different parameters (angle, submergence depth, porosity). This project would suit a student with a good background in fluid mechanics and numerical methods. Experience with the Linux operating system is an advantage and if not a current skill, a willingness to learn is needed.

These projects will also involve interaction with a colleague in Perth working on a different aspect of CFD modelling as well as a

Ph.D. student who is conducting experimental measurements.

2) Understanding trub separation in brewing (*Solid-liquid separation in the whirpool*)

(1 project 12 CP)

Brewing practice dates back to at least the ancient Sumerians and Egyptians and has a long and detailed history in Europe that is well documented over many centuries. The basic process has remained the same over a long time:

- malt the grain
- mash (or steep) in hot water to convert the starch to sugars (forming the wort),
- add bittering and other flavour additions,
- boil to sterilise, isomerize the hop acids and help precipitate the proteins and tannins (forming trub, a fluffy particulate material),
- cool and then ferment.

Separation of the trub from the wort is highly desirable as its presence in the fermentation can result in off flavours. This is usually accomplished by stirring the wort for a time until it is rotating quickly in the kettle, and then waiting for the solids to accumulate at the centre of the tank floor. This process is termed “whirlpooling” (for obvious reasons).

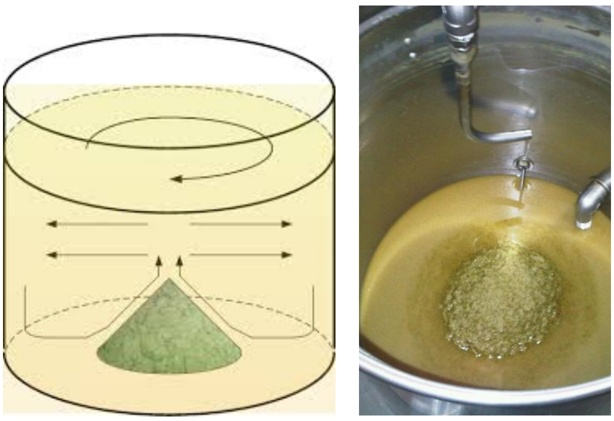

Figure 2 Schematic of the whirlpool flow (left) and photo of the trub cone (right) in a kettle.

This project aims to use CFD to investigate the whirlpool and understand the effect of the tank base geometry on the flow and subsequent trub separation. Conical, flat, inverse conical and round-based tanks will be considered.

This project would best suit a student who has some experience already with CFD and mesh generation (although it is not essential). Other requirements are some background in fluid mechanics, numerical methods and good physical intuition. An interest in, and experience with, brewing beer is a definite plus, but is in no way essential!

3) Mining waste disposal 1 (*Turbulent flow in an open channel*)

(1 project 12 CP)

Flow in an open channel occurs in geophysical situations such as rivers and magma tubes, in agricultural applications such as irrigation canals and in industrial processes where fluid is transported from one part of a plant to another via gravity. In all but a few specialised cases, these flows are highly turbulent. An exception occurs when the fluid is non-Newtonian, and here, industrially relevant flows can occur in the laminar, transitional or weakly turbulent regime.

In this project, we utilise a high-order CFD package *semtex* (developed by Prof Hugh Blackburn) to study the effect of fluid parameters, channel geometry and Reynolds number on the turbulent flow of Newtonian and non-Newtonian fluids in open channels.

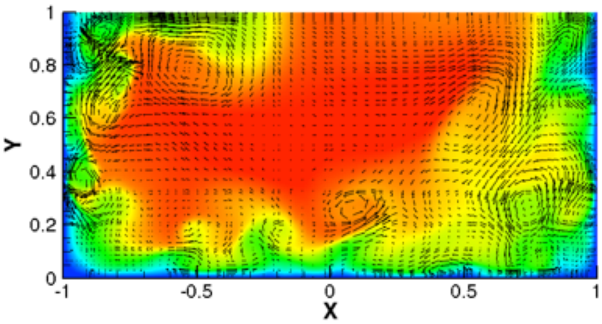

Figure 3 Instantaneous snapshot of turbulent non-Newtonian flow in a rectangular channel.

For some applications, the presence of a yield stress could determine the shape of the channel itself, and if time allows a study of how to implement a self-formed channel in the software will also be considered.

This project would suit a student with a good background in fluid mechanics and numerical methods. Experience with the Linux operating system is an advantage and if not a current skill, a willingness to learn is essential.

4) Mining waste disposal 2 (*Mixing in turbulent non-Newtonian flows*)

(1 project 12 CP)

One means of aiding dewatering of mining waste streams is to add solutions of long chain polymers known as flocculants. There has been recent interest in adding these materials to a partially dewatered tailings stream, which is usually a mud-like material that is often non-Newtonian. How these polymers mix in transitional and turbulent flow inside a pipe is currently unknown.

In this project, the student will utilise the *semtex* software (developed by Prof Hugh Blackburn) to study the effect of non-Newtonian rheology on the mixing and dispersion of tracer materials in non-Newtonian flows inside pipes

5) Simulation of splashing drops

(1 project 12 CP)

The impact of a liquid drop onto a liquid surface is as complex as it is beautiful. Despite being studied for over one hundred and twenty years, there are still unanswered questions.

One important phenomenon is the entrainment of small air bubbles during some drop impact that is believed to play a role in mass transfer of CO_2 and O_2 into the oceans.

In this project, the student will use a bespoke CFD code to investigate the effect of droplet oscillation on the transition between different splashing regimes. In particular how does drop oscillation at impact modify the entrapment of small bubbles. Developing an understanding of this might explain the discrepancy between different sets of experimental results and add to the fundamental understanding of splash. A successful project is likely to result in submission of a journal paper to a good fluids journal (if that interests you).

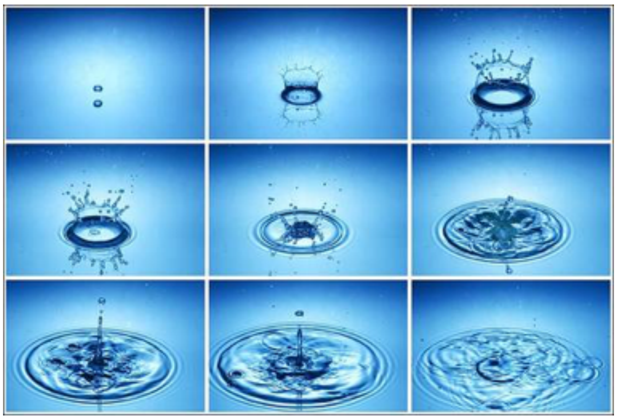

Figure 4 Experimental images of a drop splash sequence on a shallow pool.

This project would suit a student with a good background in fluid mechanics and numerical methods. Experience with the Linux operating system is an advantage and if not a current skill, a willingness to learn is essential.

6) Dune formation in oil and gas pipelines (*Modelling solid-liquid transport in fluids*)

(1 project 12 CP)

The formation of sand dunes via wind is well known and something we see every time we visit the beach. Less obvious but just as common, is the formation of dunes under the ocean via the action of ocean currents flowing over granular material like sand. Far less well-known is that dunes can also form in gas and oil pipelines as a result of the inevitable creation of sand in the extraction process.

This can lead to all kinds of problems in operating these pipelines including wear, corrosion, high pressure drops and complete pipe blockage.

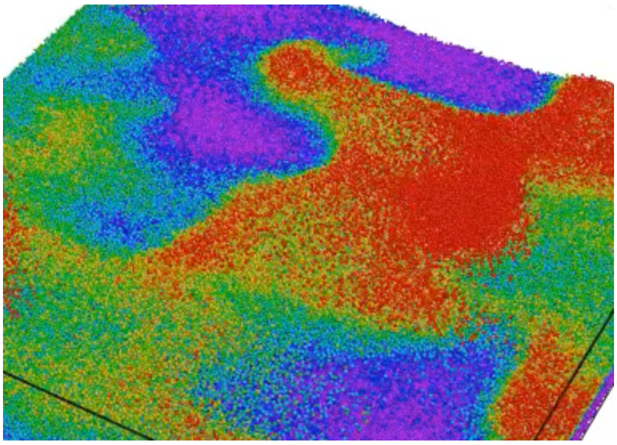

Figure 5 Coupled CFD/DEM simulation of solid particle transport in a bed.

In this project, software developed at CSIRO will be used to solve the coupled equations that describe the flow of a fluid (oil or gas) through a porous media (a sand bed) and the flow of a granular material (individual sand grains). In particular the conditions that occur near the transition between flow regimes (i.e. fully suspended, sliding bed, settled bed) will be identified, and the conditions under which dunes form will be identified. Key criteria such as wavelength, dune amplitude and shape as a function of grain size, shape, in-line sand concentration and pressure drop will be determined.

This project will be done in conjunction with a an external CSIRO supervisor (Dr James Hilton) and will suit a student with good physical intuition, good computer skills and ideally some familiarity with the Unix/Linux environment.

7) A novel gas-solid contactor design

(1 student, 12 Credit Points)

Heating and cooling of solid particles in a gas stream and/or undertaking chemical reactions on their surface or using them as a catalyst is a fundamental part of many processing applications including the production of alumina, cement, milk powders and potentially in the catalytic conversion of waste products such as CO_2. Devices such as rotary kilns, fluidised beds and spray driers are extensively utilised in practice, however all have some deficiencies.

In this project, a two-dimensional CFD model will be developed of a novel "Cross-flow" gas-solid contactor with special emphasis on trying to predict heat transfer to/from solid particles from/to a gas stream. The proposed contactor is elegantly simple and its design allows for a set of contactors to be built in series in a space only marginally larger than a single contactor.

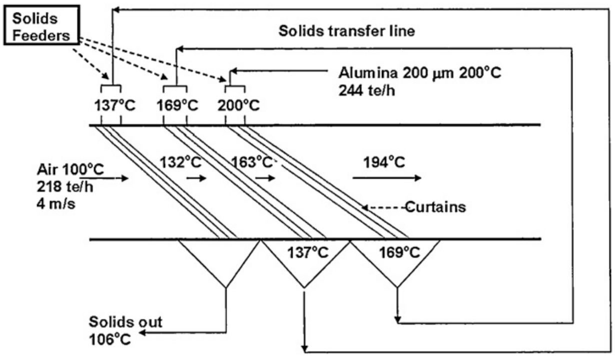

Figure 6 Schematic of gas-solid contactor for use in alumina production.

This project will suit a student with knowledge of fluid mechanics, turbulence modelling and heat transfer. Familiarity with ANSYS software Fluent of CFX is desirable, although not essential. A willingness to learn as part of the project is essential. The project will also provide the student with exposure to a new technology development and the joys (and pain) that such a venture can entail. This project will be undertaken in collaboration with Emeritus Professor Owen Potter (Monash) and Mark Latham (Latham Solutions).

FLUID CHAOS PROJECTS

Chaos is a term commonly viewed as equivalent to disorder or randomness. However, in fluids the common understanding is often incorrect! Chaotic advection produces structures in fluids that can be highly ordered – but not simple. Chaos in fluids is pervasive and generates the template on which all heat, mass and reactive transport occurs. One of the fascinating aspects of fluid chaos is that it can occur in flows that are known exactly in space and time.

The concept of Lagrangian chaos is central in both fluid mixing at low Reynolds number and in turbulent flow containing coherent structures. Applications include nanoparticle deposition in alveoli in the lung, viscous mixing in industrial processes, mixing in microfluidic devices, dispersion of nutrients in large scale structures in the ocean and ore body formation in porous geological media.

In the following three projects, we consider different types of flows and investigate the conditions under which they are chaotic and could mix.

8) A novel reactor design (*Chaos in Taylor-Couette Flow*)

(1 student, 12 Credit Points)

The flow between two rotating cylinders is known as Taylor-Couette flow. On first sight this flow might seem rather bland, but over twenty different flow types have been categorised, each with unique properties. Couette, Taylor-vortex, wavy vortex, modulated vortex, twist vortex, wavy-inflow, wavy-outflow, turbulent vortex, featureless turbulence ..... the list goes on.

In this project, the student will use an advanced CFD code to study some of these Taylor-Couette flow regimes in detail and apply Lagrangian particle tracking to understand which might give rise to chaotic fluid flow. In particular, "twist", "wavy inflow" and "wavy outflow" flow regimes hold promise for dividing the flow into "reaction units" that do not mix with each other, but which mix well internally. The students will develop an appreciation for supercomputing and the use of sophisticated computer codes to understand the real world.

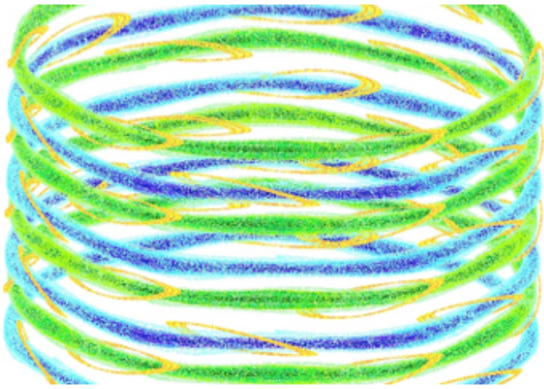

Figure 7 Non-mixing vortex cores in wavy vortex flow (green and blue structures), encircled by inclined non-mixing rings (yellow structures).

The project is computationally based, and will suit students with a strong interest in fluid mechanics and numerical methods. It will continue a project started in 2016.

9) Developing a new in-line mixer (*Fluid flow in a swirl pipe*)

(1 student, 12 Credit Points)

Turbulent flow mixes well because it is "randomly chaotic" and fluid elements are continually stretched and folded in unpredictable and random ways. Very low Reynolds number flows can also mix, but only if they are carefully "programmed" to introduce what is known as "Lagrangian chaos". The most problematic mixing regime occurs for intermediate Reynolds numbers that are too low to be turbulent but too high to be effectively driven by viscous forcing. Such flows are laminar, usually steady and often mix poorly.

In this project we consider application of a "swirl pipe" to mixing. The swirl pipe geometry is shown in the following figure for a 4-lobe design. The slowly spiralling lobes impart non-uniform swirl and radial motion to the flow as it moves down the pipe, and thus moves fluid from the wall toward the pipe centre and *vice versa*. The superposition of the flows from the 4 lobes has the potential to induce Lagrangian chaos in a programmed way. However a student project in 2016 found this was not possible unless the pipe was modified with the placement of internal "fins".

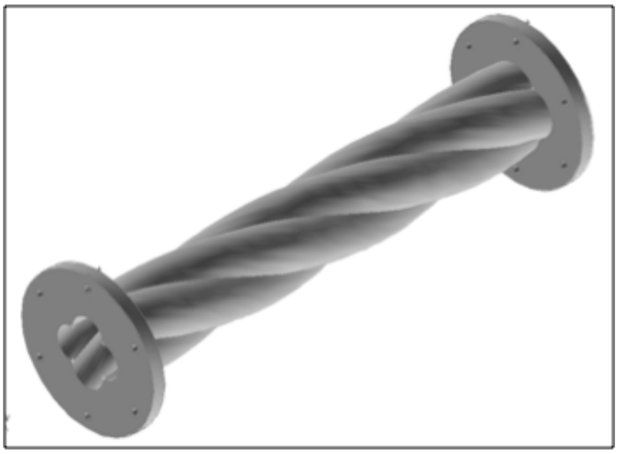

Figure 8 Four Lobe swirl pipe design**.**

This project will focus on generating the flow in different design pipes (2, 3, 4 lobe) with different spiral ratios and understand how internal fins can be used to ensure the entire pipe flow becomes chaotic.

The CFD will be done with Fluent (or CFX). This project will suit a student who has good fluid mechanics background. Some experience with CFD (using one of the packages above) is a definite advantage.

10) Experimental chaos

(1 student, 12 Credit Points)

Visualisation of chaotic flows has produced many beautiful flow images that also offer detailed insight into flow structure and mixing processes.

In this project, a new experimental apparatus that has been designed and partially built in the Monash Engineering workshop will be completed and commissioned with significant input from the student. Once operational, a series of flow visualisations will be undertaken.

The project aligns with an existing Ph.D. project that is computationally based and would suit a student with good mechanical skills, an interest in experimental fluid mechanics. A knowledge of electronics is an advantage, but not essential.

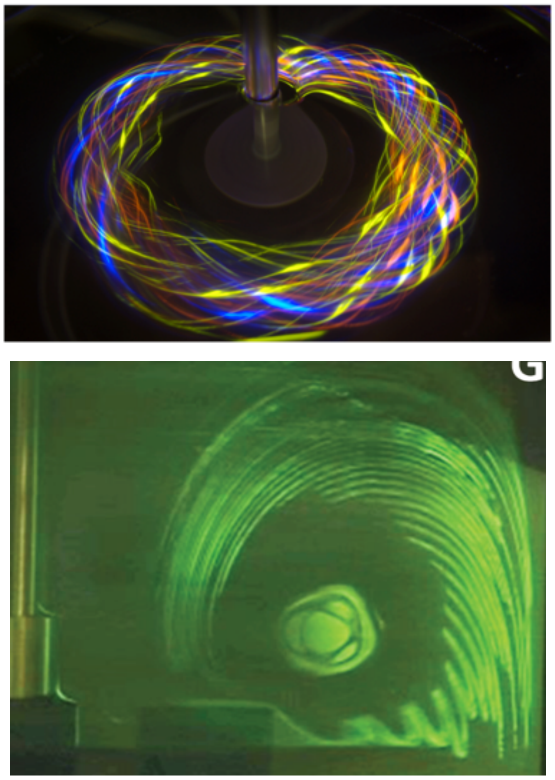

Figure 9 Mixing/segregation in a stirred tank. Top is the attractor vortex that draws particles into a torus near the impellor and bottom is a visualization of invariant surfaces and manifolds.

11) Building agitation

(1 student, 12 Credit Points)

Mixing is a crucial part of many manufacturing processes. In food, in biotechnolgy, in microfluidic lab-on-a-chip diagnostic devices, etc, mixing produces the transformation of material properties that is the hallmark of successful value-added manufacturing.

During this project you will have an excellent opportunity to get hands on experience designing, building and analyzing an industrial chaotic mixing device and measuring the fluid mechanical mechanisms by which it functions. You will also write, implement, and test various mixing control protocols. The project is suited to those interested in fluid mechanics, mechatronics, industrial design, experimentation, or control.

A video of a similar type of flow device (the blinking vortex flow) can be seen at: http://www.eg.bucknell.edu/~tsolomon/mpg/blinkvortexdye.mpg