Imperial College London

Introduction to the examples

Richard Ferrier

Department of Earth Science and Engineering, Imperial College London

Outline

CFD Examples

- Advection of a top hat
- Lid-driven cavity
- Backward facing step
- Flow past a sphere
- Water collapse
- Tephra settling

GFD Examples

- The lock–exchange
- Hokkaido-Nansei-Oki tsunami
- Rotating periodic channel
- Restratification following open ocean deep convection
- Tides in the Mediterranean Sea
- Stokes square convection

Outline

CFD Examples

- Advection of a top hat
- Lid-driven cavity
- Backward facing step
- Flow past a sphere
- Water collapse
- Tephra settling

GFD Examples

- The lock–exchange
- Hokkaido-Nansei-Oki tsunami
- Rotating periodic channel
- Restratification following open ocean deep convection
- Tides in the Mediterranean Sea
- Stokes square convection

Advection of a top hat

- Top hat distribution of a tracer, advected with prescribed velocity
- Compares CG, CV, DG discretisations
- Simple, fast: run time 2 min.

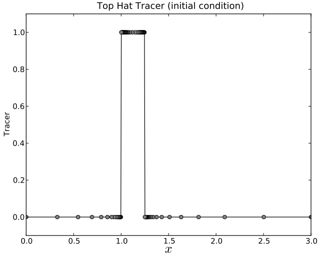
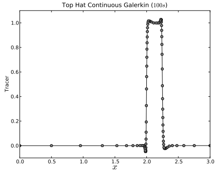


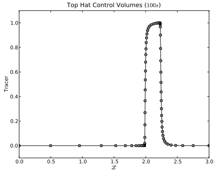
Figure: Initial top hat distribution.

Imperial College London

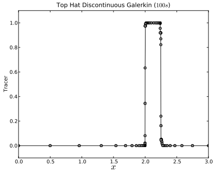
Advection of a top hat



Continuous Galerkin



Control Volume



Discontinuous Galerkin

Imperial College London

Continuous Galerkin

- Basic finite element discretisation
- Not good for advection of sharp discontinuities
- Not conservative
- SUPG stabilisation applied in this instance

Control Volume

- Simple and efficient, sometimes diffusive
- Need to choose an interpolation method. Here, ‘FiniteElement’ interpolation with Sweby limiter is used.

Discontinuous Galerkin

- Popular for advection problems
- Slope limiters still needed near discontinuities to prevent overshoots

Exercises

- Turn off SUPG stabilisation for the CG case and see what it does.
- Change the resolution of the adapted meshes.
- Change the temporal and spatial discretisations to get a less diffusive CV advection scheme.

Lid-driven cavity

- Simple test case often used for verification and validation
- Laminar in this instance
- Run time: 7 hr. for all four mesh resolutions

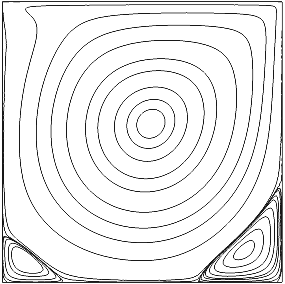
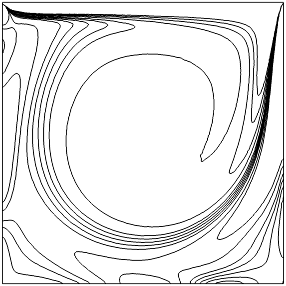


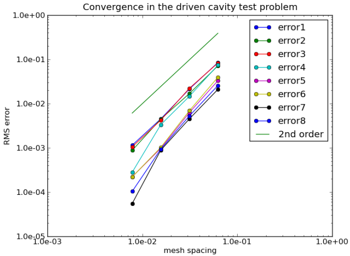
Figure: Streamfunction contours in converged solution for $h = 1/128$.

Imperial College London

Lid-driven cavity



(a) vorticity contours in converged solution for $h = 1/128$



(b) error vs. experiment

Figure: The error metrics are listed in the manual.

Backward facing step (2D and 3D)

- Classic CFD test case
- Experimental and numerical data available for comparison
- Test of numerical methods or turbulence models

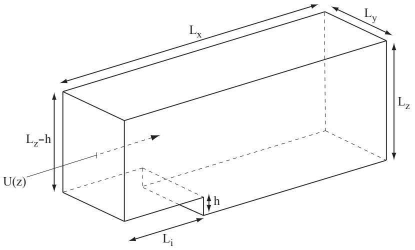


Figure: Geometry of 3D backward facing step

Imperial College London

Backward facing step (2D and 3D)

Run times:

- 25 min. for 2D reference
- 6 hr. for 2D with k-ε
- 5 hr. (8 cpu) for 3D

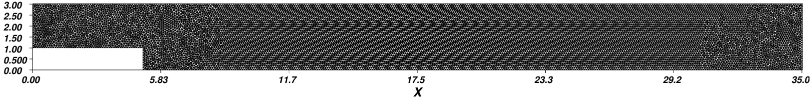


Figure: Geometry of 2D backward facing step showing the mesh.

Backward facing step, 2D results

- Reattachment point estimation
- Profile evolution in time/space

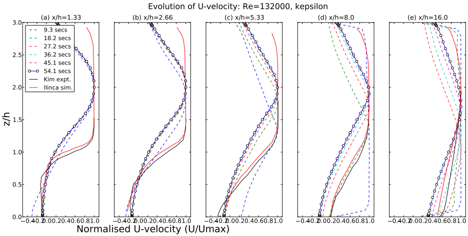


Figure: Streamwise velocity profiles at several points downstream of the step showing the converged solution and comparing to experimental and other numerical data.

Backward facing step, 3D results

- Recirculation bubble and reattachment point

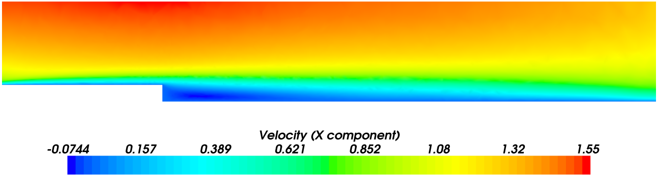


Figure: Velocity cut plane through centre of 3D geometry, time $= 50\,\mathrm{s}$.

Backward facing step, exercises

- Increase the Reynolds number.
- Add adaptivity options.

Flow past a sphere

- Drag calculation at different Reynolds numbers
- Run time: 9 hr. (8 cpu)
- Adaptive mesh resolves dynamics near surface

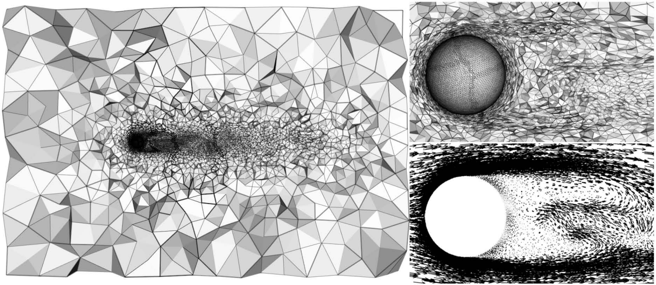
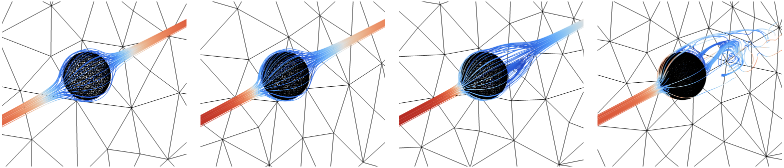


Figure: Sphere at $Re = 1000$. Clockwise from left: Cut plane showing anisotropic mesh, close up of sphere, velocity vectors.

Imperial College
London

Flow past a sphere

- Plot streamlines by processing vtu files in Paraview
- Time series of pressure and viscous drag force integrals available in stat file



(a) Re = 1 (b) Re = 10 (c) Re = 100 (d) Re = 1000

Figure: Streamlines showing transition from laminar to turbulent wake with increasing Reynolds number.

Flow past a sphere

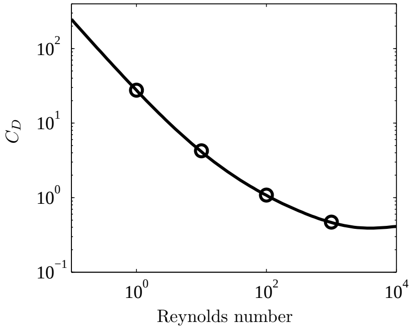


Figure: Plot of drag coefficient vs. Reynolds number showing effect of turbulence. Circles: Fluidity data, line: correlation to experimental data from Brown and Lawler (2003), Journal of Environmental Engineering, 129(3).

Flow past a sphere, exercises

- Write an offline python diagnostic to calculate the drag coefficient from the vector force on the sphere.
- Investigate impact of different discretisations and adaptivity parameters on the drag.
- Investigate different shaped objects.

Water collapse

- Barrier removed instantaneously at $t = 0$; flow driven by gravity
- Inviscid, multi-material flow, mesh adaptivity
- Static detectors to compare with benchmark lab data
- Run time: 2 hr.

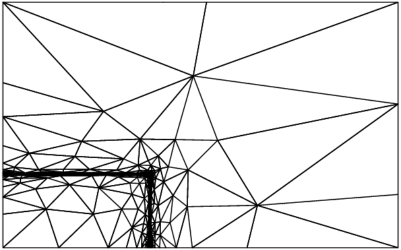
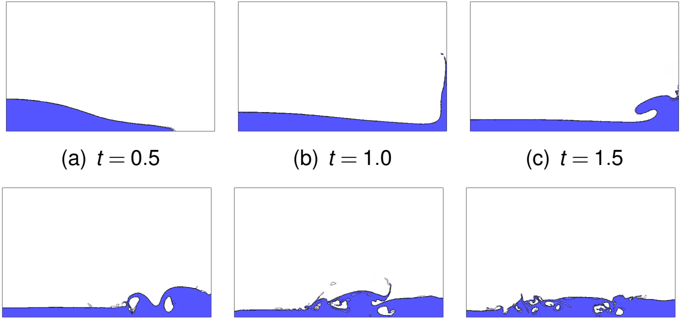


Figure: Initial unstructured adapted mesh for water collapse problem.

Imperial College London

Water collapse



(a) $t = 0.5$ (b) $t = 1.0$ (c) $t = 1.5$

(d) $t = 1.75$ (e) $t = 2.25$ (f) $t = 2.5$

Figure: The evolution of the water volume fraction over several timesteps. The presence of water is indicated as a blue region and the interface to the air is delineated by the contours (in black).

Water collapse

- Post-processing scripts output several plots, e.g. comparison of water depth to experimental data

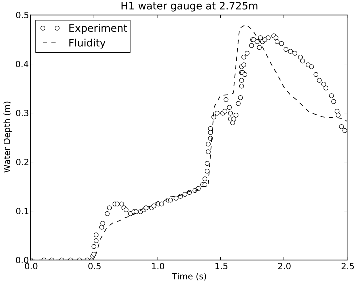


Figure: Comparison with experimental depth gauge data at $x = 2.725\,\mathrm{m}$.

Water collapse, exercises

- Disable the adaptivity option to run on a fixed mesh.
- Alter the water/air viscosity/density.
- Modify the tank geometry.

Tephra settling

- Replicates a laboratory experiment of tephra (fine volcanic ash particles) settling through a tank of water (Carey, 1997)
- Small tephra particles can settle either individually according to Stokes' law, or collectively as a cloud of particles (a plume)
- Dispersed multiphase approach, adaptive timestepping
- Run time: 1 hr.

Tephra settling - Numerical results (1)

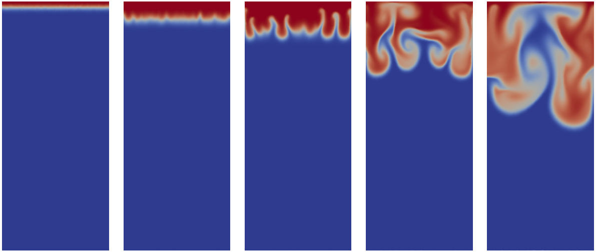


Figure: Simulation visualisations at $t = 10, 30, 50, 80$ and 110 seconds. Tephra particles initially settle individually, but as more tephra fluxes in, the layer eventually becomes unstable and plumes begin to form.

Tephra settling - Numerical results (2)

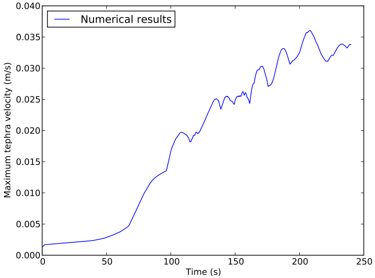


Figure: Plot of the maximum tephra phase velocity against time. Tephra particles initially settle at approximately 0.0017 ms^{-1}, as predicted by Stokes' law. Plumes begin to form after approximately 30 seconds, resulting in settling velocities over 10 times greater than that of an individual particle.

Tephra settling - Exercises

- Decrease the characteristic element size to better resolve the plume behaviour.
- Alter the particle size to observe its effect on plume formation.
- Add a second particle phase (with a different particle size).

Outline

- CFD Examples
  - Advection of a top hat
  - Lid-driven cavity
  - Backward facing step
  - Flow past a sphere
  - Water collapse
  - Tephra settling
- GFD Examples
  - The lock–exchange
  - Hokkaido-Nansei-Oki tsunami
  - Rotating periodic channel
  - Restratification following open ocean deep convection
  - Tides in the Mediterranean Sea
  - Stokes square convection

Imperial College
London

The lock–exchange

- Fluids of different densities (temperature) separated by a barrier. As the barrier is removed, the denser fluid collapses under the lighter.
- Boussinesq flow, control volume discretisation, mesh adaptivity
- Run time: 10 min.

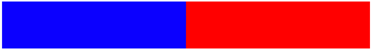


Figure: Lock-exchange initial temperature (colour) distribution.

The lock–exchange

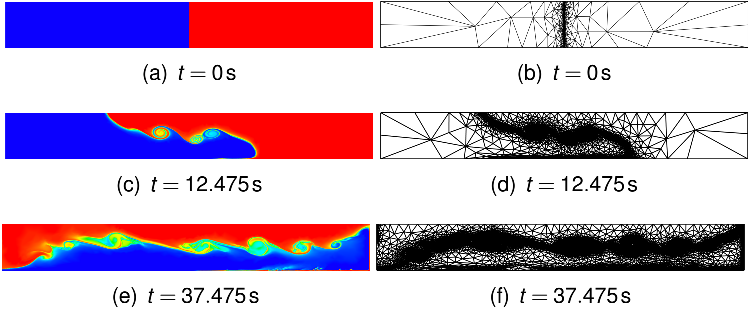


Figure: Lock-exchange temperature distribution (colour) with meshes, over time (t).

The lock–exchange, diagnostics

- Front speed (or Froude number)
- Mixing given by domain fraction of fluid in specified temperature classes

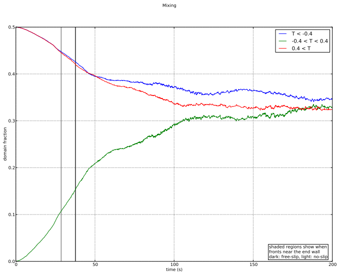


Figure: Time evolution of fraction of domain that contains fluid in three temperature classes. Blue: cold, red: warm, green : mixed

The lock–exchange, exercises

- Increase the simulation time from the default settings (to e.g. 30 secs) and calculate the average Froude number
- Play with the adaptivity options.
- Change the diffusivity and viscosity values.
- Run with a fixed mesh (this will require making a new input mesh)
- Try adding some detectors to visualise the particle trajectories.

Imperial College
London

Hokkaido-Nansei-Oki tsunami

- Okushiri island, Japan, 1993
- Runup height of up to 30m
- Simulation based on a 1/400 laboratory setup
- Uses the free-surface and wetting and drying facilities of Fluidity as well as detectors.
- Run time: 1.5 hr. (4 cpu)

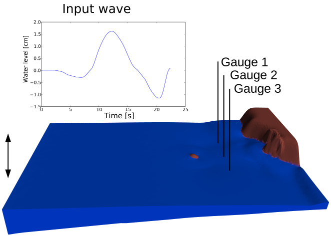


Figure: The domain and the three gauge stations.

Imperial College
London

Hokkaido-Nansei-Oki tsunami

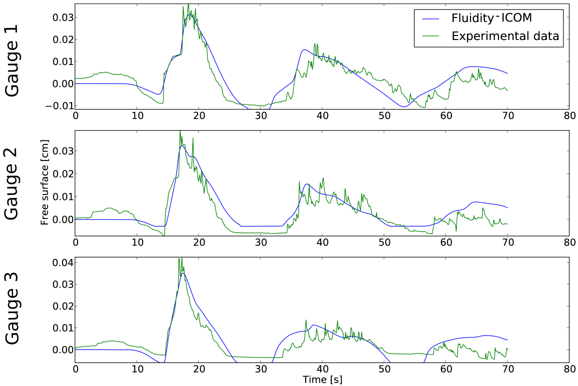


Figure: The numerical and experimental results for the three gauge stations.

Rotating periodic channel

- Unit square domain, periodic in zonal direction and zero-slip at North and South boundaries. Coriolis forcing.
- The flow is driven by a velocity source term:

$$\vec{F} = \begin{bmatrix} y^3 \\ 0 \end{bmatrix}$$

- Provides a convergence test for the $P_{1DG}P_2$ element pair.
- A good example of using python state for online diagnostics and analysis, and also using python for setting initial conditions.
- Run time: 10 min.

Imperial College London

Rotating periodic channel

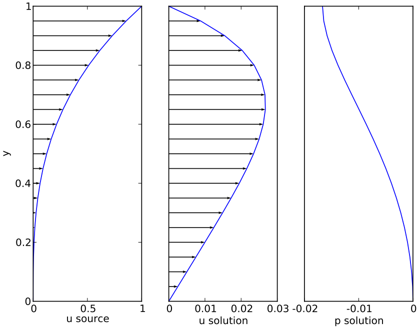


Figure: Velocity forcing term and analytic solutions for velocity and pressure for the rotating periodic channel test case. Note that each of these quantities is constant in the x direction.

Rotating periodic channel

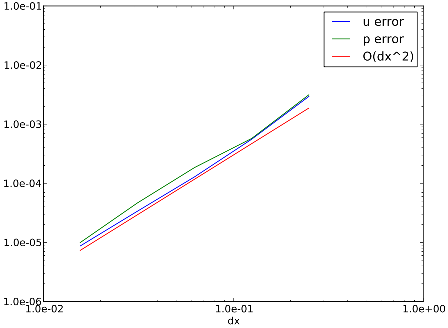


Figure: Error in the pressure and velocity solutions for the rotating channel as a function of resolution.

Rotating periodic channel, exercises

- Understand the use of analytic forcing functions in Fluidity using Python.
- For the Continuous Galerkin example, see what the effect is of removing the SUPG stabilisation
- Change the resolution of the adapted meshes
- Change the spatial and temporal discretisations to get a less diffusive advection scheme

Restratification following open ocean deep convection

- Idealised model of the restratification phase of OODC using $P_{1DG}P_2$ and an extruded mesh.
- Run time: 20 hr. (32 cpu)

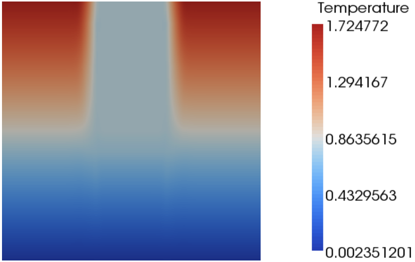
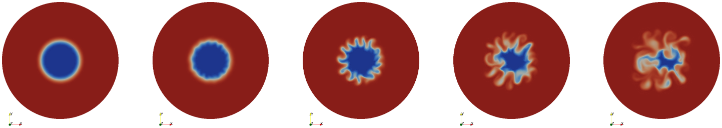


Figure: A vertical slice throughout the domain showing the initial temperature stratification. The domain is a cylinder of radius 250 km and height 1 km.

Restratification following open ocean deep convection



(a) 0 days (b) 10 days (c) 20 days (d) 30 days (e) 40 days

Figure: The temperature cross section at a depth of 40 m.

Restratification following open ocean deep convection, exercises

- Work out the kinetic and potential energies using the vtus or stat file.
- Try running with different resolutions and look at the effect on the eddies.

Tides in the Mediterranean Sea

- Tidal modelling is a widely used method for validating free surface implementations.
- Flow is driven by both astronomical and co-oscillating boundary tide forcing for the four main tidal constituents: M_2, S_2, K_1 and O_1.
- Run time: 5 hr. (64 cpu)

Tides in the Mediterranean Sea

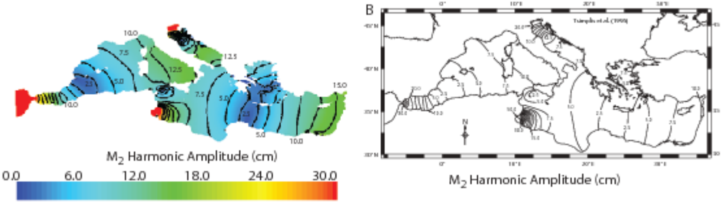


Figure: Plots of the M_2 tidal harmonic amplitude in the Mediterranean Sea from Fluidity–ICOM and the high resolution model of M. N. Tsimplis *et al.* (1995), J. Geophys. Res. 100(C8).

Tides in the Mediterranean Sea

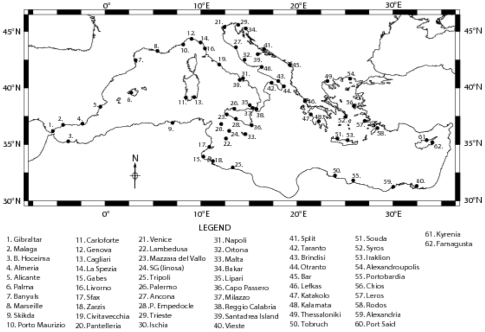


Figure: Locations of 62 tide gauges in the Mediterranean Sea used to calculate the root mean square error.

Stokes square convection

- Steady-state isoviscous convection at a Rayleigh number (Ra) of 10^5 , in a two-dimensional square domain
- Comparison against a well-established two-dimensional cartesian geometry benchmark result for Stokes flow
- Run time: 15 min.

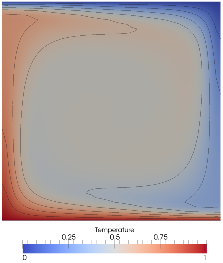


Figure: Steadystate temperature field, on a uniform structured mesh of 48 $\times$ 48 elements. Contours are spaced at intervals of 0.1.

Stokes square convection

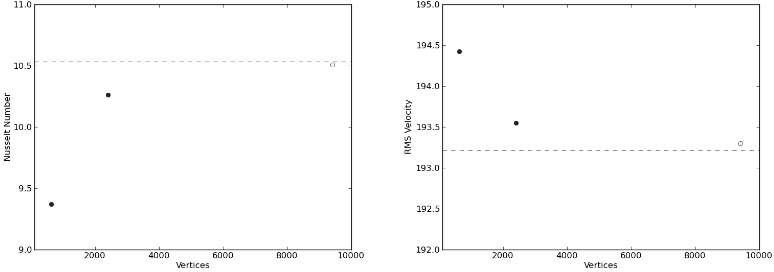


Figure: Results from 2-D, isoviscous Stokes square convection benchmark cases: (a) Nusselt number vs. number of triangle vertices, at $Ra = 1 \times 10^5$, (b) RMS velocity vs. number of triangle vertices, at $Ra = 1 \times 10^5$. Benchmark values are denoted by horizontal dashed lines. Note that the highest resolution case is not included in the example.

Stokes square convection, exercises

- Verify that results do indeed converge towards the benchmark values at higher resolution.
- Alter the initial condition for temperature to verify that, excluding the location of upwelling flow at $x = 0$ or $x = 1$, results are insensitive to this initial condition.
- Change the Rayleigh number to $Ra = 1 \times 10^4$

Imperial College London

Getting started

- The examples have been run in advance and the output can be found in `/scratch/examples`
- You may need to specify your Fluidity binaries folder `export PATH=<<fluidity dir>>/bin/:$PATH`
- and Python tools folder `export PYTHONPATH=<<fluidity dir>>/python/:$PYTHONPATH`
- If you get ‘permission denied’ running a postprocessing script, try `chmod u+x <<script>>`