

Summary of the CFD examples

Pablo Brito Parada

Department of Earth Science and Engineering, Imperial College London

Outline

Advection of a top hat

Lid-Driven Cavity

Backward facing step

Flow past a sphere

Water collapse

Tephra settling

The top hat

- ▶ Top hat distribution of a tracer, advected with prescribed velocity.
- ▶ Simple, fast: ideal laboratory to experiment and play.
- ▶ Compare CG, CV, DG discretisations.

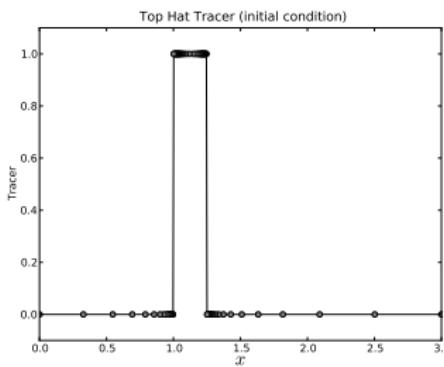
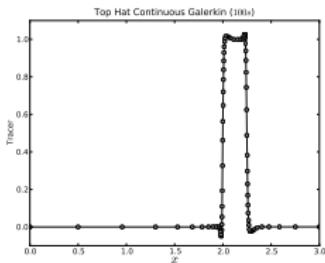
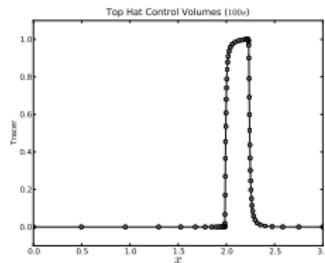


Figure: Initial top hat distribution.

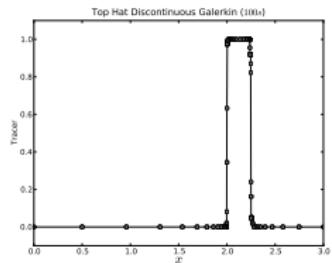
The top hat



CG



CV



DG

Continuous Galerkin

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

Control Volume

- ▶ Simple and efficient, sometimes diffusive
- ▶ Need to choose an interpolation method
- ▶ Here, “FiniteElement” interpolation with Sweby limiter 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 control volume interpolation method to HyperC.

Lid-Driven Cavity - Setup

- ▶ Simple test case often used for verification and validation.
- ▶ 2D flow with no slip boundaries on the bottom and sides.
Constant rightwards velocity imposed on the lid.
- ▶ Example file and data are for $Re = 1000$.

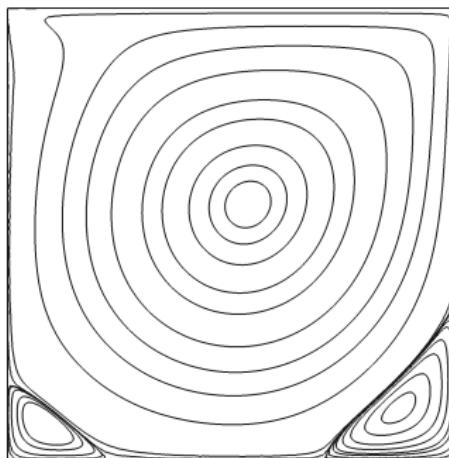
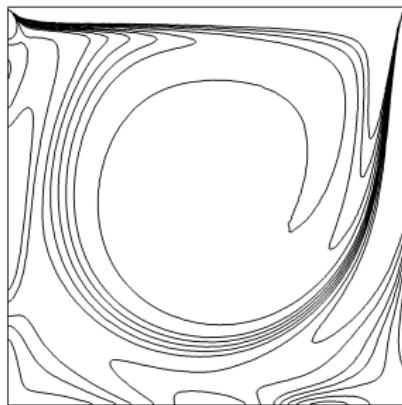
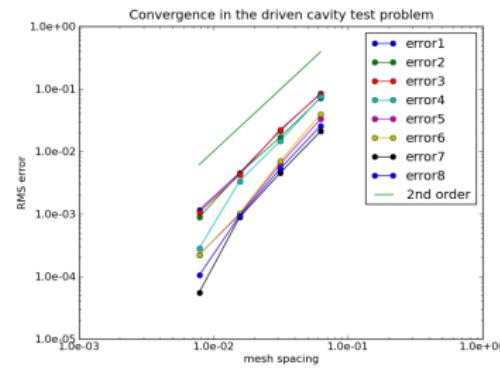


Figure: Streamfunction contours in converged solution for $h = 1/128$.

Lid–Driven Cavity - Output



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



(b) error vs. mesh spacing for various metrics from the literature

Figure: The error metrics are described in the manual, section 10.4.

Lid–Driven Cavity - Exercises

- ▶ What about mesh adaptivity?
- ▶ Boundary condition on the lid is set to avoid issues in the upper corners. What happens if we modify this? Use a small mesh first!
- ▶ What happens at higher Re ?
- ▶ What happens to wall clock time if you run in parallel?

Backward facing step (2D and 3D)

- ▶ Classic CFD benchmark test case,
- ▶ Experimental and numerical data available for comparison,
- ▶ Test of numerical methods or turbulence models,
- ▶ Serial and parallel simulations.

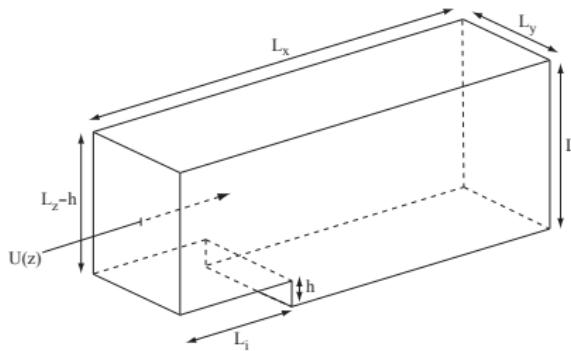


Figure: Geometry of the 3D backward facing step.

Backward facing step (2D and 3D)

- ▶ Simulation run in serial.

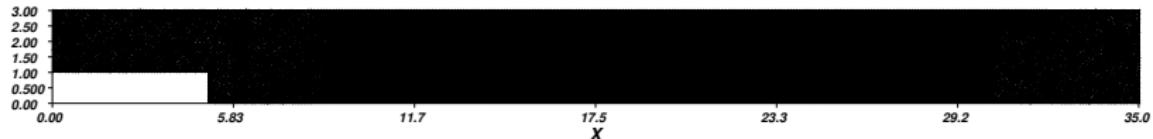


Figure: Geometry of 2D backward facing step showing the mesh.

Backward facing step, 2D results

- ▶ Reattachment point estimation,
- ▶ Profile evolution in time and space.

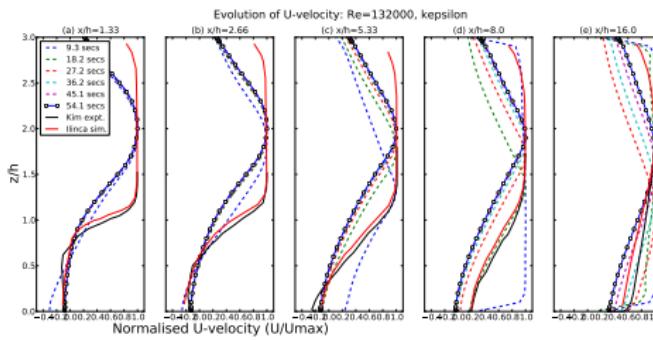


Figure: Streamwise velocity profiles at several points downstream of the step showing the converged solution. This is compared to experimental and other numerical data.

Backward facing step, 3D results

- ▶ Recirculation bubble and reattachment point.

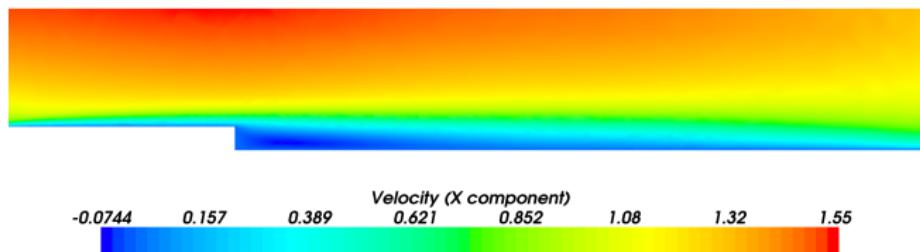


Figure: The velocity on a cut plane through the centre of the 3D geometry,
time = 50 s.

Backward facing step, exercises

- ▶ Increase the Reynolds number.
- ▶ Add adaptivity options for both 2D and 3D.

Flow past a sphere

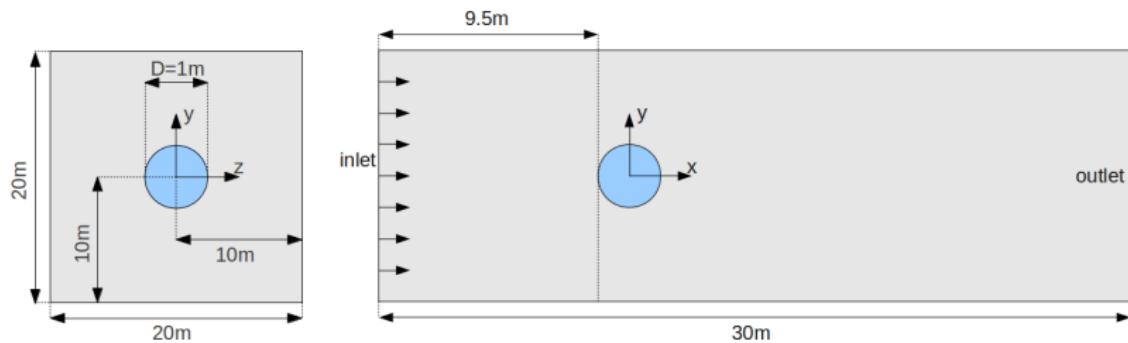
- ▶ Fluidity is used compute the drag force on an isolated sphere, see Chapter 10.7.2 and Equation (10.2) in the manual for more details.

- ▶ The results are compared against a curve optimised to fit a large amount of experimental data (Brown and Lawler, 2003), see Chapter 10.7.3 and Equation (10.3) in the manual for more details.

Flow past a sphere - Setup

- ▶ The sphere is modelled as a void space in the mesh.
- ▶ The flow is assumed to be incompressible
- ▶ Free-slip boundary conditions are applied to the top, bottom and side walls, and a no-slip condition to the surface of the sphere.

Flow past a sphere - Computational domain



Flow past a sphere - Results

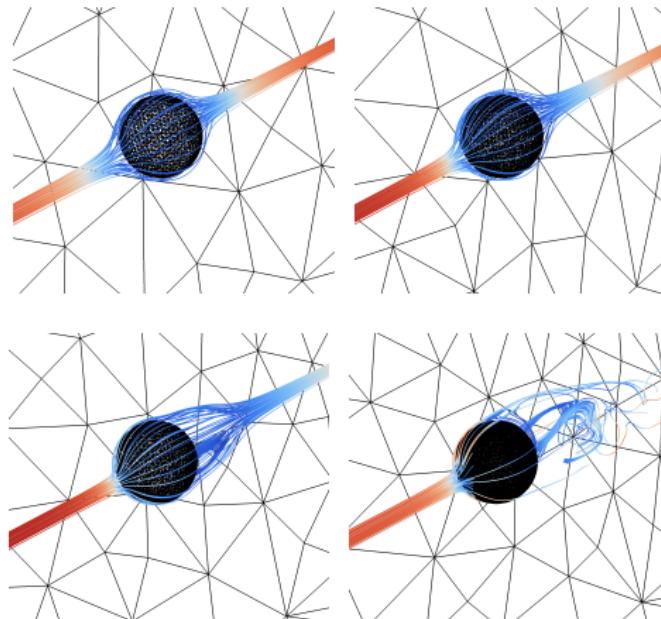
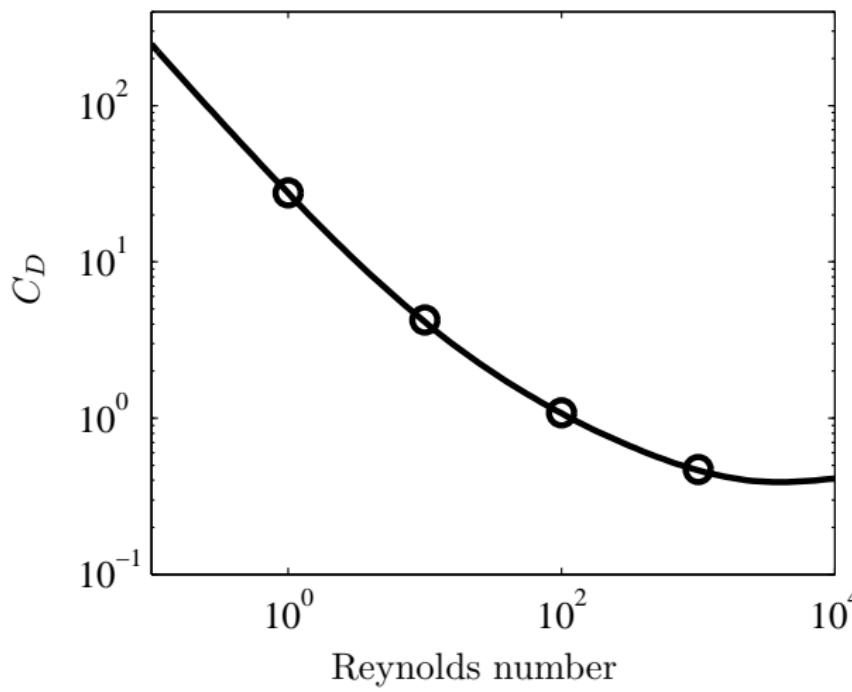


Figure: Streamlines and surface mesh in the flow past the sphere example.
Top-left to bottom-right show results from Reynolds numbers
 $Re = 1, 10, 100, 1000$.

Flow past a sphere - Results



Flow past a sphere - Exercises

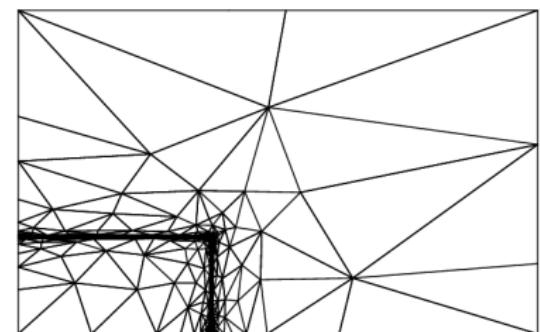
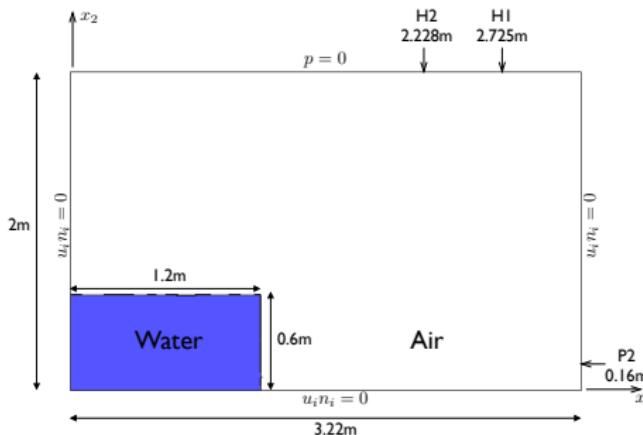
- ▶ We actually compute the (vector) force on the sphere and output this to the stat file. This can then be converted to the drag coefficient via (10.2). Write a Python function to do this conversion and the error from the correlation (10.3).
- ▶ Try varying some of the discretisation and adaptivity parameters to see what the impact on the accuracy of the calculated drag is.
- ▶ Try changing the shape of the object, e.g. benchmark data is also available for flow past a cylinder [Schäfer et al., 1996].

Water collapse

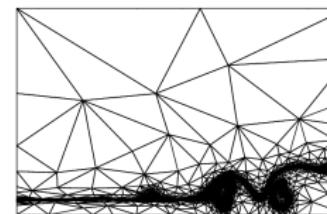
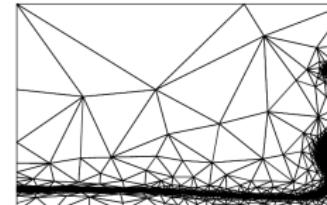
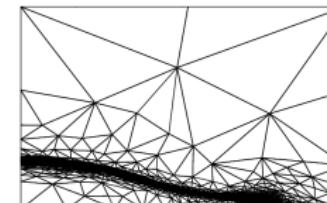
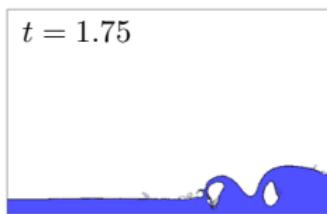
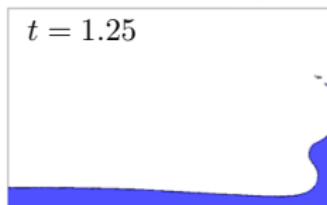
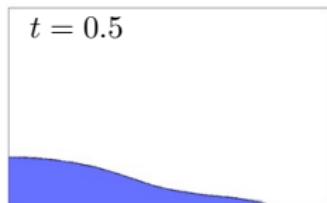
- ▶ Fluidity is used to replicate a laboratory experiment of a collapsing column of water within an atmosphere of air (Lakehal et al., 2002).
- ▶ A reservoir of water is initially held behind a barrier.
- ▶ The water column collapses and floods the rest of the tank when the barrier is quickly removed.

Water collapse - Simulation setup

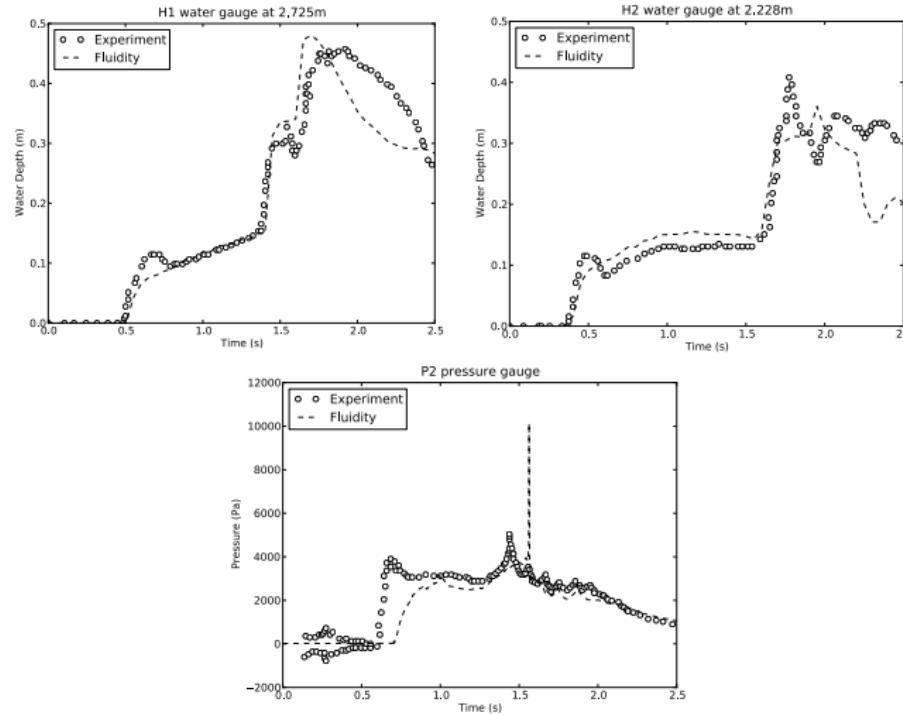
- ▶ The multi-material approach is used to represent the two fluids.
- ▶ The flow is assumed to be incompressible, inviscid, and 2D.
- ▶ Free-slip boundary conditions on the bottom and sides, and open top.



Water collapse - Numerical results (1)



Water collapse - Numerical results (2)



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

- ▶ Fluidity is used to replicate a laboratory experiment of tephra (fine volcanic ash particles) settling through a tank of water (Carey, 1997).
- ▶ Small tephra particles can settle either individually, or collectively as a cloud of particles (a plume).
- ▶ Plumes are generated when the bulk density of the tephra-water mixture becomes large enough, yielding settling velocities much greater than those expected of single particles (which settle at a velocity given by Stokes' law).

Tephra settling - Simulation setup

- ▶ The simulation uses a 0.3×0.7 metre domain, replicating the cross-section of the water tank used in the experiments.
- ▶ No normal flow boundary conditions are weakly imposed along with a zero velocity initial condition.
- ▶ The influx of particles from the air above is simulated using a flux boundary condition at the top of the domain. This allows tephra to flux in at a rate of $0.472 \text{ gm}^{-2}\text{s}^{-1}$.

Tephra settling - Numerical results (1)

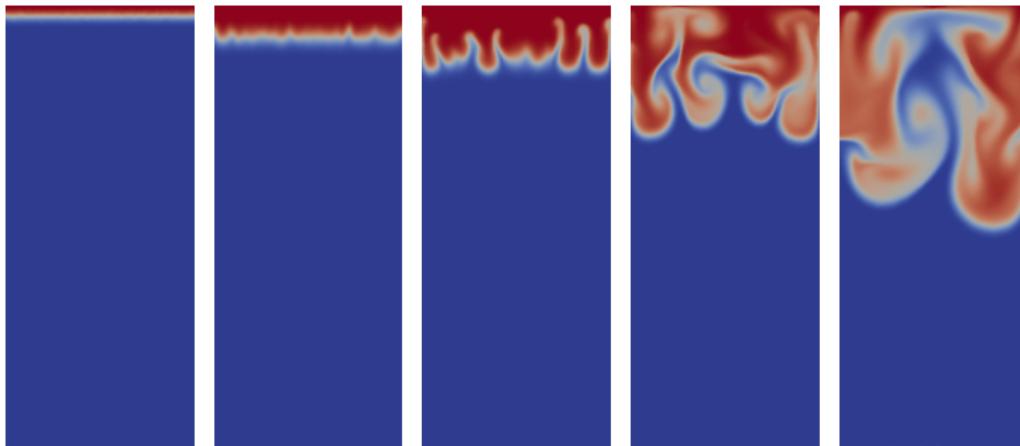


Figure: Simulation visualisations at $t = 10, 30, 50, 80$ and 110 seconds.
Tephra particles initially settle individually, but as more tephra fluxes in, the layer eventually becomes unstable and plumes begin to form.

Tephra settling - Numerical results (2)

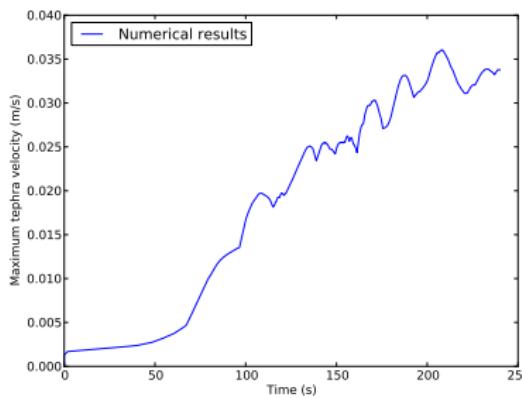


Figure: Plot of the maximum tephra phase velocity against time. Tephra particles initially settle at approximately 0.0017 ms^{-1} , as predicted by Stokes' law. Plumes begin to form after approximately 30 seconds, resulting in settling velocities over 10 times greater than that of an individual particle.

Tephra settling - Exercises

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

Data

The examples have been run in advance
and the output can be found in

/scratch/Training2012/

Summary of the GFD examples

Adam Candy

Department of Earth Science and Engineering, Imperial College London

Outline

Lock-exchange

Tsunami

Rotating periodic channel

Restratification after open-ocean deep convection

Tides in the Mediterranean Sea

Setting up simulations of geophysical processes

The lock-exchange

- ▶ Flat bottomed tank, separated into two partitions by a barrier.
- ▶ Each half is filled with fluids of different density (temperature).
- ▶ The barrier is removed, and the denser fluid collapses under the lighter.
- ▶ The mesh changes in time as the dynamics evolves.

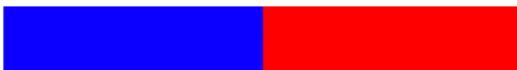


Figure: Lock-exchange initial temperature distribution. Blue labels dense fluid, and red lighter.

The lock-exchange

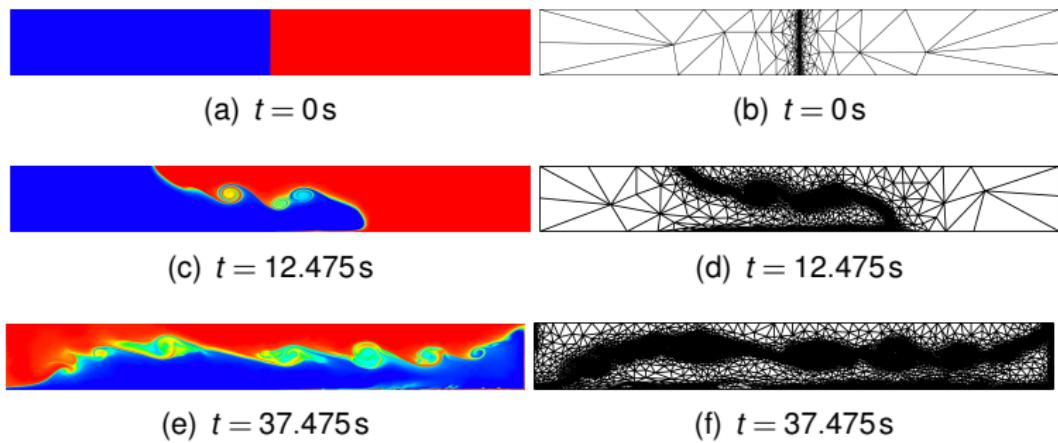


Figure: Lock-exchange temperature distribution (colour) with meshes, over time (t).

The lock-exchange, diagnostics

- ▶ Front speed (or Froude number).
- ▶ Mixing indicated by domain fractions of fluid in specified temperature classes.

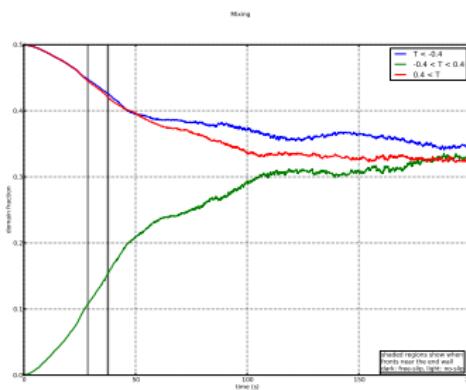


Figure: Time evolution of fraction of domain that contains fluid in three temperature classes. Blue: cold, red: warm, green: mixed

The lock-exchange, exercises

- ▶ Experiment with the adaptivity options.
- ▶ Try adding some detectors to visualise the particle trajectories.

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 functionality of Fluidity.

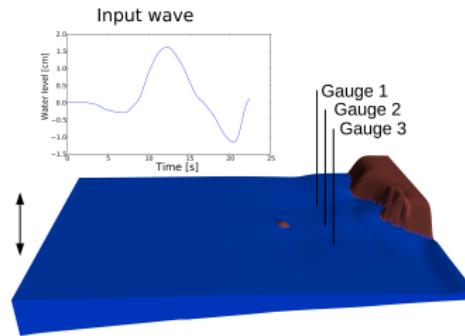


Figure: The domain and the three gauge stations.

Hokkaido-Nansei-Oki tsunami

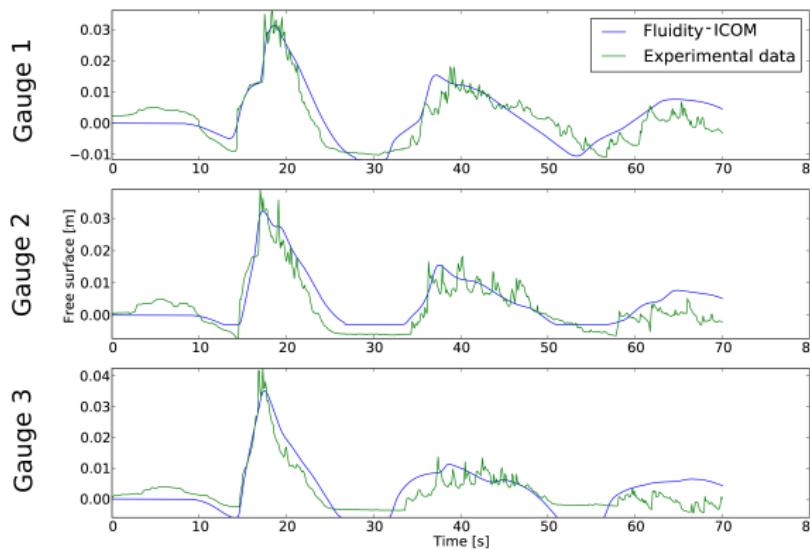


Figure: The numerical and experimental results at the three gauge stations.

Hokkaido-Nansei-Oki tsunami - Exercises

- ▶ Add more detectors.
- ▶ Check how increasing the wetting and drying threshold parameter affects the results.
- ▶ Try changing the viscosity value (How does it affect the inundation of the tsunami event?).

Rotating periodic channel

- ▶ Domain: Unit square, periodic in zonal direction.
- ▶ Boundary conditions: No-slip at the North and South boundaries.
- ▶ Coriolis applied.
- ▶ Forcing applied by a Python function.
- ▶ The flow is driven by a velocity source term:

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

- ▶ Parameters are chosen such that the solution converges to a steady state with a known analytical solution.

Rotating periodic channel

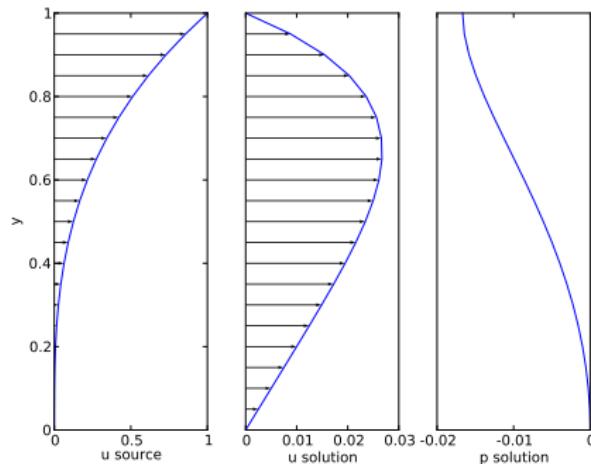


Figure: Velocity forcing term and analytic solutions for velocity and pressure.
Each of these quantities are constant in the x direction.

Rotating periodic channel

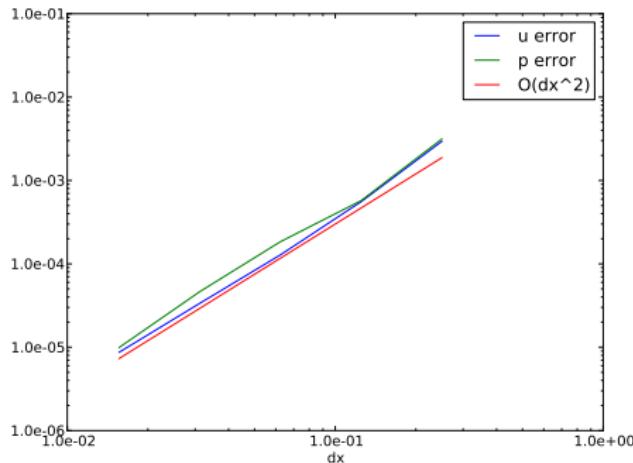


Figure: Error in the pressure and velocity solutions as a function of mesh resolution.

Rotating periodic channel, exercises

- ▶ Understand the use of analytic forcing functions in Fluidity using Python (e.g. take a look at `channel_tools.py` and `plot_theory`).

Restratification - Setup

- ▶ This example is an idealised model of the restratification phase of open-ocean deep convection (OODC).
- ▶ The setup is taken from Rousset et al. [2009].
- ▶ Cylinder, radius 250km, height 1km.

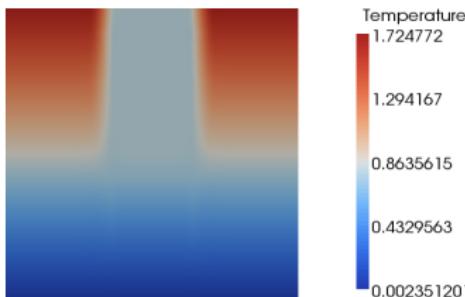
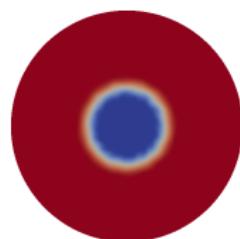
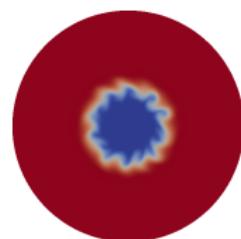


Figure: Cross section of initial temperature distribution.

Restratification - Output



(a) 10 days



(b) 20 days



(c) 30 days



(d) 40 days

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

Restratification - Exercises

- ▶ How does mesh adaptivity influence the solution?
- ▶ Fixed mesh resolution size - what happens as you increase/decrease the resolution?
- ▶ Discretisation of temperature - what changes between DG, CG and CV?
- ▶ Scaling - how far can you push the parallelisation of this run, with and without adaptivity?

Tides in the Mediterranean Sea

- ▶ Tidal modelling is a widely used method for validating free surface implementations.
- ▶ Tides introduced by an astronomical body forcing, and a
- ▶ Co-oscillating boundary tide forcing.
- ▶ This example considers the four main tidal constituents:
 M_2 , S_2 , K_1 and O_1 .

Tides in the Mediterranean Sea

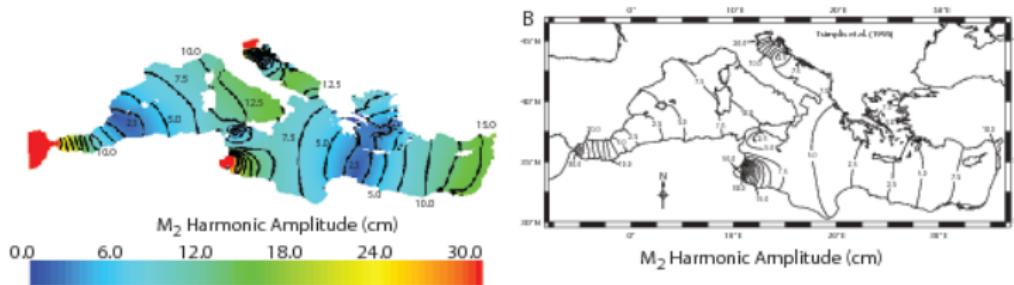


Figure: The M_2 tidal harmonic amplitude from (left) Fluidity–ICOM and (right) a high resolution tidal model[†].

[†] M. N. Tsimplis *et al.* (1995), J. Geophys. Res. 100 (C8).

Tides in the Mediterranean Sea

The amplitude of each of the tidal components is considered, and a RMS error of the difference of these to observed tide guage data is calculated. The locations of these tide guages is shown below.

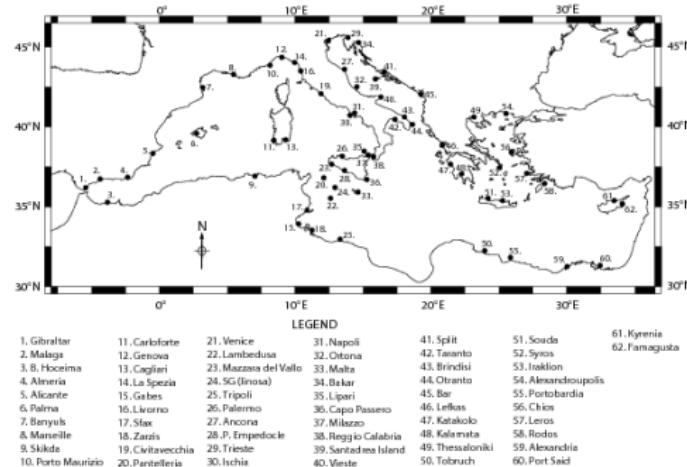


Figure: Locations of 62 tide gauges in the Mediterranean Sea used to calculate the root mean square error.

Tides in the Mediterranean Sea - exercises

- ▶ Consider the forcing tidal components contained in the netCDF file 'med.nc' (e.g. using ncview).
- ▶ Examine the mesh features (e.g. open the '.msh' file in Gmsh).
- ▶ Look at how to apply the forcing of different tidal components in the simulation.
- ▶ Limit the error calculation by region, are the errors greater in some parts of the Mediterranean?

Setting up simulations of geophysical processes, an overview

The following aspects will be outlined:

- ▶ Meshing
 - Gmsh is used to produce a mesh on a spherical manifold,
 - An extrusion, creating a 3D mesh is carried out within fluidity, controlled by options in Diamond.
- ▶ Additional options in Diamond for:
 - Gravity force,
 - Coriolis pseudo-force,
 - Bottom drag,
 - Free surface,
 - Tidal forcing.



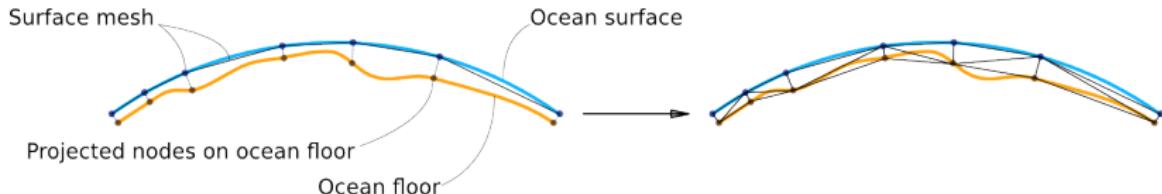
2D horizontal mesh



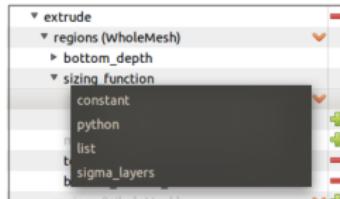
- ▶ Mesh is generated on a spherical shell using Gmsh.
Tutorial http://amcg.ese.ic.ac.uk/files/gmsh_tutorial.pdf OR
http://perso.uclouvain.be/jonathan.lambrechts/gmsh_ocean/
- ▶ Coastlines extracted from the GSHHS dataset.
- ▶ Mesh available in
trunk/examples/tides_in_the_Mediterranean_Sea/med.msh

Extruding in Fluidity

- ▶ 2D horizontal mesh is extruded vertically within Fluidity at runtime.



- ▶ Flexibility to choose layered σ or z -coordinates.
- ▶ Bathymetry extracted from a NetCDF file (e.g. GEBCO dataset).



Extrusion: Representation of the surface of the spherical Earth

- ▶ Linear mapping
 - Element faces on sea surface are flat,
 - Appropriate choice for P1 grids,
 - Suitable for barotropic simulations.
- ▶ Super-parametric mapping
 - Element faces on sea surface are described by n^{th} order polynomials,
 - Order of polynomial equal to the degree of the mesh,
 - Suitable for baroclinic simulations (currently up to 2nd order in parallel).

Gravity, Coriolis and bottom drag

- ▶ Gravity force and Coriolis pseudo-force are set under physical parameters
 - Gravity magnitude and direction,
 - Coriolis (f –, β –plane, on-sphere).
- ▶ Ocean floor: no-normal-flow & drag wall-law

$$\frac{\partial u}{\partial z} \propto D_{bottom} = \frac{1}{2} C_D \rho u^2,$$

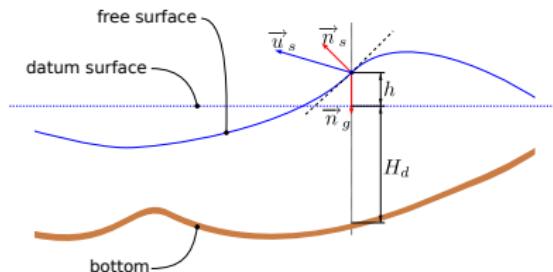
set as boundary condition on velocity:

▼ boundary_conditions (drag_on_bottom_and_sides)	■
surface_ids	
▼ type (drag)	▼
constant	▼
▼ quadratic_drag	▼

Setting the free surface

- ▶ Free surface - kinematic boundary condition on velocity.
 - The velocity of a particle at the surface is tangential to the surface.

$$-\vec{n}_s \cdot \vec{n}_g \frac{\partial h}{\partial t} = \vec{n}_s \cdot \vec{u}$$



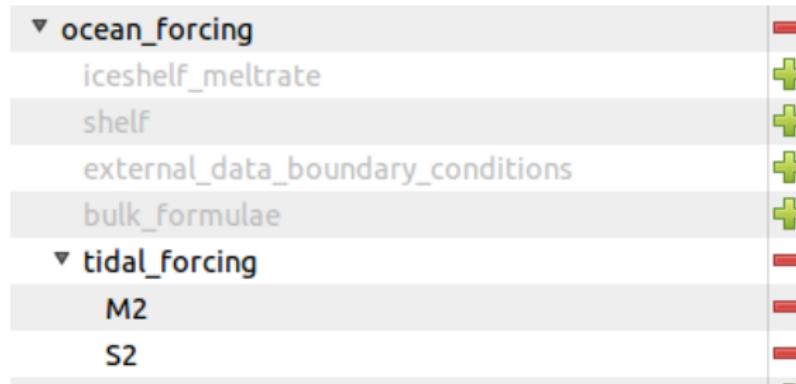
▼ boundary_conditions (FreeSurface)

surface_ids

► type (free_surface)

Setting the tidal forcing-body forcing

- ▶ Astronomical body forcing.
 - Specify the tidal components.



Setting the tidal forcing-free surface elevation at boundary

- ▶ Dirichlet boundary condition on pressure at open boundaries.
 - Pressure set for free-surface elevation to match observational data.
- ▶ User has to specify:
 - Which tidal components,
 - NetCDF file to read free-surface elevation from,
 - Name of phase and magnitude variables in file.

The screenshot shows a software interface for managing simulation parameters. On the left, there is a tree view of configuration sections:

- repair_stiff_nodes
- atmospheric_pressure
- scheme
- solver
- initial_condition (WholeMesh)
- boundary_conditions (BoundaryTide)
 - surface_ids
 - type (dirichlet)
 - apply_weakly
 - from_file
 - tidal (M2) (highlighted in orange)
 - tidal (S2)
 - tidal (K1)
 - tidal (O1)
 - tidal

On the right, there is a detailed view of the 'from_file' section for the 'tidal (M2)' entry, showing its attributes:

Attributes	Name	Value
file_name	med.nc	
variable_name_amplitude	M2amp	
name	M2	
variable_name_phase	M2phase	

Below the attributes table, there is a 'Data' section which currently says 'No data'. At the bottom, there is a 'Comment' section with the text 'No comment'.

Data

The examples have been run in advance
and the output can be found in

/scratch/Training2012/

Stokes Example

“Team Stokes”

Stokes Equation:

$$\nabla \cdot [\mu (\nabla \tilde{u} + \nabla \tilde{u}^T)] - \nabla p = Ra T \hat{k}$$

Instantaneous balance (no inertia).

Example considered:

Stokes flow driven by thermal buoyancy.
2D square domain.

*Conservation of mass and temperature
advection-diffusion equations also solved.*

Setup

$T = 0$ (Fixed)

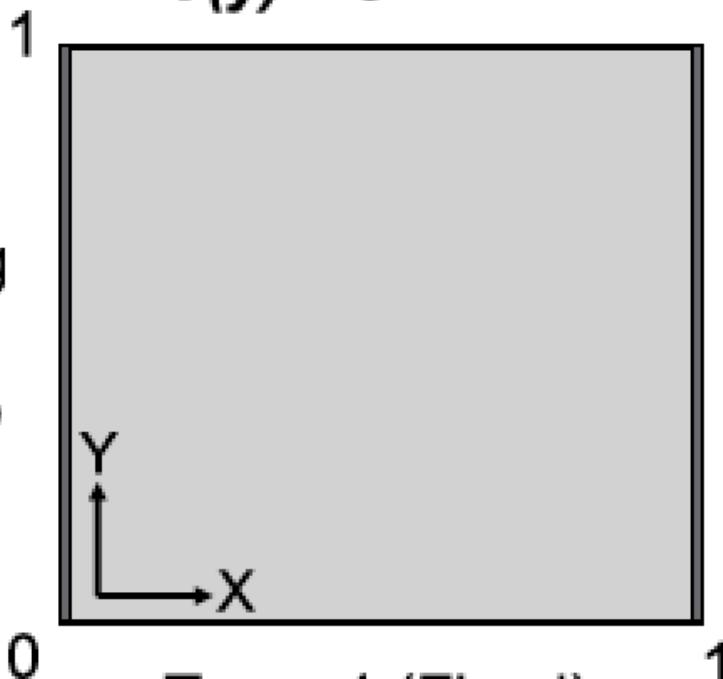
$\mathbf{V}(x) = \text{Free Slip}$

$\mathbf{V}(y) = 0$

$T = \text{Insulating}$

$\mathbf{V}(x) = 0$

$\mathbf{V}(y) = \text{Free Slip}$



$T = 1$ (Fixed)

$\mathbf{V}(x) = \text{Free Slip}$

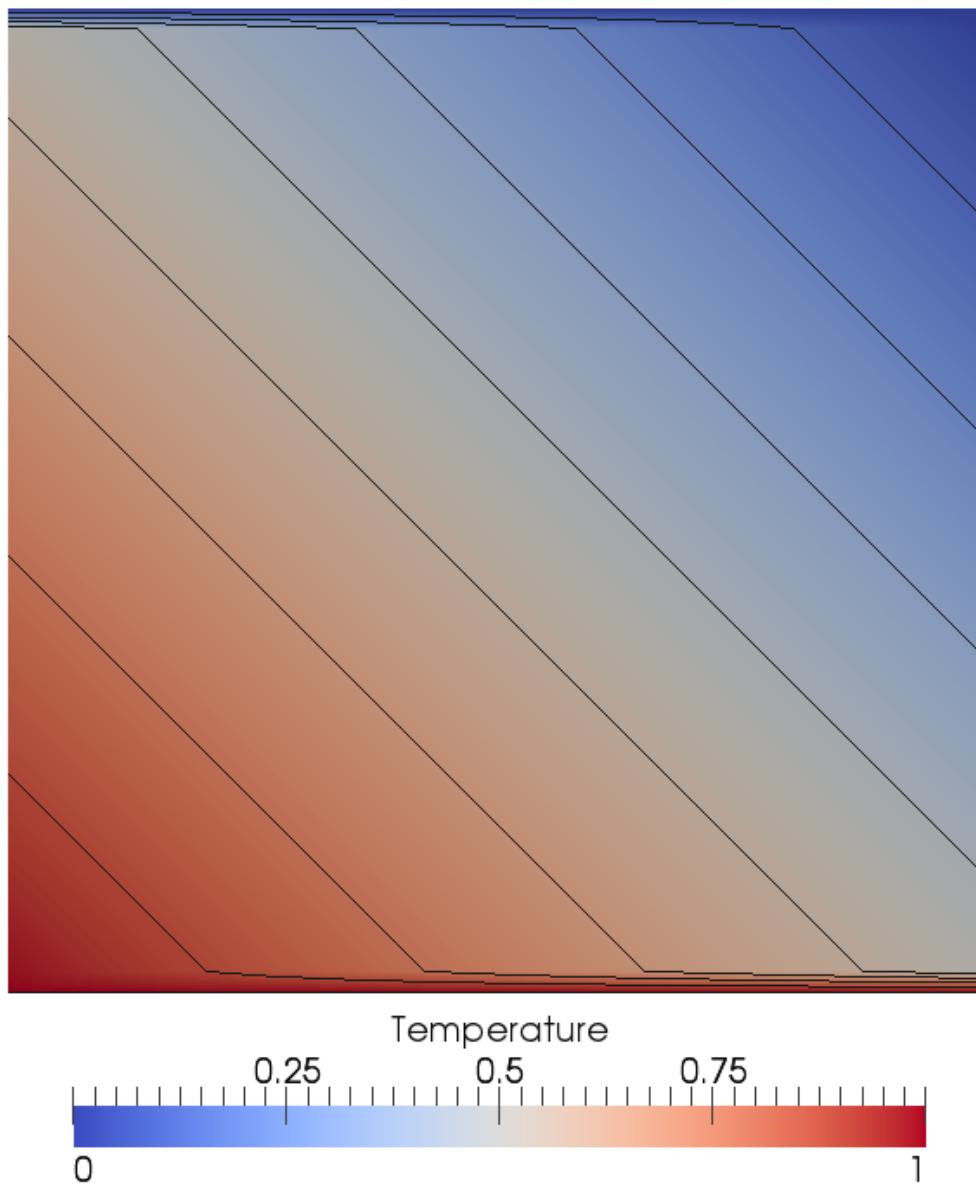
$\mathbf{V}(y) = 0$

$T = \text{Insulating}$

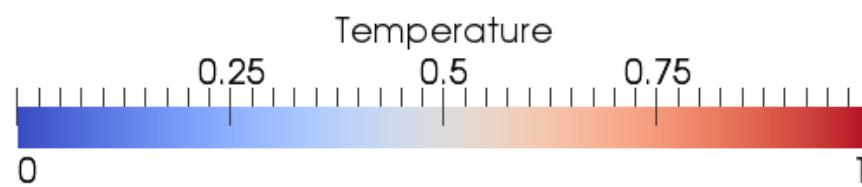
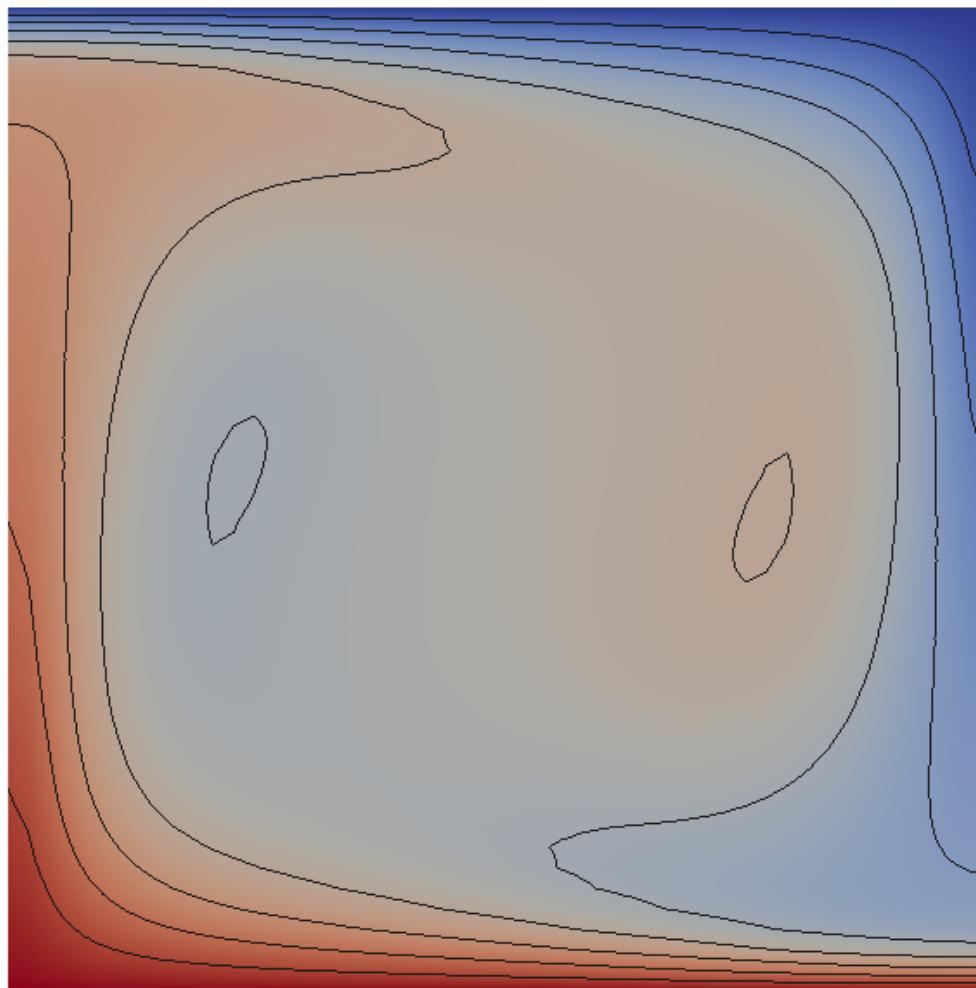
$\mathbf{V}(x) = 0$

$\mathbf{V}(y) = \text{Free Slip}$

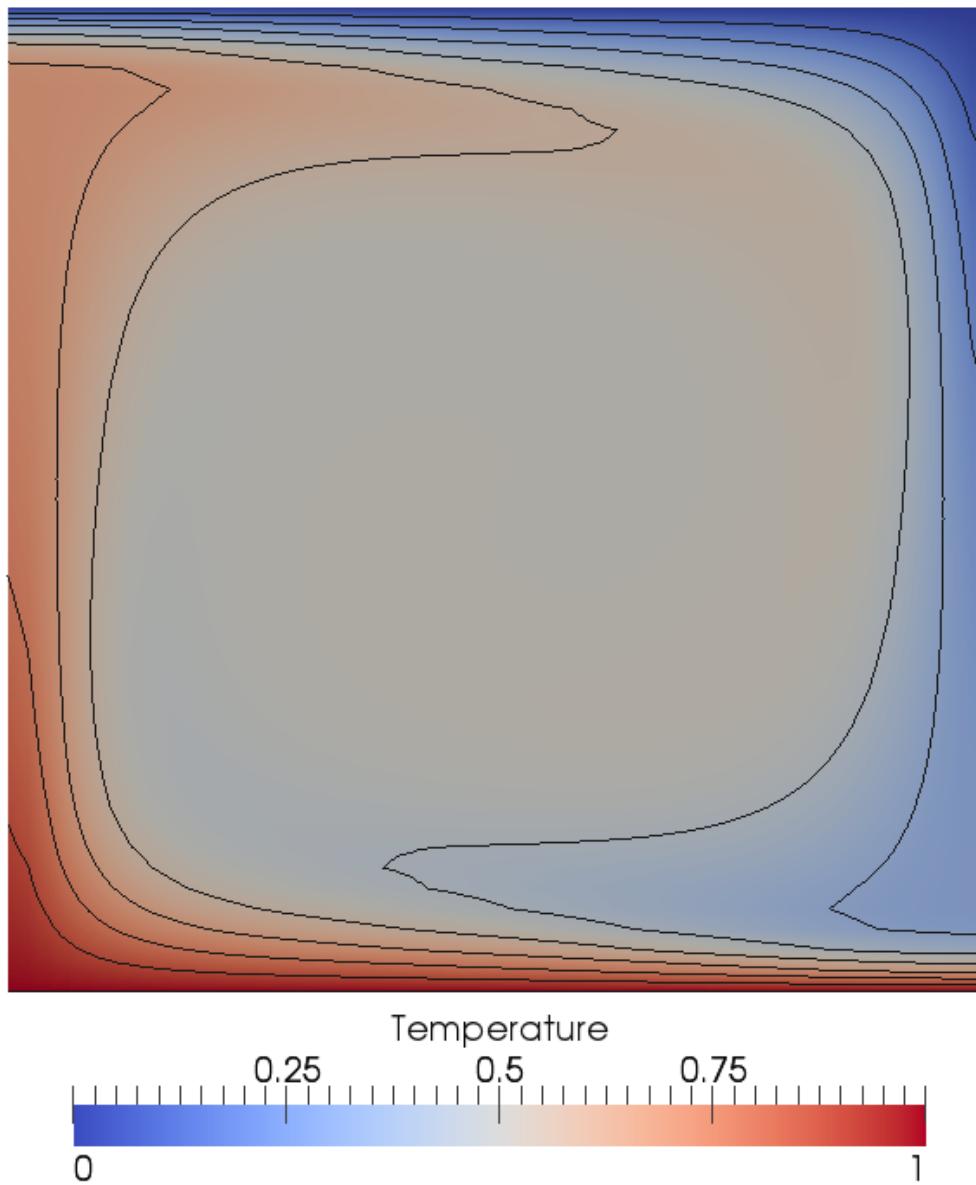
Initial condition



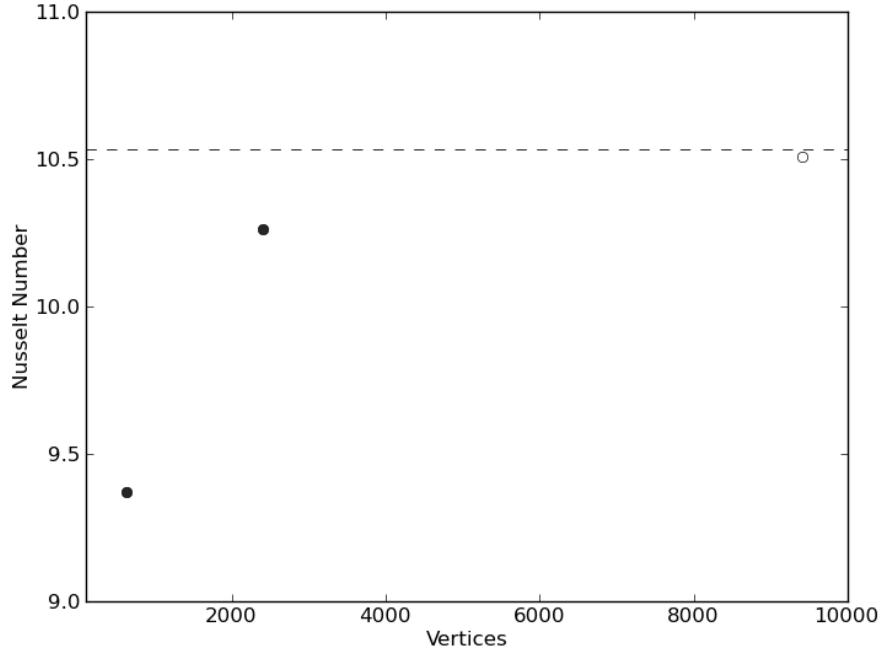
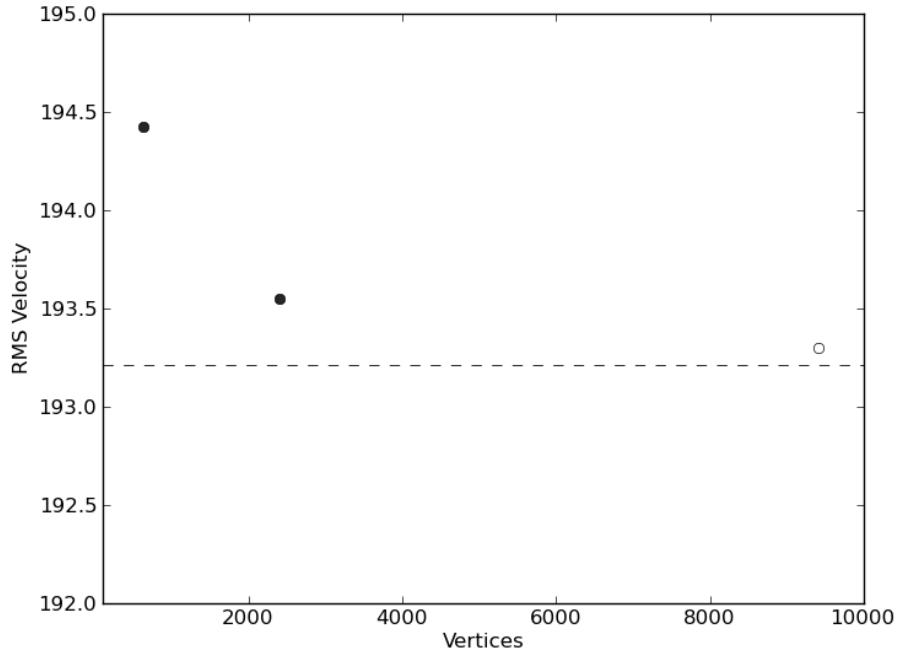
Intermediate:



Approaching steady-state:



Diagnostics:



With increasing grid resolution, results converge towards benchmark solutions (dashed lines), as expected.