Case C3.6: Flow over a 2D periodic hill

DNS results at Re = 2,800

D. Moxey, J. Peiro, S. Sherwin

Department of Aeronautics,
Imperial College London

2nd International Workshop on High-Order CFD Methods
Cologne, 28th May 2013

Overview

- Outline of numerical discretisation
- Representative results
- Summary

Numerical discretisation

Simulations performed with open-source code *Semtex*.

- Joint continuous discretisation: 2D nodal spectral element (quads only), 1D Fourier pseudospectral.
- High-order velocity correction discretisation of incompressible Navier-Stokes equations.
- Three-step splitting scheme: explicit advection, implicit pressure/velocity solves.
- Direct static-condensation on each 2D plane.
- Volumetric flowrate imposed through Stokes correction function.
- Parallelisation in Fourier direction using MPI (expensive!)

Numerical discretisation

Simulations performed with open-source code *Semtex*.

- Joint continuous discretisation: 2D nodal spectral element (quads only), 1D Fourier pseudospectral.
- High-order velocity correction discretisation of incompressible Navier-Stokes equations.
- Three-step splitting scheme: explicit advection, implicit pressure/velocity solves.
- Direct static-condensation on each 2D plane.
- Volumetric flowrate imposed through Stokes correction function.
- Parallelisation in Fourier direction using MPI (expensive!)

Numerical discretisation

Simulations performed with open-source code *Semtex*.

- Joint continuous discretisation: 2D nodal spectral element (quads only), 1D Fourier pseudospectral.
- High-order velocity correction discretisation of incompressible Navier-Stokes equations.
- Three-step splitting scheme: explicit advection, implicit pressure/velocity solves.
- Direct static-condensation on each 2D plane.
- Volumetric flowrate imposed through Stokes correction function.
- Parallelisation in Fourier direction using MPI (expensive!)

Numerical discretisation

Simulations performed with open-source code *Semtex*.

- Joint continuous discretisation: 2D nodal spectral element (quads only), 1D Fourier pseudospectral.
- High-order velocity correction discretisation of incompressible Navier-Stokes equations.
- Three-step splitting scheme: explicit advection, implicit pressure/velocity solves.
- Direct static-condensation on each 2D plane.
- Volumetric flowrate imposed through Stokes correction function.
- Parallelisation in Fourier direction using MPI (expensive!)

Numerical discretisation

Simulations performed with open-source code *Semtex*.

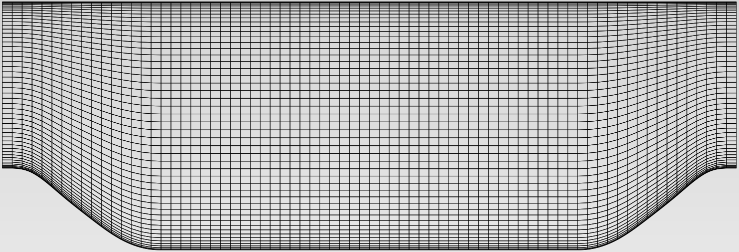
- Joint continuous discretisation: 2D nodal spectral element (quads only), 1D Fourier pseudospectral.
- High-order velocity correction discretisation of incompressible Navier-Stokes equations.
- Three-step splitting scheme: explicit advection, implicit pressure/velocity solves.
- Direct static-condensation on each 2D plane.
- Volumetric flowrate imposed through Stokes correction function.
- Parallelisation in Fourier direction using MPI (expensive!)

Numerical discretisation

Simulations performed with open-source code *Semtex*.

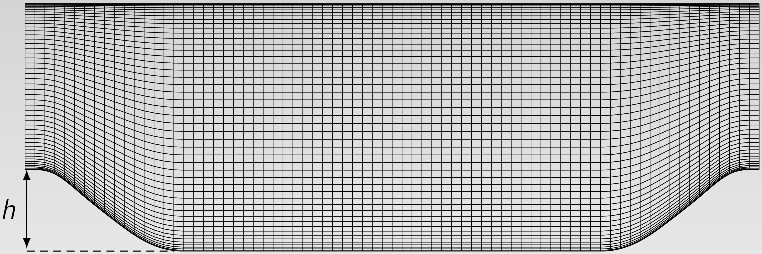
- Joint continuous discretisation: 2D nodal spectral element (quads only), 1D Fourier pseudospectral.
- High-order velocity correction discretisation of incompressible Navier-Stokes equations.
- Three-step splitting scheme: explicit advection, implicit pressure/velocity solves.
- Direct static-condensation on each 2D plane.
- Volumetric flowrate imposed through Stokes correction function.
- Parallelisation in Fourier direction using MPI (expensive!)

Simulation resolution



- Very sensitive to choice of resolution
- 2D mesh: 3,626 elements, polynomial order $P = 6$.
- 160 Fourier planes
- $\sim$ 28 million degrees of freedom
- Custom mesh generation

Simulation resolution



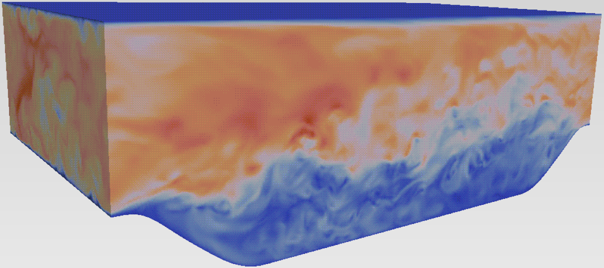
- Very sensitive to choice of resolution
- 2D mesh: 3,626 elements, polynomial order $P = 6$.
- 160 Fourier planes
- $\sim$ 28 million degrees of freedom
- Custom mesh generation

Simulation parameters

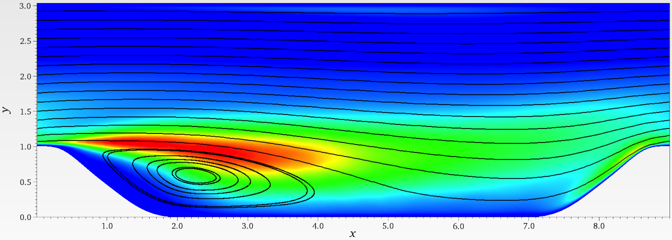
Parameter	Value
Δt	$0.001 t_c$
T_{avg}	$275 t_c$
Re_b	2,800
N_{proc}	80
T_{TauBench}	9.281s

- Simulation uses around $32k$ work units per t_c ($\sim$ 12 days for T_{avg}).
- Normalisation by bump height h and bulk velocity u_b.
- Convective time $t_c := t u_b / h$

Flow features

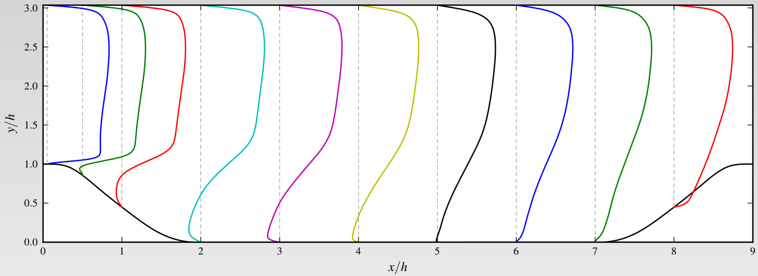


Inst. velocity

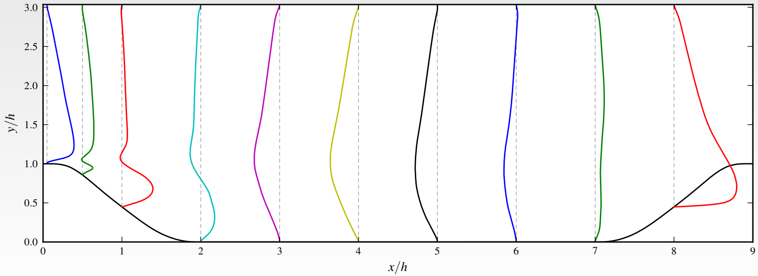


k, $\langle u \rangle$ s-lines

Velocity profiles



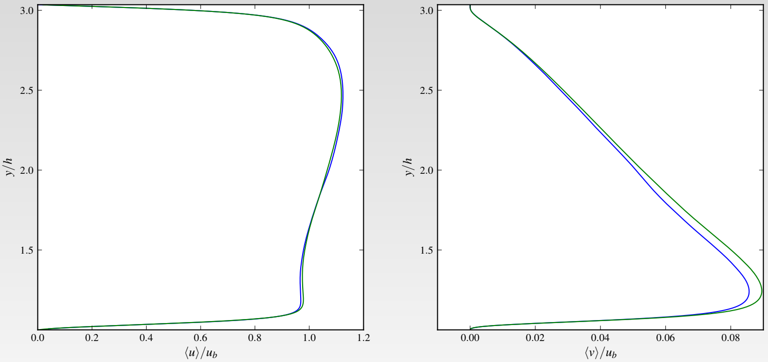
$\langle u \rangle / u_b$



$\langle v \rangle / u_b$

Velocity profiles: comparison

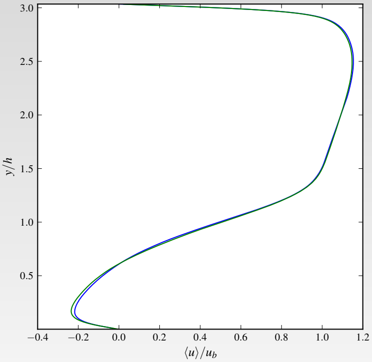
Comparison of simulation vs. reference solutions (LESOCC), available on QNET CFD wiki. At $x/h = 0.05$:

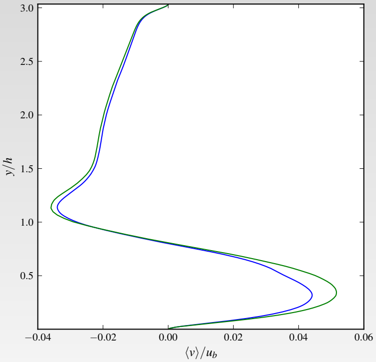


Good convergence at inlet

Velocity profiles: comparison

Comparison of simulation vs. reference solutions (LESOCC), available on QNET CFD wiki. At $x/h = 2$:

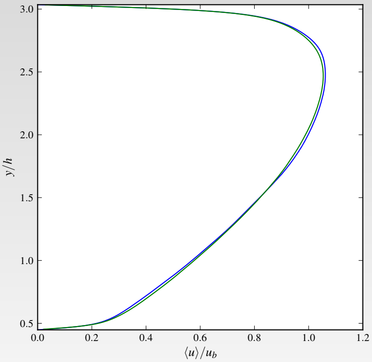


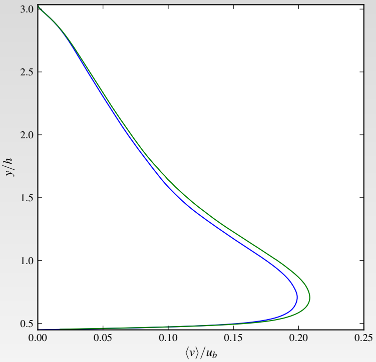


Not so great in recirculation region (time-averaging/resolution)

Velocity profiles: comparison

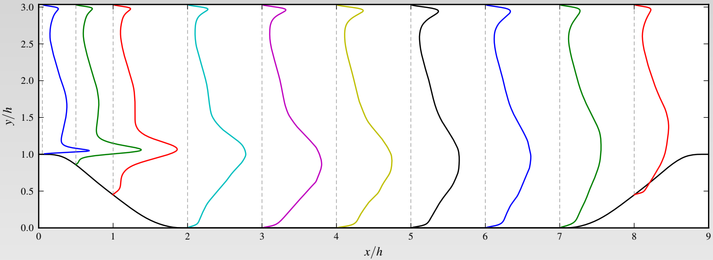
Comparison of simulation vs. reference solutions (LESOCC), available on QNET CFD wiki. At $x/h = 8$:



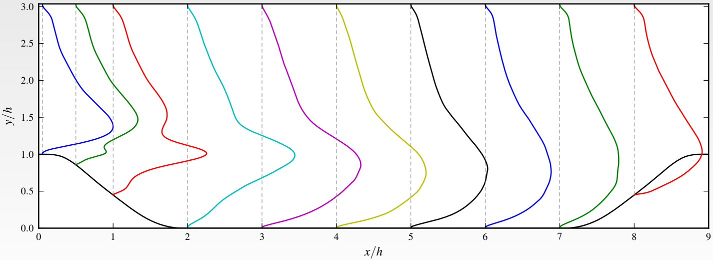


Improves as we continue downstream

Reynolds stresses



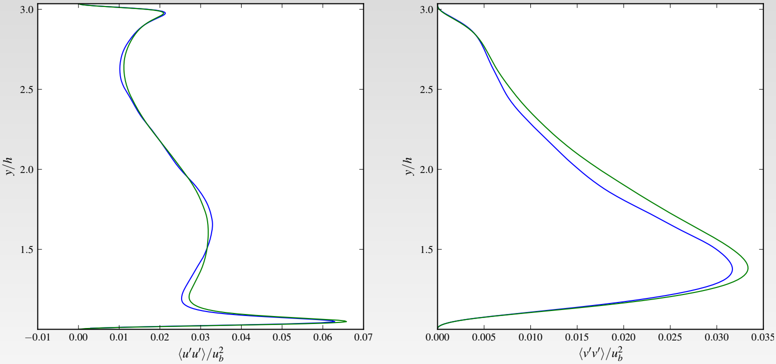
$\langle u'u' \rangle / u_b^2$



$\langle v'v' \rangle / u_b^2$

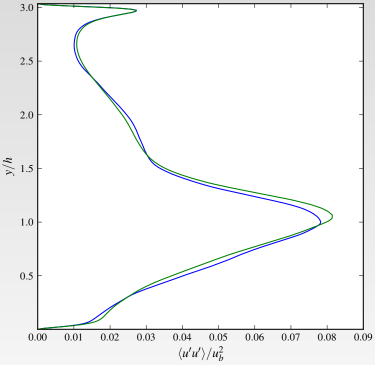
Reynolds stresses: comparison

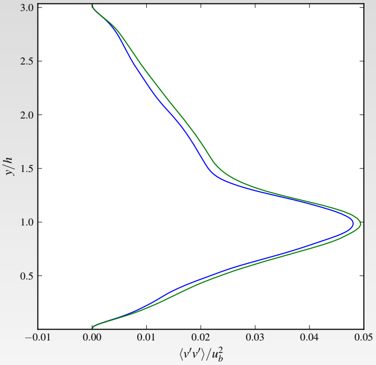
Comparison of simulation vs. reference for Reynolds stresses show a similar picture. At $x/h = 0.05$:



Reynolds stresses: comparison

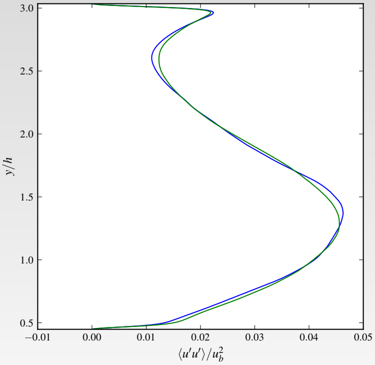
Comparison of simulation vs. reference for Reynolds stresses show a similar picture. At $x/h = 2$:

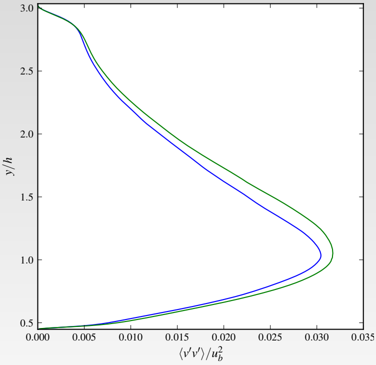




Reynolds stresses: comparison

Comparison of simulation vs. reference for Reynolds stresses show a similar picture. At $x/h = 8$:





Summary

- Results show fairly good agreement with existing linear profiles. . .
- . . . however still probably requires further temporal averaging for statistical convergence.

Future work:

- Investigate meshing options to reduce number of degrees of freedom required.
- Comparison against experimental data at $\mathrm{Re} = 5{,}600$.
- Implement parallelism in spectral element planes for more efficient communication pattern (Nektar++).