2D periodic hill database 2021

Sylvain Laizet, Imperial College London

April 2021

New database with 27 simulations (3 different heights, 3 different streamwise extents and 3 different hill shapes

Flow configuration

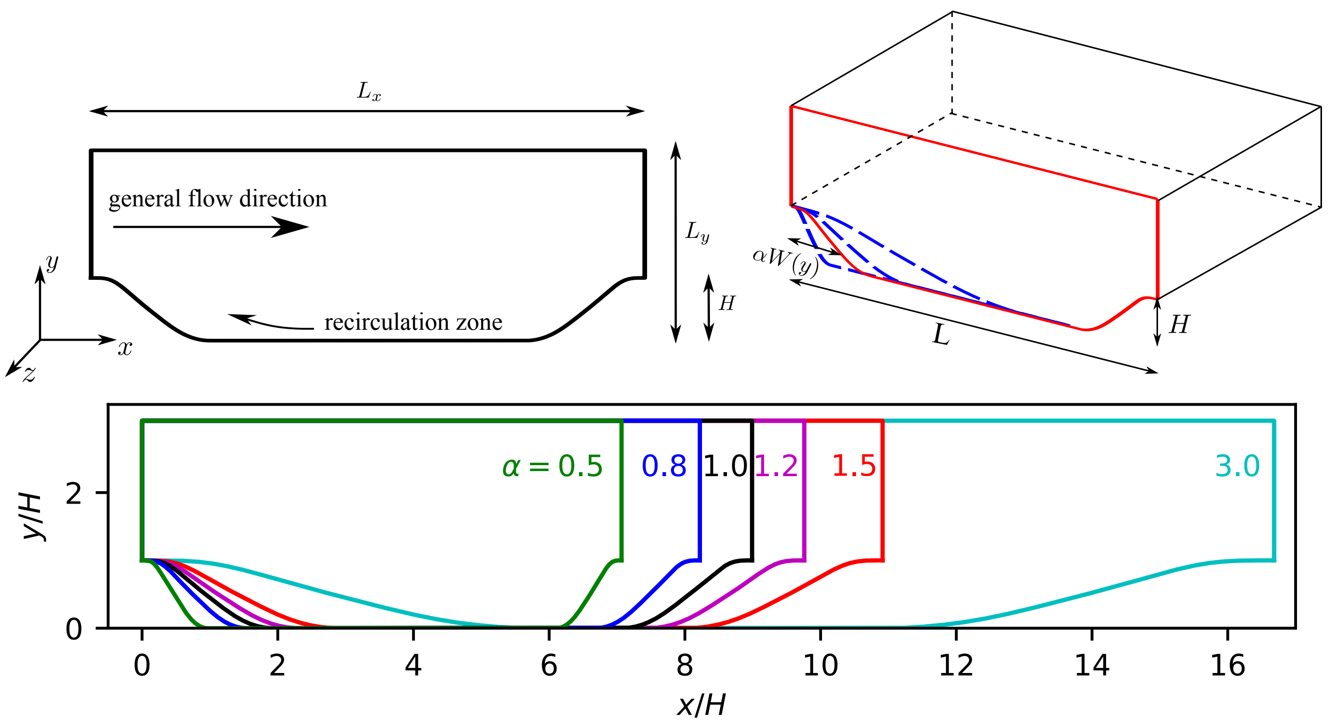


Figure 1: Schematic of the periodic hill geometry. The x, y and z coordinates are aligned with streamwise, wall-normal, and spanwise, respectively. The dimensions of the original geometry are normalized with H with $L_x/H = 9$, $L_y/H = 3.036$, and $L_z/H = 4.5$. Geometry variations obtained by scaling the width of the hill by a factor of $\alpha = 0.5$, 1.0 (baseline), and 1.5. Three different lengths for the flat section are investigated, with the total horizontal length of the domain equal to $L_x/H = 3.858\alpha+2.142$, $L_x/H = 3.858\alpha+5.142$ and $L_x/H = 3.858\alpha+8.142$. Three different domain height ($L_y/H = 2.024,\ 3.036$ and 4.048 are investigated.

Parameters

Name folder: `alphXX-YYYY-ZZZZ` where `XX` is the parameter $\alpha = 0.5, 1.0$ or 1.5. `YYY` is the streamwise dimension of the domain $L_x/H = 6, 9, 12$ for $\alpha = 1$, $L_x/H = 7.929, 10.929, 13.929$ for $\alpha = 1.5$, $L_x/H = 4.071, 7.071, 10.071$ for $\alpha = 0.5$. **Content:**

- `yp.dat`: Refined mesh in the vertical direction. In the other directions, $\Delta x = L_x/n_x$ and $\Delta z = L_z/n_z$.
- `mean.vtr`: Paraview file for umean, vmean, wmean, pmean, dissmean and vortmean (mean dissipation and vorticity fields).
- `rms_1.vtr`: Paraview file for rms of velocity and pressure fields.
- `rms_2.vtr`: Paraview file for Reynolds stresses (uv,uw and vw).
- `mean_files.dat`: ASCII file for umean, vmean, wmean, pmean (with X and Y coordinates).
- `rms_1.dat`: ASCII file for rms of velocity and pressure fields.
- `rms_2.dat`: ASCII file for Reynolds stresses (uv,uw and vw).
- `diss_vort_files.dat`: ASCII file for dissmean, vort (with X and Y coordinates).

Example:

```
open(10,file='mean_files.dat',status='unknown',form='formatted')
do j=1,ny
do i=1,nx
read(10,*) y1(i),y2(j),umean(i,j,1),vmean(i,j,1),wmean(i,j,1),pmean(i,j,1)
enddo
enddo
close(10)
```

Numerical Methods

The 27 Direct Numerical Simulations (DNS) are performed by solving the forced incompressible Navier-Stokes equations for a fluid with a constant density ρ:

$$\frac{\partial \mathbf{u}}{\partial t} = -\frac{1}{\rho}p - \frac{1}{2}\left[(\mathbf{u}\otimes\mathbf{u}) + (\mathbf{u}\cdot)\mathbf{u}\right] + \nu^2\mathbf{u} + \mathbf{f} \tag{1}$$

$$\nabla\cdot\mathbf{u} = 0 \tag{2}$$

where $\otimes$ indicates outer-product of vectors; $p(\mathbf{x},t)$ is the pressure field and $\mathbf{u}(\mathbf{x},t)$ the velocity field; $\mathbf{x}$ and t are spatial and temporal coordinates, respectively. The forcing field $\mathbf{f}(\mathbf{x},t)$ is used to enforce boundary conditions through an immersed boundary method and to ensure a specified mass flux. Note that convective terms are written in the skew-symmetric form, which allows the reduction of aliasing errors while conserving the energy.

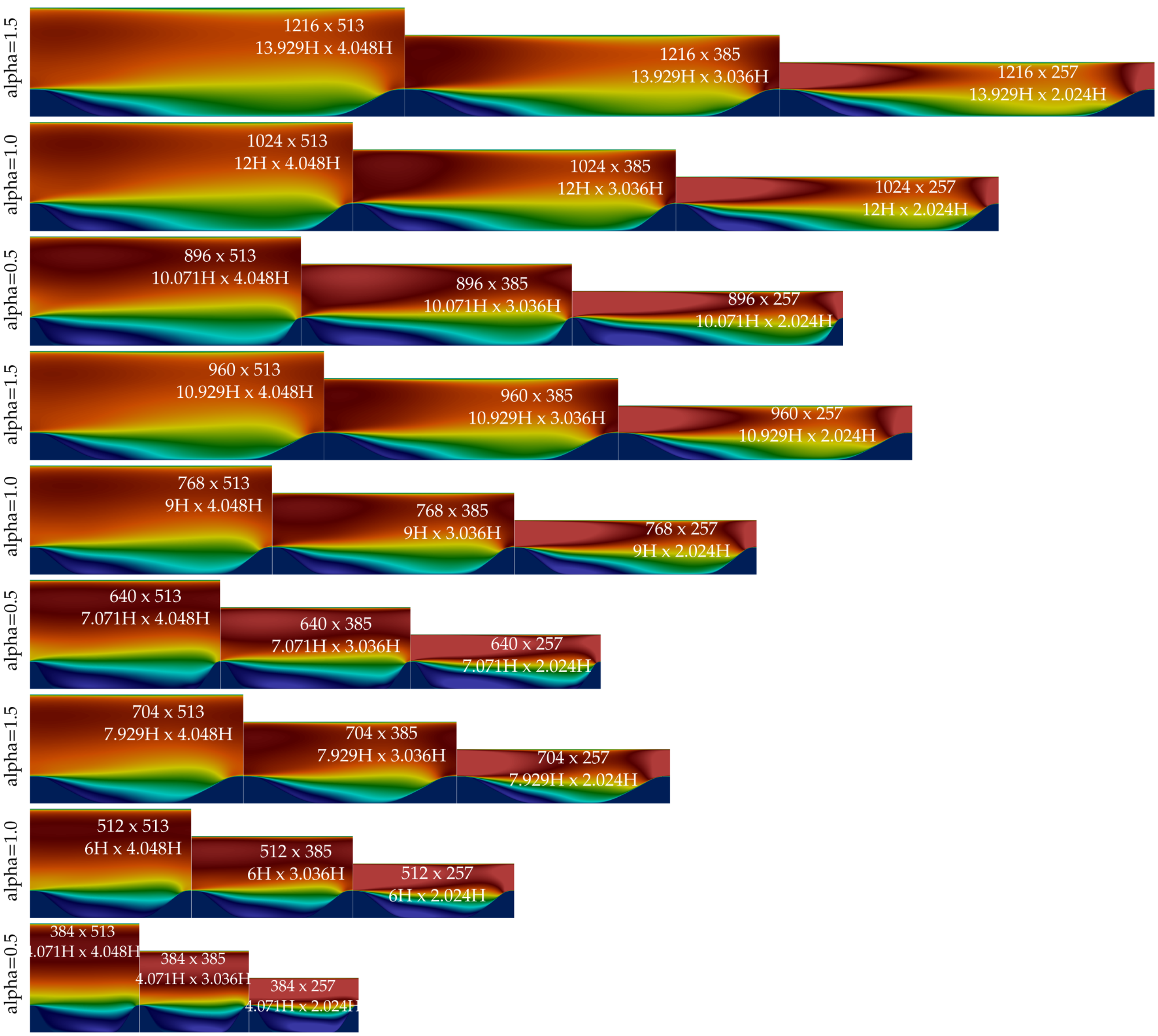
These equations are solved on a Cartesian mesh with the high-order flow solver `Incompact3d`, which is based on sixth-order compact schemes for spatial discretization and a third-order Adams-Bashforth scheme for time advancement. The code is publicly available on GitHub [1] and is the flagship solver of the `Xcompact3d` framework, a suite of high-order finite-difference flow solvers dedicated to the study of turbulent flows on supercomputers [2].

To treat the incompressibility condition, a fractional step method is used, which requires to solve a Poisson equation and it is solved fully in the spectral space. The divergence-free condition is ensured up to machine accuracy by utilizing the concept of modified wavenumber. The pressure mesh is staggered from the velocity mesh by half a mesh point to avoid spurious pressure oscillations. The modelling of the hill geometry is performed with a customized immersed boundary method based on a direct forcing approach that ensures a zero-velocity boundary condition at the wall. Following the strategy of [3], an artificial flow is reconstructed inside the 2D hill in order to avoid any discontinuities at the wall. More details about the present code and its validation can be found in [4]. The high computational cost of the present simulations requires the parallel version of this solver. The computational domain is split into a number of sub-domains (pencils) which are each assigned to an MPI-process. The derivatives and interpolations in the x-, $y-$, and z- directions are performed in X-, Y-, Z-pencils, respectively. The 3D FFTs required by the Poisson solver are also broken down as series of 1D FFTs computed in one direction at a time. The parallel version of `Incompact3d` can be used with up to one million computational cores [5].

Periodic boundary conditions are used in both the streamwise and spanwise directions while a non-slip condition is used both at the top and bottom wall of computational domain. The initial condition for the streamwise velocity field is given by

$$u(y) = 1 - \left(\frac{y}{H}\right)^2$$

while the initial conditions for the vertical and spanwise velocity field are equal to zero. The mass flow rate for all the simulations is kept constant in time via the imposition of a constant pressure gradient at each time step. The data are collected after a transitional period, from the point when the flow is fully developed. A simulation time step $\Delta t = 0.0005H/U_b$ is used. For all the calculations, turbulent statistical data have been collected over a time period $T = 750H/U_b$. Data are averaged in time and in the spanwise direction for which $L_z/H = 4.5$ is always discretised with $n_z = 192$. The resolution in the streamwise and vertical directions is provided in figure



References

[1] S. Laizet, Incompressible Navier-Stokes equations solver with multiple scalar transport equations, `https://github.com/xcompact3d/Incompact3d`.

[2] P. Bartholomew, G. Deskos, R.A. Frantz,F.N. Schuch, E. Lamballais, S. Laizet, Xcompact3d: An open-source framework for solving turbulence problems on a cartesian mesh 12 (2020), 100550.

[3] R. Gautier, S. Laizet, E. Lamballais, A DNS study of jet control with microjets using an immersed boundary method, Int. J. of Computational Fluid Dynamics 28 (6-10) (2014) 393–410.

[4] S. Laizet, E. Lamballais, High-order compact schemes for incompressible flows: A simple and efficient method with the quasi-spectral accuracy, J. Comp. Phys. 228 (2009) 5989–6015.

[5] S. Laizet, N. Li, Incompact3d: A powerfull tool to tackle turbulence problems with up to O(10^5) computational cores, Int. J. Num. Methods in Fluids 67 (2011) 1735–1757.