Seventh International Conference on Computational Fluid Dynamics (ICCFD7), Big Island, Hawaii, July 9-13, 2012

ICCFD7-1604

Simulating Flow over Periodic Hills Using a Dual-Mesh Hybrid Solver with High-Order LES

H. Xiao, Y. Sakai, R. Henniger, P. Jenny*
Corresponding author: jenny@ifd.mavt.ethz.ch

Institute of Fluid Dynamics, ETH Zürich, Switzerland

Abstract: In this work, we present a hybrid solver coupling a high-order LES solver with Cartesian mesh and a RANS solver with body-fitting mesh. This hybrid solver is developed within a dual-mesh consistent hybrid framework, where LES and RANS simulations for the same flow are conducted simultaneously on different computational domains and different meshes. In the LES, an immersed boundary method with relaxation forcing is used with the Cartesian mesh to enforce non-slip boundary conditions at the curved boundaries. The flow over periodic hills at Reynolds number $Re = 2800$ is simulated using the new solver. The adequacy of the boundary representation and forcing strategy is demonstrated. It is concluded that dual-mesh consistent hybrid framework is successful in the context of coupling a high-order LES solver on Cartesian mesh and a general-purpose RANS solver on body-fitting mesh. The approach explored in the new hybrid solver could be used to take advantage of the potential of many existing academic codes to simulate practical flows in industry and in nature, where complex geometries and wall boundaries currently represent major hurdles.

Keywords: Hybrid LES/RANS method, Turbulence Modeling, High-Order Method, Immersed Boundary Method, Parallel Computing

1 Introduction

Large Eddy Simulations (LES) for industrial flows are mostly conducted on unstructured grids using second-order numerical schemes. This is to take advantage of the existing infrastructures available in Reynolds Averaged Navier Stokes (RANS) equation solvers. However, in many cases it is desirable to perform LES with high-order numerical schemes with low dissipation and on massively parallel computers. In these scenarios, high accuracy and high efficiency are essential, and the general-purpose CFD tools developed for RANS simulations are not suitable. Consequently, in academic research large amounts of efforts have been invested in developing such high-order LES solvers with good scalability on parallel computers with tens of thousand of processors. A recently developed solver IMPACT is such an example [1, 2]. Compared to LES solvers with low-order numerical schemes on unstructured meshes, these solvers have very little numerical dissipation, which is critical for many detailed investigations of fluid dynamics such as noise generation in turbulence flows. Conceptually, they also offer the advantage of being able to distinguish the numerical approximation errors from the Sub-Grid Scale (SGS) modeling errors [3]. Another practically important advantage is the high efficiency of these solvers on parallel computers, largely due to the use of structured meshes which leads to lean memory consumption and fast memory access.

Despite numerous advantages of the high-order LES solvers on structured meshes mentioned above, a major drawback of these methods is the difficulty of dealing with flows with complex boundaries. The difficulty comes from two related but distinct issues: (1) the representation of immersed complex boundaries in LES with Cartesian mesh; and (2) the modeling of near-wall turbulence in LES. The first issue has been addressed by several authors using immersed-boundary methods [4]. To deal with the second difficulty, the commonly used approach is the hybrid LES/RANS methods, where RANS simulations are performed in the near-wall region while LES is performed in the free-shear region.

Recently, Xiao and Jenny [5] proposed a dual-mesh consistent hybrid framework, where LES and RANS are conducted simultaneously in the entire domain on separate meshes. In this model, relaxation forces are used to ensure the consistency between the two solutions in terms of velocity and turbulent quantities. As a proof-of-concept of the consistent hybrid framework, they developed a hybrid solver using the LES and RANS solvers based on the CFD platform OpenFOAM. Both solvers used body-fitting mesh covering computational domains identical to the physical flow domain.

The objective of this work is to present an extended version the original framework using a high-order LES solvers on Cartesian meshes. The LES solver developed by Henniger et al. [1] is chosen for this study, although other LES codes can also be used. The same RANS solver based on OpenFOAM and the same coupling schemes as in ref. [5] are used. A simple immersed-boundary method with relaxation forcing is used in the LES solver to represent the irregular domain.

The rest of the paper is organized as follows. Section summarizes the hybrid framework and the implementation of the new hybrid solver. Section 3 presents numerical simulations of the flow over periodic hills at Reynolds number $Re = 2800$. Based on these simulations, conclusions are drawn in Section 4.

2 Hybrid LES/RANS Framework and Its Implementation

2.1 Dual-Mesh Consistent Hybrid LES/RANS Framework

In incompressible flows with constant density, the momentum and pressure equations for the filtered quantities and the Reynolds-averaged quantities can be written in a unified form as follows [5]:

$$\frac{\partial U_i^*}{\partial t} + \frac{\partial \left(U_i^* U_j^*\right)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p^*}{\partial x_i} + \nu \frac{\partial^2 U_i^*}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}^*}{\partial x_j} + Q_i^* \tag{1a}$$

$$\text{and} \quad \frac{1}{\rho}\frac{\partial^2 p^*}{\partial x_i \partial x_i} = -\frac{\partial^2}{\partial x_i \partial x_j}\left(U_i^* U_j^* + \tau_{ij}^*\right) + \frac{\partial Q_i^*}{\partial x_i}, \tag{1b}$$

where t and x_i are time and space coordinates, respectively; ν is the kinematic viscosity, ρ is the constant fluid density, and p^* is the pressure. In filtered equations, U_i^*, p^*, and τ_{ij}^* represent filtered velocity $\overline{U}_i$, filtered pressure $\overline{p}$, and residual stresses τ_{ij}^{sgs}, respectively. In Reynolds-averaged equations, U_i^*, p^*, and τ_{ij}^* represent Reynolds-averaged velocity $\langle U_i \rangle$, Reynolds-averaged pressure $\langle p \rangle$, and Reynolds stresses τ_{ij}, respectively. The Reynolds stress τ_{ij} is defined as the correlations of velocity fluctuations $\langle u_i' u_j' \rangle$, although the apparent Reynolds stresses are actually $-\langle u_i' u_j' \rangle$ (see, e.g., Chapter 4 of ref. [6]). Q_i^* represents the drift forces applied in the filtered equations (Q_i^L) and in the Reynolds-averaged equations (Q_i^R) to ensure consistency between the two solutions. This term will be detailed in Equations (3) and (4).

In this hybrid framework, the filtered equations and the Reynolds-averaged equations are solved simultaneously in the entire domain but on separate meshes. This leads to some redundancy, and the consistency between the two solutions is enforced via relaxation forcing terms in the respective equations.

We first define Exponentially Weighted Average (EWA, or simply referred to as *average* hereafter) quantities including velocity, dissipation, and turbulent stress for the LES as

$$\langle \overline{U}_i \rangle^{\text{AVG}}(t) = \frac{1}{T}\int_{-\infty}^{t} \overline{U}_i(t') e^{-(t-t')/T} dt', \tag{2a}$$

$$\langle \tau_{ij} \rangle^{\text{AVG}}(t) = \frac{1}{T}\int_{-\infty}^{t} \left[u_i''(t') u_j''(t') + \tau_{ij}^{\text{sgs}}(t')\right] e^{-(t-t')/T} dt', \quad \text{and} \tag{2b}$$

$$\langle \varepsilon \rangle^{\text{AVG}}(t) = \frac{1}{T}\int_{-\infty}^{t} \left[2\nu \overline{S}_{ij}(t') \overline{S}_{ij}(t') - \tau_{ij}^{\text{sgs}}(t') \overline{S}_{ij}(t')\right] e^{-(t-t')/T} dt', \tag{2c}$$

respectively, where T is the averaging time; $u_i'' = \overline{U}_i - \langle \overline{U}_i \rangle^{\text{AVG}}$ is the fluctuating velocity with respect to the exponentially weighted average; $\overline{S}_{ij}$ is the filtered rate-of-strain tensor. The terms inside the integrals in Equations (2b) and (2c) are the total turbulent stress and the total dissipation rate in LES including the resolved and modeled parts.

The consistency between the two solutions requires that the exponentially weighted average quantities and the Reynolds-averaged quantities be approximately equal, e.g., $\langle \overline{U}_i \rangle^{\text{AVG}} \approx \langle U_i \rangle$ for the velocities and $\langle \tau_{ij} \rangle^{\text{AVG}} \approx \tau_{ij}$ for the turbulent stresses. The regions well-resolved by the LES mesh are classified as LES regions where the LES solution would dominate, and the under-resolved regions are called RANS regions where the RANS solution should prevail. The consistency and the selective dominance mechanism are enforced via drift forces (Q_i^L in filtered equations and Q_i^R in Reynolds-averaged equations) defined as follows:

$$Q_i^L = \begin{cases} (\langle U_i \rangle - \langle \overline{U}_i \rangle^{\text{AVG}})/T^{(\text{L})} + G_{ij} u_j''/T^{(\text{G})} & \text{in RANS regions} \\ 0 & \text{in LES regions,} \end{cases} \tag{3}$$

and

$$Q_i^R = \begin{cases} 0 & \text{in RANS regions} \\ (\langle \overline{U}_i \rangle^{\text{AVG}} - \langle U_i \rangle)/T^{(\text{R})} & \text{in LES regions,} \end{cases} \tag{4}$$

where

$$G_{ij} = \frac{\tau_{ij} - \langle \tau_{ij} \rangle^{\text{AVG}}}{\langle \tau_{kk} \rangle^{\text{AVG}}}, \tag{5}$$

and $T^{(\text{L})}$, $T^{(\text{G})}$, $T^{(\text{R})}$ are the relaxation time scales. Similarly, to ensure consistency on the turbulent quantities in RANS simulation, in the well-resolved (LES) regions they are relaxed towards the corresponding LES quantities via the following drift terms. Detailed solution algorithms and the choice of the parameters are presented in ref. [5].

2.2 Implementation

The hybrid framework is implemented for incompressible flows based on the open source CFD platform OpenFOAM [7, 8] and a high-order LES/DNS solver IMPACT (Incompressible Turbulent flows with Compact differentiation on Massively PArallel Computers) recently developed by Henniger et al. [1]. In the current study, we use a combination of an open-source general purpose CFD solver and an in-house academic code to demonstrates the flexibility of the hybrid framework, as well as its potential to unleash the power of many existing high-order high-efficiency academic codes.

The LES solver IMPACT is a high-order high-efficiency solver for incompressible turbulent flows developed for massively parallel computers. Structured grids are used to allow for fast memory access. Spatial derivatives are discretized on staggered grids with compact finite differences of arbitrary convergence order. A sixth-order finite difference scheme is used for all the spatial derivative terms in this study. Temporal derivatives are discretized explicitly using a third-order Runge-Kutta scheme with efficient storage. Parallelization is achieved via three-dimensional domain decomposition in space. More details can be found in refs. [1] and [2]. The ADM-RT (Approximate Deconvolution Model with Relaxation Term) is used for the SGS turbulence modeling [9, 10].

In RANS simulations, the continuity and momentum equations for incompressible turbulent flows are solved using the PISO (Pressure Implicit with Splitting of Operators) algorithm on unstructured meshes [11]. Collocated grids are used with the Rhie and Chow interpolation being employed to prevent the pressure–velocity decoupling [12]. Spatial derivatives are discretized with the finite volume method using the second-order central scheme for both convection and diffusion terms. A second-order implicit time-integration scheme is used to discretize the temporal derivatives. For the turbulent modeling, the k–ε two-equation model of Launder–Sharma [13] is used as closure for the RANS equations.

The dual-mesh hybrid framework introduced above assumes that the LES and the RANS simulations solve the same flow on identical computational domains. To use an LES solver with Cartesian mesh in the framework, the irregular domains need to be taken into account. In this study, a simple immersed boundary method is employed to account for the solid boundaries. At the beginning of each simulation, the locations of the points in the Cartesian mesh are examined. The points falling outside the fluid domain are identified and flagged. During the simulations, a damping force is applied on momentum equations corresponding to all the points outside the fluid domain. The forcing Q_i^{IB} has an opposite direction as the velocity and has a magnitude proportional to the velocity, i.e.,

$$Q_i^{\mathrm{IB}} = -\alpha U_i. \tag{6}$$

where α is a positive constant with the dimension of frequency. Consequently, the forcing effectively drives the velocities on these points to zero, imposing correct non-slip boundary conditions at walls.

3 Numerical Simulations

To demonstrate the capability of the developed hybrid solver, the flow over periodic hills is simulated. This configuration features a massive separation and recirculations. Extensive experiments and well-resolved DNS and LES have been conducted for this configuration at various Reynolds numbers to provide benchmark data [14, 15]. We choose the flow at Reynolds number $Re = 2800$.

The LES and RANS meshes used for this case are shown in Figure 1. Spacial and time resolution information can be found in Table 1. The same set of algorithmic parameters as in ref. [5] are used: $T = 2.2H/U_b$, $\tau_l = 0.28H/U_b$, $\tau_r = 0.28H/U_b$, and $\tau_g = 0.07H/U_b$. For simplicity, the RANS regions are pre-specified as consisting of all cells with distance smaller than $D = 0.2H$ from the nearest wall. To ensure numerical stability, a linear ramp function ranging from 0 to 1 is multiplied on all drift terms during an initial simulation period of $2T$.

As in ref. [5] for the original solver hybridLRFoam, the internal consistency of the hybrid solver impactFoam. The mean streamwise velocities obtained from the LES and from the RANS simulation in the hybrid solver are compared in Figure 2(a). The results show very good consistency between the velocities in the LES and those in the RANS simulations. The data points outside the physical fluid domain (i.e., in the solid region) are kept in the plot as indications of the quality of enforcing immersed boundary conditions. The mean velocity profiles from pure LES and pure RANS simulations are shown in Figure 2(b) as reference. It can be seen that the LES solver and the RANS solver give very different velocity profiles in standalone simulations.

The streamwise mean velocities at nine cross sections of the channel are presented in Figure 3(a). The mean velocities are obtained by performing time averaging and spanwise averaging of the LES velocities. However, due to the internal consistency between the mean LES velocity and the velocities in the RANS simulation, the distinction between mean LES velocities and mean RANS velocities are not important. The overall good predictions of velocity profiles in this simulation suggest that both for the pure LES and for the hybrid solver the boundary representation and forcing do not pose a major difficulty. It can also be seen that the LES mesh resolution is rather good for this Reynolds number. The pure LES results show some deviations from the benchmark simulation results particularly in the free-shear region (from $y/H = 1.5$ to 2.5) and the reattached flow regions (from $x/H = 4$ to 7). Since the hybrid solver has much better predictions in these regions, and the hybrid solver differs from the pure LES only in near-wall modeling, it can be inferred that the deviations in the pure LES results are mostly likely caused by the inadequate near-wall resolution, and not due to the immersed boundary representation and forcing.

4 Conclusion

In this paper, we present a hybrid solver coupling a high-order LES solver with Cartesian mesh with a RANS solver with body-fitting mesh. With this solver, numerical simulations of the flow over periodic hill have been performed. The simulation results demonstrate the adequacy of the boundary representation and forcing strategy, as well as the accuracy of the pure LES solver. More importantly, the simulations also suggest that the consistent hybrid framework is successful in the new hybrid solver with high-order LES on Cartesian mesh and RANS solver on body-fitting mesh.

The approach explored in the new hybrid solver could be used to take advantage of the potential of many existing academic codes to simulate practical flows in industry and in nature, where complex geometries and wall boundaries currently represent major hurdles.

References

[1] R. Henniger, D. Obrist, and L. Kleiser. High-order accurate solution of the incompressible Navier–Stokes equations on massively parallel computers. *Journal of Computational Physics*, 229(10):3543–3572, 2010.

[2] R. Henninger. IMPACT simulation code. Retrieved July 2012.

[3] R. Pasquetti and C.J. Xu. High-order algorithms for large-eddy simulation of incompressible flows. *Journal of Scientific Computing*, 17(1):273–284, 2002.

[4] A. Posa, A. Lippolis, R. Verzicco, and E. Balaras. Large-eddy simulations in mixed-flow pumps using an immersed-boundary method. *Computers and Fluids*, 47:33–43, 2011.

[5] H. Xiao and P. Jenny. A consistent dual-mesh framework for hybrid LES/RANS modeling. *Journal of Computational Physics*, 231(4):1848–1865, 2012.

[6] S. B. Pope. *Turbulent Flows*. Cambridge University Press, Cambridge, 2000.

[7] OpenCFD Ltd. The open source CFD toolbox, 2011.

[8] H. G. Weller, G. Tabor, H. Jasak, and C. Fureby. A tensorial approach to computational continuum mechanics using object-oriented techniques. *Computers in Physics*, 12(6):620–631, 1998.

[9] S Stolz and N.A. Adams. An approximate deconvolution procedure for large-eddy simulation. *Physics of Fluids*, 11(7):1699–1701, 1999.

[10] S. Stolz, N.A. Adams, and L Kleiser. An approximate deconvolution model for large-eddy simulation with application to incompressible wall-bounded flows. *Physics of Fluids*, 2001.

[11] R. I. Issa. Solution of the implicitly discretised fluid flow equations by operator-splitting. *Journal of Computational Physics*, 62:40–65, 1986.

[12] C. M. Rhie and W. L. Chow. A numerical study of the turbulent flow past an isolated airfoil with trailing edge separation. *AIAA*, 21(11):1525–1532, 1983.

[13] B. E. Launder and B. I. Sharma. Application of the energy dissipation model of turbulence to the calculation of flow near a spinning disc. *Letter of Heat Mass Transfer*, 1:131–138, 1974.

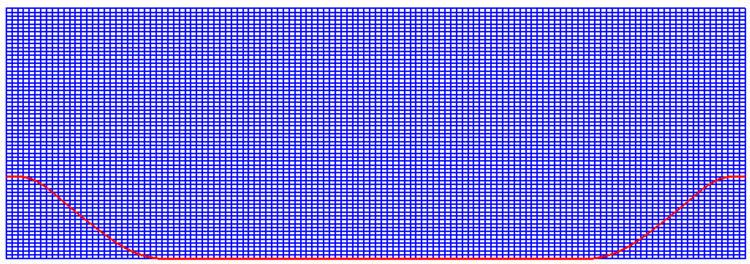
[14] M. Breuer, N. Peller, Ch. Rapp, and M. Manhart. Flow over periodic hills: Numerical and experimental study in a wide range of Reynolds numbers. *Computers & Fluids*, 38(2):433–457, 2009.

[15] S. Saric, S. Jakirlic, M. Breuer, B. Jaffrezic, G. Deng, O. Chikhaoui, J. Fröhlich, D. von Terzi, M. Manhart, and N. Peller. Evaluation of detached eddy simulations for predicting the flow over periodic hills. In *ESAIM: Proceedings*, volume 16, pages 133–145, 2007. doi: 10.1051/proc:2007016.

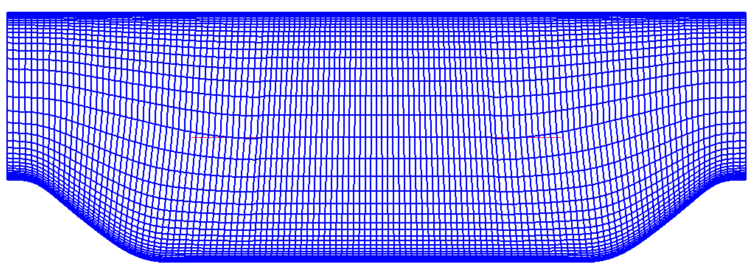
Table 1: Domain and mesh parameters for the flows over periodic hill at $Re = 2800$. x, y, z are aligned with streamwise, wall-normal, and spanwise directions, respectively.

Domain size ($L_x \times L_y \times L_z$)	$9H \times 3.036H \times 4.5H$
Simulation time-span[b]	$50\, T_{thr}$
$N_x \times N_y \times N_z$ (LES)	$128 \times 64 \times 32$
$N_x \times N_y \times N_z$ (RANS)	$128 \times 37 \times 16$
$\Delta x \times \Delta y \times \Delta z$ in y^+ (LES)	$13 \times 9 \times 27$
First grid point (RANS)[c]	Below $0.7y^+$
Time-step size	$2.8 \times 10^{-3}\ H/U_b$

[a] Through-time T_{thr} is defined as L_x/U_b.
[b] The wall unit is defined as $y^+ = \nu/u_\tau = \nu/\sqrt{\tau_w/\rho}$.

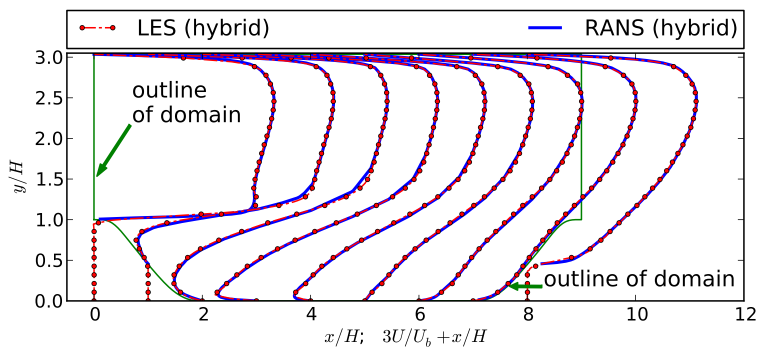


(a) LES mesh with immersed boundary

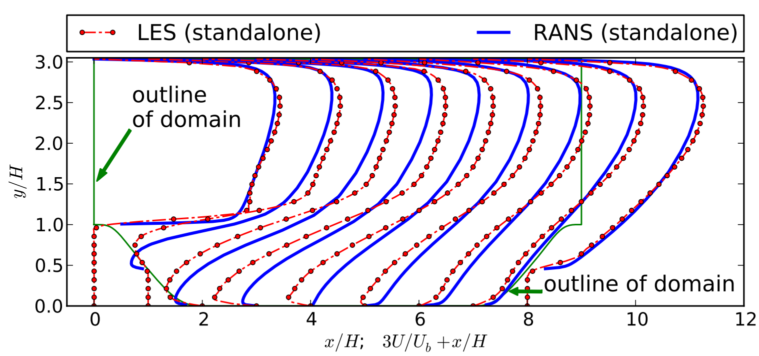


(b) RANS mesh

Figure 1: Illustration of (a) the Cartesian LES mesh for finite differencing and (b) the RANS body-fitting mesh for finite-volume discretization, using the periodic hill case as example. The LES mesh has uniform spacing with the immersed boundary indicated. The RANS mesh is refined in the wall-normal direction near the wall. The free-shear region is very coarse.

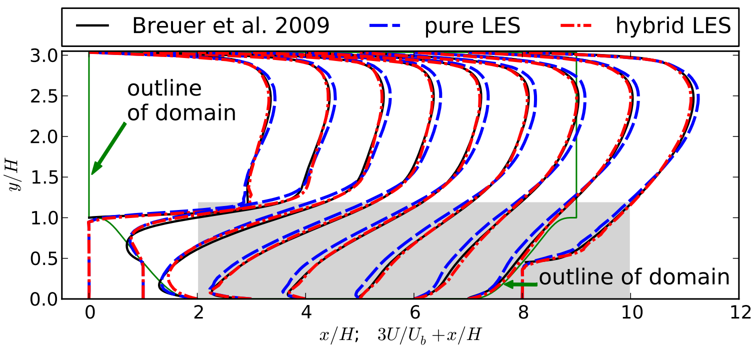


(a)

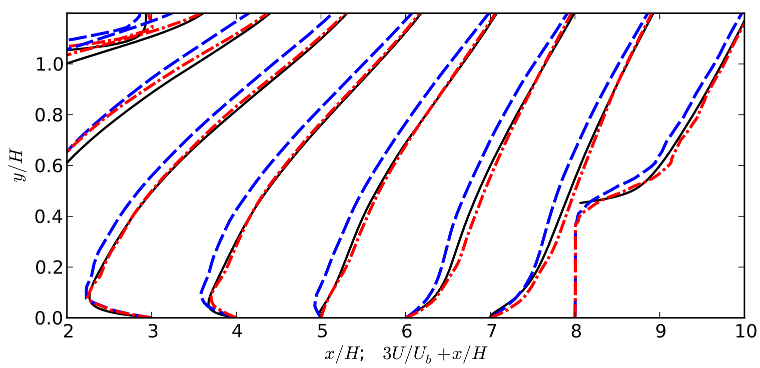


(b)

Figure 2: Internal consistency between the LES and the RANS: mean streamwise velocity from the LES and that from the RANS are compared for the flow over periodic hills at $Re = 2800$. (a) LES and RANS velocities from the hybrid solver with coupling; (b) LES and RANS velocities from standalone LES and RANS solvers without coupling. The velocities are shown in the solid zone for both cases. For the LES results, the lines pass through all data points, but markers are only shown for every seventh points for clarity.



(a)



(b)

Figure 3: Mean velocities in the streamwise direction in the flow over periodic hills at $Re = 2800$, with comparison among hybrid simulation, pure LES simulation, and the benchmark simulation of Breuer et al. [14]. The reattached region (shaded) in plot (a) is zoomed-in and shown in plot (b).