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 coupling a high-order LES solver on Cartesian mesh and a RANS solver on body-fitting mesh.

Keywords: Hybrid LES/RANS method, Turbulence Modeling, Immersed Boundary Method

1 Introduction

Recently, Xiao and Jenny [4] 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. 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 hybrid framework is implemented for incompressible flows based on the open source CFD platform OpenFOAM [3] and a high-order LES/DNS solver IMPACT (Incompressible Turbulent flows with Compact differentiation on Massively PArallel Computers) recently developed by Henniger et al. [2]. A simple immersed-boundary method with relaxation forcing is used in the LES solver to represent the irregular domain.

2 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 [1]. The streamwise mean velocities at nine cross sections of the channel are presented in Figure 1. 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. 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.

3 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 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.

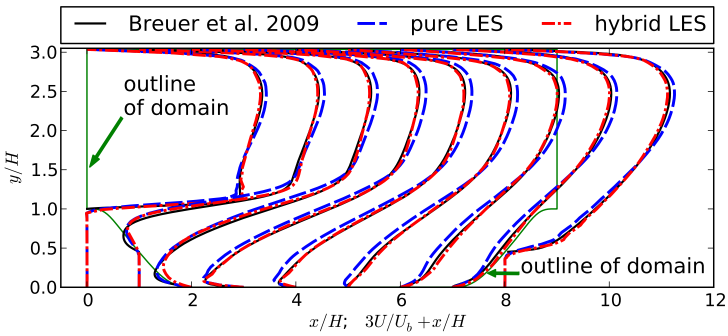

Figure 1: Mean velocities in the streamwise direction in the flow over periodic hills at $Re = 2800$, with comparison among hybrid simulation, pure LES simulation, and the benchmark simulation of Breuer et al. [1].

References

[1] 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.

[2] 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.

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

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