Theoretical & Applied Mechanics Letters 6 (2016) 302–305

Contents lists available at ScienceDirect

Theoretical & Applied Mechanics Letters

journal homepage: www.elsevier.com/locate/taml

Letter

A large eddy simulation of flows around an underwater vehicle model using an immersed boundary method

Shizhao Wang [a], Beiji Shi [a,b], Yuhang Li [a,b], Guowei He [a,b,*]

[a] The State Key Laboratory of Nonlinear Mechanics, Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190, China
[b] School of Engineering Sciences, University of Chinese Academy of Sciences, Beijing 100049, China

HIGHLIGHTS

- The velocity self-similarity of wake is predicted by using large-eddy simulation.
- Diffuse interface immersed boundary method is coupled with large eddy simulation.
- The flow solver with IB method shows nearly linear parallel scalabilities.

ARTICLE INFO

Article history:
Received 2 November 2016
Accepted 7 November 2016
Available online 22 November 2016
*This article belongs to the Fluid Mechanics

Keywords:
Underwater vehicle
SUBOFF
Immersed boundary method
Large eddy simulation
Adaptive mesh refinement

ABSTRACT

A large eddy simulation (LES) of the flows around an underwater vehicle model at intermediate Reynolds numbers is performed. The underwater vehicle model is taken as the DARPA SUBOFF with full appendages, where the Reynolds number based on the hull length is 1.0×10^5. An immersed boundary method based on the moving-least-squares reconstruction is used to handle the complex geometric boundaries. The adaptive mesh refinement is utilized to resolve the flows near the hull. The parallel scalabilities of the flow solver are tested on meshes with the number of cells varying from 50 million to 3.2 billion. The parallel solver reaches nearly linear scalability for the flows around the underwater vehicle model. The present simulation captures the essential features of the vortex structures near the hull and in the wake. Both of the time-averaged pressure coefficients and streamwise velocity profiles obtained from the LES are consistent with the characteristics of the flows pass an appended axisymmetric body. The code efficiency and its correct predictions on flow features allow us to perform the full-scale simulations on tens of thousands of cores with billions of grid points for higher-Reynolds-number flows around the underwater vehicles.

The modern underwater vehicles have untraditional appendages to achieve high maneuverability at intermediate to high Reynolds numbers [1,2]. This raises two challenges for a full-scale simulation of the flows around the underwater vehicles: the first one is to handle the complex geometric and moving boundaries; the second one is to calculate the characteristics of viscous flows near the boundaries and in the wake [3,4]. Recently, the immersed boundary (IB) method in combination with large eddy simulation has been developed to simulate turbulent flows with complex geometric and moving boundaries [5–7]. The IB method is a non-body conformal method and circumvents the generation of body-fitting grids, where an artificial force is added to the Navier–Stokes equations to represent the boundary effect on flows, This method has been widely used in cardiovascular flows, bio-locomotion, and wind-turbines [8–10] with great successes.

Recently, Posa and Balaras [11] have used the hybrid immersed boundary method and large eddy simulation to simulate the wake of an axisymmetric body with appendages. They choose a sharp interface IB method to simulate the turbulent wakes. The sharp interface IB method treats the boundaries on the Eulerian meshes by using complex local flow field reconstructions or the cut cell techniques, which are usually time consuming for a body with complex geometry. Instead of reconstructing the cell near boundaries, the diffuse interface IB method spreads the effects of solid boundaries onto a band of cells near boundaries. This method ensures the efficiency and robustness of the implementation. The diffuse interface IB method has been successfully utilized in laminar flows, but the grid resolution near the wall often limits its application to turbulent flows. The diffuse interface IB

* Corresponding author.
E-mail address: hgw@lnm.imech.ac.cn (G. He).

http://dx.doi.org/10.1016/j.taml.2016.11.004

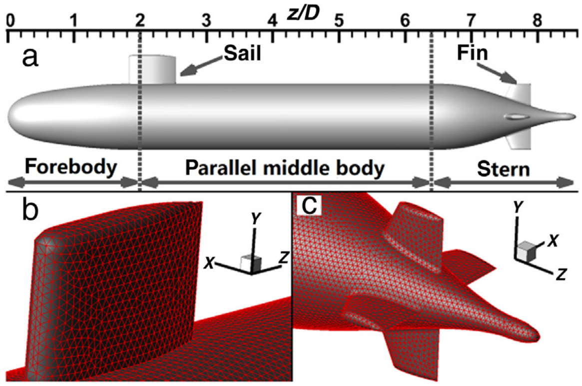

Fig. 1. DARPA SUBOFF with full appendages (a) and the Lagrangian mesh near the sail (b) and fins (c).

method cannot refine the grid only along the wall normal direction, since it is a non-body conformal method. The adaptive mesh refinement is an efficient way to locally refine the mesh, and can be utilized to reduce the number of mesh cells in the diffusive IB method. Furthermore, the diffusive IB method needs to be combined with the large eddy simulation to avoid resolving all flow structures in turbulence. However, the combinations of the diffuse interface IB method, adaptive mesh refinement, and large eddy simulation might not guarantee their accuracy and efficiency, since they have different theoretical bases and numerical implement techniques. The simulations of turbulent flows with complex geometric boundaries are required to investigate the validation and efficiency of the combinations of the diffuse interface IB method, adaptive mesh refinement, and the large eddy simulation.

The objective of the present work is to investigate the validation and efficiency of the hybrid diffuse interface IB method, adaptive mesh refinement and large eddy simulation for turbulent flows with complex geometric boundaries. The advantages and disadvantages of the method will also be reported. The simulated model is taken as the flows around an underwater vehicle. We will use the moving-least-squares reconstruction on a block structured mesh with the adaptive mesh refinement technique. We will first introduce the underwater vehicle model and the numerical method that will be used. The efficiency of our code will be discussed and numerical results will be presented. Finally, we will summarize the results and future work.

In the present work, the DARPA SUBOFF is used as the underwater vehicle model. The model consists of an axisymmetric hull, a sail and four fins, as shown in Fig. 1. The axisymmetric hull is composed of a bow forebody, a parallel middle body section, and a curved stern. The hull has a maximum diameter D and a length $L/D = 8.6$. The details of the used model can be found in the Ref. [12]. The appendages raise the challenges in both handling with the complex geometric boundaries and capturing the flow features (such as boundary layer, junction flows, tip flows, and their interactions), which provide a sufficient complex model for investigating the capability of the diffuse interface IB method in combination of large eddy simulation and the adaptive mesh refinement.

The present work focuses on deep-submergence underwater vehicle, where the effects of free surface on the flows near the model are ignored. The flows around the model are governed by the Navier–Stokes equations for single phase incompressible flows. The governing equations for large eddy simulation are given by

$$\frac{\partial \tilde{u}_i}{\partial x_i} = 0, \tag{1}$$

$$\frac{\partial \tilde{u}_i}{\partial t} + \frac{\partial \tilde{u}_i \tilde{u}_j}{\partial x_j} = -\frac{\partial \tilde{p}}{\partial x_i} - \frac{\partial \tilde{\tau}_{ij}}{\partial x_j} + \frac{1}{Re}\frac{\partial^2 \tilde{u}_i}{\partial x_j \partial x_j} + f_i, \tag{2}$$

Table 1
Strong scalability of the flow solver on a mesh of about 50 million cells. The notations "N_{core}" and "N_{cell}" denote the number of cores and the number of cells, respectively. "T_{step}" denotes the wall-clock time cost per step.

N_{core} (million)	N_{cell} (million)	$N_{\text{cell}}/N_{\text{core}}$	T_{step}(s)
96	50	0.52	6.1
192	50	0.26	3.0
384	50	0.13	1.6

Table 2
Weak scalability of the flow solver with a mesh of about 0.26 million cells per core. The notations "N_{core}" and "N_{cell}" denote the number of cores and the number of cells, respectively. "T_{step}" denotes the wall-clock time cost per step.

N_{core} (million)	N_{cell} (million)	$N_{\text{cell}}/N_{\text{core}}$	T_{step}(s)
192	50	0.26	3.0
1536	403	0.26	3.2
12288	3.2	0.26	3.7

where $\tilde{u}_i$ $(i = 1, 2, 3)$ and $\tilde{p}$ are the filtered velocity components and pressure, respectively. The sub-grid stresses $\tilde{\tau}_{ij}$ is represented by the wall-adapting local eddy-viscosity model with $C_w = 0.6$ [13]. f_i $(i = 1, 2, 3)$ are the volume forces that represent the effects of boundaries on the flows in the IB method. Re is the Reynolds number.

Equations (1) and (2) are discretized on a Cartesian Eulerian mesh and solved by using a projection method. The second-order central difference is used for the spatial derivatives, and the second-order Adams–Bashforth method is used for the time advance. Figure 1 presents the Lagrangian mesh near the sail and fins on the SUBOFF. A diffuse interface IB method based on the moving-least-squares reconstruction is used to represent the effects of the model surface on flows. [14,15]. The computational domain is $[-4.3D,\ 4.3D] \times [-4.3D,\ 4.3D] \times [-2.6D,\ 23.2D]$. The uniform upstream flow boundary condition is used at the inlet, and convective outflow boundary condition is used at the outlet. The non-slip boundary conditions are used on the immersed surfaces. The slip boundary conditions are used at the outer boundaries. A trip wire is located at the $0.25D$ downstream of the model nose. The Reynolds number based on the upstream flow velocity and the length of the model is $Re_L = U_\infty L/\nu = 1.0 \times 10^5$, corresponding to a Reynolds number based on the maximum diameter of $Re_D = U_\infty D/\nu \approx 1.16 \times 10^4$. Here U_∞ is the uniform free stream flow velocity and ν is the kinematic viscosity of the fluid.

In the present simulation, we utilize the block-structured mesh with adaptive mesh refinement. The parallel scalability of the flow solver is tested on meshes with different levels of refinement. Table 1 gives the wall-clock time cost of the flow solver on a mesh of about 50 million cells, which decreases as increasing the number of cores; Table 2 gives the wall-clock time cost of the flow solver on a mesh of about 0.26 million cells per core, which keeps nearly constant as increasing the number of cores. They show the strong and weak scalabilities of the parallel solver, respectively. In this letter, we report the preliminary results on the mesh of 50 million cells with a minimum grid length of $dh = 0.0336$. The minimum grid length is about 300 wall units, where the wall unit is estimated based on the turbulent boundary layer over a flat plate. The grid independence is checked to guarantee the sufficient resolution for the time-averaged pressure coefficient on the hull and the streamwise velocity profiles in the wake. It is worth to mention that the grid resolution is not fine enough to directly calculate the wall shear stress. A wall model is usually utilized to correctly obtain the wall shear stress in the LES with such a near-wall grid resolution. We calculate the time-averaged pressure coefficient on the hull and the streamwise velocity profiles in the wake in the present letter. The simulations with wall models and the distribution of wall shear stress will be carried on in future.