

Numerical Study of Flow Phenomena Concerning Interaction of Flow and Complex or Moving Boundaries Using Immersed Boundary Method

张, 晏榕

<https://hdl.handle.net/2324/1959133>

出版情報：九州大学, 2018, 博士（工学）, 課程博士
バージョン：
権利関係：

氏 名 : 張 晏榕 (チョウ アンヨン)

論 文 名 : Numerical Study of Flow Phenomena Concerning Interaction of Flow and Complex or Moving Boundaries Using Immersed Boundary Method
(埋込み境界法を用いた流れと複雑・移動境界の干渉に関する流体现象の数値解析的研究)

区 分 : 甲

論 文 内 容 の 要 旨

Research Background and Objectives

Interaction of flow and complicated or moving boundaries are common in normal life as well as various engineering applications, e.g., free surface of the sea, flapping wings of a bird, a beating tail of a fish or a rotating propeller of an aircraft. Owing to the remarkable development of the computer technique, the computational fluid dynamics (CFD) has now become an important strategy to study various flow phenomena as well as the experimental study. To perform the computational simulation for the problems concerning complicated or moving boundaries, dealing with the boundaries is a crucial point. The immersed boundary method that belongs to the category of the Eulerian approach is a promising one to deal with such boundary problems in CFD, in which the effect of the boundaries is imposed by introducing an external force into the fluid. Remarkable development has been made since it was proposed and a number of applications have been performed. Despite of the obtained achievement, however, applications of the immersed boundary method are found usually confined to problems at relatively low Reynolds numbers or on relatively simple movement, while applications to turbulent flows or those including moving boundaries with relatively complicated configuration or self-induced movement are considered to be limited.

In this study, an analysis method introducing the immersed boundary method is developed, where the force feedback mechanism is also considered to simulate the movement of the boundaries controlled by the hydrodynamic force. The present method is applied to several problems concerning rough or moving boundaries. First, the performance of the immersed boundary including the force feedback mechanism is validated and then three-dimensional flow phenomena in the interaction of the fluid and boundaries are investigated, the present test cases include:

1. Validation case for static immersed boundary: flow around a circular cylinder.
2. Validation case for moving boundary involving the force feedback: a single settling particle in fluid.
3. Application of the immersed boundary method to the study on the effect of an anisotropy-resolving subgrid-scale (SGS) model on large eddy simulation (LES) of turbulent channel flow with wall roughness.
4. Three-dimensional simulation of self-propelled fish-like body swimming in a channel where the fish model and its movement are defined in a structured grid system using the immersed boundary method.

Computational Methods

In the present study, the mass and momentum conservation equations for an incompressible flow are solved. To deal with the immersed boundaries, the direct forcing approach is adopted.

The present simulations are performed using the open-source CFD code “FrontFlow/Red” that was originally developed in the project of “Revolutionary Simulation Software” started in 2005 as a part of the

next-generation IT program sponsored by the Ministry of Education, Culture, Sports, Science and Technology. The immersed boundary method and force feedback mechanism are newly introduced in this program code as user subroutines, where explicit schemes are adopted to implement the direct forcing approach and force feedback mechanism. For the case of turbulent flow with wall roughness, an anisotropy-resolving SGS model that has been recently developed is adopted.

Results and Discussion

For the validation case of flow around a circular cylinder, the obtained drag and lift coefficients as well as the Strouhal number were compared with those from previous studies, and good agreement was yielded. Three-dimensional vortex structures were also captured well for the case of $Re=1000$, showing the feasibility of the present program code using the direct forcing approach.

For the validation case of a single settling particle in fluid, the comparison of the limit velocity showed good agreement with previous results. The dependency of the time step and grid resolution for the use of the third-order upwind scheme and the first-order time marching method, respectively, was carefully examined and their effects on the computational results were found to be insignificant.

For the LES of turbulent channel flow with wall roughness, by comparing the results with those from other smooth and rough DNSs, the present simulation provided reasonable trends that the presence of roughness caused a downward shift of the mean-velocity profile without any obvious change in slope in the logarithmic law region. In addition, it was found from the turbulent stress profiles that there is a shift away from the wall as well as a lower peak value for the streamwise turbulent stress compared with a smooth-wall case. An important conclusion from the present results is noted that the grid dependency was largely reduced for a coarse grid-resolution case by using the anisotropy-resolving SGS model. The present anisotropic SGS model successfully provided the predictions of the mean velocity and the Reynolds shear stress better than a canonical isotropic eddy-viscosity type of SGS model does.

For the simulation of three-dimensional self-propelled fish swimming, it was confirmed that the obtained swimming velocity increased almost linearly with the tail-beat frequency. Since this relationship showed no conflict with those indicated in some previous studies, the present results are considered to be reliable and then the obtained flow fields were investigated in more detail. Wake vortices were shed from the tail and formed a Karman vortex street, which played an important role in thrust generation. The results obtained for various phase differences among neighboring rigid body parts showed that the swimming was considerably more efficient with a small phase difference than with a large one. There existed an optimal body amplitude to obtain a maximum average longitudinal velocity when the other two parameters (the tail-beat frequency and the phase difference) were fixed. The discrepancy between the changing tendencies of the oscillation and translational movement in the longitudinal direction as the amplitude increases implied a decrease of the swimming efficiency. The fluctuation in the vertical velocity implies an importance of such as pectoral fins for the swimming stability.

Conclusions

In the present study, the immersed boundary method was applied to problems concerning relatively complicated or moving boundaries for investigating the flow phenomena involved. The performance of the program code introducing the immersed boundary method that includes a force feedback mechanism was validated firstly through the simulations of flow around a circular cylinder and a single settling particle in fluid. Further application cases of turbulent channel flow with wall roughness and three-dimensional self-propelled fish swimming were then carried out. The obtained results verified the feasibility and effectiveness of the application of the immersed boundary method to the concerned problems. Furthermore, several useful insights about the flow phenomena involved were obtained.