High fidelity simulations of moving objects in a turbulent flow using a Cartesian mesh

ARCHER2-eCSE01-6 Technical Report

Dr Sylvain Laizet, Dr Arash Hamzehloo, Dr Athanasios Giannenas (Imperial College London)

Abstract:
In recent years Immerse Boundary Methods (IBMs) have been gaining popularity within the scientific community in computational fluid mechanics for simulation of complex objects immersed in a turbulent flow. They eliminate issues relating to mesh quality and re-meshing when simulating moving objects. These issues can potentially introduce a significant computational overhead when associated with body-conforming methods. All IBMs rely on the addition of an extra forcing term in the governing equations to impose the appropriate Dirichlet or Neumann boundary conditions on the immersed solid surfaces. Most IBMs available in the literature rely on complicated interpolation or extrapolation schemes for the calculation of the forcing term that could potentially increase the computational overhead in the context of high-performance computing. Here, a novel and simple Alternating Direction Forcing Immersed Boundary Method (ADR-IBM) is presented, implemented, and validated in the high-order finite-difference framework Xcompact3d. It is a framework dedicated to the study of turbulent flows using CPU-based supercomputers. The proposed IBM allows a sharp representation of the immersed boundary interface and ensures the imposition of Dirichlet boundary conditions. A Fluid Structure Interaction module is available, for problems where one or more complex solid structures interact and modify the behaviour of the surrounding fluid. FSIs are commonly encountered both in nature and in many engineering fields such as aerospace, engine blades, energy (fixed and floating wind turbines, Wave Energy Converters (WECs)) and biomedical (heart valves among others).

The IBM developed in this project employs a series of one-dimensional cubic spline reconstructions to interpolate an artificial flow field inside the solid that satisfies the Dirichlet boundary condition at the desired location with second order accuracy. Additionally, the reconstruction ensures that the interpolated function remains continuous everywhere, which in turn reduces the Gibbs oscillations significantly. The aim of this 1D approach is to be compatible with the 2D domain decomposition strategy implemented in Xcompact3d. The computational domain is divided in pencils, to allow for implicit quasi-spectral derivations and interpolations in one direction at a time. Naturally, the consecutive reconstructions are performed before each derivative calculation in the appropriate direction. Hence, the velocity function is reconstructed in the x-direction before the calculation of the derivative in x-pencils and similarly for the y- and z-directions. Xcompact3d can scale with up to one million CPU cores [1,2].

The governing equations are the forced incompressible Navier-Stokes equations, with an extra forcing term (related to the IBM) only acting in the vicinity of the immersed boundaries to impose the required boundary conditions for the velocity field. A fractional

step method is employed for the time integration of the momentum equation while sixth-order finite-difference compact schemes are utilised for the spatial discretization. The Poisson equation, which is required to satisfy the incompressibility condition, is solved in spectral space to avoid expensive iterative solvers. More details about Xcompact3d can be found in [1,2,3]. Thanks to a previous eCSE project, it is now possible to simulate two-phase flows in Xcompact3d, for instance air and water. The stability of the two-phase flow solver has been improved during this project to allow for more complex simulations involving a moving object immersed at the interface of two fluids [4].

The ADR-IBM has been designed to simulate fluid flow problems with arbitrarily complex 2D and 3D immersed fixed and moving objects. The representation of the boundary surface should be sufficiently flexible in order to avoid limiting the type of geometries that can be simulated. Xcompact3d offers two options depending on the geometrical complexity of the considered object(s). Objects with simple geometries (cylinder, NACA profile, Ahmed body) can be directly immersed in the computational domain using analytical equations. If the objects cannot be represented with equations (analytically describing the objects), a Computer-Aided Design (CAD) interface has been designed to make it possible to import objects generated by CAD software with the Stereolithography stl format. This format is a widely used file type in the CAD community. The surface of the object is described by breaking it into a collection of triangles and listing the location of the triangle vertices as well as which side of the triangle faces outwards. An example of simulations of a drone (imported with the CAD interface) is presented in figure 1.

Figure 1: Visualisation of vortical structures for a multi-rotor unmanned aerial vehicle in hover.

One of the novelties of this work is the extension of the ADR-IBM to Fluid-Structure Interaction (FSI) problems. Here, the loose coupling is preferred over the strong one due to its considerably lower computational cost. When a strong coupling is employed, the governing equations are solved implicitly by introducing sub-time-steps within the physical time-steps until the coupled fluid and solid solutions converge. These sub-iterations are responsible for the increased computational overhead. It should be noted that unlike loosely coupled algorithms, strongly coupled ones do not suffer from numerical instabilities when the fluid and solid densities are comparable. However, the use of loose coupling is sufficient for the purposes of this study and can provide accurate results as only large, reduced mass values have been considered.

The performance of Xcompact3d when combined with the new IBM are presented in figure 2. It can be seen that the strong scaling is excellent with up to 65k cores on ARCHER and on Irene-Rome (a supercomputer in France with similar hardware as ARCHER2).

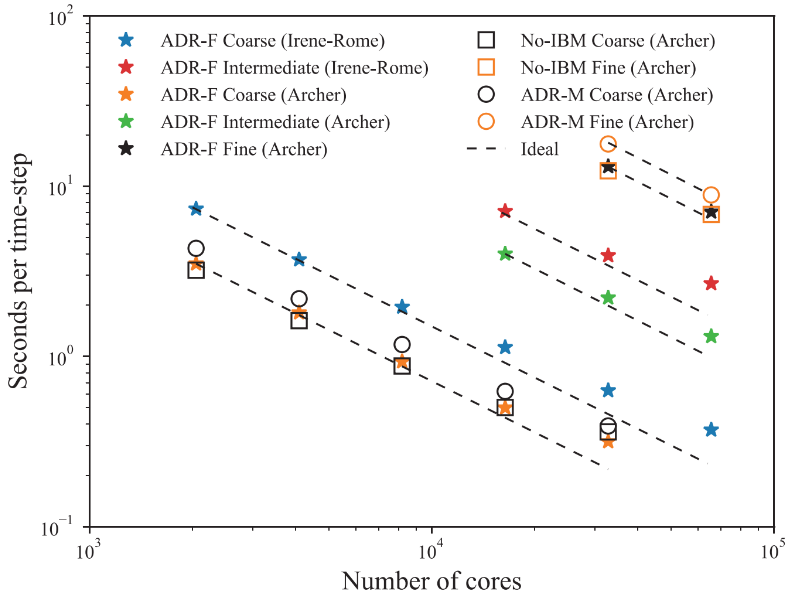

Figure 2: Strong scaling of Xcompact3D with ADR-F, ADR-M and No-IBM for various resolutions on the HPC facilities of Archer and Irene-Rome

Finally, the potential of the method in handling multiple moving objects for practical applications is demonstrated with the control of a square bluff body wake by two rear pitching flaps, see figure 3. The wake is generated by a square and is characterised by laminar vortex shedding. Two forcing strategies were examined corresponding to in-phase ``snaking'' and out-of-phase ``clapping''. The effects of the bluff body aspect ratio, flapping frequency, flapping amplitude, flap length and Reynolds number were investigated and the results are currently under review in a manuscript.

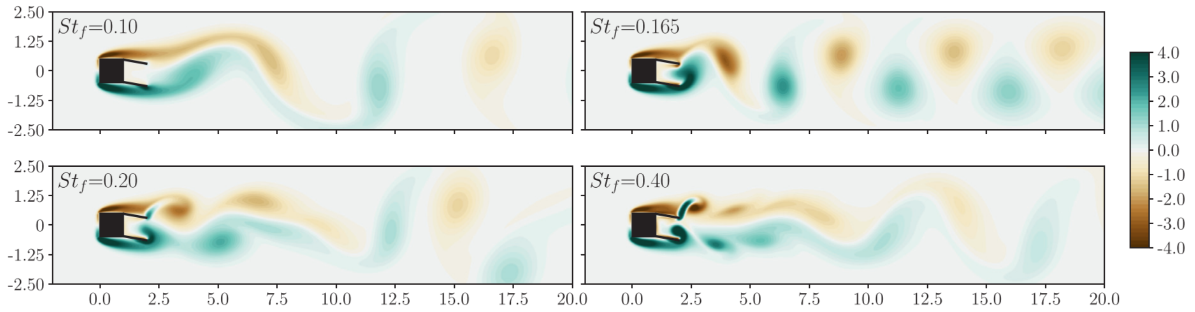

Figure 3: Instantaneous visualisations of the vorticity field for the forced flow over a square cylinder using rear pitching flaps.

The new capabilities implemented thanks to this project are presented in three papers (two published and one under review):

1- Giannenas, A. E., & Laizet, S. (2021). A simple and scalable immersed boundary method for high-fidelity simulations of fixed and moving objects on a Cartesian mesh. Applied Mathematical Modelling, 99, 606-627.
2- Hamzehloo, A., Bartholomew, P., & Laizet, S. (2021). Direct numerical simulations of incompressible Rayleigh–Taylor instabilities at low and medium Atwood numbers. Physics of Fluids, 33(5), 054114.
3- Giannenas, A. E., Bempedelis N., Schuch F.N. & Laizet, S., A Cartesian immersed boundary method based on 1D flow reconstructions for high-fidelity simulations of incompressible turbulent flows around moving objects, under review in Flow, Turbulence and Combustion, 2022

Conclusion:

In this project, a simple and scalable Alternating Direction Reconstruction Immersed Boundary Method (ADR-IBM) was developed and used for high-fidelity simulations of incompressible turbulent flows with fixed and moving objects, with or without Fluid-Structure Interaction (FSI) problems. It can be combined with a Computer-Aided Design (CAD) interface to study fluid flow problems at scale with complex geometries.

Acknowledgement:

This work was funded under the embedded CSE programme of the ARCHER2 UK National Super-computing Service (https://www.archer2.ac.uk/ecse/).

References:

[1] Laizet, S., & Lamballais, E. (2009). High-order compact schemes for incompressible flows: A simple and efficient method with quasi-spectral accuracy. Journal of Computational Physics, 228(16), 5989-6015.
[2] Laizet, S., & Li, N. (2011). Incompact3d: A powerful tool to tackle turbulence problems with up to O (105) computational cores. International Journal for Numerical Methods in Fluids, 67(11), 1735-1757.
[3] Bartholomew, P., Deskos, G., Frantz, R. A., Schuch, F. N., Lamballais, E., & Laizet, S. (2020). Xcompact3D: An open-source framework for solving turbulence problems on a Cartesian mesh. SoftwareX, 12, 100550.
[4] Hamzehloo, A., Bartholomew, P., & Laizet, S. (2021). Direct numerical simulations of incompressible Rayleigh–Taylor instabilities at low and medium Atwood numbers. Physics of Fluids, 33(5), 054114.