Joint EUROMECH / ERCOFTAC Colloquium 549
"Immersed Boundary Methods: Current Status and Future Research Directions"
Leiden, The Netherlands, 17-19 June 2013

Assessment of Immersed Boundary Method toward high accuracy

Nicolas James[1], Damien Biau[2], Julien Dambrine[1], Morgan Pierre[1] and Eric Lamballais[2]

[1] Lab. de Mathématiques, Université de Poitiers - UMR CNRS 7348, Téléport 2 - Bd Marie et Pierre Curie - BP 30179, 86962 Futuroscope Chasseneuil Cedex, France, nicolas.james@univ-poitiers.fr

[2] Institut Pprime, CNRS-Université de Poitiers-ENSMA, Téléport 2-Bd. Marie et Pierre Curie - BP 30179, 86962 Futuroscope Chasseneuil Cedex, France

ABSTRACT

In this study, a validation tool is used to investigate the numerical errors associated with an immersed boundary method (IBM). The goal is to suggest new strategies to increase the accuracy of IBM in the context of finite difference (FD) and finite volume (FV) schemes used on a structured mesh. This particular framework is believed to be a powerful tool to perform efficiently direct and large eddy simulation (DNS/LES) of turbulent flows in realistic geometry while dealing with the constraints of massively parallel computing. To the authors' experience, it is quite easy to implement rudimentary IBM in a DNS/LES code if a significant loss of numerical accuracy is accepted. Conversely, it is very difficult to reach high-accuracy using an IBM, and to the authors' knowledge, a robust high-order IBM is not available yet. To improve accuracy, two strategies are considered in this work. First, the benefit offered by a cut-cell technique is assessed using a second-order accurate Marker and Cell (MAC) method [5]. Then, the ability to increase the formal order of the discretization while using an IBM is discussed in the light of new results obtained using a direct forcing technique.

The assessment procedure is based on a reference solution for the classical problem of the flow over a circular cylinder at Reynolds number $Re = 40$ (see figure 1 for an illustration). Very recently, this reference solution has been obtained by [4] who have shown that its accuracy enables a test of spatial convergence up to a threshold of 10^{-11} and 10^{-7} for the L_2-norm of the velocity and pressure respectively. The reference solution is available at any location up to 40 diameters from the cylinder centre. A spectral interpolation, preserving the data accuracy, is provided as a user-friendly script for GNU Octave or Matlab software.

In the first part, two types of IBM are compared using the same MAC discretization on a Cartesian grid. For the first method, the direct forcing technique of [3] is used. For this technique, it was shown by [4] that the forcing of the mesh nodes just next to the immersed boundary (using a second-order interpolation scheme) enables a second-order spatial convergence for the $Re = 40$ cylinder problem, as far as velocity data are concerned. However, it was observed that the pressure convergence was less favourable, suggesting a spurious role of the projection method used to ensure the incompressible condition. The velocity and pressure convergence is illustrated in figure 2. To improve the discretization in the near-cylinder region, [1] have proposed a new cut-cell method that ensures conservation properties following the technique of [2]. The location of the velocity components is, as in [2], adapted to the cut-cell geometry. However, the pressure node is placed at the centre of the Cartesian cells for both fluid-cells and cut-cells (seee [1] for more details). The convergence of this method is exhibited for the $Re = 40$ cylinder problem. As shown in figure 2, a remarkable reduction of the numerical errors is obtained using this cut-cell method by comparison with the direct forcing technique of [3]. This reduction is almost by one order of magnitude for the pressure. For the velocity, a second-order convergence is recovered, as expected. Concerning the pressure, the second-order convergence is less clear. This point will be discussed at the colloquium. The analysis of error maps presented in figure 3 shows clearly that the main contribution of the error is located near the cylinder boundary, especially for the pressure. It is worth noting that the errors are higher in the upstream region of the near-cylinder.

The second part of the study deals with the role of the projection method in IBM, a key point regarding the concentration of pressure errors in the near-cylinder region. The use of high-order schemes (on a collocated mesh) is found unable to reduce these errors that seem to be mainly connected to the incompressibility treatment. In other words, even if a sixth-order scheme is used, a second-order convergence is observed without any significant gain in accuracy. As a first step, it is proposed in this study to investigate the behaviour of the solution of a passive scalar equation (with a Peclet number $Pe = 40$). The corresponding convection/diffusion equation, using the reference velocity field, does not require any incompressibility treatment, enabling to focus on the discretization errors of the convective and diffusive operators free from spurious contribution of the pressure. For that purpose, a reference passive scalar solution has been computed for the $Re = 40$ cylinder problem using an isothermal condition on the surface cylinder (see figure 1 for an illustration). This solution is provided in the same form as the velocity and pressure data, i.e. as a GNU Octave or Matlab script. At the Colloquium, the convergence of the passive scalar using IBM will be discussed using the two methods [3, 1]. Then, a specific forcing technique combined with high-order finite difference schemes will be presented in order to increase the formal accuracy of the method.

As a conclusion, reference solutions for the velocity, pressure and passive scalar in the $Re = 40$, $Pe = 40$ cylinder problem can be very useful to test the accuracy of new IBM developments. The authors would like to share these reference solutions with the other Colloquium participants.

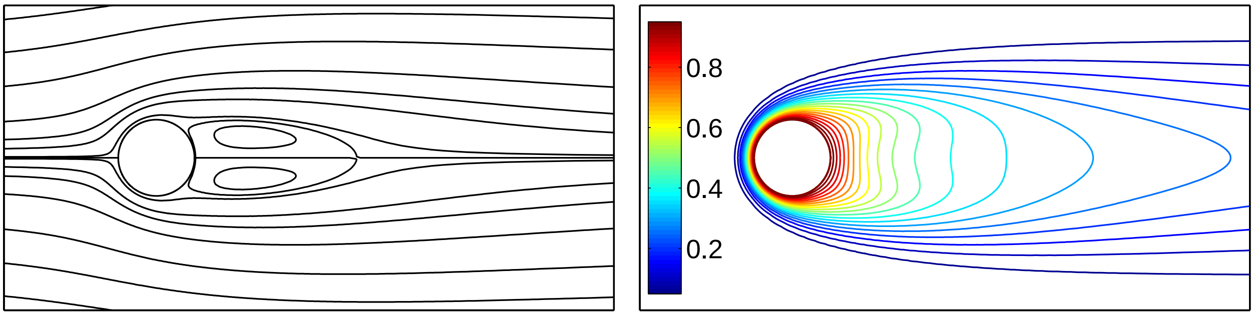

Figure 1: Streamlines (left) and passive scalar isocontours (right) for the flow over a circular cylinder at $Re = 40$ and $Pe = 40$ (reference solutions).

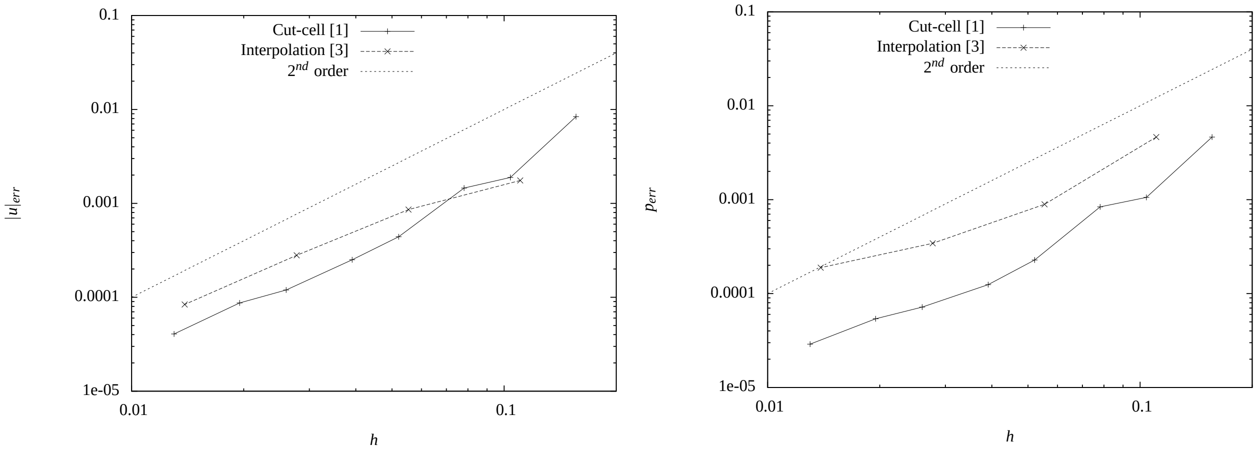

Figure 2: Error decrease with the spatial resolution. Left: velocity error. Right: pressure error.

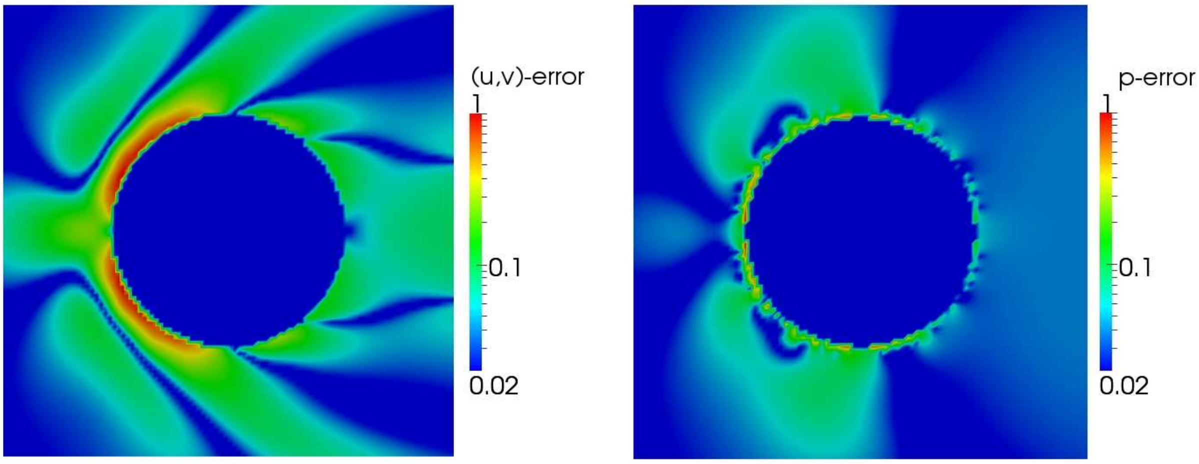

Figure 3: Maps of normalized error at the resolution 1024^2 in the subdomain $[-D,D]^2$. Left: velocity error. Right: pressure error.

REFERENCES

[1] F. Bouchon, T. Dubois, and N. James. A second-order cut-cell method for the numerical simulation of 2D flows past obstacles. *Comput. Fluids*, 65:80–91, 2012.

[2] Y. Cheny and O. Botella. The ls-stag method: A new immersed boundary/level-set method for the computation of incompressible viscous flows in complex moving geometries with good conservation properties. *J. Comp. Phys.*, 229:1043–1076, 2010.

[3] E. A. Fadlun, R. Verzico, P. Orlandi, and J. Mohd-Yusof. Combined immersed-boundary finite-difference methods for three-dimensional complex flow simulations. *J. Comp. Phys.*, 161:35–60, 2000.

[4] R. Gautier, D. Biau, and E. Lamballais. A reference solution of the flow over a circular cylinder at $Re = 40$. *Comput. Fluids*, 75:103–111, 2013.

[5] F. H. Harlow and J. E. Welch. Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface. *Phys. Fluids*, 8(12):2182–2189, 1965.