Geosci. Model Dev. Discuss.,
https://doi.org/10.5194/gmd-2018-7-AC2, 2018

Geoscientific
Model Development
Discussions

Interactive comment on "Implementation of an Immersed Boundary Method in the Meso-NH model: Applications to an idealized urban-like environment" *by* Franck Auguste et al.

Franck Auguste et al.

franck.auguste@cerfacs.fr

Received and published: 3 November 2018

Introduction

We thank the Referee 2 for his/her interest in our work and his/her positive appreciation of the manuscript. We are glad that the Referee 2 gives some suggestions to improve the manuscript. Our response is split in two sections. The first section answers to the similar comments done both by the Referee 1 and by the Referee 2. The second section gives the responses related to the specific comments done by the Referee 2. As the discussion progresses, we invite the Referee 2 to look after the revised version

of the manuscript sent with the present document. A color code applied to the text highlights the modifications: the red color is used when modifications/corrections are done; the green color is used when new insertions are proposed. Note the modification of the title following the GMD requirement.

1 Common response to Referee 1 and Referee 2

The Referees compliment the extensive work but feel that the details of the numerical implementation are not clearly expressed. Moreover, the numerical implementation suffers from a lack of details.

To propose an immersed boundary method (IBM) in the Meso-NH (MNH) code able to model the ground or topography interaction with an atmospheric flow, as it mentions by the Referees, the number of numerical developments and associated validations has to be high. It induces a long description which could be problematic regarding the format of a manuscript. For this reason, the authors decided to condense the ample information running the risk of loosing a part. The authors agree with the Referees observations. Therefore, in the proposed revised version, an important effort to give additional details is done. About the organization, Referee 1 suggests either to split the paper in two parts and/or to place the current Sections 4, 5 and 6 (which are pointed out in the limit of the GMD scope) in Appendix. Referee 2 requires more details on the cylinder case at moderate Reynolds number (Sect. 6). The authors share the same view and propose to preserve one test case dedicated to the forcing of the pressure solver and to place the others test cases of Sections 4 and 5 in Appendix. Following the Referee 2 and because of the GCT validation, Section 6 is preserved in the core of the paper. The new structure of the paper allows to detail the numerical methods (Sect. 2 and 3) without an increase of the paper volume.

In particular, the Referees make it clear the lack of details on the use of the image points in the Ghost-Cell Technique (GCT) and on the Level-Set Function (LSF). The discussion on GCT in Section 3 is therefore reinforced. Concerning LSF and in the present paper, this function is built for academic bodies and their theoretical solutions are known. The intensive work we had done to implement an accurate LSF was related to the modeling of an interface only known in a discrete way (such as the data of a real urban topography). This work is presented in another paper: Auguste et al.(submitted to Atmospheric Environment). That's said, the LSF presentation in the present paper is reinforced in Section 3.

The Referees mention that the simulation of the Taylor-Green vortex does not have to appear in the section dedicated to the potential flows.

The Referees are absolutely right. At the short time scales, the viscous influence vanishes and the Taylor-Green vortex solution is associated to an inviscid flow. The confusion (and mistakes) of the authors to put this test case in a "Potential Flows" category comes from an abusive use of the "Taylor-Green vortex" term. Even if the flow structure solution presents an array of vortex similar to the Taylor-Green ones, we do not have the right to use this term. The studied potential flow (the velocity field derives from $\pi^{-1}cos(\pi lx)sin(\pi my)$) is solution of the Poisson equation (Popinet, 2003). This case testing the pressure solver moves into Appendix of the proposed revised manuscript.

The Referees mention a lack of details and/or confusion on the molecular diffusion used in the Direct Numerical Simulations.

The molecular diffusion is taken into account in the cases presenting a low Reynolds number ("low" compared to most of Re of atmospheric applications). This term is associated to the fluid kinematic viscosity ν_f. Therefore, $\nu_f \Delta u$ is explicitly added

in a physical purpose (Navier-Stokes equations resolution, Eq. 3). The numerical implementation is the most simple: in 1D for example, its contribution on the $\frac{\partial u_i^{n+1}}{\partial t}$ is computed for uniform Cartesian grids such as $\nu_f/\Delta^2(u_{i+1}^n - 2u_i^n + u_{i-1}^n)$ where Δ is the space step. The explicit-in-time resolution induces the respect of the stability condition $\mathcal{O}(\nu_f/\Delta^2)$. Some additional comments about the fluid viscosity are inserted in Section 2. Even if this type of flow is far from the atmospheric applications, this is a robust way to test and validate the implemented GCT. For example, this study makes us confident in the forcing of the Reynolds stresses $\nabla.(\nu_e \nabla \overline{u})$ near an immersed wall (ν_e the turbulent viscosity). In the same spirit and for future thermodynamic applications, the authors mention that another study was carried out on the parabolic heat equation to confront (and validate) the GCT to a 1D pure diffusion problem $\frac{\partial T_f}{\partial t} = (\lambda_f/\rho C_p)\Delta T_f$.

2 Specific response to Referee 2

Section 3. In Figure 1 (a), what does the triangle mean?

The triangle symbol indicates an arbitrary type of nodes (P/U/V/W for example). This symbol does not appear in the new Fig.1-a. This symbol is defined in Fig.2-b.

The authors should give the mathematical expression of the level set function and show how it is used to identify the interface by one example, as did in the cited paper of Tseng and Ferziger (2013).

To improve the introduction of LSF, a new Fig. 1-a is used. In addition, the definitions of the vector normal to the interface and the curvature are given.

In the expression $GI = 2phi_G n$, the variable phi_G is not defined.

As it is suggested, the definitions of ϕ_G and ϕ_I are added.

The use of "original" for the proposed method to differ from the classical method would be misleading and inappropriate. The author could use another word like "new", "novel" or "proposed" instead.

We agree with the Referee and the "original" term is substituted by "new" or "proposed".

In the flow over a stationary cylinder test, the authors have studied different meshes but not indicated the domain size, while which has much greater impact to the final solution. Comparison could be made to the reference paper "Moving immersed boundary method, International Journal for Numerical Methods in Fluids, 2017". A convergence study could be performed for the convergence rate of velocity, as it never shows in the other parts.

The domain size is indicated: "The limit of the numerical domain is $10D_{cyl}$ upstream the obstacle for the inlet condition (U_∞, the uniform incoming velocity) and lateral condition (slip condition), $15D_{cyl}$ for the outlet condition allowing the vorticity evacuation." Our domain size choice followed the Auguste (2010) one. That's said, the domain size can have a dramatic influence studying an unbounded Stokes regime for example and we agree with the Referee about the possible domain size influence in the 2D presented case at moderate Re. Numerical effects of the inlet/outlet conditions can weakly affect our results at $Re = 40$ as it is demonstrated in Cai et al. (2017); this is one of the reason to use the $\approx$ symbol in Table 1. Even if a convergence study based on a variable such as an axial velocity in the wake of the body does not appear in the proposed revised version, the paper indicates the ability to simulate the physical problem with a good description of the vortex structure in the near-wake of the cylindrical body. To compensate the non-appearance of a convergence study, the proposed revised version of the paper is enriched with a supplementary materials which shows the ability of MNH-IBM

to simulate a physical problem (Straka et al., 1993) governed by thermal effects and viscous effects (the used ν_f value is 10^4 higher than the atmosphere one). This study compares with the literature results (body conformal grid method) the velocity and the spread of a density current.

References

Auguste, F. (2010). Instabilités de sillage et de trajectoire dans un fluide visqueux. Ph.D. thesis, University of Toulouse.

Auguste, F., Paoli, R., Lac, C., Masson, V., and Cariolle, D. (submitted to Atmospheric Environment). Large eddy simulations devoted to the health impact of pollutant dispersions in cities: the case of the NO2 plume due to the AZF explosion in Toulouse (21/09/01).

Brennen, C. E. (1982). A Review of Added Mass and Fluid Inertial Forces. Naval Civil Engineering Laboratory, Port Hueneme, CA. CR 82.010.

Cai, S.-G., Ouahsine, A., Favier, J., and Hoarau, Y. (2017) Moving immersed boundary method. Int. J. Numer. Methods Fluids, 85(5), 288-323.

Cuxart, J., Bougeault, P., and Redelsperger, J.-L. (2000). A turbulence scheme allowing for mesoscale and large-eddy simulations. Quart. J. Roy. Meteor. Soc., 126(562), 1-30.

Goldstein, D., Handler, R., and Sirovich, L. (1993). Modeling a no-slip flow boundary with an external force field. J. Comput. Phys., 105(2), 354-366.

Kempe, T., Frölich, J. (2012). An improved immersed boundary method with direct forcing for the simulation of particle laden flows. Journal of Computational Physics, 231(9), 3663-3684.

Kim, J., Kim, D., and Choi, H. (2001). An immersed-boundary finite-volume method for simulations of flow in complex geometries. J. Comput. Phys., 171(1), 132-150.

Lac, C., Chaboureau, J.-P., Masson, V., Pinty, J.-P., Tulet, P., Escobar, J., Leriche, M., and others (2018). Overview of the Meso-NH model version 5.3 and its applications.

Lafore, J. P., Stein, J., Asencio, N., Bougeault, P., Ducrocq, V., Duron, J., Fisher, C., Hèreil, P., Mascart, P., Masson, V., Pinty, J. P., Redelsperger, J.-L., Richard, E., and Vilà-Gueau de Arellano, J. (1998). The Meso-NH Atmospheric Simulation System. Part I: adiabatic formulation and control simulations. Scientific objectives and experimental design. Annales Geophysicae, 16, 90-109.

Lundquist, K. A., Chow, F. K., and Lundquist, J. K. (2010). An immersed boundary method for the Weather Research and Forecasting model. Mon. Wea. Rev., 138(3), 796-817.

Lundquist, K. A., Chow, F. K., and Lundquist, J. K. (2012). An immersed boundary method enabling large-eddy simulations of flow over complex terrain in the WRF model. Mon. Wea. Rev., 140(12), 3936-3955.

Mittal, R., and Iaccarino, G. (2005). Immersed Boundary Methods. Annu. Rev. Fluid Mechs, 37:239-261.

Popinet, S. (2003). Gerris: a tree-based adaptive solver for the incompressible Euler equations in complex geometries. J. Comput. Phys., 190(2), 572-600.

Straka, JM, Wilhelmson, R. B., Wicker, L. J., Anderson, J. R., and Droegemeier, K. K. (1993). Numerical solutions of a non-linear density current: A benchmark solution and comparisons. Int. J. for Num. Methods in Fluids, 17(1), 1-22.

Please also note the supplement to this comment:
https://www.geosci-model-dev-discuss.net/gmd-2018-7/gmd-2018-7-AC2-supplement.zip

---

Interactive comment on Geosci. Model Dev. Discuss., https://doi.org/10.5194/gmd-2018-7, 2018.