Geosci. Model Dev. Discuss.,
https://doi.org/10.5194/gmd-2018-7-AC1, 2018
© Author(s) 2018. This work is distributed under
the Creative Commons Attribution 4.0 License.

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 1 for his/her interest in our work and his/her positive appreciation of the manuscript. We are glad that the Referee 1 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 1. As the discussion progresses, we invite the Referee 1 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 1

"This study describes the numerical implementation, verification and validation of an Immersed Boundary Method (IBM) in the atmospheric solver Meso-NH for applications to urban flow modelling". Need some reference.

The IBM annual review of Mittal and Iaccarino (2005) and the MNH reference papers (Lafore et al., 1998; Lac et al. , 2018) are inserted.

"Covering all possible cases is obviously impossible but from a fluid mechanics standpoint one can invoke the principle of similarity which permits, for example, to observe von Kármán streets in the wake of a centimeter-scale cylinder as well as in the cloud layout behind an island." I understand the thought of the authors, but in both cases the Reynolds number (which characterize the "principle of similarity") is much different.

As it is suggested by the Referee 1, the use of the "principle of similarity" is abusive. The sentence is rephrased : "Covering all possible cases is obviously impossible but from a fluid mechanics standpoint one can invoke the persistence of flow behaviors whatever the scales. It allows for example, to observe von Kármán streets in the wake of a centimeter-scale cylinder as well as in the cloud layout behind an island."

"Even if the physical application in our mind is the atmospheric mesoscale reaction to perturbations induced by urban cities, the more the obstacles are considered as a part of the scales numerically resolved the more the results accuracy is." Is this sentence is useful ?

The sentence is rephrased: "The physical application in our mind is the atmospheric mesoscale reaction to perturbations induced by urban cities and the more the obstacles are considered as a part of the scales numerically resolved the more the results accuracy is."

"This approach and its variant developed by Goldstein et al. (1993) for a rigid interface can suffer from the lack of stiffness (fluid-solid interface is generally spread over few cells) which can be problematic to recover the boundary layer." The main issue with Goldstein et al. (1983) approach comes from the fact that the time step should be very low if we want to enforce a rigid boundary condition.

The sentence is rephrased: "This approach and its variant developed by Goldstein et al. (1993) for a rigid interface can suffer from the lack of stiffness (fluid-solid interface is generally spread over few cells) and the time step restriction."

"Depending on how to resolve the partial differential equations, Cartesian grid methods (Ye et al., 1999) are written for finite-volume discretizations (Cut-Cell Technique, CCT) and for finite-difference discretizations (Ghost-Cell Technique, GCT) as in Tseng and

Ferziger (2003)." This sentence could be clarified.

A sentence is added: "CCT reshapes the cell cut by the interface to preserve mass, momentum and energy. Using GCT, the local spatial reconstruction is done inside the solid region."

"Another argument in favor of discrete forcing is that it does not introduce source terms in the conservation equations." Source terms are introduced but in a discrete way rather than in a continuous way.

We agree with the Referee. The authors discuss about a GCT characteristics and apply incorrectly this to all discrete forcing types. The sentence is rephrased: "An argument in favor of GCT is that it does not introduce source terms in the conservation equations."

The Meso-NH code at a glance. The overall section is understandable but some definitions are missing and should be explicitly given or not mentioned. Indeed for a significant part of the paper (potential flow, DNS, ...) Coriolis forces, molecular diffusion does not play any role. Moreover more details about the LES technique could be given, for instance what is the filter size ? Finally, it is interesting to know that the advective term can be solved using four different schemes to understand section 5

The terms which are not used in the present paper disappears in the text and in the equations. To improve the presentation of the MNH turbulence scheme (Cuxart et al., 2000), Section 2.3 is reinforced. To improve the introduction of the IBM forcing of the turbulence scheme, Sect. 3.3 is reinforced.

Other authors (Kempe and Froelich JCP 2012) are using LS method to follow the shape of moving immersed bodies.

The reference Kempe and Frölich (2012) is added during the LSF presentation.

"l3-l8" - This paragraph should be clarified. Is it necessary to mention the Heaviside function as function of the LS function ? The authors could say that there are only looking at point closed to the interface.

The heaviside function was used to clearly show that the forcing acts only in the solid region and the conservation laws in the fluid region is preserved. This use is dispensable. Equation 7 disappears and Fig. 1 is modified in consequence.

"A problematic case regularly met in the mirror interpolation is the vicinity of ghosts with the interface". Why ?

A ghost in the vicinity of the interface has an image point also closed to the interface. This is not a problem when the boundary condition is independent of the fluid information. This could be a problem when wall models (based on the turbulent Reynolds number for example) are used. In the worst case scenario, the ghost and its associate mirror point coincide.

"The definition of several images per ghost permits the access to a building of the normal profile of the fluid information by an 1D quadratic interpolation". This point could be clarified.

The authors hope the clarity of Section 3.1 is improved in the revised version.

"Note the $l\phi/2$ approximation on the location of the derivative term". This point could be clarified.

The formulation is exact when $\frac{\partial\overline{\psi}}{\partial n}|_b = \frac{\partial\overline{\psi}}{\partial n}|_{l\phi/2}$

"First looking at the RHS of (5), the ... law coming from the resolution of the explicit-in-time schemes near the interface and in the solid regions badly affects the ... computation (note that the GCT operates after the step projection)." To be clarified.

If the CCT is not used to correct the pressure solver, the computation of the RHS of Eq. (5) will perceive the GCT. Some old tests had shown that the impermeability satisfaction using GCT was not convincing. An imposition of a null velocity in the solid region was more convincing but it raised the question of the mass and energy conservations in the truncated cells. These observations induced the motivation to correct the pressure solver with a volume approach and a reconstruction of the truncated cells.

The meaning of the symbol tilde could be clarified.

The tilde symbol was used to distinguish the tangent velocity at the interface without and with the pragmatic limitation. To avoid some possible confusion, the use of the symbol disappears in the revised paper.

Why the disk radius is split in 8. Why not 4 or 16 ? Figure 4 (b) is misunderstanding since the arbitrary surface is a square !

The choice of the splitting was decided regarding the number of adjacent points and desiring an isotropic behavior of the surface integration. The Referee correctly points out that this choice is an arbitrary one and a study of the impact of the splitting type should be perspicacious. The good results of the initial choice do not motivate this sensitivity study in the present paper. This study could occur in the future when this technique will be generalized to non uniform Cartesian grids.

"This number is limited by a convergence criterion (compromise between incompressibility satisfaction and CPU cost)". What is the CPU cost of this step ?

With a low number of transported species, the numerical schemes in MNH responsible of the CPU cost are mainly the advection of the mean wind and the pressure solver. Depending on the flow, the Poisson equation in the pressure solver is resolved by $[2:4]$ iterations. The characteristic CPU cost of the pressure solver is about $[20:30]\%$ the global cost.

The subsection 3.3 could be clarified. The referee is not fully accustomed to wall turbulence modelling. Nevertheless the discussion is quite hard to follow.

The authors hope the clarity of Section 3.3 is improved in the revised version.

Since the proposed methods are closed to the ones proposed by Kim, Kim and Choi (JCP2001) the authors could mention it in the introduction.

The reference Kim et al. (2001) is added.

Potential flows. "With a change of Galilean reference frame, this study corresponds to an uniform body acceleration ab in a fluid initially at rest The steady uniform flow past a sphere is not equivalent to the accelerated flow past a sphere". How the added mass is computed ?

The potential scalar Ψ^* evolves in the solid region and induces a modification of the velocity field. For an acceleration during one time step (the body translating in a longitudinal direction), Ψ^* corresponds to a longidutinal gradient solution and $\mathcal{A}m_f a_b = \int_{\mathcal{V}_s} \frac{\partial \rho_f u}{\partial t} d\mathcal{V}$; the equation is completed (Sect. 4.1).

"Unsurprisingly Acyl increases with the confinement". Reference ?

The reference Brennen (1982) is added.

"To conclude, this section validates the modification to the pressure solver. IBM appears less accurate than BFM when the ground presents low curvature in regard of the space resolution. It seems more pertinent than BFM to model high interfaces such as sharp edges or corners". More details should be given to explain this behaviour.

The sentence is inserted: "The minimum pressure which should aim to $-\infty$ is not perceived or smoothed by IBM and allows the pressure solver not to diverge. Note that Lundquist et al. (2010, 2012) using compressible WRF model observe similar behaviors." This sentence appears now in Appendix related to the pressure solver test.

"The adopted strategy with IBM is to model the advection term with a low order scheme near the interface (Sect. 3). To avoid numerical diffusion, why a low order scheme is used ?"

The order in space of the numerical scheme dedicated to the mean wind is decreased near the immersed interface such as the scheme order is decreased in BFM. The decrease in IBM is not essential but allows to limit the number of ghosts in the solid region, limit the communications during a parallel computation. Indeed, the chosen implementation implies that the associated images and ghosts points have to be localized in the integration volume of each processor. The cells thickness to communicate between processors is currently four. A part of this comment is added in the Appendix A.

"The vorticity equation for a 2D inviscid flow reveals no production in time". To be clarified.

The vorticity equation (w, the vorticity) in the studied case (inviscid flow, uniform density, incompressible) is $\frac{\partial w}{\partial t} + (\overline{u}.\nabla)w = -(w.\nabla)\overline{u}$. The LHS term vanishes in a 2D case.

"To fit as well the potential solution, a non-trivial condition is employed on the tangent velocity" Why such a complicated boundary condition is used.

The velocity field $\overline{u}$ evolves such as $\mathcal{O}(1/r^2)$ with r the radial distance to the cylinder interface. To not impose a linear evolution of $\overline{u}$ and $\frac{\partial \overline{u}}{\partial r}$, we use a simple wall model acting such as a weak condition. This case is also a good example of the interest to define several images points.

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-AC1-supplement.zip

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