Proceedings of 34nd International Conference on Offshore Mechanics and Arctic Engineering
OMAE'15
May 31-June 5 , 2015, St.John's, Canada

OMAE2015-42195

DYNAMIC DOMAIN DECOMPOSITION STRATEGY COUPLING LATTICE BOLTZMANN METHODS WITH FINITE DIFFERENCES APPROXIMATIONS OF THE NAVIER-STOKES EQUATIONS TO STUDY BODIES IN CURRENT

Giuseppina Colicchio
CNR-INSEAN, Rome, Italy. Centre for Autonomous Marine Operations and Systems (AMOS), Department of Marine Technology, NTNU, Trondheim, Norway. Email: giuseppina.colicchio@cnr.it

Claudio Lugni
CNR-INSEAN, Rome, Italy. Centre for Autonomous Marine Operations and Systems (AMOS), Department of Marine Technology, NTNU, Trondheim, Norway. Email: claudio.lugni@cnr.it

Marilena Greco
Centre for Autonomous Marine Operations and Systems (AMOS), Department of Marine Technology, NTNU, Trondheim, Norway. CNR-INSEAN, Rome, Italy. Email: marilena.greco@ntnu.no;marilena.greco@cnr.it

Odd M. Faltinsen
Centre for Autonomous Marine Operations and Systems (AMOS), Department of Marine Technology, NTNU, Trondheim, Norway. Email: odd.faltinsen@ntnu.no

ABSTRACT

A Domain-Decomposition (DD) strategy is proposed for problems involving regions with slow variations of the flow (A) and others where the fluid features undergo rapid changes (B), like in the case of steady current past bodies with pronounced local unsteadiness connected with the vortex shedding from the structures. For an efficient and accurate solution of such problems, the DD couples a Finite Difference solver of the Navier-Stokes equations (FD-NS) with a Multiple Relaxation Time Lattice Boltzmann method (MRT-LBM). Regions A are handled by FD-NS, while zones B are solved by MRT-LBM and the two solvers exchange information within a strong coupling strategy. Present DD strategy is able to deal with a dynamic change of the sub-domains topology. This feature is needed when regions with vorticity shed from the body vary in time for a more flexible and reliable solution strategy. Its performances in terms of accuracy and efficiency have been successfully assessed by comparing the hybrid solver against a full FD-NS solution and experimental data for a 2D circular cylinder in an impulsively started flow.

INTRODUCTION

Computational fluid dynamics (CFD) is becoming more and more popular for hydrodynamics studies because of its potentialities in terms of capabilities and limitation of computational costs with respect to experiments. However there is no method able to solve any generic problem efficiently and accurately (see *e.g.* discussion in [1]). This is why there is an important effort in developing Domain-Decomposition (DD) strategies where different solvers are used in time-space zones of the flow evolution so to optimize the simulation outcomes in terms of reliability, flexibility and costs.

The authors have already contributed to this research attempt by developing various DD strategies, in the context of violent water-structure interactions involving slamming and other nonlinear phenomena, such as breaking and fragmentation of the air-water interface, entrapment of gas cavities (see *e.g.* [2], [3], [4] and [5]). Here this approach is proposed for studying problems associated with multiple time-scale phenomena. In the present implementation, the framework is a body in a single-phase unbounded or bounded fluid domain under the action of a steady

current which has practical applications, for instance, for subsea structures. In this case, depending on the body geometry and the involved Reynolds number *Re*, the current-structure interaction leads to local rapid changes in the flow features due to vortex shedding from the body. As the vortical structures leave the body they tend to diffuse due to viscosity and the flow variations tend to become less strong and slower. To properly estimate the current induced loads on the body it is necessary to properly describe both the injection of vorticity in the fluid and the further development of the wake. Present DD proposes to examine the slowly varying regions with a classical Finite Difference (FD) approximation of the Navier-Stokes (NS) equations and to treat small unsteady regions with Multiple Relaxation Time-Lattice Boltzmann method (MRT-LBM). The latter solves the discrete Boltzmann equation from the statistical mechanics and has been used more and more commonly in the last decades to simulate fluid dynamic problems. Examples of applications for marine problems are for instance [6], [7]. This method has shown high suitability for problems associated with large Knudsen numbers (*Kn*), where the *Kn* is the ratio between Mach and Reynolds (*Re*) numbers. It means that it represents a valuable tool of analysis for microfluids. Moreover, the explicit time marching scheme, based on an advection and a collision step, makes it much more efficient when dealing with unsteady problems with respect to the classical CFD solvers that have to solve algebraic systems resulting from the discretization of the NS equations (see [8]). On the other hand, the method has a weakly-compressible nature and so it is more computationally expensive for almost steady fluid problems. In those cases the weakly compressibility effects have to exit the computational domain, requiring longer times of simulation and numerical strategies have to be used to filter out those effects at the boundaries. This makes the classical CFD methods more efficient. Differently from [9] here the DD strategy is able to deal with a dynamic change of the sub-domains topology, allowing a much more flexible tool for bodies moving freely inside the whole computational domain. This numerical feature is needed when the sub-domains, with different time-scale, modify. For example at high *Re* the regions where the LBM contribute is needed, *i.e.* those with relevant shed vorticity, change in time. On the other hand this represents a core numerical challenge especially because both solvers operate on a multi-blocks decomposition of the domain with a local mesh refinement. The major difficulties are in the FD-NS side because it is required the solution of a Poisson equation for the pressure and the available solvers have difficulties when applied on multi-blocks with varying cell sizes, changing in time within a dynamic grid algorithm.

In the next section the two methods are briefly outlined, then an important part of the work is devoted to the description of the developed DD strategy. More in detail, the adopted procedure for the dynamic change of the domains sizes is examined, highlighting the criteria for the choice of the solver, the tracking of the sub-domains boundaries and the exchange of information across the sub-domains interface. The hybrid method is then applied to the problem of a current past a body and the results are verified against other numerical solutions and experimental data.

METHODS USED IN THE DOMAIN-DECOMPOSITION

Here a fluid-dynamic problem involving fluid regions with phenomena governed by different time scales is examined by means of a Domain-Decomposition strategy. In order to optimize the solution in terms of accuracy and efficiency two methods have been selected. Both solvers are accurate to the second order in space and time and are briefly outlined in the following.

The Lattice Boltzmann method The first is a Multiple Relaxation Time-Lattice Boltzmann method (MRT-LBM), which models the fluid as particles in lattice cells associated with an evolving probability distribution, and so it is well suited to handle small-scale spatial and temporal flow features. The relaxation time leads to a weak-compressible modelling of the fluid. The distribution function $\bar{f}(\bar{r},t)$ of each particle, at position $\bar{r}$ and time t, enables to represent the average macroscopic properties of the flow. Here the number of components of $\bar{f}$ are equal to the possible directions of particle advection, including the rest state. Following the work in [10], the molecular motion of real fluids is simplified and represented through cubices. In each of them, the evolution equation of $\bar{f}$ at the center is formally written as

$$\bar{f}(\bar{r}+\bar{c}_\alpha\Delta t, t+\Delta t) - \bar{f}(\bar{r},t) = -\mathbf{S}\cdot[\bar{f}(\bar{r},t) - \bar{f}_{eq}(\bar{r},t)]. \quad (1)$$

Practically the problem is subdivided into two steps: 1) a Collision step, where the right hand side of the equation is implemented to take into account the viscous effects. Practically the distribution function $\bar{f}$ is updated as a function of its distance from the equilibrium conditions $\bar{f}_{eq}$; 2) an Advection step, where the new distribution functions are convected to the new position with the velocity $\bar{c}_\alpha$.

The speed $\bar{c}_\alpha$ expresses all possible advections of the particle in the lattice. In the present case, $\bar{f}$ has 19 components and the corresponding speeds include the zero value ($\alpha = 0$) and the speeds bringing the particle to any face ($\alpha = 1-6$) and side ($\alpha = 7-18$) center of the cube. The adopted 3D Lattice arrangement is shown in figure 1 together with the velocity directions. The LBM is based on the Boltzmann equation, a molecular description of a fluid directly incorporating physical terms due to the interaction between molecules. Its discrete formulation can be handled numerically more easily than its Navier-Stokes counterpart that, however, can be derived from the Boltzmann equation in the continuous limit and for small Kn numbers.

The method preserves the mass and linear momentum and it can be seen as a finite difference method with central derivation and explicit time integration scheme, with a second order accuracy.

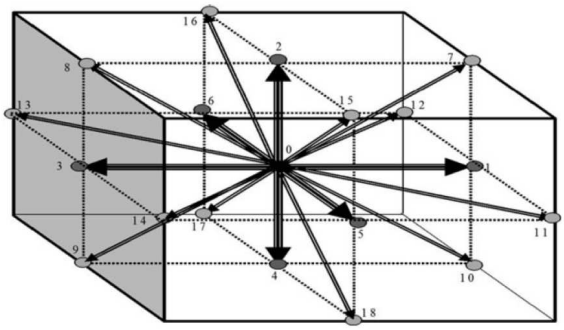

FIGURE 1. MRT-LBM SOLVER: ARRANGEMENT OF A LATTICE FOR THREE-DIMENSIONAL PROBLEMS.

In the present MRT-LBM [10], the single relaxation time is substituted with a collision matrix **S**, whose eigenvalues are chosen so that the relaxation times of the non-conserved quantities (*e.g.* energy for athermal fluids) are shorter than the hydrodynamic time scale. The velocity moments are written as a linear function of $\bar{f}$ and introduced in equation (1). This leads to a source term in the right-hand side with some coefficients directly linked to the shear viscosity and others directly connected to the bulk viscosity. In this way the two viscous terms can be decoupled in the solution algorithm leading to a larger stability with respect to other classical Bhatnagar-Gross-Krook approaches [11]. To help reducing the round-off errors in the density calculations, the density ρ is decomposed in its average part ρ and its fluctuation $\delta\rho$, where $\delta\rho = \sum_i f_i$. At a body surface a no-slip condition is implemented using the Bounce Back (BB) method [12], [13], based on the idea that a particle incident to the wall is automatically bounced back into the flow and its tangent component is switched in the opposite direction. The BB method is quite easy to implement; in case of curved bodies, it involves interpolations and, in the present implementation, it preserves the second-order accuracy of the solver.

The LBM method leads to uncoupled equations between the cells describing the fluid domain, so that it is easy to parallelize the algorithm and to nest and Adaptive Mesh Refinement algorithm in it. In particular here, PARAMESH is used [14], [15], [16]. This package builds a hierarchy of sub-grids to cover the computational domain, with spatial resolution varying to satisfy the local features of the problem. The sub-grid blocks form the nodes of a tree data-structure (quad-tree in 2D or oct-tree in 3D). The local refinement halves the mesh size, this means that the local time step has to be halved as well as some of the relaxation matrix components have to be modified consistently. Very briefly Collision and Advection steps are performed $2^{ref_{lev}-1}$ times for each ref_{lev} refinement level ($ref_{lev} = 1$ is the coarsest level and the level increases as the mesh size is halved) and the new information are exchanged and propagated to the levels with equal or higher refinement.

The Finite-Difference Navier-Stokes method In this case the fluid is assumed as incompressible, viscous and in laminar and isothermal conditions. It means that the governing equations are the conservation of fluid mass and of fluid momentum for the unknowns velocity $\bar{u}$ and pressure p *i.e.*

$$\begin{cases} \nabla \cdot \bar{u} = 0 \\ \dfrac{\partial \bar{u}}{\partial t} + (\bar{u} \cdot \nabla)\bar{u} = -\dfrac{\nabla p}{\rho} + \dfrac{2\nabla \cdot (\mu \mathbf{D})}{\rho} \end{cases} \tag{2}$$

using an Earth-fixed coordinate system. Here μ is the dynamic viscosity and **D** the strain tensor and ρ and $\bar{u}$ are, respectively, the fluid density and velocity. Equations 2 are solved in time, for given initial and boundary conditions, by a Finite-Difference Navier-Stokes solver based on an approximated Projection method and combined with a Predictor-Corrector scheme for the time integration. No body force is present, because it has been neglected also in the MRT-LBM formulation. The fixed grid is staggered with the scalar variables (*i.e.* ρ and p) defined at the cell center and the velocity components on the grid faces. The method is accurate to the second order in time and space and involves the solution of a pressure Poisson equation for each subset of the time-integration algorithm. The details of the solver can be found, for instance, in [17].

DOMAIN-DECOMPOSITION STRATEGY

The two solvers described in the previous section have been strongly coupled within the developed Domain-Decomposition (DD) strategy. Local information are provided through the common boundaries of their sub-domains which modify in time when a suitable criterion is satisfied. These DD features are described in detail next. To make the solution efficient in time, the problem is solved in a multi-block grid using an adaptive mesh refinement according to the strategy in [18].

Division of the domain between the two solvers The features of the AMR algorithm are extensively used in the DD strategy. First of all, because the LBM is used to describe the generation of vorticity and its later evolution, it needs to be discretized on a more refined grid. So the level of refinement is used to identify the division between the two domains. In particular a threshold level of refinement $lev_{treshold}$ defines the boundaries between the two solvers. All the leaves blocks, blocks at the local highest refinement level, characterized by refinement level greater than $lev_{treshold}$ are used for the LBM solution, those with level of refinement equal to $lev_{treshold} + 1$ and whose neighbours have a boundary shared with a block of lower level are considered

boundary blocks. The parents of these boundary blocks and all the leaves with refinement level less than $lev_{treshold}+1$ identify the part of the domain where the NS algorithm is applied.

Information exchange between the solvers The two sub-domains overlap on the parents of the boundary blocks so that the solution can be enforced to smoothly go from the particle- to the continuous-fluid solver description. This feature has been found useful for hybrid-solver strategies. Figure 2 gives an example of fluid domain decomposition for the flow around a circula cylinder: the MRT-LBM portion contains the body and the flow regions interested by relevant shed vorticity while the remaining domain is described by the FD-NS solver. The two solvers have different time steps satisfying their own stability requirements and describe the fluid in a different manner. The time step of the LBM solver has stricter constraints so that the Δt_{LBM} is obtained dividing by an integer value α the outer time step Δt. As already

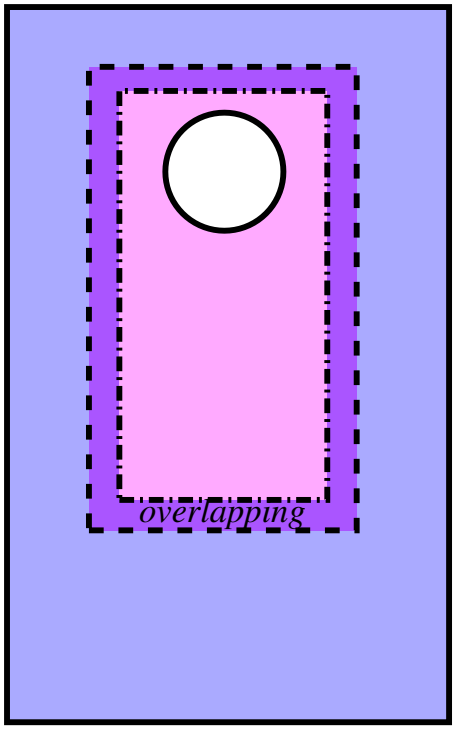

FIGURE 2. SKETCH OF THE PROBLEM: DEFINITION OF DD SUB-DOMAINS. FOR THE FLOW AROUND THE CYLINDER, THE REGION CLOSE TO THE BODY IS MODELLED WITH A MRT-LBM, THE OUTER REGION WITH A NS, WITH LARGE. SPATIAL DISCRETIZATION.

found stable for other DD strategies [3], the values of pressure inside the FD-NS domain on the overlapping regions are used to extrapolate the boundary pressure conditions for the MRT-LBM at the n-th sub-time steps $t+n\Delta t/\alpha$; vice versa, the values of pressure and velocity at $t+\Delta t/2$ obtained from the LBM evolution are given as boundary conditions to the NS solver (see figure 3), that can thus calculate the velocity field at the $t+\Delta t$. The two solvers equations are not written on the same variables,

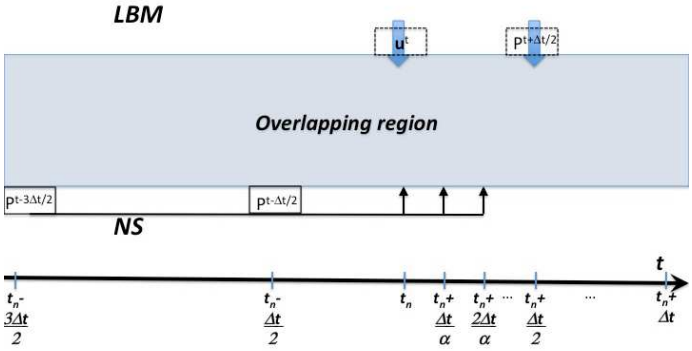

FIGURE 3. EXCHANGE OF VARIABLES ACROSS THE OVERLAPPING REGION.

the FD-NS is based on pressure and velocity, the MRT-LBM on the distribution function. The variables are then *translated* from LBM to NS using the properties

$$\begin{aligned} \Sigma_{i=0}^{18} f_i &= \rho \\ \Sigma_{i=0}^{18} f_i \bar{c}_i &= \rho \bar{u} \\ p &= \rho c_s^2 \end{aligned} \tag{3}$$

where c_s is the speed of sound. The pressure boundary conditions from the NS to the LBM are written using the equilibrium momentum

$$\bar{m} = \bar{m}^{eq}(\rho, \bar{u}_{bound}) \tag{4}$$

as described in [10] and where $\bar{u}_{bound}$ is the velocity in the cell next to the considered boundary location.

Dynamic evolution of the two-solvers boundary As already said, the MRT-LBM domain deals with the vorticity shed from bodies and, for high Reynolds numbers, the regions where its contribute is needed changes in time. Two threshold values of vorticity ω_l and ω_u are associated to each refinement level higher than $lev_{treshold}$. If the minimum value of $|\nabla \times \bar{u}|$ inside the MRT-LBM block is lower than ω_l then the block is tagged for derefinement, if $|\nabla \times \bar{u}| > \omega_u$, the block is tagged for refinement,

otherwise nothing happens. Once this operation has been performed for each of the LBM blocks, the PARAMESH solver is able to rearrange the blocks and refine/derefine the domain and eventually change the boundary between the two solvers dynamically. Nonetheless it respects some properties in the refinement/derefinement process, for example during the refinement process, the refinement level is not allowed to jump by more than 1 refinement level at any location in the spatial domain. This procedure is computationally expensive so it is performed only every N time steps, where N is to be assessed according to the dimensions of the blocks, *i.e.* ensuring that, if the vorticity moves with the reference velocity, it cannot travel for a length larger then the minimum block size.

APPLICATION

The developed hybrid solver is applied here to the case of a 2D circular cylinder in an impulsively started flow. This choice soon highlights the advantages of the DDD. In fact the NS solver is applied on the coarsest refinement level on the whole fluid domain for the first time instants, when no vorticity is present in the flow field. As soon as the vorticity develops, causing the detachment of the flow from the cylinder, the LBM solution is initialized. This allows a direct solution of the pressure impulse phase. Pressure and velocity, calculated by the NS, are transformed in distribution functions using the initial distribution weights and the equations(3) [10]. Moreover, this shortens massively the time to get almost to the steady state, in fact, in the MRT-LBM, the impulsive start corresponds to the activation of shock waves that propagate from the body and have to be filtered at the boundary with the FD-NS.

Simulations of the flow around the cylinder have been performed for five Reynolds numbers, respectively equal to $Re = 10, 20, 40, 100, 200$. For low Reynolds numbers, the time saving with respect to the full Navier-Stokes solver is very limited because the grid does not need to be very fine to capture the details of the flow. The time saving starts to be important for $Re = 100$ when the vorticity structures elongate and break up. There the coarse discretization of the Navier-Stokes equation fails to capture this behaviour of the flow and the use of a finer mesh becomes expensive. With the DD strategy, the LBM allows to describe the evolution of the flow as shown in figure 4, where the vorticity contours and the streamlines are displayed for the flow at $Re = 100$. The mesh size of the coarsest level for the displayed example is $\Delta x = R/5$, where R is the radius of the cylinder and there are four levels of refinement, this means that on the finest level $\Delta x = R/40$. The chosen ratio between FD-NS time step and MRT-LBM time step is $\alpha = 10$, to be sure that compressible effects are limited. Despite this, the computational time to simulate a period of oscillation of the wake with the full FD-NS on the coarsest fine mesh is only 1.5 times smaller than that of the DD with four levels of refinement and the details that can be captured with the DD are way more than those that can be captured with a coarse FD-NS solver. An example of the accuracy of the captured details can be appreciated in figure 5 where the average detachment angle of the flow is plotted against a spectral element method and experimental data form [19]. The present hybrid solver fits well the other data and shows similar accuracy as the numerical method by Wu and coauthors. For the largest Re examined (Re=200), the DD overestimates the experimental data while the full FD-NS solver underestimates them. It implies a slightly lower drag force by the present solution. Moreover, the good comparison with other numerical experimental data proves that the initial condition does not influence the final-steady flow regime but only the initial evolution of the wake.

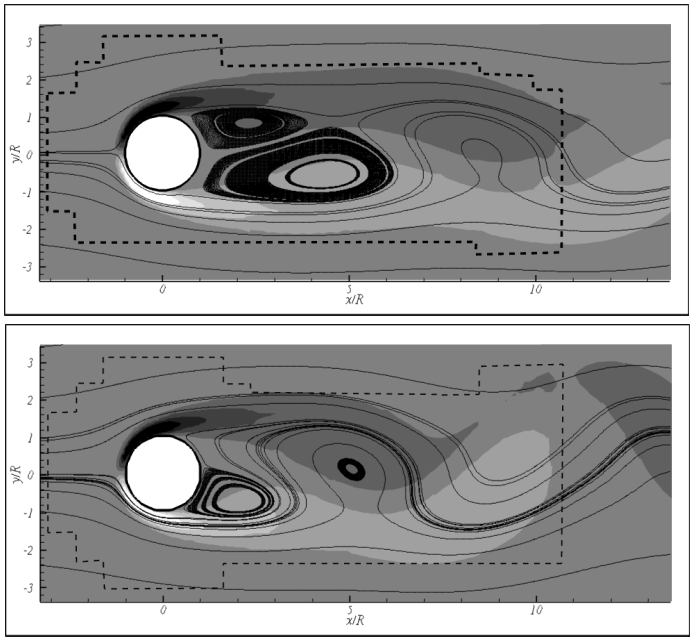

FIGURE 4. FLOW BEHIND A CYLINDER AT RE=100 FOR TWO DIFFERENT TIMES; THE DASHED LINE REPRESENTS THE BOUNDARY BETWEEN THE MRT-LBM AND THE FS-NS.

Still some limitations of the present strategy can be spotted in figure 4. There the MRT-LBM/FD-NS boundary is plotted at two time steps with a bold dashed line. The highest vorticity levels are in the MRT-LBM subdomain but a careful analysis of the vorticity contours shows some local oscillations. It is easier to spot them on the downstream vertical boundary that is kept fixed throught the computation. There, it seems that a part of the pressure signal is reflected back altering the local velocity field. The same happens on all the boundaries impinged by the vorticity but, because they are continuously changing in time, the effects appear more limited and are, in a way, smoothed out by the later evolution. This suggests that a suitable strategy must

be identified in terms of the vorticity values ω_l and ω_u and of the time-step ratio α so to minimize possible stability issues. To perform an exhaustive investigation and to ensure solution insensitivity from the numerical choices, different relevant problems must be examined within the DD strategy.

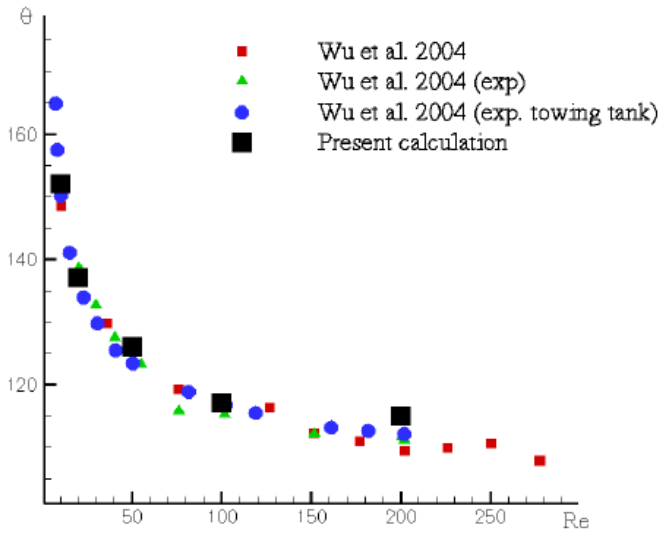

FIGURE 5. DETACHMENT ANGLE OF THE FLOW AT DIFFERENT REYNOLDS NUMBERS.

SUMMARY

An efficient and flexible Domain-Decomposition strategy has been proposed for fluid dynamic problems involving quickly-varying and slowly-varying flow regions. The hybrid solver couples a Multiple Relaxation Time Lattice Boltzmann method (MRT-LBM) with a classical Finite Difference approximation of the Navier-Stokes equations (FD-NS). The MRT-LBM solves the discrete Boltzmann equation from the statistical mechanics. It has been used more and more commonly in the last decades to simulate fluid dynamic problems. In particular, it has shown its suitability to deal with low Knudsen numbers (Kn), where the Kn is the ratio between Mach and Reynolds numbers; this has made it a valuable tool of analysis for microfluids. Moreover, the explicit time marching scheme, based on an advection and a collision step, makes it much more efficient in respect to the classical CFD solvers that have to solve algebraic systems for unsteady problems. On the other hand, the weakly compressibility nature of the method makes it less efficient to deal with almost steady fluid problems. Thus, within the DD, slowly varying regions are solved by the FD-NS. Both the solvers operate on a multi-blocks decomposition of the domain with a local mesh refinement. The DD strategy is able to deal with a dynamic change of the sub-domains topology, allowing a much more flexible tool for bodies moving freely inside the whole computational domain. All features of the method have been described. These include the adopted procedure for the dynamic change of the domains sizes, criteria for the choice of the solver, the tracking of the sub-domains boundaries and the exchange of information across the sub-domains interface. The method was then applied to the problem of a cylinder in an impulsively started current and its performance was compared against another numerical solution and experimental data. The choice of the application of the 3D solver to a 2D problem was done to make the comparison with other data easy and above all to visualize efficiently the dynamic evolution of the solvers interface which represents the key element of our proposed DD algorithm. Both the objectives were achieved.

ACKNOWLEDGMENT

This research activity is partially funded by the Research Council of Norway through the Centres of Excellence funding scheme AMOS, project number 223254, and partially by the Flagship Project RITMARE - The Italian Research for the Sea - coordinated by the Italian National Research Council and funded by the Italian Ministry of Education, University and Research within the National Research Program 2011-2013.

REFERENCES

[1] Faltinsen, O. M., and Timokha, A. N., 2009. *Sloshing*. Cambridge University Press, Cambridge, UK.

[2] Greco, M., Colicchio, G., and Faltinsen, O. M., 2009. "Bottom slamming for a very large floating structure: coupled global and slamming analyses". *J. of Fluids and Structures,* ***25***, pp. 420–430.

[3] Colicchio, G., Greco, M., and Faltinsen, O. M., 2006. "A bem-level set domain decomposition strategy for nonlinear and fragmented interfacial flows". *J. for Numerical Methods in Engineering,* ***67***(10), pp. 1385–1419.

[4] Colicchio, G., Greco, M., and Faltinsen, O. M., 2011. "Domain-decomposition strategy for marine applications with cavities entrapments". *J. of Fluids and Structures,* ***27***(4), pp. 567–585.

[5] Greco, M., Colicchio, G., Lugni, C., and Faltinsen, O. M., 2014. "A domain-decomposition strategy for a compressible multi-phase ow interacting with a structure". *Int. J. for Numerical Methods in Engineering,* ***98***(11), pp. 840–858.

[6] Modini, S., Colicchio, G., Lugni, C., and Graziani, G., 2011. "Lattice boltzmann simulation of 3d vortical flows". In Proc. of AIMETA.

[7] Kristiansen, D., and Faltinsen, O., 2012. "A lattice-boltzmann method applied to simulate the flow past a circular cylinder and a flat plate". In Proc. of the 9th European Fluid Mechanics Conference (EFMC9).

[8] Velivelli, A., and Bryden, K., 2006. "Parallel performance and accuracy of lattice boltzmann and traditional finite difference methods for solving the unsteady two-dimensional burgers' equation". *Physica A: Statistical Mechanics and its Applications,* ***362***(1), pp. 139–145.

[9] Velivelli, A., and Bryden, K., 2014. "Domain decomposition based coupling between the lattice boltzmann method and traditional cfd methods-part i: Formulation and application to the 2-d burger equation". *Advances in Engineering Software,* ***70***, pp. 104–112.

[10] dHumières, D., Ginzburg, I., Krafczyk, M., Lallemand, P., and Luo, L., 2002. "Multiple-relaxation-time lattice boltzmann models in three dimensions". *Phil. Trans. R. Soc. Lond. A,* ***360***, pp. 437–451.

[11] Qiam, Y., dHumières, D., and Lallemand, P., 1992. "Lattice bgk models for navier-stokes equations". *Europhys. Lett.,* ***17***, pp. 479–484.

[12] Chen, S., and Doolen, G., 1998. "Lattice boltzmann method for fluid flow". *Ann. Rev. Fluid Mech.,* ***30***, pp. 329–364.

[13] B.Ahrenholz, Tolke, J., and Krafczyk, M., 2005. "Lattice-boltzmann simulations in reconstructed parametrized porous media". *Int. J. of Comp. Fluid Dynm.*, pp. 1–15.

[14] MacNeice, P., Olson, K., Mobarry, C., deFainchtein, R., and Packer, C., 2000. "Paramesh : A parallel adaptive mesh refinement community toolkit.". *Computer Physics Communications,* ***126***, pp. 330–354.

[15] Olson, K., and MacNeice, P., 2005. "An over of the paramesh amr software and some of its applications",in adaptive mesh refinement-theory and applications". *Proceedings of the Chicago Workshop on Adaptive Mesh Refinement Methods, Series: Lecture Notes in Computational Science and Engineering,* ***41***.

[16] Olson, K., 2006. "Paramesh: A parallel adaptive grid tool". *Parallel Computational Fluid Dynamics 2005: Theory and Applications: Proceedings of the Parallel CFD Conference*.

[17] Colicchio, G., 2004. "Violent disturbance and fragmentation of free surfaces". PhD thesis, University of Southampton, Southampton, UK.

[18] MacNeice, P., Olson, K. M., Mobarry, C., deFainchtein, R., and Packer, C., 2000. "Paramesh : A parallel adaptive mesh refinement community toolkit.". *Computer Physics Communications,* ***126***, pp. 330–354.

[19] Wu, M., Wen, C., Yen, R., Weng, M., and Wang, A., 2004. "Experimental and numerical study of the separation angle for flow around a circular cylinder at low reynolds number". *Journal of Fluid Mechanics,* ***515***, pp. 233–260.