Simulating an exterior domain for drag force computations in the lattice Boltzmann method

Jonas Latt [a], Yannick Grillet [b], Bastien Chopard [a], Peter Wittwer [c]

[a] *Computer Science Department, University of Geneva, Switzerland*

[b] *Engineering School of Geneva, HES-SO, Switzerland*

[c] *Physics Department, University of Geneva, Switzerland*

Abstract

The simulation of a stationary fluid flow past an obstacle by means of a lattice Boltzmann method is discussed. The problem of finding appropriate boundary conditions on the boundaries of the truncated numerical domain is addressed by a method recently discussed in the literature, based on a truncated expansion of the solution. The iterative process at the heart of this method is coupled with the iteration steps of a progressive grid refinement technique that allows a rapid convergence towards a well resolved stationary state. It is shown that this combination results in a highly efficient numerical tool which can speed up the resolution process in a substantial manner.

Key words: Drag force computation, artificial boundary condition, lattice Boltzmann method, nested grid technique

1 Introduction

We consider the task of simulating a stationary incompressible fluid flow past a rigid obstacle. The fluid is assumed to fill the whole two-dimensional (2D) or three-dimensional (3D) space, the obstacle is placed at the origin of the coordinate system, and the fluid velocity is asymptotically constant far from

Email address: `latt@cui.unige.ch` (Jonas Latt).

Preprint submitted to Elsevier Science *31 October 2005*

the obstacle. Thus, the problem consists in the resolution of the stationary Navier-Stokes equation for the velocity field $\vec{u}(\vec{x})$ with boundary condition

$$\lim_{|\vec{x}|\to\infty} \vec{u}(\vec{x}) = \vec{u}_0. \tag{1}$$

A major difficulty of this problem stems from the necessity to truncate the infinite domain for numerical purposes, and to find an artificial boundary condition for the boundaries of the truncated domain. A straightforward approach consists in using the asymptotic condition $\vec{u} = \vec{u}_0$ on the numerical domain boundaries. Although this method is easy to implement, it appears to be quite inappropriate for the needs of numerical modeling, as it requires the use of excessively large domains. Indeed, it will be shown in the present paper that the structure of the flow is strongly influenced by the shape of the obstacle even far from the center. Other approaches to this problem use extrapolation schemes on the boundaries so as to ensure a vanishing gradient perpendicular to the boundaries, for the velocity or other physically relevant quantities. The drawback of those approaches is that they are insufficient for imposing the asymptotic velocity $\vec{u}_0$ on the fluid and therefore cannot be used on all boundaries. Furthermore, they make it difficult to ensure conservation of mass and momentum across domain boundaries.

For those reasons, we introduce an alternative technique that has been described recently in the literature [2,1,8]. In this method, an explicit vector field is proposed that can be used to implement a Dirichlet boundary condition for the fluid velocity in a region reasonably far from the center. The expression for this vector field is obtained from a truncated asymptotic development of a solution to the stationary Navier-Stokes equation and approximates the structure of the flow with considerably higher precision than the constant approximation $\vec{u} = \vec{u}_0$. The drawback of this method is that it depends on the drag and lift coefficients of the obstacle which are *a priori* unknown. Therefore, the solution process involves a series of iteration steps during which the formula of the boundary condition is updated on ground of the drag coefficients of the obstacle measured at this state of the simulation. A brief overview of the method is found in Section 2.

The numerical solver used for the present work uses a discrete kinetic scheme known as lattice Boltzmann (LB) BGK method [7,3,4]. Compared to traditional solvers that simulate the dynamics of the hydrodynamic quantities velocity and pressure, LB methods describe the evolution of kinetic quantities, the particle distribution functions in phase space $f(\vec{x}, \vec{v}, t)$. They have been successfully used in the past decades both for the numerical solution of the plain Navier-Stokes equation and for the simulation of complex fluids such as multiphase or multiscale physics. Note that the BGK variant of the LB methods simulates the evolution of a compressible fluid, but does not offer a way to tune the bulk viscosity and the equation of state of the fluid. It is often used in a low Mach number regime for the simulation of incompressible fluids.

The present paper contains a case study for the numerical evaluation of a drag coefficient, and serves three main purposes. First, it presents an introduction to the boundary condition of Ref. [2,1,8] and demonstrates its efficiency and simplicity in the context of LB simulations. Second, it shows that this method can be coupled with a numerical technique based on iterative grid refinement. Finally, it is argued that although the boundary condition of Ref. [2,1,8] has been developed for incompressible flows, it also proves useful for simulations of compressible flows at low Mach numbers. For further precision, the influence of the fluid compressibility on the computation of a drag force is analyzed.

2 Boundary condition

In Ref. [1], the solution to the 2D incompressible Navier-Stokes equation is expanded in a finite series, as a function of formal parameters depending on the drag and lift coefficients of the obstacle. The corresponding theory for the 3D case is presented in Ref. [8]. It is recognized that at a certain distance from the center, the structure of the flow doesn't depend for the specific details of the obstacle geometry, but only on 2 (2D) or 3 (3D) distinct parameters. These considerations result in the prescription of an explicit vector field that can be used as a boundary condition on the numerical domain boundaries. The zeroth- and first-order terms of the expansion for a 2D flow on the velocity field $\vec{u} = (u, v)$ read

$$
\begin{aligned}
u(x,y) &= u_\infty \left(1 + l\frac{d}{\pi}\frac{x}{x^2+y^2} + l\frac{b}{\pi}\frac{y}{x^2+y^2} - \theta(x)\sqrt{l}\frac{d}{\sqrt{\pi}}\frac{1}{\sqrt{x}}e^{-\frac{y^2}{4lx}}\right), \qquad (2)\\
v(x,y) &= u_\infty \left(l\frac{d}{\pi}\frac{y}{x^2+y^2} - l\frac{b}{\pi}\frac{x}{x^2+y^2} - \theta(x)\sqrt{l}\frac{d}{2\sqrt{\pi}}\frac{y}{x^{3/2}}e^{-\frac{y^2}{4lx}}\right)
\end{aligned}
$$

where $d = F_x/(2\rho l u_\infty^2)$ and $b = F_y/(2\rho l u_\infty^2)$ are the drag and the lift coefficient, and $l = \nu/u_\infty$ is the viscous length, dependent on the dynamic fluid viscosity ν. The formula also uses the Heaviside function $\theta(x)$, defined as $\theta(x) = 1$ for $x > 0$ and $\theta(x) = 0$ for $x < 0$. Without loss of generality, the asymptotic velocity $\vec{u}_\infty = (u_\infty, 0)$ has been taken to be parallel to the x-axis. This boundary condition is implicit in the sense that it depends on two constants, d and b that are in general unknown before the execution of the simulation.

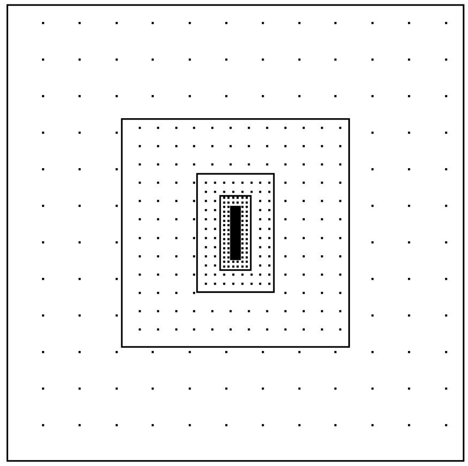

Fig. 1. Structure of the numerical grid close to a rectangular obstacle. One dot on the figure represents a square of 8×8 grid nodes.

3 Numerical solver

The numerical solver that has been used runs on a structured, homogeneous grid. However, the problem under consideration requires a higher resolution of the grid close to the center, given that in this region, the fluid is subject to sharp pressure and velocity variations. Therefore, a grid refinement technique, such as the one presented in Ref. [5] is applied with a hierarchy of nested grids that have a successively finer resolution as they approach the system center (*cf.* Fig. 1). Those nested grids are deployed progressively in such a way as to speed up the convergence of the simulation towards a stationary state. At first, the simulation is run on a small domain close to the obstacle. Then, at chosen time intervals, the size of the physical domain represented in the simulation is enlarged by implementing a new, coarser grid. This is a convenient way of doing to turn the solver for the dynamic Navier-Stokes equation into an efficient solver for the stationary state.

By now, the presented numerical approach contains two separate iterative processes, one for the implementation of the boundary condition, as explained in Section 2, and one for the convergence to the stationary state, as explained in the current section. It is observed that both processes take place at comparable time scales, and can therefore be coupled in a simple manner. In the present simulations, the drag and lift coefficients needed for the implementation of the boundary condition are updated once at each grid enlargement. These two parameters are obtained from the numerical data by evaluating the exchange of momentum along the surface of the obstacle.

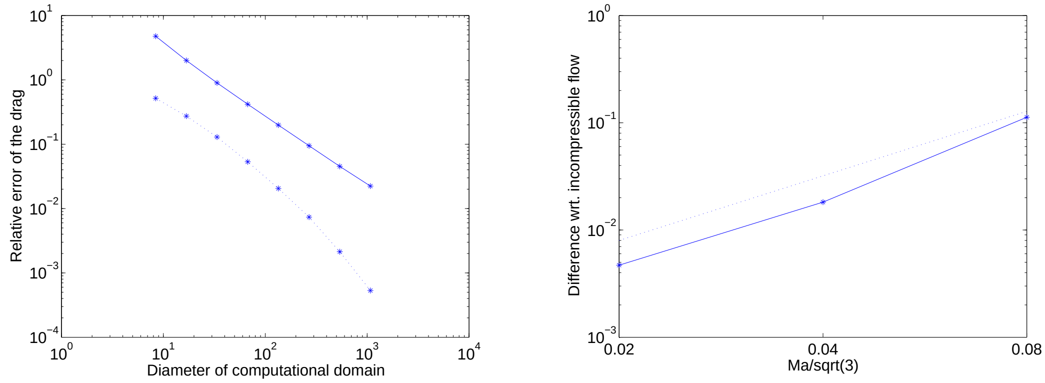

Fig. 2. (left) Drag coefficient as a function of the system size. The x-axis represents the ratio between the system size and the height of the obstacle. Solid line: asymptotic boundary condition $\vec{u} = \vec{u}_0$. Dotted line: first-order accurate boundary condition described by Eq. (2). (right) Solid line: drag coefficient, function of the Mach number. Dotted line: reference curve, representing a power law with slope -2.

4 Simulation results

For illustration purposes, we present the simulation results of a 2D flow across a rectangular obstacle with a ratio of the width to the height of 5 : 1, as it is shown of Fig. 1. The simulations are run on quadratic domains of varying size, up to three orders of magnitude larger than the obstacle. The Reynolds number $Re = A/l$, defined with respect to the height A of the obstacle, is fixed at $Re = 1$, and the Mach number at $Ma = 0.02 \cdot \sqrt{3}$. On the domain boundaries, both the constant boundary condition $\vec{u} = \vec{u}_0$, and the boundary condition described in Section 2 are tested. The measured drag coefficient d is plotted on the left part of Fig. 2 as a function of the domain size, compared to a reference solution $d = -5.0268$ that was computed on a substantially larger domain. This figure shows that the accuracy of the drag coefficient converges approximately at the same rate in the presence of either boundary condition. However, using the boundary condition described by Eq. (2) gains almost an order of magnitude on the accuracy of the drag coefficient, independently of the system size.

As a conclusion, the combination of a nested grid technique with an appropriate boundary condition results in an efficient numerical scheme. Indeed, the computation of drag forces as those shown on the left part of Fig. (2) were performed within one day on a common desktop computer. It must also be noted that at the high level of accuracy that is applied here, compressibility effects play an important role. This statement is quantified by a plot on the right part of Fig. 2, which shows the difference between the drag force exerted by a compressible fluid and a reference solution for an incompressible fluid found in Ref. [1]. As it is expected, this difference decreases at roughly a second order rate with respect to the Mach number. It is however not possible to implement fluids at an arbitrary small Mach number in the LB BGK model used here, as the choice of this parameter is limited by CPU time constraints. To complete the picture, it would therefore be interesting to reproduce the simulations on a LB model for incompressible flows (*e.g.* Ref. [9]), or a more appropriate LB model for compressible flows akin to the one presented in Ref. [6], in which the bulk viscosity of the fluid can be varied.

5 Conclusion

A numerical method has been introduced for the computation of the flow in an exterior domain and the evaluation of drag coefficients, based on a LB method. A 2D numerical study has been presented to discuss the influence of the choice of the boundary condition and the fluid compressibility on the results of the simulation. We particularly conclude that the boundary condition recently exposed in Ref. [2,1,8] leads to highly accurate results and can easily be combined with a nested grid technique within the LB framework.

Acknowledgment. This work was supported by the Swiss National Science Foundation (SNF).

References

[1] S. Bönisch, V. Heuveline, and P. Wittwer. Submitted to J. Fluid Mech., 2004.

[2] S. Bönisch, V. Heuveline, and P. Wittwer. *J. Fluid Mech.*, 7:87–107, 2005.

[3] B. Chopard and M. Droz. *Cellular Automata Modeling of Physical Systems*. Cambridge University Press, 1998.

[4] B. Chopard, P. Luthi, A. Masselot, and A. Dupuis. *Advances in Complex Systems*, 5:103, 2002. http://cui.unige.ch/∼chopard/FTP/CA/acs.pdf.

[5] A. Dupuis and B. Chopard. Theory and applications of alternative lattice boltzmann grid refinement algorithm. *Phys. Rev. E*, 67:066707, 2003.

[6] A. J. S. Ladd and R. Verberg. *J. Stat. Phys.*, 104:1191, 2001.

[7] S. Succi. *The Lattice Boltzmann Equation, For Fluid Dynamics and Beyond*. Oxford University Press, 2001.

[8] P. Wittwer. *J. Math. Fluid Mech.*, 2005.

[9] Q.S. Zou, S.L. Hou, S.Y. Chen, and G.D. Doolen. *J. Stat. Phys.*, 81:35–48, 1995.