Tuning a lattice-Boltzmann model for applications in computational hemodynamics

D.R. Golbert\textsuperscript{a,b,*}, P.J. Blanco\textsuperscript{a,b}, A. Clausse\textsuperscript{c}, R.A. Feijóo\textsuperscript{a,b}

\textsuperscript{a} LNCC, Laboratório Nacional de Computação Científica, Av. Getúlio Vargas 333, Quitandinha, 25651–075 Petrópolis, RJ, Brazil
\textsuperscript{b} Instituto Nacional de Ciência e Tecnologia em Medicina Assistida por Computação Científica, Petrópolis, RJ, Brazil
\textsuperscript{c} Universidad Nacional del Centro de la Provincia de Buenos Aires, Tandil, Argentina

\begin{abstract}
The interest in lattice-Boltzmann models in the computational hemodynamics realm has increased in recent years. In this context, the correct choice of numerical parameters for the appropriate simulation of blood flows in major arteries is a crucial aspect. For this reason, we present three parameter-tuning strategies that allow us to reproduce correctly the pulsatile time-dependent flow of an incompressible fluid under physiological regimes. These strategies are studied for a model based on a single-relaxation-time approach in combination with second order boundary conditions for both velocity and pressure, and proper equilibrium distributions that take care of the incompressible behavior exhibited by the fluid. The implementation is validated with the three-dimensional Womersley flow benchmark. As well, the simulation of blood flows in a curved artery, in an anastomosed vessel, in a patient specific vertebral artery and in an aneurysmal region are presented in order to show how the method and the setting of the numerical parameters are applied to different realistic hemodynamics problems.

© 2011 IPEM. Published by Elsevier Ltd. All rights reserved.
\end{abstract}

1. Introduction

Since its introduction in the 80s the lattice-Boltzmann method (LBM) has overcome several refinements and extensions, and has become a promising numerical scheme for simulating complex fluid dynamics in the most varied applications. In particular, LBM has been systematically employed, during the last few years, in several computational hemodynamics applications. Some examples are the simulation of blood flows in heart valves [1,2], in coronaries [3], in aneurysms [4–6] and in the abdominal aorta [7], among others.

Differing from the conventional methods, based on the discretization of continuum macroscopic Navier–Stokes equations, LBM is a mesoscopic particle based method derived from the lattice–gas cellular automata method (LGCA) [8] and the Boltzmann equation [9]. Its basic idea consists in building a simplified kinetic model that incorporates the essential physics of microscopic processes, such that the macroscopic properties of the system are governed by a certain set of equations.

A key point when implementing the LBM for fluid flow simulations is the careful choice of numerical parameters, especially in time dependent problems, so that one can obtain good approximations of the physical phenomena and predict accurately the fluid behavior for different flow regimes. Particularly, in hemodynamics applications this turns out to be crucial due to (i) the characteristic physiological regimes of blood flow in the arteries and (ii) the geometrical features of the spatial domains in which the simulations are to be set up (high aspect ratio).

Previous contributions that dealt with the characterization, to some extent, of the main variables in the LBM are in Refs. [10,11]. Particularly, in Ref. [10] a complete study of the impact of the numerical parameters and their influence in the final solution is performed. Nevertheless, the precise characterization of such parameters in blood flow problems, taking into account the peculiarities of the hemodynamics field, is still missing. With this motivation in mind, the contribution of the present work is to perform a study of some relevant non-dimensional numbers, in order to provide clear orientation concerning the setting of the numerical parameters in hemodynamics simulations. In this framework, we also apply these parameter-tuning strategies to different applications of relevance in computational hemodynamics.

The structure of the paper is as follows. In Section 2, the governing equations of LBM with BGK approximation and the chosen methods for implementing boundary conditions are presented. In Section 3, an analysis of the parameters of the method and of...
the proposed parameter-tuning strategies is performed. In Section 4, we present the numerical validation and some study cases of the Womersley flow as well as an analysis of the results obtained from the simulation of blood flow in four different hemodynamic scenarios. More specifically, the blood flow in a curved artery, in an anastomosed vessel, in a vertebral artery and in an aneurysmal region are studied. This last problem is also used to study the relation between the main physical quantities that govern the characteristics of the pulsatile flow and its behavior. Finally, the conclusions are outlined in Section 5.

2. Methods

From the numerical point of view, the lattice-Boltzmann Method with BGK approximation (referring to the work of Ref. [12]) is an explicit method. It is based on the movement and collision of micro-particles distributions described by the lattice-Boltzmann equation (LBE):

\[
f_j(\vec{x} + \Delta\vec{e}_i, t + \Delta t) - f_j(\vec{x}, t) = \frac{1}{\tau} [f_j^{eq}(\vec{x}, t) - f_j(\vec{x}, t)], \quad i = 0, \ldots, \ell,
\]

where, \(f_j(\vec{x}, t)\) represents the micro-particles distribution density at position \(\vec{x}\) and time \(t\), moving towards the direction \(\vec{e}_i\) of the lattice, \(\Delta x\) is the lattice spacing, \(\Delta t\) is the time step and \(\ell\) is the dimension of the discrete velocity space. The relaxation parameter \(\tau\) is related to the kinetic viscosity of the fluid \(\nu\) through the expression: \(\nu = (2\tau - 1)\Delta x^2/(6\Delta t)\). It has been demonstrated that the LBE approximates the Navier-Stokes equations by means of an asymptotic expansion [13].

The equilibrium distribution chosen for this work was introduced by Ref. [13], and is devised to minimize the compressibility effects of the method. It is described as:

\[
f_j^{eq} = \omega_i \left\{ \rho + \rho_0 \left[ \frac{3 (\vec{v} \cdot \vec{u})}{v^2} + \frac{9 (\vec{v} \cdot \vec{u})^2}{v^4} - \frac{3 (\vec{u} \cdot \vec{u})}{v^2} \right] \right\}, \quad i = 0, \ldots, \ell,
\]

where, \(\nu = \Delta x/\Delta t\) is the particles speed, the weights \(\omega_i\) are lattice dependent, \(\rho_0\) is the average density (constant), \(\rho\) and \(u\) are the density and velocity of the fluid, calculated from the distribution density as:

\[
\rho(\vec{x}, t) = \sum_{i=0}^{\ell} f_i(\vec{x}, t) \quad \text{and} \quad \rho_0\vec{u}(\vec{x}, t) = \sum_{i=0}^{\ell} \vec{v}_i f_i(\vec{x}, t).
\]

In this model, the normalized pressure is calculated as function of the density: \(P = c_s^2 \rho/\rho_0\), where \(c_s = \sqrt{\nu/3}\) is the speed of sound in the lattice.

The characteristic directions of the three-dimensional lattice model D3Q19, implemented in this work, are (see Fig. 1): \(e_0 = (0, 0, 0), e_1 = (1, 0, 0), e_2 = (-1, 0, 0), e_3 = (0, 1, 0), e_4 = (-1, 1, 0), e_5 = (0, -1, 0), e_6 = (0, 0, 1), e_7 = (1, 0, 0), e_8 = (-1, 1, 0), e_9 = (0, -1, 0), e_{10} = (1, 0, 0), e_{11} = (0, -1, 0), e_{12} = (-1, 1, 0), e_{13} = (-1, 0, 1), e_{14} = (0, 1, 0), e_{15} = (0, 0, 1), e_{16} = (0, 1, 0), e_{17} = (-1, 0, 1), e_{18} = (0, -1, 0). This model allows particles to stand still or to move in other 18 directions. The mentioned weights for the equilibrium distribution (see Eq. (2)) of this lattice model are: \(\omega_0 = 1/3, \omega_{1, 4} = 1/18, \omega_{7, 18} = 1/36\).

In hemodynamics simulations it is a common practice to impose the value of the pressure at the inlets/outlets and a no-slip condition at the arterial walls. Thus, in order to impose the pressure boundary conditions, we adopted the model propose by Ref. [14], designed for flat surfaces. This model makes use of the conservation Eq. (3) and the reflection of some symmetric non-equilibrium distributions \(f_{i, \text{ref}} = f_i - f_i^{eq}\) to impose the desired pressure. In addition, we chose an interpolation model, proposed by Ref. [15], to impose the boundary conditions for the velocity field on curved surfaces. The use of an interpolation model is interesting for imposing the boundary conditions on surfaces that do not cross the lattice nodes exactly, as in the case of arbitrarily curved arterial walls.

3. Parameters tuning

Like most numerical approximations, LBM requires a careful selection of the numerical parameters to optimize the trade-off between the computational cost and the accuracy of the results. In order to achieve accurate results, one has to choose (estimate) the parameters accordingly and, after that, modify (tune) them, if needed.

The main control parameter is the relaxation parameter, denoted by \(\tau\), which determines the numerical stability of the scheme. The minimum acceptable value of \(\tau\) is problem dependent. We obtained stable simulations in all tested cases with \(\tau > 0.52\). It is important to keep in mind that this parameter decreases as \(\nu\) and \(\Delta t\) decrease and \(\Delta x\) increases, hence the refinement of the lattice is sometimes necessary to increase \(\tau\). Moreover, the lattice has to be fine enough to capture the spatial gradients of the solution. On the other hand, it is important to keep the Mach number (\(\text{Mach} = u_{\text{max}}/c_s\), where \(u_{\text{max}}\) is the maximum physical fluid speed) as small as possible, since the correct representation of the Navier–Stokes equations is based upon this assumption (see again Ref. [13]). Usually, keeping Mach smaller than 0.15 is enough to ensure that the moments of the micro-particles distribution reflect the macroscopic behavior of incompressible fluids.

In the context of time-dependent problems, the so-called characteristic time \(T_c\), which could be an oscillation period in the case of time periodic problems, has to be much bigger than the propagation time \((L/c_s)\) of a sound signal over the characteristic length \(L\) of the problem, so that no spurious compressibility effects take place. We will refer to this relation as \(\theta_t = T_c/(L/c_s)\). Another hypothesis of the asymptotic expansion of the lattice-Boltzmann equation that leads to the Navier–Stokes equations [13] is based on the assumption of high value of \(\theta_t\). This is a crucial issue in computational hemodynamics because of the geometric features of the spatial domains in which the numerical simulations are carried out. Therefore, precise orientation concerning this value is of the utmost importance for our purposes. Our numerical experiments indicate that \(\theta_t\) should
be larger than 30. Hence, in all our applications this is the threshold that constraints the setting of the numerical parameters.

The parameter tuning is performed based on adjusting the time step and the lattice spacing. In Table 1 we summarize the effect of three simple tuning strategies on the more relevant parameters in a time-dependent problem, with constant physical values (viscosity, dimensions and others). This table is based on the one presented by Ref. [10], with modifications on the strategies and on the parameters analyzed. The strategies consist in reducing the time step, refining the lattice or doing both. In this table, the symbol apostrophe (’) indicates the relative change of a parameter, \( \tau_r = \tau - 0.5 \), \( N_T \) and “cost” represent the number of time steps and a relative computational cost, respectively, of the simulation of a fixed time period (say a cycle) and (’) indicates the 3D case (shown only when the number differs from the 2D case). This table shows that reducing the time step has a small cost and has the benefits of reducing the Mach number and increasing \( \theta_t \), but it may lead to instabilities with the decrease of \( \tau_r \). This can be reverted by refining the lattice, at a greater computational cost. But notice that refining the lattice without reducing the time step along is not convenient, because it leads to worsening the values of \( \theta_t \) and of the Mach number, in the sense that they become smaller and greater, respectively. In the next section, one can observe some examples that make use of these strategies.

### 4. Numerical results

This section starts with the study of a benchmark problem for validation and analysis purposes. In the sequence, the implementation and the numerical results of four situations of interest in computational hemodynamics are analyzed. More specifically in the Section 4.2 we reproduce a numerical simulation (proposed in Ref. [16]) of a pulsatile blood flow in a curved cylindrical artery to test the application of the method in a problem with high aspect ratio, Reynolds and Womersley numbers; in Section 4.3 we reproduce numerically a steady state in vitro experiment of the blood flow in an anastomosed vessel to qualitatively validate the obtained results; in Section 4.4 we simulate the blood flow in patient specific vertebral artery and compare this results with the ones retrieved from a validated finite element method code; in Section 4.5 we analyze the behavior of a hemodynamic quantity in a set of simulations of the blood flow in a cylindrical artery with an aneurysm. In these four examples the blood is modeled as a Newtonian fluid with kinematic viscosity of 0.035 cm²/s. This approach is generally considered valid in hemodynamics of large arteries, see for example Refs. [4,5].

#### 4.1. Analysis and characterization of \( \theta_t \) using the Womersley flow

The Womersley flow is a well-known benchmark that consists in the flow of an incompressible fluid confined in a cylindrical tube with a reference pressure, say \( p_{\text{out}} \), on the right extremity and an oscillatory pressure, say \( p_{\text{in}} \), on the left one. This problem has analytical solution [17] for an oscillatory pressure gradient described by the function:

\[
\Delta p(t) = \frac{p_{\text{out}} - p_{\text{in}}(t)}{L_s} = -A \cos \left( \frac{2\pi t}{T} \right). \tag{4}
\]

The described problem is characterized by the non-dimensional Womersley number, calculated as \( Wo = R \sqrt{2\pi / (Tv)} \), where \( R \) is the radius of the tube. In the present work, several cases, with Womersley numbers ranging from 3.5 to 20, are simulated, considering a fixed Reynolds number (\( Re = u_{\text{max}} 2R / \nu \)) equal to 1 (as the solution is only scaled by the Reynolds number). The solutions are obtained using the lattice-Boltzmann model presented in Section 2 and studying the sensitivity of the solution following the strategies discussed in Section 3. In Fig. 2 the velocity profiles obtained with the LBM simulations are displayed (dashed black lines) for two regimes and at different time instants. The figure on the left corresponds to \( Wo = 3.5 \), and was computed with a lattice of size \( 64 \times 32 \times 32 \), while the figure on the right corresponds to \( Wo = 20 \) on a lattice of size \( 256 \times 128 \times 128 \). Note the good agreement between the numerical solutions and the analytical solutions (colored lines). As this problems were defined based on dimensionless quantities the velocity and the diameter scales where set to 1 for simplicity.

In order to retrieve as much information as possible from the simulations and from the tuning process we construct Table 2 using a test case with \( Wo = 3.5 \). Such table presents the behavior of the relative error with respect to the analytical solution (denoted by \( e_\text{c} \), calculated according to the following norm:

\[
||A - B||_2 = \sqrt{\sum_{i=1}^{N} (A_i - B_i)^2 / \rho_i^2}, \tag{5}
\]

where \( A_i^2 = (A_i \cdot A_i) \), \( N \) is the number of nodes of the lattice and \( A_i \) and \( B_i \) are the values of the vector fields \( A \) and \( B \) at node \( i \), respectively. As well, the table shows the error due to compressibility effects (denoted by \( e_\text{c} \)), measured as the difference in the flow rate between inlet and outlet, with respect to the inlet flow. Each row in the table stands for a modification either in the time step (where \( N_T = T/\Delta t \)) or in the lattice size. As expected, in some cases the sole reduction of the time step is not enough to reduce the relative error, although it is the way to minimize the compressibility error. In such situations a refinement of the lattice becomes necessary, at a higher computational cost, which implies a further reduction in the time step in order to accommodate the compressibility error to smaller values. In addition, when recomputing the correctly tuned cases with a Reynolds number of 300 we checked that the errors were of the same order, as the Mach number was still small and the \( \tau \) number was kept the same.

This problem is also used to analyze the sensitivity of the compressibility error with respect to the aspect ratio of the tube (length/diameter) and to the quantity \( \theta_t \). This study is developed for a fixed Womersley number of 3.5 and aspect ratios ranging from 2/1 to 16/1. We started the simulations with 1312 time steps per cycle and doubled this quantity until the compressibility error reaches a value close to 1%. In Table 3, the compressibility error and the dimensionless quantity \( \theta_t \) are shown for all cases. Here the symbol “=“ indicates an unstable simulation and “−" indicates a case not simulated. From this table one can see that there is an inverse direct relation between the quantity \( \theta_t \) and the compressibility error, as expected. The most interesting conclusion here is the fact that, for this problem, with \( \theta_t \approx 50 \) it is possible to guarantee that the compressibility errors are of the order of 1%. But, in the following examples, we will use \( \theta_t = 30 \) and achieve compressibility errors smaller than 2%.

#### 4.2. Curved artery

The second problem was extracted from Ref. [16] and consists of modeling the blood flow in a cylindrical curved artery with a high length/diameter ratio of 18/1 (see Fig. 3). The flow is driven by a pulsatile inflow condition at the inlet (superior extremity).
Table 1
Strategies to tune a transient simulation and their effect on main the parameters. Here, the symbol apostrophe (’) indicates the relative change of a parameter, \( N_t \) represents the number of time steps of a fixed time period and ( ) indicates the 3D case.

<table>
<thead>
<tr>
<th>Strategy</th>
<th>( N_t' )</th>
<th>( \Delta t' )</th>
<th>Nodes’</th>
<th>( \Delta x' )</th>
<th>( \theta_x' )</th>
<th>( r_e' )</th>
<th>Mach’</th>
<th>cost’</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reduce ( \Delta t )</td>
<td>( n )</td>
<td>( 1/n )</td>
<td>1</td>
<td>1</td>
<td>( n )</td>
<td>( 1/n )</td>
<td>( 1/n )</td>
<td>( n )</td>
</tr>
<tr>
<td>Refine lattice</td>
<td>1</td>
<td>1</td>
<td>( n^2 \ (n^4) )</td>
<td>( 1/n )</td>
<td>( 1/n )</td>
<td>( n^2 )</td>
<td>( n )</td>
<td>( n^2 \ (n^4) )</td>
</tr>
<tr>
<td>Do both</td>
<td>( n )</td>
<td>( 1/n )</td>
<td>( n^2 \ (n^4) )</td>
<td>( 1/n )</td>
<td>1</td>
<td>( n )</td>
<td>1</td>
<td>( n^2 \ (n^4) )</td>
</tr>
</tbody>
</table>

Fig. 2. Velocity profiles along a cycle obtained with the LBM simulations (dashed black lines) and with the analytical solutions (colored lines). The left subfigure represents \( Wo = 3.5 \) and the right one \( Wo = 20 \). The aspect ratio of the tube is 2/1. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of the article.)

Table 2
Relative error with respect to the analytical solution and compressibility error of a study case with \( Wo = 3.5 \).

<table>
<thead>
<tr>
<th>Lattice size</th>
<th>( N_t )</th>
<th>Cost</th>
<th>( \tau )</th>
<th>( e_r )</th>
<th>( e_v )</th>
<th>( e_\theta )</th>
</tr>
</thead>
<tbody>
<tr>
<td>64 ( \times ) 32 ( \times ) 32</td>
<td>1312</td>
<td>1</td>
<td>0.8</td>
<td>7.66%</td>
<td>16.7%</td>
<td></td>
</tr>
<tr>
<td>64 ( \times ) 32 ( \times ) 32</td>
<td>5248</td>
<td>4</td>
<td>0.575</td>
<td>2.68%</td>
<td>0.88%</td>
<td></td>
</tr>
<tr>
<td>64 ( \times ) 32 ( \times ) 32</td>
<td>10496</td>
<td>8</td>
<td>0.5375</td>
<td>2.85%</td>
<td>0.24%</td>
<td></td>
</tr>
<tr>
<td>64 ( \times ) 32 ( \times ) 32</td>
<td>20992</td>
<td>16</td>
<td>0.51875</td>
<td>Unstable</td>
<td>Unstable</td>
<td></td>
</tr>
<tr>
<td>128 ( \times ) 64 ( \times ) 64</td>
<td>5248</td>
<td>32</td>
<td>0.8</td>
<td>1.90%</td>
<td>3.64%</td>
<td></td>
</tr>
<tr>
<td>128 ( \times ) 64 ( \times ) 64</td>
<td>10496</td>
<td>64</td>
<td>0.65</td>
<td>0.89%</td>
<td>0.88%</td>
<td></td>
</tr>
<tr>
<td>128 ( \times ) 64 ( \times ) 64</td>
<td>20992</td>
<td>128</td>
<td>0.575</td>
<td>0.85%</td>
<td>0.22%</td>
<td></td>
</tr>
</tbody>
</table>

The inflow condition is imposed by considering a parabolic velocity profile and has a mean velocity profile given by:

\[
\tilde{\mathbf{u}}_{in}(t) = U \left[ 1 - \cos \left( \frac{2\pi t}{T} \right) \right],
\]

where \( T = 0.75 \text{ s} \) represents the period of the pulse and \( U = 16.9 \text{ cm/s} \) is the averaged fluid velocity at the inlet along a period (as proposed in Ref. [16]). On the other extremity of the geometry, a constant reference pressure is imposed. From these parameters, we conclude that this problem is characterized by a Womersley number of 4.8 and a Reynolds number of 900. The high length/diameter ratio requires a very refined lattice, with approximately 90 nodes over the artery diameter (more than 11 million nodes in total), in order to keep the relation \( T/(L/c_t) \) above 30 and, therewith, the compressibility error is lower than 2%. The parameters used were \( \tau = 0.5219 \) and \( N_t = 75760 \).

In Fig. 4, the blood flow behavior is shown in detail. The left part of the figure presents the velocity profiles along the artery at the top speed. Here we notice that the parabolic profile is distorted along the bend and recovered at the outlet. The right part of the same figure features the characteristics of the secondary flow in the bend at the same instant.

Another complementary characterization of the flow behavior is given in Fig. 5, which shows the velocity profiles along a mid longitudinal section. These results are in good agreement with the reference solution of Ref. [16] obtained from a finite element method implementation, giving more evidences of the capabilities of a lattice-Boltzmann simulation when the method is suitably tuned.

4.3. Anastomosed vessel

The goal of this example is to simulate the hemodynamics of an anastomosed vessel and compare it to two in vitro experiments presented by Ref. [18]. This situation is of high interest in the medical

Table 3
Compressibility errors of simulations in pipes with different aspect ratios.

<table>
<thead>
<tr>
<th>( N_t/1312 )</th>
<th>1</th>
<th>2</th>
<th>4</th>
<th>8</th>
<th>16</th>
<th>32</th>
</tr>
</thead>
<tbody>
<tr>
<td>A.R.</td>
<td>( e_r )</td>
<td>( \theta_x )</td>
<td>( e_r )</td>
<td>( \theta_x )</td>
<td>( e_r )</td>
<td>( \theta_x )</td>
</tr>
<tr>
<td>1/1</td>
<td>2.91%</td>
<td>24</td>
<td>0.70%</td>
<td>48</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>2/1</td>
<td>16.7%</td>
<td>12</td>
<td>3.67%</td>
<td>24</td>
<td>0.88%</td>
<td>48</td>
</tr>
<tr>
<td>4/1</td>
<td>166%</td>
<td>6</td>
<td>18.5%</td>
<td>12</td>
<td>4.00%</td>
<td>24</td>
</tr>
<tr>
<td>8/1</td>
<td>*</td>
<td>3</td>
<td>168%</td>
<td>6</td>
<td>19.1%</td>
<td>12</td>
</tr>
<tr>
<td>16/1</td>
<td>*</td>
<td>1.5</td>
<td>*</td>
<td>3</td>
<td>167%</td>
<td>6</td>
</tr>
</tbody>
</table>
community due to the impact of the procedure followed to perform the anastomosis (connecting angle, among other factors) in the hemodynamics and further re-vascularization after such intervention. Fig. 6 shows in blue the geometry of the problems proposed by Ref. [18], and in green the outflow vessel included in the present study to impose outflow pressure boundary conditions ($p_{out}$) in a plane orthogonal to a Cartesian axis (as the pressure imposition method requires). Is this case, we are interested in studying the steady-state flow when a parabolic velocity profile is imposed at the inlet ($u_0$ at the center).

Fig. 7 shows the velocity profiles along the channel for Reynolds numbers 299 and 564, noting that the parabolic profile is recovered before the bend, in agreement with Ref. [18]. Also, Fig. 8 compares the numerical and experimental streamlines (the latter taken from Ref. [18] with permission of ASME) at the mid longitudinal section. It can be seen that the results are qualitatively similar, as the main vortex structures observed in the experiments are reproduced by the lattice-Boltzmann simulation. More specifically, we notice that not only the two vortices are reproduced, but, as in the experiment, they expand and move rightwards as the Reynolds number increases. Moreover, in the upper branch the simulation renders the same streamline patterns as in the experiment for the two tested Reynolds numbers. The simulations with Reynolds numbers 299 and 564 were respectively performed using 50 and 60 nodes over the arterial diameter and setting parameters such that $\tau = 0.53$ and $\tau = 0.5255$, which results in $Mach < 0.14$ for both cases. It should be stressed that, as in the previous cases, the strategies discussed in Section 4.1 were followed to calibrate the numerical parameters.

4.4. Physiological blood flow at the vertebral artery

This case represents a physiological simulation of the blood flow in a segment of a patient specific left vertebral artery. In this geometry (extracted from a 3D volume dataset whose source is a DICOM image), we impose physiological pressure data (retrieved from the numerical results given by the simulation of a 1D arterial network [19]) at the inlet and outlet of the arterial segment. The solution provided by the LBM implementation is compared with the one obtained using the finite element method (FEM) (in-house software with second order finite difference discretization in time and linear tetrahedral elements with bubble enrichment for the velocity field [20]). Fig. 9 shows the geometry of the arterial segment, the pressure boundary conditions at the proximal and distal locations and the flow rates of the second cardiac cycle obtained from
the LBM and FEM simulations. To minimize the density variation (remember it is proportional to the pressure) we imposed only the pressure gradient, instead of the actual pressure values, as it is the responsible for driving the flow in rigid pipes.

The challenge here is to deal with the complex geometry and with the highly pulsatile behavior of the pressure gradient. As said before, the LBM is an explicit method, so the time step has to be small enough such that the pressure changes in one extreme are propagated through the entire domain as fast as possible in order to minimize compressibility effects. In other words, we need to calculate the parameter denominated $\theta_t$ not based on the period of the simulation, as done in the previous examples, but based on the characteristic time in which the pressure gradient changes the most. As seen in Fig. 9), the main variation in the pressure gradient happens in approximately 0.04 s (20 times smaller than the period), which is considered the characteristic time in this problem. We use the indicated value of $\theta_t = 30$ in a very refined mesh, with approximately 80 nodes along the diameter of the segment (about 8 million nodes).

Fig. 8. Streamlines of the velocity field in the problem of the anastomosed vessel for $Re = 299$ (above) and $Re = 564$ (below). The results correspond to the LBM simulations (left) and to the physical experiments (right), extracted from Ref. [18] (with permission from ASME).

Fig. 9. Illustration of the vertebral artery segment and description of the pressure boundary conditions. In the top right subfigure, we also present the flow rates obtained from the simulations via LBM and FEM.
nodes in total), and this resulted in 1.1 million time steps per period and a compressibility error of less than 1%. For this particular case we used $r = 0.5037$ (smaller than the value indicated above of 0.52), which in other problems led to instability.

The mesh used in the LBM simulation represents a subset of the segment of the vertebral artery that was used in the FEM simulation. The LBM simulation was set up by imposing the average pressure retrieved from the FEM simulation at the proximal and distal sections of the arterial segment. This procedure was mandatory for two reasons: (i) to reduce computational time and (ii) to have inlet/outlet boundaries aligned with the cartesian axes, avoiding the need to impose pressure boundary conditions over arbitrarily

---

**Fig. 10.** Velocity profiles from LBM (full lines) and FEM (dashed lines) over four lines distributed along the artery at three different instants in the cardiac period (indicated by colors in the bottom center subfigure). (For interpretation of the references to color in this figure legend, the reader is referred to the web version of the article.)

**Fig. 11.** Three-dimensional velocity profiles along the artery at the top flow rate instant. In the zoomed subfigures are compared some velocity profiles obtain from the FEM (left) and LBM (right) simulations.
oriented transversal sections. The fact that the FEM simulation and the LBM simulation are not identical is the main reason for the differences observed in the flow rates shown in Fig. 9. However, it can be noticed that the LBM result follows the FEM result consistently throughout the entire cardiac cycle. Here, because of the highly pulsatile boundary condition, the LBM presented the disadvantage of requiring a very small time step and, as consequence, a 5 times more refined mesh than the FEM. On the other hand, the LBM results are much more detailed in the temporal and spatial domains. The computational time to run two cardiac cycles with a LBM code with OpenMP shared memory parallelization and a performance of 8MLUPS (millions of lattice updates per second) in a Altix-XE 340 machine with 2 processors Intel Xeon E5520 2.27 GHz Quad Core and 24 GB DDR3 RAM memory was 27.2 days. The FEM simulation was performed in a different architecture, with a distributed memory parallelization scheme.

Fig. 10 presents a detailed comparison between the results obtained by LBM and FEM. The velocity profiles over four lines distributed along the artery are compared at three different instants in the cardiac period (see the bottom center subfigure). This shows a good agreement of the velocity profiles between the solutions presented by both methods. The differences observed in the profiles are consistent with the differences observed in the flow rate in Fig. 9.

We also compare in Fig. 11 some three dimensional velocity profiles obtained by LBM and FEM. These profiles show excellent agreement and put in evidence the complexity of the flow and the influence of the curvatures.

4.5. Artery with an aneurysm

Saccular aneurysms are pathological alterations commonly encountered in the anatomy of the cerebral vasculature. Understanding the hemodynamics in aneurysmal regions is of the utmost importance to analyze the influence of the fluid dynamics variables on the pathology, such as thrombus formation and damage over the endothelium, produced by shear stresses. With this motivation in mind, the goal of this example is to characterize a hemodynamic quantity of interest in an aneurysmal region by ranging within the physiological regime the two main dimensionless parameters that govern the pulsatile blood flow, namely the Womersley number and the Reynolds number. Fig. 12 shows a diagram of the study case, which consists of a spherical volume emulating a saccular aneurysm attached to a cylindrical artery with aspect ratio 10/1. The flow is driven by a pulsatile pressure drop between inlet and outlet given by:

\[ p_{\text{in}}(t) = p_{\text{out}} + C(\cos(\mu t) + 1), \]

where, the constants \( C \) and \( \mu \) have to be chosen in order to set up the Womersley and Reynolds numbers in the problem.

Fig. 13 displays, for a simulation with \( Re = 400 \) and \( Wo = 4.5 \), the iso-surfaces of the velocity magnitude (left figure) and the features of the secondary blood flow (right figure). The images are taken when the fluid reaches the minimum (bottom figure) and maximum (top figure) velocities.

From a set of 36 simulations with physiological Reynolds numbers of 50, 100, 200 and 400 and Womersley numbers ranging from 1 to 5 (0.5 increment), the main component of the vorticity averaged over the volume containing the aneurysm (see Fig. 12) was studied. We have chosen to analyze the behavior of the vorticity because it is associated with the vortex intensity and, along with its oscillation, this factor may play an important role the aneurysms rupture risk (see Refs. [21,22]). Fig. 14 shows the stationary oscillations induced on the aneurysm vorticity by the imposed pressure oscillation. The left column shows the mean vorticity component during one cycle, which characterizes the behavior of this quantity for each Reynolds number and as a function of the Womersley number. Note that, for each Reynolds number, the vorticity component reaches its peak with a different Womersley number. This behavior can be noticed in the complementary figures presented in the right column of Fig. 14, where the same results are shown in...
Fig. 14. Main component of the averaged vorticity around the aneurysm, presented during a period, with Reynolds numbers 50, 100, 200 and 400, respectively.
5. Conclusions

In this work we have performed a study about the applicability of a lattice-Boltzmann model to simulate pulsatile blood flow in arterial vessels under physiological regimes. Such study was focused on the characterization of the different errors present in the simulations, taking into account the specific flow conditions as well as the geometrical features of the spatial domains representing the arterial vessels.

The lattice-Boltzmann model was able to reproduce the phenomenology of all tested cases, with the Reynolds number ranging between 1 and 900, and Womersley numbers between 3.5 and 20. Furthermore, the lattice-Boltzmann method proved to deliver accurate solutions even in physiological regimes accounting for blood flow in patient-specific arterial geometries.

The four main parameters that affect the stability and accuracy of the simulation are the relaxation parameter \( \tau \), the Mach number, the relation \( \theta_{\tau} \) and the lattice refinement. The parameter \( \tau \) does not have a well established minimum value, but, in the examples analyzed, keeping it above 0.52 was enough to ensure numerical stability. This value depends, among other factors, on the chosen models for the equilibrium distribution and for the boundary conditions. It has also been verified that reducing the Mach number below 0.15 did not imply into significant changes in the results. As for the relation \( \theta_{\tau} \), above examples indicate that it must be kept above 30 in order to guarantee compressibility errors smaller than 2%. In addition, it is crucial to correctly choose a representative characteristic time of the problem, especially in the cases that present highly pulsatile boundary conditions such as the one presented in Section 4.4.

Acknowledgments

This work was partially supported by the Brazilian agencies CNPq and FAPERJ. Also, we would like to thank the great support provided by the National Institute of Science and Technology—Medicine Assisted by Scientific Computing (INCT-MACC). Finally, the first author was supported by the Brazilian agency CNPq through process 140234/2009-8. The support of these agencies is gratefully acknowledged.

Conflict of interest

None of the authors has a conflict of interest regarding this work.

References


**Fig. 15.** Left subfigure presents the upper and lower bounds of the average main vorticity component in the aneurysm induced by the pressure oscillation. The colors indicate the Reynolds number: 50 (black), 100 (red), 200 (green), 400 (blue). And, right subfigure shows the dependence of the main resonance frequency on the Reynolds number. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of the article.)


