HPC Simulation of Particles Deposition in Human Airways

B.Eguzkitza, C. Butakoff, G.Houzeaux, M.Vázquez, B.Mignot

Abstract

The goal of this project was to optimise the Alya HPC simulation code for the problem of airflow and particle deposition in human airways, validate its results and establish a pipeline for the use of HPC in the design of treatments for respiratory diseases. High-resolution human airway model was constructed using computed tomography (CT) images of a patient. Using the patient data, it is possible to assess how inhaled drugs (represented by particles) are transported and identify the areas of deposition, which can be compared to the deposition data acquired in vivo. The focus is given to the assessment of performance measured in terms of efficiency of the code in HPC context and validation. This project has been developed as collaboration between Fluidda and the Barcelona Supercomputing Center (BSC) within the SME HPC Adoption Programme in Europe (SHAPE) programme of PRACE.

1. Introduction

The flow in the human airways can be highly complex due to a rapidly increasing number of branches and decreasing of their size. As a consequence a large variety of Reynolds numbers (Re) can be observed. Additionally with the high amplitude of the inhalation and the rapid acceleration occurring during the sniff, the air flow can become transitional and turbulent. Considering all these issues, the use of large-scale simulations becomes a must, and includes a robust turbulence model.

When dealing with analysis of drug delivery to the lung, additional challenges need to be taken into account: particle deposition. Depending on where the inhaled particles are, different deposition mechanisms exist. Air passing through the nose firstly passes through the nasal hairs filtering particles larger than 10-15um in diameter, additionally most of the particles larger than 10um are deposited in nasal septum or turbinates. More particles are deposited on tonsils and adenoids, where the immune system takes care of any biologically active material. Few particles larger than 10um enter trachea and most of these deposit on the carina or within the bronchi. Particles in the range 2-5um deposit by sedimentation under gravity in the smaller airways. Therefore most of the particles >2um are trapped and removed by the mucociliary escalator of the upper airways, trachea, bronchi, bronchioles. Smaller particles reach alveolar ducts and alveoli. Some particles (smaller than 0.1um) are deposited via Brownian motion, other particles between 0.1 and 0.5um mainly stay suspended as aerosols and about 80% of them get exhaled.

In this report we summarise the high performance computing (HPC) simulations of particle deposition in human airways, including basic validation. For a given patient, high resolution airways geometry was extracted from computed tomography (CT) images whereupon simulations were carried out. The results were compared with the flow rates and particle deposition information obtained by the company. The simulations were carried out using Alya, a part of the PRACE benchmark suite [1]. We have all the ingredients to solve this kind of multi-physics problems but robustness and efficiency have been improved.

* Corresponding author email address: beatriz.eguzkitza@bsc.es, guillaume.houzeaux@bsc.es, cbutakoff@elem.bio, mariano@elem.bio, Benjamin.Mignot@fluidda.com
2. General description of the problem

2.1 Airway geometry and properties

The patient geometry is shown in Figure 1 and is subdivided into different anatomical regions (the green part shows the inhaler geometry that is attached to the airway geometry):

- Two inlets inside of the inhaler. The particles are injected through one of the inlets.
- The upper airway (UA)
- The central airways (CENTRAL)
- Right upper lobe airways (RUL, red)
- Right middle lobe airways (RML, yellow)
- Right lower lobe airways (RLL, orange)
- Left upper lobe airways (LOL, blue)
- Left lower lobe airways (LLL, purple)
- *Ending of the airways, serving as outlets (b[0-9]r,l{1,2}[r,l], e.g. b1r, b12l, b8l, b10r, ...)*

Air was modelled as fluid with density of 1.225 kg/m³ and viscosity of 1.7894e-05 kg/(m.s). Discrete particles with density 1000 kg/m³ and diameters in a range from 0.9 to 30µm are injected and one-way coupled with the air.

The aerodynamic diameters of the particles range from 0.9 to 30µm as provided by Fluida. The density is 1000 kg/m³. Acting on particles are the drag, gravity and buoyancy.

All the simulations were carried out with constant inflow velocity.

2.2 System Discretisation

The mesh, created using ICEM software (ANSYS) has 6,697,178 elements. Figure 2, Figure 3 and Figure 4 show different aspects of the mesh including element size variation throughout the domain. Of particular note are narrowing present in the bronchi, indicative of the airway obstruction in this patient. The mesh was generated making sure there are several elements in the cross section in the narrow areas, which is a necessary condition for the FEM method to be able to solve the flow in the interior of the domain.
2.3 Modelling approach

The main algorithms involved in this work are the fluid solver and Lagrangian particle tracking. Specific details of the formulation are described in [2].

The numerical model of the Navier–Stokes solver is a stabilised finite-element method, based on the Variational MultiScale method. The velocity and pressure unknowns, respectively $u$ and $p$ are split into grid scales and subgrid scale. Then, the Galerkin method is applied to velocity and pressure and the subgrid scales are approximated using an algebraic model. A convection tracking strategy is also considered. The resulting algebraic monolithic system is solved with a split strategy. We first extract the pressure Schur complement equation. This equation is then solved using an Orthomin (1) algorithm, which requires solving twice the momentum equation and once the continuity equation. This is the momentum preserving Orthomin (1) method, as after one shot of this algorithm, the momentum is preserved.

The flow in the upper airways is mostly turbulent and/or transitional in nature. Therefore, accurate predictions of the flow field require a suitable turbulence model or sufficiently fine mesh. The numerical discretisation used in this work is based on a low-dissipation numerical scheme described in [3]. Lagrangian approach treats the dispersed phase as a set of individual point-particles in a continuous carrier phase. The particles are tracked through the flow field by solving the equations of motion for each particle with the relevant forces acting on it. Particles are transported solving the Newton’s second law, and by applying a series of forces: drag, lift, gravity and buoyancy. Brownian motion can be considered as well. The time integration is based on the Newmark’s method. Due to parallelisation, performance and accuracy issues, there exist some constraints on the time step. First, particles are not allowed to cross more than one element from one time step to the next one, for two reasons: parallelisation constraint, consistency with the discretisation scheme used to solve the Navier–Stokes equations. Second, if the Newmark/Newton–Raphson does not converge, time is automatically decreased. Third, in order to control the error, an adaptive time step strategy is applied based on an error estimate. The parallelisation in Alya code is achieved at two levels. At the MPI level, a classical substructuring technique is employed by partitioning the mesh into subdomains. Communication between neighbouring subdomains is essentially achieved by means of non-blocking send–receive after matrix–vector products, and global communications are used to compute scalar products. The second level of parallelism is obtained through OpenMP pragmas in the main loops of the code. A colouring strategy and multidependences technique are also considered to avoid the overhead implied by the ATOMIC pragma during the assembly of element matrices and right-hand sides into the global ones. The coloring technique consists in assigning a color to each element. Elements that share a node cannot have the same color such that the elements of the same color can be computed in parallel. The main drawback of the coloring approach is that it hurts spatial locality since contiguous elements are not computed by the same thread. For multidependences approach the domain assigned to each MPI process is divided into subdomains. We map each subdomain into an OpenMP task. Knowing that two subdomains are adjacent if they share at least one node, we can use the information about the adjacency of subdomains provided by Metis to know which OpenMP tasks cannot be executed at the same time. In [4] and [5] the authors describe with detail all methods.

2.4 Coupling CFD with particle simulations

The coupling of CFD with the particle simulations implies problems related to the load imbalance and synchronisation as particles are not uniformly distributed in the domain and frequently move from one subdomain to another. To address
this problem, we have adapted DLB library [2], dynamic load balancing, to balance computational load due to particles at runtime and multi-code strategy to achieve coarse grain asynchronism.

Additionally, we saw that the respiratory motion is periodic, leading to periodic gross flow behaviour inside the geometry. This means that in majority scenarios, the CFD needs to be solved only once, whereupon the obtained velocity fields can be used repeatedly to test the transport of different particle types, effect of different forces acting on particles, different deposition models at a significantly reduced computational cost. However, for large geometries solved at small time steps, this requires efficient handling of the precalculated velocity fields, which needed to be integrated into the hybrid parallelisation scheme of Alya.

2.4 Model parameterisation

To parameterise the model, the flowrates at different bronchi for this patient, were provided by Fluidda. The common approach would be to impose these flowrates at the outlets of the computational domain. However, that will usually lead to a problem of misrepresenting the real flow. Many times the flowrate measurements are estimated based on assumptions and contains measurement errors. Additionally, the pulmonary model used does not represent the complete lung. As a consequence, if the geometry does not support the flows that are imposed at the outlets, the generated flow will not be representative of the reality. A typical example is if there is a Y-shaped geometry with 2 outlets and one inlet. If the combined flowrate at the outlets is larger than the one supported by the inlet, there will immediately be flow not only through the inlet but also between the outlets, which is not characteristic of the airflow in the lungs.

Instead of this typical approach, the provided flowrates were summed up and applied at the inlets of the model, keeping the outlets open. The flowrates at the outlets were observed and compared to the ones measured. Afterwards, the pressure boundary condition was applied at the outlets to adjust the flowrates as to match the measured flowrates.

In particular, we observed that while the flowrates in all the branches approximately matched the measured flowrates, in one particular pulmonary branch (R1-3) the flowrate was significantly higher than the measured. After adjusting the pressures, the flowrates matched the measured ones. The curves can be seen in Figure 9 where blue curve, labelled as Fluidda, shows the flowrate measurements, “Alya open” are the flowrates with all the outlets open (no pressure boundary conditions). The yellow “Alya Pressure R1-3 1500 Barye” represents the flowrates after the parameterisation, showing that the geometry that is being analysed, does not represent the whole relevant patient information, as there is an important airflow obstruction somewhere down the branch R1-3, which is not present in the geometry. Figure 9 also shows flowrates as a function of time at each of the branches (blue), as compared to the measurements provided by Fluidda (black line) when the outlets were open.

The boundary conditions were:

- Normal velocities at the inlets are 1649 cm/s and 1674 cm/s (splitting the flowrate XXX between the inlets according to their surface area, see Figure 5).
- Non-slip walls everywhere except the inhaler
- For the inhaler, elastic bouncing condition was programmed
- Pressure of 1500 Barye was imposed in RUL outlet region (grey in Figure 6).

Figure 10 shows how these boundary conditions affected the pressure distribution throughout the domain.
3. Numerical Results

The simulation of one second of flow with the time step 2.5e-04s (largest time step where the flow would not diverge) took 336 CPUs (using MPI) and 20 hours.

Between 3M and 7M discrete inert particles are injected into the domain through the inlet 1 (Figure 5), uniformly distributed over the inlet surface with zero initial velocity. The particle sizes and distributions in time were provided by Fluidda. The maximum number of particles floating in the domain (not deposited) are not higher than 250K.

Table 1 shows the deposition percentages for 1\(\mu\)m and 6\(\mu\)m particles. Figure 7 illustrates the trajectories for some particles as well as the pressure field in the domain, while Figure 11 shows the velocity around the narrow’s areas. Figure 8 represents a deposition map of the total number of particles coloured by the particle type (1 represents the particles with the smallest diameter, 21 -- the largest).
Figure 7: Pressure field and some trajectories

Figure 8: Deposition map for all the particles

Figure 10: Flowrates at different outlets

Figure 9: Pressure field
Table 1: Deposition in % for 1µm and 6 µm particle diameter

<table>
<thead>
<tr>
<th></th>
<th>central</th>
<th>devise</th>
<th>mouth</th>
<th>UA</th>
<th>LLL</th>
<th>LUL</th>
<th>RLL</th>
<th>RML</th>
<th>RUL</th>
<th>OUT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1µm</td>
<td>0.09539</td>
<td>0.0</td>
<td>91.79</td>
<td>8.0735</td>
<td>0.0251</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0150</td>
<td>0.0</td>
</tr>
<tr>
<td>6µm</td>
<td>0.11564</td>
<td>0.0</td>
<td>91.7035</td>
<td>8.1657</td>
<td>0.01</td>
<td>0.0</td>
<td>0.0</td>
<td>0.2</td>
<td>0.0050</td>
<td>0.0</td>
</tr>
</tbody>
</table>

Figure 12: Velocity field in narrowing surroundings

4. Benefits for SME

Accurate computer simulations are a very important success factor before taking prototypes to full-scale medical devices. They provide the best way to improve the technology before manufacturing and could also give us deeper knowledge of the physics governing the device and its behaviour in patients.

With the help of HPC resources, optimal services will be at hand. Fluidda is a company that offers services to medical devices manufacturers and pharmaceutical industry to improve their inhalers and nebulisers. So far, they have been using commercial simulation software to provide their services. However, it falls short when accuracy in large-scale problems is required. In addition, this service scenario is more and more frequent. Moreover, when commercial software is used, particle transport and deposition is both inaccurate and inefficient. The idea of this work is to assess how the HPC-code called Alya covers all these issues.

A successful completion of the project could lead to new customers and services. With it, the company will demonstrate to biomedical industry the importance of the use of such a simulation package to attack this problem, which is directly linked to improving life quality of so many people. It will wide open Fluidda's market and services span.

5. Conclusions and Outlook

This paper is the result of a PRACE SHAPE project where our aim was to demonstrate the ability of Alya code to solve HPC simulation of particles deposition in human airways. The code has been optimised and one of the ideas of the work has been to validate its results and to establish a pipeline to enable the use of HPC resources in the company workflow.
To conclude the paper, we summarise the main difficulties that we had to overcome in this work:

1. In the process of executing the project, BSC has discovered that the provided geometry appeared to have narrowing of the distal bronchi in some areas of the lung. That led to a necessity to consider several strategies of constructing computational mesh. Given that, Alya uses FEM method care that must have been taken to ensure sufficient mesh resolution in the interior of the narrowings.

2. An initial attempt at implementing flowrate outlet boundary conditions has shown us the shortcomings of that methodology and it was decided to impose pressure boundary conditions instead, to mimic elevated resistance to the airflow in the real lungs. The latter allowed matching the observed flowrates. This observation allowed us to see whether the patient might have some pulmonary problem in a specific lobe by comparing the observed flowrates to the measured flowrates.

3. Initial simulations were carried out using slip-wall conditions of the inhalation device geometry. However, due to a very high velocity and complexity of the flow in the device, the majority of the particles had difficulty advancing along the walls. Therefore, the reflection wall law had to be programmed (particles hitting the wall would bounce back). The latter allowed for a much more accurate particle behaviour making all the particles leave the device into the mouth cavity.

Acknowledgements

This work was financially supported by the PRACE project funded in part by the EU’s Horizon 2020 research and innovation programme (2014-2020) under grant agreement #823767.

And also thanks to INSPIRe, (FIS2017-89535-C2-1-R), under grant agreement Programa Estatal de I+D+i Orientada a los Retos de la Sociedad from Ministerio de Ciencia, Innovación y Universidades.

References


