Brolin Daniel, 940901-5014, danbro-3@student.ltu.se

Kask Nathalie, 930722-3686, natkas-2@student.ltu.se

Last revision: March 7, 2018

Luleå Tekniska Universitet
F7024T Multifysik, simulering och beräkning

Assignment 6: Convection and diffusion in a branched pipe

With supervisor
Hans Åkerstedt

Abstract

This report is the last part of a written documentation of the laboratory exercises made in the course Multiphysics, Simulation and Computation at Luleå University of Technology. This exercise serves as practice in formulating mathematical models to describe physical and technical problems. For analytic and visual support COMSOL is used to solve the laboratory exercises.

To practice rudimentary calculations and plots within COMSOL Multiphysics the convection and diffusion in a branched pipe is studied.
Increasing the inflow field velocity increases the kinetic energy in the particles, which pushes them to to the surface walls; which is why the concentration of particles on the wall increases with the velocity.

1 Introduction

COMSOL Multiphysics® is a general-purpose software platform based on advanced numerical methods. It is a powerful tool useful to simulation of flow, fields, force and such in models provided either by files of built directly in COMSOL. To practice rudimentary calculations and plots, COMSOL is used to solve the laboratory exercises numerically.

This report is a part of a written documentation of the laboratory exercises made in the course Multiphysics, Simulation and Computation at Luleå University of Technology. This exercise serves as practice in formulating mathematical models to describe physical and technical problems in way that is suitable for implementation of the finite element method.

In this last assignment the convection and diffusion of particles in a branched pipe, as for instance, the trachea and the bronchi in the human body. The respiratory tracts in the human body transports gases to, and from the lungs. In addition, the airways adjust the temperature and humidity of the air to prevent the delicate tissue in the lungs from being damaged. Model simulation has for many years been focused on tests on vital capacity single-breath washout, with the purpose to act as a tool for early detection lung diseases such as cigarette smoke induced airway changes.

Convection works to bring the inhaled air and the molecules contained within it into contact with the lining of the respiratory tract, but also to remove the air out of the respiratory tract during expiration. The rate, at which, material (air) is delivered to a region of the respiratory tract by airflow is restricted by convection. Diffusion occur independent from convection. The driving force of diffusion is the thermal motion of molecules. All molecules in a gas display random motion resulting from the fluctuation of forces exerted by surrounding molecules. This behaviour is called Brownian motion and results in a net-migration of molecules from regions of high concentration to regions of low concentration. The net-migration is called diffusion.

Convection transports material along and through the airways, while diffusion cause parts of the material to move from the center of the airways, closer to the walls and into the tissue. This results in a layer of air, immediately adjacent to the airway surface, which moves relatively slow due to friction. Furthermore, since molecules and particles are nonzero-diffusive in reality, this problem is usually solved with the convection-diffusion equation, where both the convective and the diffusive contributions to the mass transport are taken into consideration[1].

Except for the simplest geometries, the equations for convection cannot be solved by hand. It is therefore vital to understand the interplay of convection and diffusion in real systems through numerical modeling.

2 Method

2.1 General

The exact method to calculate the coefficients and simulate the system is detailed and well explained in the instructions[2], but the general way to go about it is the same as most COMSOL projects.

1. Choose system type (fluids, laminar, $2D$).
2. Introduce global parameters.
3. Build geometry.
4. Set study specifications for your system type:
    - Set fluid parameters.

- Set Boundary conditions.
- Set initial conditions.

5. Build a mesh grid of your geometry.
6. Compute system.

By the application of the Navier-Stokes equations together with the equation for convection-diffusion, as presented in equation 1, the concentration, c, can be obtained.

The Navier-Stokes equation is given by

$$\frac{D\mathbf{u}}{Dt} = \left(\frac{\delta \mathbf{u}}{\delta t} + (\mathbf{u}\cdot \bigtriangledown)\mathbf{u}\right) = -\frac{1}{\rho} \bigtriangledown p + \nu \bigtriangledown^2 \mathbf{u} \tag{1}$$

where $\bigtriangledown \cdot \mathbf{u} = 0$ and ν is the kinematic viscosity of air.

Furthermore, the convective-diffusion equation is given by

$$\frac{\delta c}{\delta t} + (\mathbf{u}\cdot \bigtriangledown)c = D \bigtriangledown^2 c \tag{2}$$

where D is the diffusion constant given by the Brownian diffusion as presented below.

$$D = \frac{\kappa T}{k_b} = \frac{\kappa T C u}{3\pi\rho\nu d_f} \tag{3}$$

Here κ is Boltzmanns constant, T the absolute temperature, ρ is the density of air and ν is the viscosity of air and a is the pipe radius.

Finally, Cu is the correction factor called Cunningham factor and is given by

$$Cu = 1 + \frac{\lambda}{d_f}(2.34 + 1.05e^{-0.39\frac{d_f}{\lambda}}) \tag{4}$$

Here λ is the mean-free path of the air molecules, which is equal to 70nm. The radius of the trachea is set to 0.3mm, corresponding to the 16th generation of the human lung. The particle diameter is set to $d_f = 20nm$. The inlet velocity is simulated to correspond to light, moderate and heavy physical activity. This is done by ranging the average velocity $\mathbf{u}$ from 1 cm/s to 10 cm/s.

The simulation profile for this assignment is provided from the 3D geometry in the file lungmodell.mphbin.

Finally, to the fraction of particles deposited to the wall is found by finding the ratio of particles flowing out of the system divided by the number of particles flowing into the system.

3 Results

In this section the result from the average velocity, $\mathbf{u}$, sweep is presented. In figure 1 the fraction of particles deposited is presented for the sweep of the average velocity, $\mathbf{u}$. In figures 2, 3 and 4, the concentration of particles for each slice in the 3D model geometry is presented. In figures 5, 6 and 7 the particles proximity to the surface walls, depending on the average velocity, $\mathbf{u}$, is presented. Finally, in figures 8, 9 and 10 the velocity magnitude is presented for the for the particles.

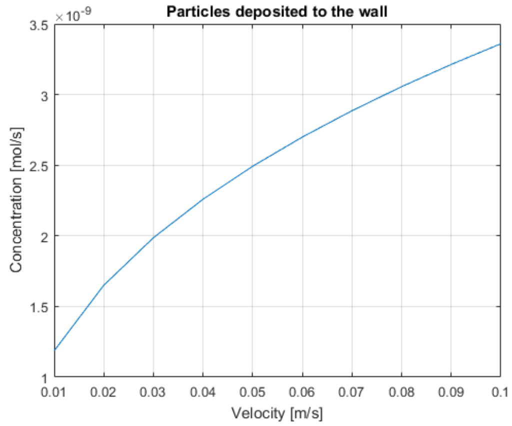

Figure 1: The fraction of particles deposited for an the average velocity sweep between 1 to 10cm/s.

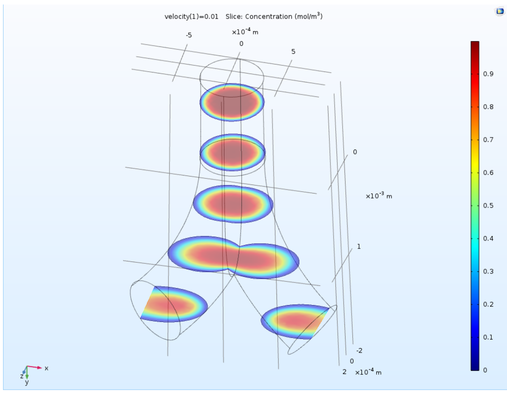

Figure 2: Particle concentration for simulation with 1cm/s average velocity.

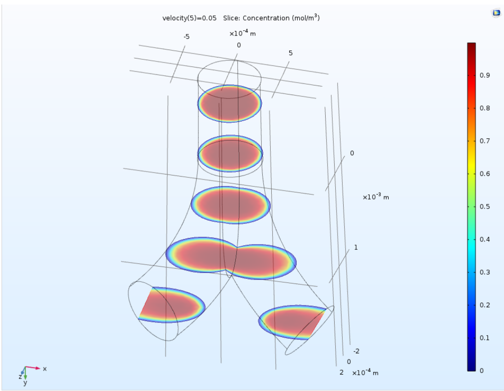

Figure 3: Particle concentration for simulation with 5cm/s average velocity.

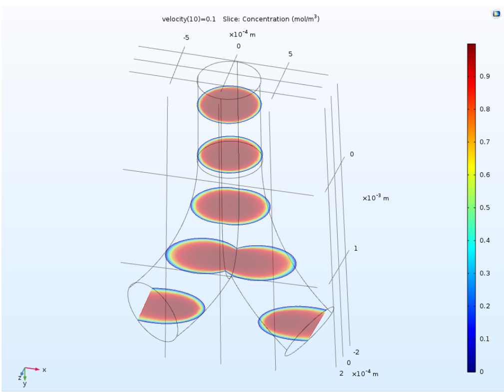

Figure 4: Particle concentration for simulation with 10cm/s average velocity.

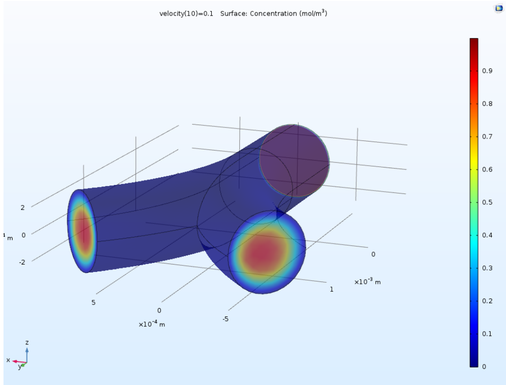

Figure 5: The particle concentration proximity to the surface wall at 1cm/s average velocity simulation.

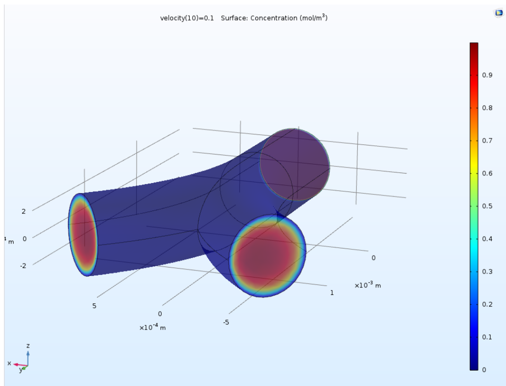

Figure 6: The particle concentration proximity to the surface wall at 5cm/s average velocity simulation.

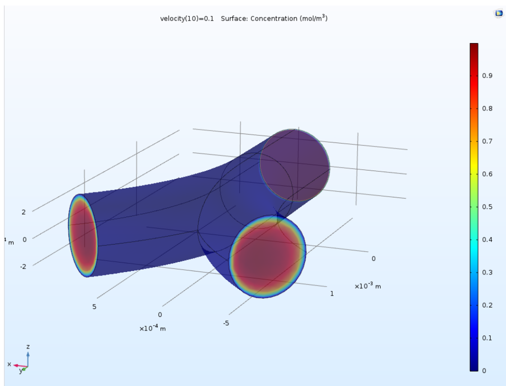

Figure 7: The particle concentration proximity to the surface wall at 10cm/s average velocity simulation.

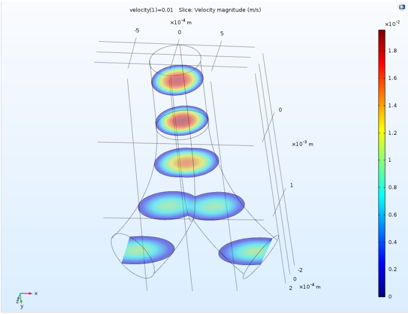

Figure 8: The velocity magnitude of the particles in simulation with 1cm/s average velocity.

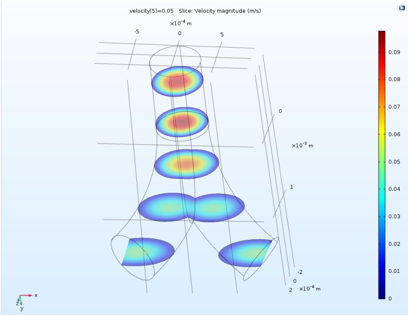

Figure 9: The velocity magnitude of the particles in simulation with 5cm/s average velocity.

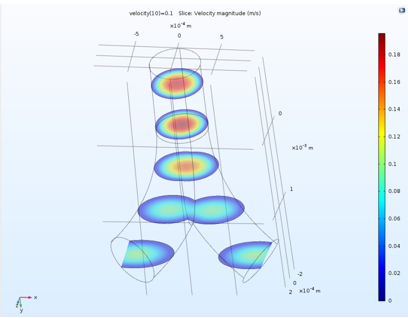

Figure 10: The velocity magnitude of the particles in simulation with 10cm/s average velocity.

4 Discussion and Conclusions

By inspecting the figures 8, 9 and 10 and compare the velocity magnitude to the particle concentrations in 2, 3 and 4, it is clear that the higher the velocity is, the higher the kinetic energy is in the particles.

As a result, the particles are forced from the center of the airway and closer to the surface walls.This can also be seen by looking in the airway outlet displayed in figures 5, 6 and 7.

From figure 1 it is clear that the amount of particles deposited to the wall increases with an increasing average velocity. The increase of the particles deposited is the result of the diffusion effects hindering some particles to pass through the airway tubes in higher velocities.

References

[1] Amos Gilat. *Convection-Diffusion Equation.* John Wiley & Sons (Asia) Pte Ltd, 2011.

[2] Multiphysics F7024T Hans Åkerstedt. Assignment #6, convection and diffusion in a branched pipe. Technical report, Department of Engineering Science and Mathematics, May 2017.