"Inspiratory Steady Flow Analysis in Symmetrically Bifurcating Lung Airways"

Nikhil Kumar Tamboli

Department of Mechanical Engineering

Indian Institute of Technology, Bombay

Abstract

The aim of this project is to study the flow characteristics of air in a human respiratory tract using open source CFD package 'OpenFOAM v7'. This flow characteristics include velocity profile at critical sections, recirculation regions and wall shear stress analysis. Since the real respiratory tract has complicated geometry, unsymmetrical bifurcations and varying flow conditions, this study is restricted to laminar flow of air under steady state condition. The geometry is also simplified to 2D circular tubes with symmetric bifurcations. The Re number of the flow is taken as 500.

1. Introduction

The respiratory system is nothing but a branching network of bronchi. The real respiratory tract has varying cross section along the length, flexible peripheral walls and unsymmetrical bifurcations. The diameter of the bronchi varies from 1 mm to 20 mm. The bifurcations in the flow path are characterized by complex geometry and unusual flow patterns. Bifurcating flow involves distinctive flow phenomena such as skewed axial velocity profiles, strong secondary motion, reversed flow, flow separation, and transition of flow from laminar to turbulent. These complex characteristics are responsible for the deposition of fatty substances and pollutants in the inner walls of the lung, thereby changing the shear stress distribution along the walls. Hence, it is important to analyse fluid dynamic effects because it is directly associated with the respiratory health disorders.

2. Problem Statement

This study focuses on a small portion of the respiratory tract where a parent tube is symmetrically bifurcated into two daughter tubes. Each of this daughter tubes is further bifurcated into two sub daughter tubes with the angle of bifurcation as 60 degrees. As far as final results are concerned velocity profiles, recirculation regions and wall shear stress calculations at the critical sections and sub-daughter tubes have been carried out in this study.

3. Governing Equations

This problem requires incompressible and steady form of continuity and momentum equations to obtain the field variables like velocity and pressure. The equations are shown as below:

Continuity equation:

$$\nabla . \boldsymbol{u} = 0$$

Momentum equation:

$$(\boldsymbol{u}.\nabla)\boldsymbol{u} = -\frac{\nabla P}{\rho} + \nu . \nabla^2 \boldsymbol{u}$$

Where, **u** is velocity vector, P is pressure, ρ is fluid density, ν is kinematic viscosity.

4. Simulation Procedure

4.1 Geometry and Mesh

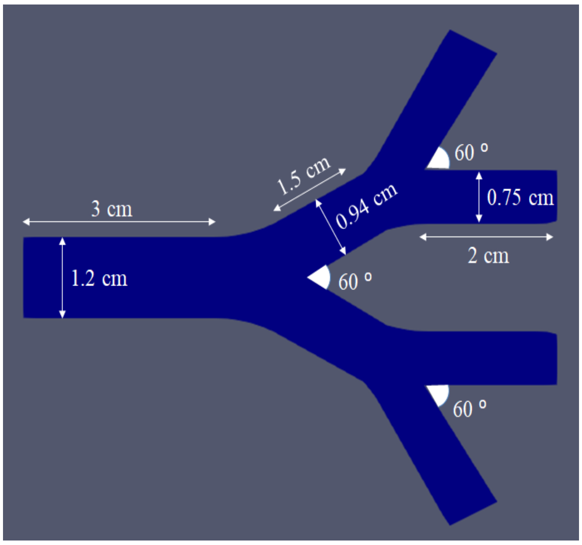
Both geometry creation and mesh generation process are carried out simultaneously using ‘blockMesh’ utility in the OpenFOAM. The geometry consists of parent tube, diameter 1.2 cm and symmetrically bifurcated two daughter tubes of diameter 0.94 cm each (generation 1) arising from this parent tube. Each of this daughter tubes is further bifurcated into two sub daughter tubes of diameter 0.75 cm each (generation 2). Angle of bifurcation is taken as 60 degrees in both these bifurcations. The total number of blocks used in the geometry is 28. The grid independence test is carried out for two cases using course, moderate and fine mesh. The grid independence test ensures that the final solution is independent of the mesh size. Course mesh contains 49248 hexahedral elements with 0.125 mm cell size, moderate mesh contains 65664 hexahedral elements with 0.125 mm cell size and fine mesh contains 82082 hexahedral elements with 0.1 mm cell size in the flow direction. Finally, moderate mesh with 65664 hexahedral elements is taken for simulations of other cases as it is comparatively accurate and computationally inexpensive than the other two.

In this study, three different cases are analysed (with and without blockage). In the last two cases other than the first case, a blockage with gradual decrease and then increase in cross section is introduced which restricts the air flow thereby changing the flow characteristics in subsequent tubes. Blockage length introduced is 1.5 cm and diameter of vena contracta is 0.4 cm.

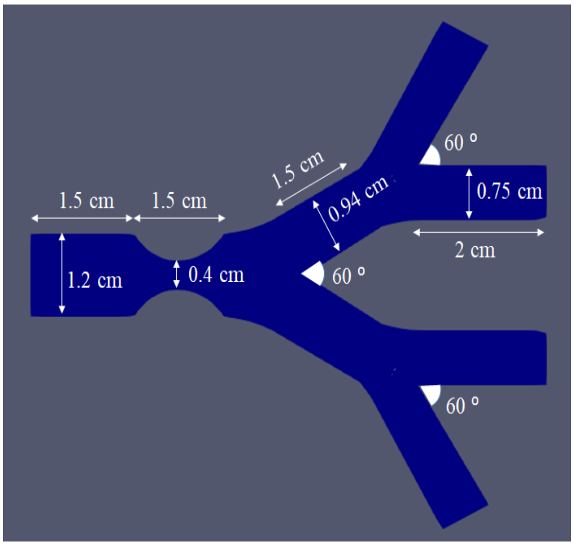
The three different cases are shown below:

Case 1 – Without blockage
Case 2 – With blockage before first bifurcation
Case 3 – With blockage after first bifurcation in the upper daughter tube.

Case 1



Case 2



Case 3

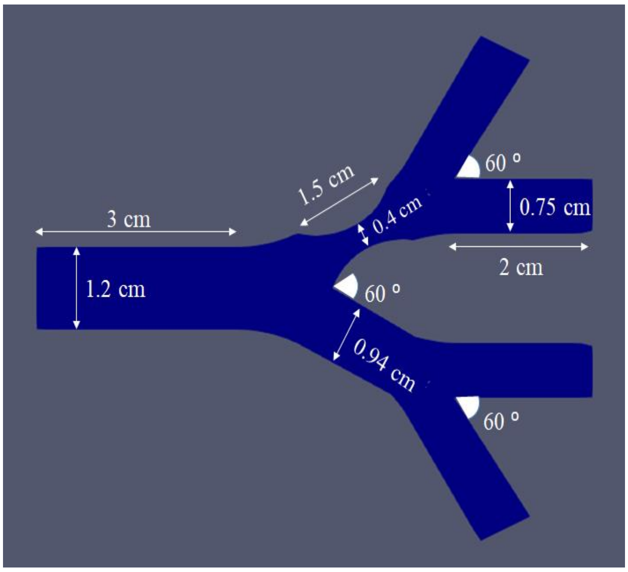


Figure 4.1 – Geometry description for different cases.

Grid Independence Test

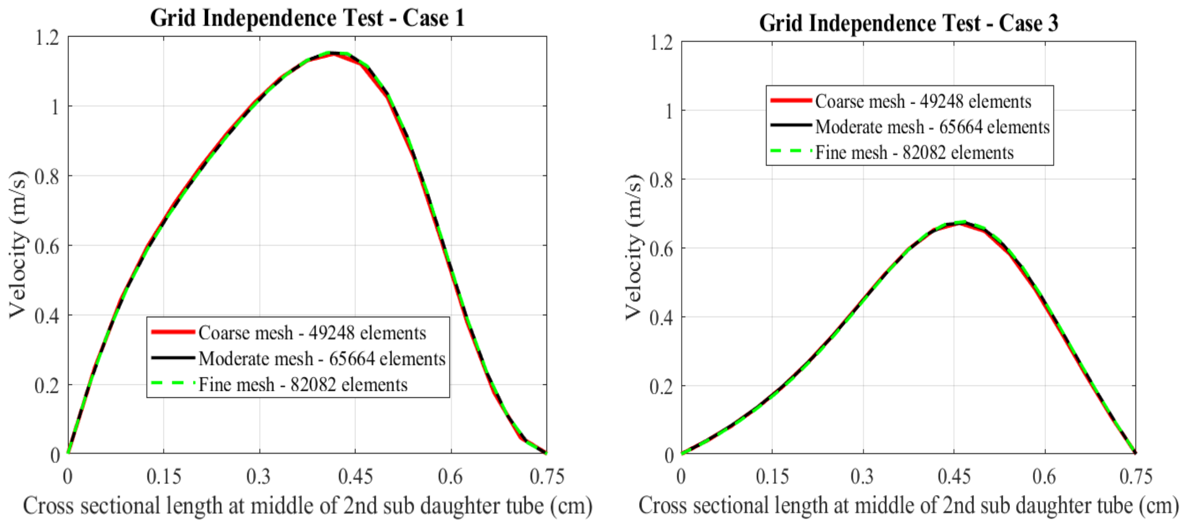


Figure 4.2 – Grid independence test for case-1 and case-3.

4.2 Initial and Boundary Conditions

There are generally two ways to approach the respiratory flow:

1) Pressure drop driven model, where atmospheric pressure is taken at the inlet and negative gauge pressure is taken at the outlet. But computationally this model is difficult to implement and hence not generally preferred.

2) Inlet velocity model, where a parabolic velocity profile is taken at the inlet and fixed zero pressure is taken at the outlet. This model is computationally easy to implement and hence preferred.

In this study, the inlet velocity model is chosen. The initial and boundary conditions used in the simulation are described in the table below:

Table-4.1 Boundary conditions

Field variable	Inlet	Wall	Outlet
Velocity	Fixed - Parabolic Velocity Profile	No Slip	Zero Gradient
Pressure	Zero Gradient	Zero Gradient	Fixed - Zero Value

Table-4.2 Flow and fluid properties

Fluid	Kinematic viscosity	Re	Inlet velocity
Air	0.00001654 m^2/s	500	U = 0.689 m/s

4.3 Solver

The problem involves 2D steady state analysis of laminar flow of air. This problem can be simulated by number of solvers. In this study, simulations are carried out using icoFoam solver, pisoFoam solver and pimpleFoam solver. icoFoam solver is generally used for transient, incompressible and laminar flow of Newtonian fluids. pisoFoam solver is used for transient, incompressible and turbulent flows. pimpleFoam solver is used for transient, incompressible and turbulent flow of Newtonian fluids. icoFoam and pisoFoam solver works on PISO algorithm to solve the governing equations. Both these solvers require a low Courant number (< 1). Whereas pimpleFoam solver works on PIMPLE algorithm and works fine even with courant number greater than 1. Apart from these, wall shear stress calculations are possible with pisoFaom and pimpleFoam solver. icoFoam solver, by default is not designed to calculate wall shear stress. The simulation is done for 2.5 seconds to achieve steady state condition. The solver preferred is icoFoam as well as pisoFoam. pisoFoam is used for wall shear stress calculations. A time step value is chosen in such a way that Courant number value is less than 1. The typical value of time step taken is lesser than or equal to 0.0001 seconds.

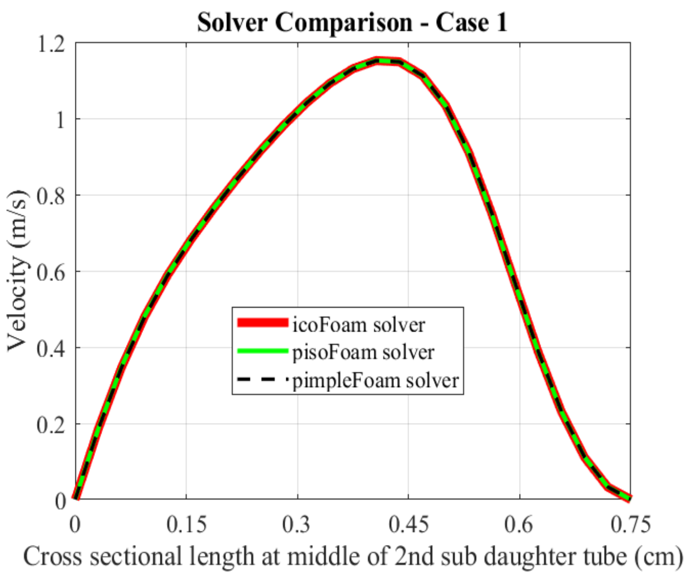
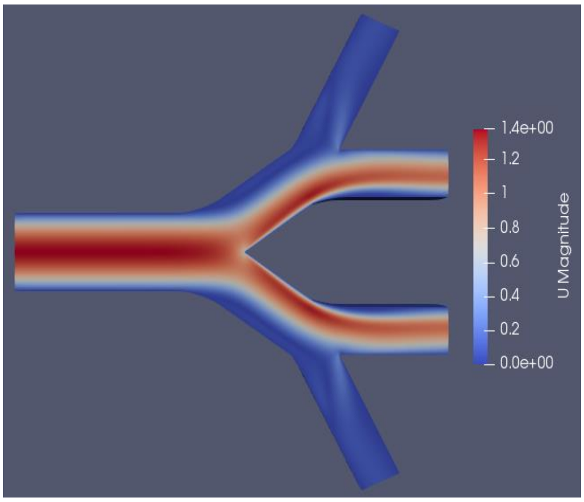


Figure 4.3 – Solver comparisons for case-1.

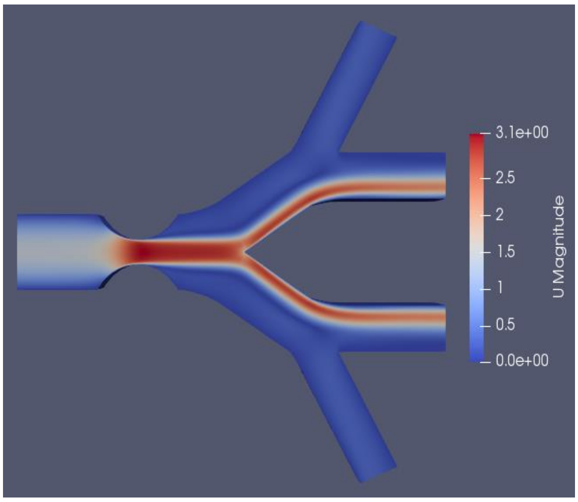
5. Results and Discussions

The following are the results obtained at the end of 2.5 seconds i.e. when steady state is achieved. The result is viewed using paraView.

Case 1



Case 2



Case 3

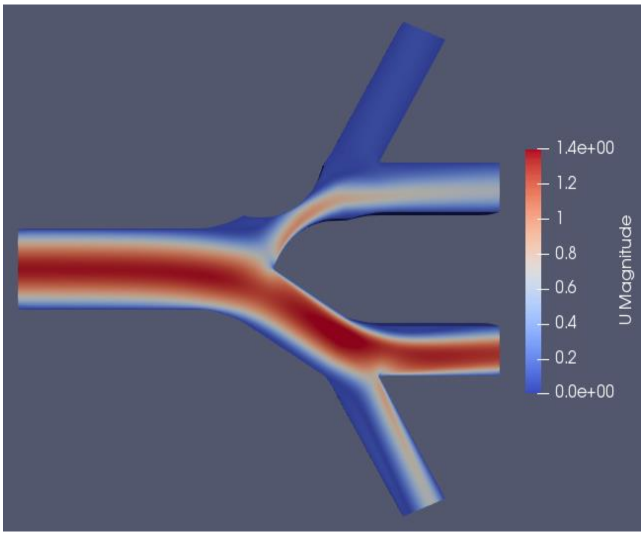


Figure 5.1 – Final results of simulation under steady state condition.

The detailed description of the results from the simulation is presented below:

5.1 Velocity profiles

Case 1

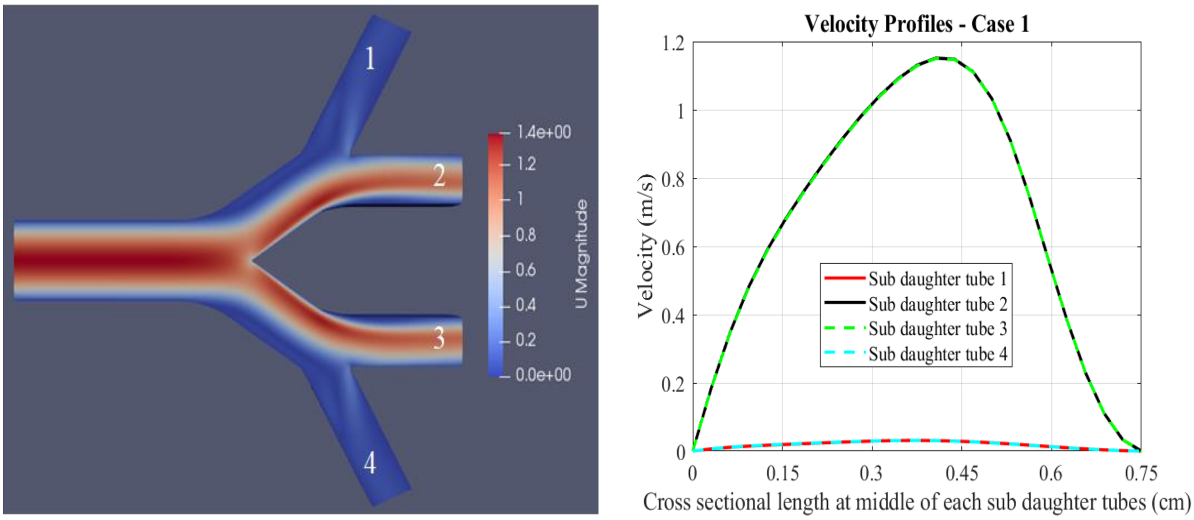


Figure 5.2 – Velocity profile across cross sectional length of each sub daughter tubes for case-1

Case 2

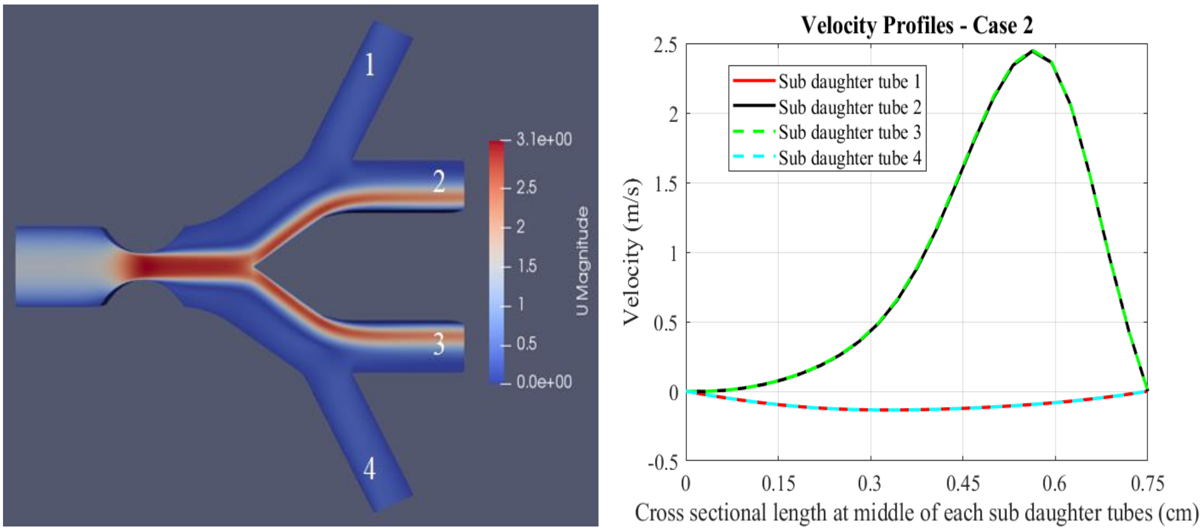


Figure 5.3 – Velocity profile across cross sectional length of each sub daughter tubes for case-2

Case 3

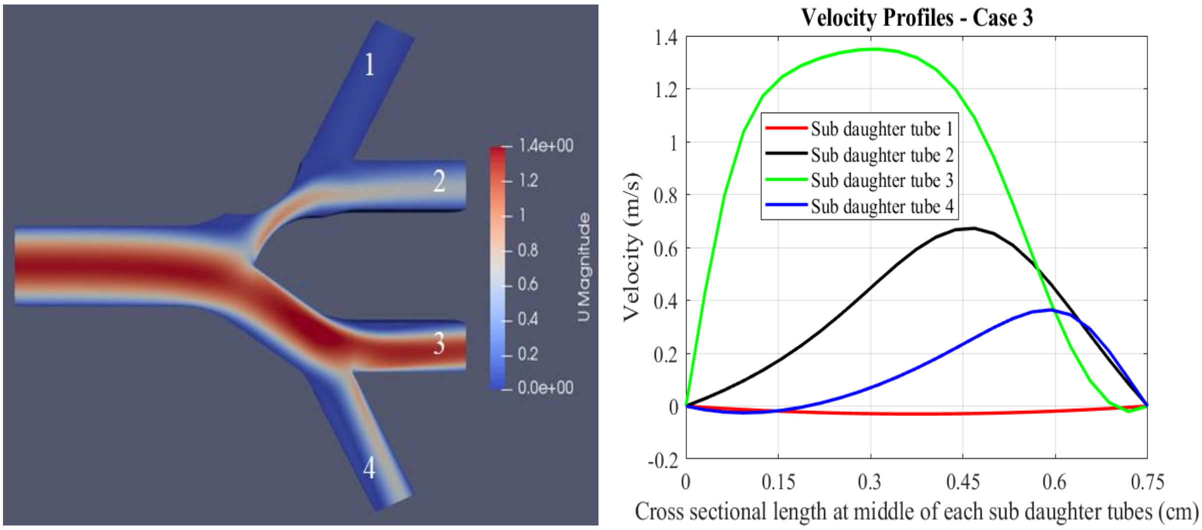
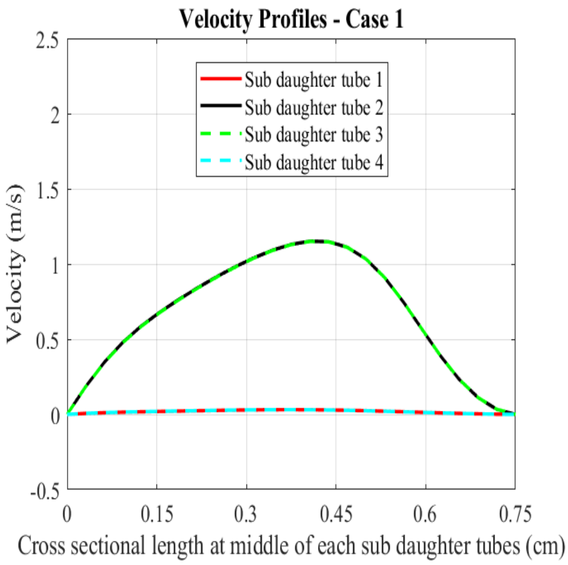
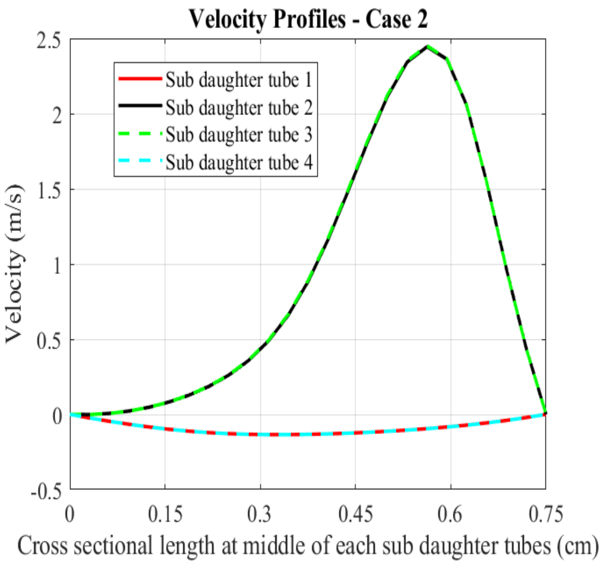


Figure 5.4 – Velocity profile across cross sectional length of each sub daughter tubes for case-3.

Overall comparisons





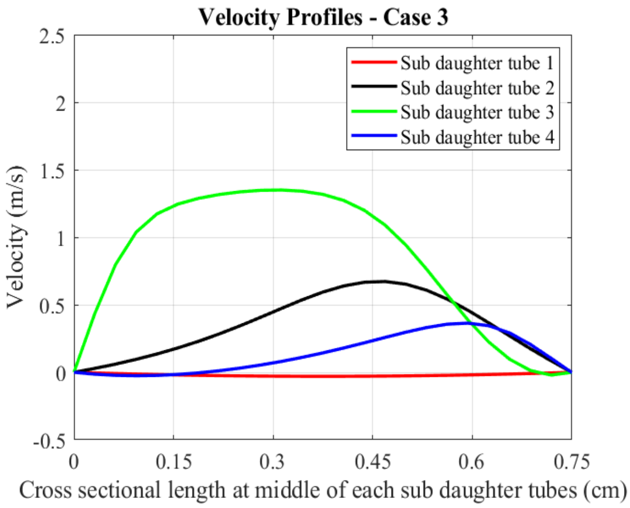


Figure 5.5 – Overall velocity profile comparison across cross sectional length of each sub daughter tubes for all cases.

5.2 Wall shear stress

Wall shear stress is the tangential force per unit area that is exerted by the flowing fluid on the peripheral wall boundary and vice-versa. The direction is the local tangent plane of the wall. Hence, it is obvious that the wall shear stress values can be positive, negative or zero.

In this study, wall shear stress calculations are done for case-1 and case-3 only. For case-3, very high velocity at the vena contracta demands very small time step which leads to failure of pisoFoam solver at any intermediate time step.

Case 1

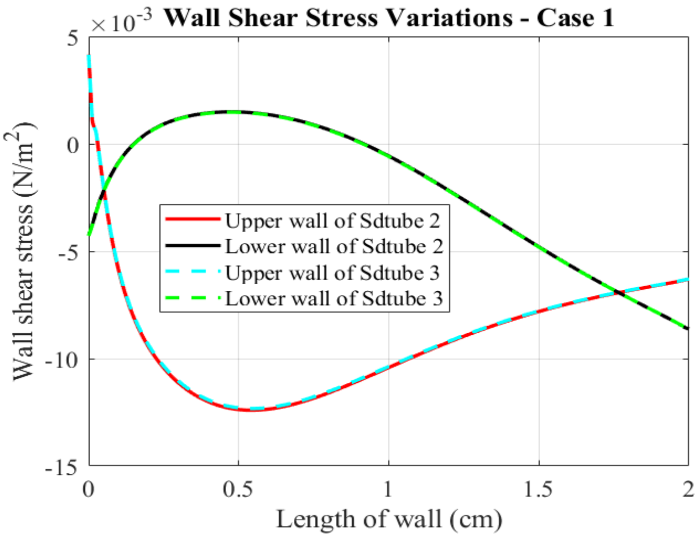


Figure 5.6 – Wall shear stress variation with the length of sub daughter tube 2 and tube 3 for case-1

Case 3

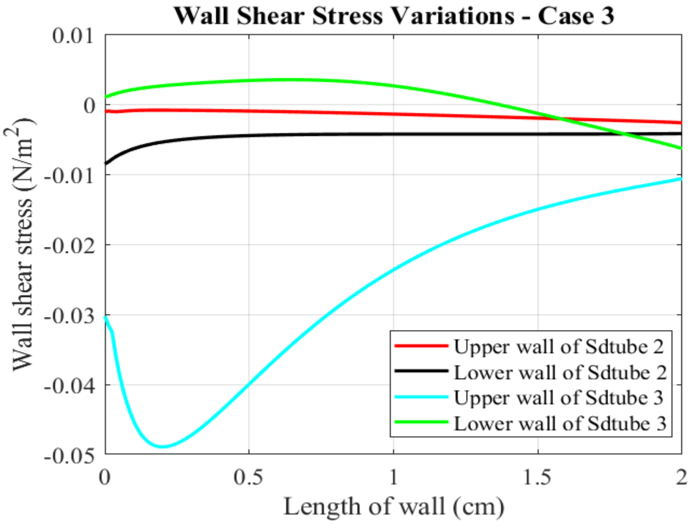


Figure 5.7 – Wall shear stress variation with the length of sub daughter tube 2 and tube 3 for case-3

Overall Comparisons

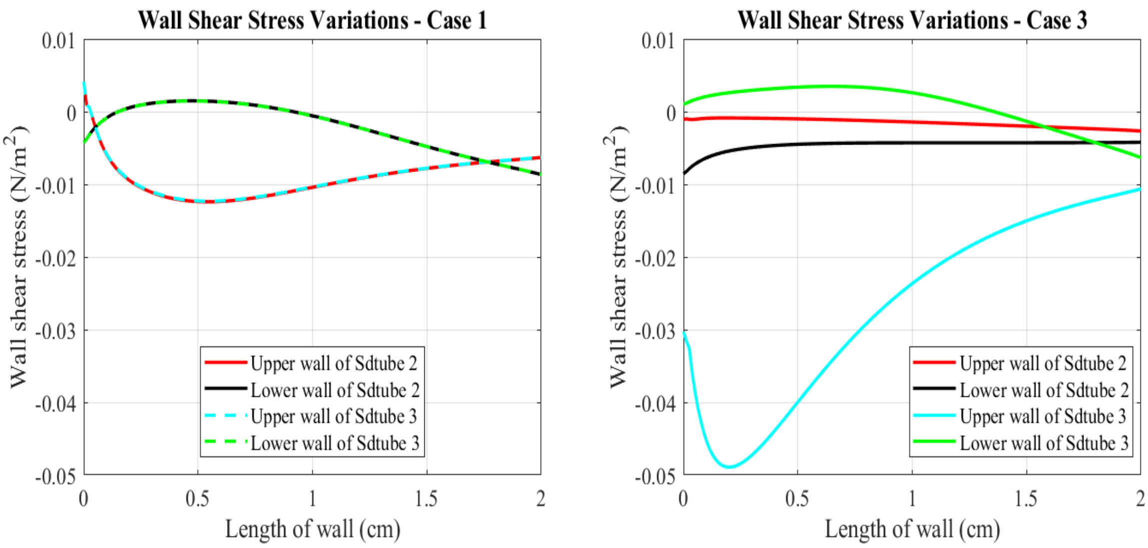


Figure 5.8 - Overall wall shear stress comparisons for case1 and case3.

It is to be noted that a positive value of wall shear stress implies that tangential force per unit area is exerted by the fluid on the walls, whereas a negative value of wall shear stress suggests that force per unit area is experienced by the moving fluid due to the walls.

5.3 Recirculation regions

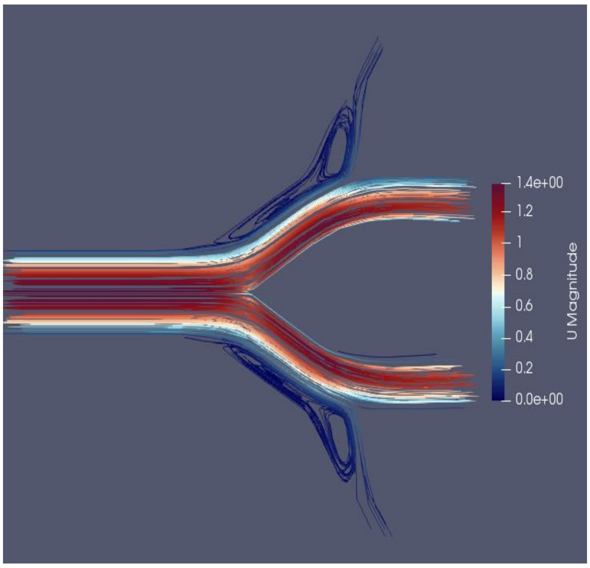
The flowing fluid exerts pressure on its surrounding. This pressure value is dependent on the geometry of the conduit in which it is flowing. Hence, the change in geometry changes the pressure value at different points inside the flow domain. Due to these, some specific regions are created where there are positive and negative pressures. This pressure difference allows the flow fluid to change direction and moves it along the circular path. The motion of fluid along the circular path is known as recirculation.

At the recirculation region, the directions of velocity are reversed. Hence, we observe both the positive values as well as negative values of the velocity across the cross section of the pipe where recirculation is present.

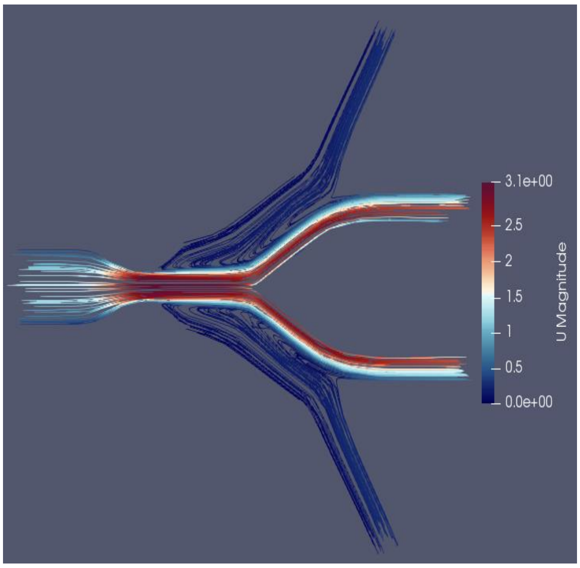
The negative value of velocity is an indicative of the reverse motion of the fluid.

The following results show the streamlines of the air flow inside the respiratory tract. The streamlines are circular where there is recirculation.

Case 1



Case 2



Case 3

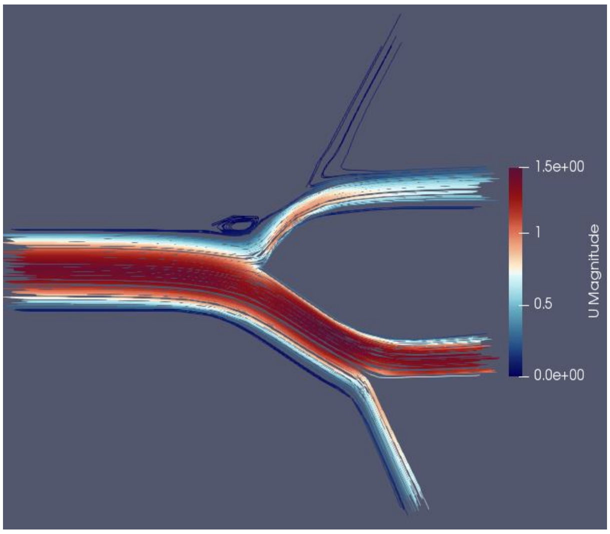


Figure 5.9 – Streamlines of flowing fluid showing recirculation regions for all cases.

Comparisons - Velocity profile to identify recirculation

- At the junction (middle) of parent tube and daughter tube.

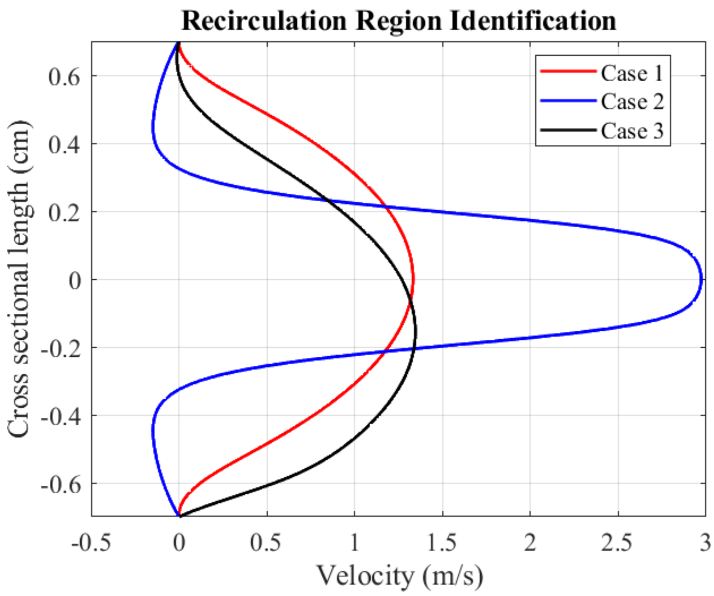


Figure 5.10 – Velocity profile across cross sectional length at the junction (middle) of parent tube and daughter tube.

From above figure, it is concluded that recirculation region exists for case2 on the either side as the flow is symmetrical. For case3, recirculation region exists on the upper side only. It is due to the unsymmetrical nature of the flow. For case1, there is no recirculation region.

- At the junction (middle) of upper daughter tube and sub daughter tube (upper 2nd generation).

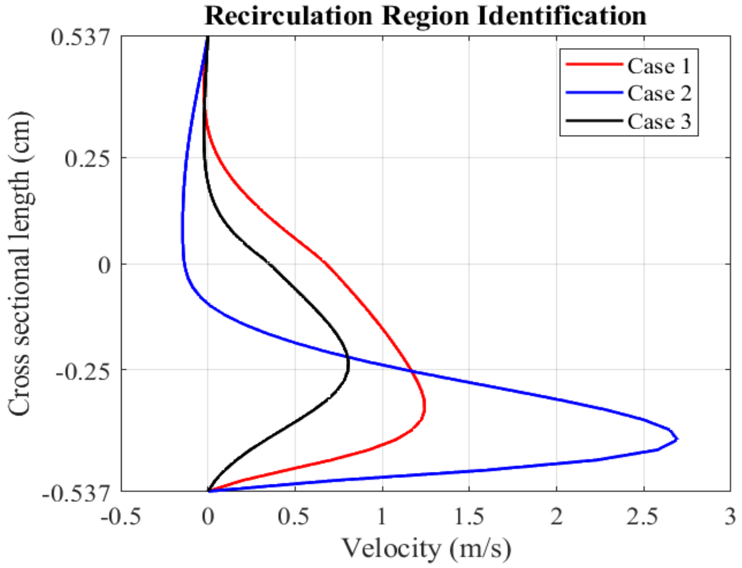


Figure 5.11 – Velocity profile across cross sectional length at the junction (middle) of upper daughter tube and sub daughter tube (upper 2nd generation).

It is concluded that, recirculation region exists for all the cases on the upper side of the cross sectional length taken. Whereas at the lower side no recirculation because the flow is attached to the wall.

- At the junction (middle) of lower daughter tube and sub daughter tube (lower 2nd generation).

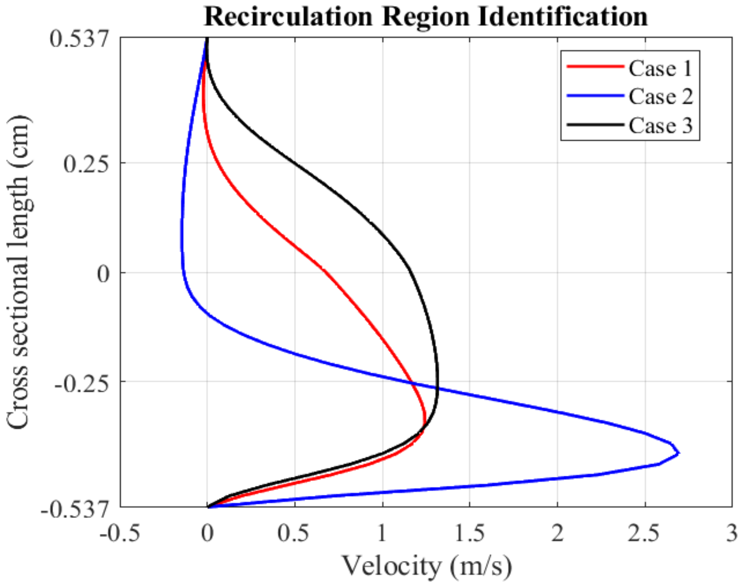


Figure 5.12 – Velocity profile across cross sectional length at the junction (middle) of lower daughter tube and sub daughter tube (lower 2nd generation).

It is concluded that, recirculation region exists for case1 and case2 at the upper side of the cross sectional length taken. Whereas at the lower side for case1 and case2 no recirculation because the flow is attached to the wall. For case3 there is negligible recirculation.

References

[1] Zhao, Y., Brunskill, C. T. and Lieber, B. B., 1997, **"Inspiratory and Expiratory Steady Flow Analysis in a Model Symmetrically Bifurcating Airway".** Department of Mechanical and Aerospace Engineering, State University of New York at Buffalo, NY 14260.

[2] Zhao, Yao, Lieber Baruch B., 1994, **"Steady Inspiratory Flow in a Model Symmetric Bifurcation".** Department of Mechanical and Aerospace Engineering, State University of New York at Buffalo, NY 14260.

[3] Augusto, L. L. X., Lopes G. C. and Gonçalves J. A. S., 2016, **"A CFD Study of Deposition of Pharmaceutical Aerosols under Different Respiratory Conditions".** Vol. 33, No. 03, pp. 549 - 558, July – September. Brazilian Journal of Chemical Engineering. ISSN 0104-6632.