SIMULATION OF THE BREATHING CYCLE IN THE EXTRATHORACIC REGION USING AN IMMERSED BOUNDARY APPROACH

Laura Nicolaou, Tamer Zaki

Imperial College London
South Kensington, London SW7 2AZ
e-mail: laura.nicolaou-fernandez03@imperial.ac.uk

e-mail: tamer.zaki@imperial.ac.uk

ABSTRACT

Simulation of flow and particle deposition in the human respiratory system [1,2] is of interest in order to examine, understand and optimise the effectiveness of inhaled drug delivery. The aerosol is generally inhaled through the mouth, passing by the throat before reaching the lungs. For effective pulmonary drug delivery, the aerosol should reach the targeted area, i.e. the lungs. Therefore, understanding the flow dynamics and deposition patterns of inhaled particles in the mouth-throat region is important in designing effective inhalation devices.

A CFD model is presented for flow in a realistic mouth-throat geometry throughout a breathing cycle. The geometry is obtained by surface extraction from in vivo MRI scans in order to achieve an accurate representation of the flow patterns in the extrathoracic region. Due to the complexity of this geometry an immersed boundary method is used [3]. This greatly simplifies the task of grid generation and discretisation of the governing equations, and eliminates the problems associated with grid quality that exist for boundary-fitted grid techniques. The method has been validated against both numerical and experimental results in the literature and shows excellent agreement with previous studies.

Simulations in the mouth-throat region are run at a tracheal Reynolds number, $Re_d = 1700$, which is an average adult's light-breathing condition. An oscillatory inflow is used to model the breathing cycle and the inhaled particle diameters range from 0.1 to $10\mu m$. Initial results for steady flow are shown in figures 1 and 2. It can be seen that the flow is strongly influenced by the geometry, with highly skewed axial flows due to centrifugal forces induced by the airway bends. The flow accelerates towards the end of the mouth region as the cross-sectional area is reduced, forming a laryngeal jet. The streamlines show recirculation in the larynx as a result of this jet. The flow then undergoes expansion as it enters the trachea with streamlines remaining closer to the posterior side due to the bend in the larynx.

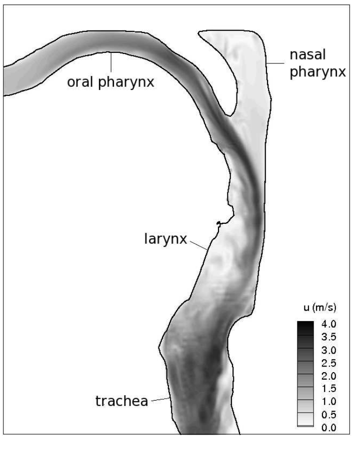

Figure 1: Mid-plane contours of velocity magnitude.

Figure 2: Mid-plane streamlines.

REFERENCES

[1] A.F. Heenan, Experimental measurements and computational modeling of the flow field in an idealized human oropharynx, *Experiments in Fluids* **35**, pp. 70-84 (2003)

[2] S.T. Jayaraju et al., Large eddy and detached eddy simulations of fluid flow and particle deposition in a human mouth-throat, *Journal of Aerosol Science* **39**, pp. 862-875 (2008)

[3] J. Kim et al., An Immersed-Boundary Finite-Volume Method for Simulations of Flow in Complex Geometries, *Journal of Computational Physics* **171**, pp. 132-150 (2001)