Data in Brief 10 (2017) 101–107

Contents lists available at ScienceDirect

Data in Brief

journal homepage: www.elsevier.com/locate/dib

Data Article

Computational fluid dynamics benchmark dataset of airflow in tracheas

A.J. Bates [a,*], A. Comerford [a], R. Cetto [a,b], D.J. Doorly [a], R.C. Schroter [c], N.S. Tolley [b]

[a] Department of Aeronautics, Imperial College, London, United Kingdom
[b] Department of Otolaryngology, St. Mary's Hospital, Imperial College Healthcare Trust, United Kingdom
[c] Department of Bioengineering, Imperial College, London, United Kingdom

ARTICLE INFO

Article history:
Received 3 September 2016
Received in revised form
17 November 2016
Accepted 23 November 2016
Available online 28 November 2016

Keywords:
Tracheas
CFD
Airflow
Goiters
LES
DNS

ABSTRACT

Computational Fluid Dynamics (CFD) is fast becoming a useful tool to aid clinicians in pre-surgical planning through the ability to provide information that could otherwise be extremely difficult if not impossible to obtain. However, in order to provide clinically relevant metrics, the accuracy of the computational method must be sufficiently high. There are many alternative methods employed in the process of performing CFD simulations within the airways, including different segmentation and meshing strategies, as well as alternative approaches to solving the Navier–Stokes equations. However, as *in vivo* validation of the simulated flow patterns within the airways is not possible, little exists in the way of validation of the various simulation techniques. The data presented here consists of very highly resolved flow data. The degree of resolution is compared to the highest necessary resolutions of the Kolmogorov length and time scales. Therefore this data is ideally suited to act as a benchmark case to which cheaper computational methods may be compared. A dataset and solution setup for one such more efficient method, large eddy simulation (LES), is also presented.

DOI of original article: http://dx.doi.org/10.1016/j.resp.2016.09.002

* Corresponding author.
E-mail address: a.bates11@imperial.ac.uk (A.J. Bates).

http://dx.doi.org/10.1016/j.dib.2016.11.091
2352-3409/

Specifications Table

Subject area	Physics
More specific subject area	Fluid mechanics & Biomechanics
Type of data	Table, Figures, CSV file & STL file
How data was acquired	Computed tomography (CT) segmentation and Computational Fluid Dynamics (CFD) simulation
Data format	STL file, CSV file & Analyzed data
Experimental factors	Airway surface data from segmented CT data
Experimental features	Flow data presented at near direct numerical simulation resolution.
Data source location	N/A
Data accessibility	Data available with article

Value of Data

- The tracheal geometry can act as a benchmark geometry for assessment of computational methods for airflow in the tracheal region.
- The extracted data (locations of extraction given) and associated refinement statistics have been provided from the DNS simulation. Researchers can compare their methods precisely with the provided case at the same locations.
- A detailed description of methods for calculating turbulent refinement statistics are outlined for future investigations.
- UA provides a simple measure for the degree of non-uniformity in the flow and is particularly useful for flow in constricted and curved tubular geometries. Benchmark values of this metric are provided in this paper.

1. Data

The dataset includes: the tracheal geometry in STL format (Supplementary 1) and a CSV file of points that represent the centerline (Supplementary 2); extracted fluid mechanical metrics and turbulent statistics (including locations of extraction) from a simulation approaching the level of DNS. Additionally, all methods to calculate fluid mechanical metrics are outlined, including turbulent statistics (Section 4.2) and UA (Section 4.3). UA data has been provided for a number of idealised flow scenarios.

2. Experimental design, materials and methods

2.1. Reference and LES validation data

Two different tracheal airflow datasets are presented: a reference solution and a large eddy simulation. The reference solution is near DNS and is computationally expensive as it tries to resolve all scales, while LES is cheaper as it introduces a model that represents the effect of the unresolved scales on the resolved scales. As the reference solution is near DNS it can be used as benchmark data to assess the validity of a turbulence model, such as LES, and also to assess the degree of turbulence within the flow.

Analysis of convergence data was performed on one geometry (case B in Bates et al. [1] and shown in Fig. 1). The data was extracted from this geometry as it demonstrates large curvature and constriction. Hence represents a challenging benchmark from a flow computation point of view. A highly