FLOW ANALYSIS OF COPD EFFECTS IN HUMAN AIRWAYS

MUHAMAD ZUHAIRI BIN SULAIMAN

A thesis submitted in fulfillment of the requirements for the award of the degree of Master of Engineering

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

NOVEMBER 2008
ABSTRACT

Airflow inside human airway is difficult to study experimentally. Therefore, Computational Fluid Dynamics has been used to study Chronic Obstruction Pulmonary Disease (COPD) inside human airways. First and second generation of human airway (trachea and both left and right bronchi) with two COPD at location T2 and L2 of the airway has been constructed and simulated by numerical method. Obstruction at each location of COPD has been varied for four sizes; 20 percent, 40 percent, 60 percent and 80 percent of the airways diameters. Laminar flow with Re = 1710 and turbulent flow with Re = 5000 with velocity inlet boundary condition were used. Pressure redistribution, velocity redistribution and fluid shear stress analysis has been observed. Both constrictions gave significant effect to the airflow inside the human airway. The correlation of these parameters plotted. For Case 1, the G2.2 region experienced velocity increase as the Tumour at L2 grows. For up to 60 percent growth of airway diameter, the velocity correlation shows a slow velocity increment. However, tumor growth above this value results in high velocity increase. Pressure distribution at the G1 to G2.2 bifurcation was observed to increase due to momentum effect of the Tumour at inlet location L2. Fluid shear stress correlation at tip of Tumour was observed to increase nonlinearly. Case 2 shows the significant pressure drop after the Stenosis at location T3. Stenosis with 80 percent diameter shows the highest pressure drop downstream. Fluid shear stress increased drastically in 80 percent of Stenosis diameter in parallel to the velocity increase and area decrease. In comparison of both COPD, Case 1 (Tumour) produce less peak pressure effect of 101329.5 Pa compare to 102000 Pa for Case 2 (Stenosis). Fluid shear stress and velocity effect was also higher in Case 2 compared to Case 1. It is observed Case 1 produces more serious COPD effect compared Case 2.
ABSTRAK

CONTENTS

CHAPTER TITLE PAGE

DEDICATION i
ACKNOWLEDGEMENTS iv
ABSTRACT v
ABSTRAK vi
TABLE OF CONTENTS i
LIST OF TABLES iv
LIST OF FIGURES v
LIST OF SYMBOLS viii
LIST OF APPENDICES xii

1 INTRODUCTION

1.1 Background 1
1.2 Computational Fluid Dynamics 3
1.3 Human Fluids Mechanics Studies 4
1.4 Objectives 8
1.5 Scope of Study 8
1.6 Project Flow Chart 9
2

LITERATURE REVIEW
2.1 Introduction 10
2.2 Previous Studies 10
2.2.1 Airways Chronic Obstruction
Pulmonary Diseases (COPD) Studies
2.3 Airways Anatomy 14
2.4 Pulmonary Behaviours 16
2.4.1 Flow, Volume, and Pressure 16
Distributions
2.4.2 Respiratory System 17
2.4.3 Airflow in Respiratory Tracts 18
2.5 Airways Diseases 19
2.6 Airway Flow Pattern 21

FLUID FLOW THEORY
3.1 Introduction 23
3.2 Incompressible Flow 23
3.3 Fluid Flow Theory 24
3.3.1 Navier-stokes Equations 24
3.3.2 Various Type of flow 25
3.3.3 Simulation Software 26
3.4 Finite Volume Method 26
3.5 K-epsilon Turbulent Model 27

METHODOLOGY
4.1 Introduction 30
4.2 Airways model 30
4.2.1 Normal Asymmetric Models 32
4.2.2 COPD Asymmetric Models (Case 1) 33
4.2.3 COPD Asymmetric Models (Case 2) 34
4.3 Air Properties 35
RESULTS AND DISCUSSIONS

5.1 Model Validation 41
5.2 Grid Independence test result 42
5.3 Results 43
5.3.1 Case 1: COPD at Location L2 (Laminar Flow) 43
5.3.2 Case 2:COPD at Location T2 (Laminar Flow) 50
5.3.3 Normal Airway flow 55

CONCLUSION AND RECOMMENDATIONS

6.1 Conclusion 58
6.2 Recommendations 59

REFERENCES 60

APPENDIX A-D 62
CHAPTER 1

INTRODUCTION

1.1 Background

Human airways flow studies have been carried out by many researchers previously either by experimental or numerical method (Table 1.1 and Table 1.2). Instead of only experimental work, simulations using numerical methods have been one of the famous alternative study methods. However, solving the numerical equation require a powerful computing hardware to solve the discretised mathematical equations. Regarding the development of the computer software and hardware capability, numerical method with make use of computer for fluid flow, which is also known as Computational Fluid Dynamic (CFD), has become well known tools in the engineering analysis. The CFD code capabilities that are available in the market nowadays are proven for providing almost sufficient and good results to solve various engineering analysis either on heat, flow and structure studies. The CFD method also provides an alternative method in engineering areas that are very difficult to study experimentally.

Geometry of human airways as Calay et al. (2002) describe that the airways network has quite small dimension and the smaller airways deep down into the lungs are inaccessible. Because of this geometry, it is difficult to simulate the highly asymmetric
branching structure together with curvatures of the bifurcation in physical models. Later, simulation by using numerical method was determined as one alternative for this kind of flow studies.

In order to study the human airway flow, human anatomy and the respiration process must be clearly understood. Respiration process occurs in human pulmonary organ for delivering the oxygen and discharge the carbon dioxide. The human airways consist of upper respiratory tract, which include the nose, mouth, pharynx, larynx and trachea. Afterwards, it divides into another two branches that are called ring and left bronchi of the lung up until the alveolar. Figure 1.1 shows the picture of human anatomy structures.

Figure 1.1: Human airway anatomy structures [20].
Human airflow intake varies with time depending on the metabolism activities. According to the literatures, normal human breathing airflow is laminar along the trachea, but becomes turbulent in the case of heavy breathing due to the increase of oxygen supply into the body during heavy exercise.

Airflow studies are important in biomedical field especially to have better understanding about several flow parameters inside the human respiratory system. The airflow studies encouraged the researchers to collect and analyse detailed variables and parameters behaviour of the flows, and later to come out with suggestions and solutions for treating and curing the airway diseases of Chronic Obstruct Pulmonary Disease (COPD), such as Stenosis and Tumour which are harder to do by experimental analysis. The studies, in the end will bring an improvement in the airways stenting technology. Along with the airflow studies, artificial human airway model developers could get precise data and information of the human airway’s physical parameters in order to design the artificial airway for treating various types of the COPD patients.

Figure 1.2 shows the first generation, G1 (trachea) and second generation, G2 (left and right bronchi) of the normal human airways.

Figure 1.2: First and second generations of human airway Calay C. K. et al. (2002)
1.2 Computational Fluid Dynamics (CFD)

Within the last decade, computational fluid dynamics (CFD) has become a popular choice among the researchers. CFD methods were used by the researchers to study the fluid dynamics and aerosol particle behaviour inside the airways. Additionally, CFD is also widely used to study the blood flow in human blood vessel. The capability of the methods is proven to give good approximation results.

CFD became a useful choice in the study of airways because of its capability to investigate the complex model that is difficult to do by an experimental study. Literature shows that the human airways consist of 25 generations and the diameter of each generation will reduce to less than 1 mm in the last generation, alveolar. Therefore, due to these complexities, CFD could give a better alternative for the studies. The researchers also could reduce research costs; with the benefits of code flexibility, the researcher can easily change the parameters of the studies accordingly to meet their study purposes.

Nowadays, the invention and innovation of the technology have brought computer hardware capability faster and cheaper which encouraged the researchers in utilising the benefits of using the CFD method. In addition, the increasing of CFD software’s development gives researchers the choice of selecting suitable commercial CFD codes to obtain the approximation of preliminary analysis result.

In this study, commercial CFD software will be used to perform various type of analysis on selected airways, and later the parameters and variables obtained in the study area are discussed to achieve the objectives of the study according to the scope of study.
1.3 Human Fluid Mechanics Studies

According to Luo X. Y. et al.(2004), the Reynolds number (Re) in the first generation of human airway (trachea) varies from 800 in normal breathing until about 9300 in heavy breathing depending on the diameter of airways. Therefore, flow in the trachea could either be experiencing laminar and turbulent flow. Usually the first two generation of trachea-bronchial airways are in turbulent state, later the flow decelerates in the subsequent branches due to the branching effect.

Table 1.1 shows the summary of the studies by previous researchers on the human airway flow studies.

Table 1.1: Overview of human airways fluid mechanic studies

<table>
<thead>
<tr>
<th>Year</th>
<th>By</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>2002</td>
<td>Calay, R.K. et al.</td>
<td>Numerical simulation of the unsteady flow pattern within human lung of three dimensional asymmetric bifurcation model of central airway base on morphological data of Horsfield. The study included the time dependent flow and up to third human airway generation.</td>
</tr>
<tr>
<td>2002</td>
<td>Nowak, N. et al.</td>
<td>CFD simulation of airflow and aerosol deposition in human lung base on Weibel model and CT scan of a cadaver lung cast. Gambit and Fluent 5 were used.</td>
</tr>
<tr>
<td>2002</td>
<td>Liu Y. et al.</td>
<td>The study covered modelling of flow in fifth and sixth generation of human airway. The model was symmetrical.</td>
</tr>
<tr>
<td>2003</td>
<td>Liu, Y. et al.</td>
<td>Modelled the human airway flow in asymmetric human</td>
</tr>
<tr>
<td>Year</td>
<td>Authors</td>
<td>Description</td>
</tr>
<tr>
<td>------</td>
<td>---------</td>
<td>-------------</td>
</tr>
<tr>
<td>2004</td>
<td>Hegedus, Cs. J. et. al</td>
<td>Study was about exact mathematical description of the geometry of human airways by elaborating numerical methods. Single and multiple airways geometries and computational grid were generated.</td>
</tr>
<tr>
<td>2004</td>
<td>Juan, R. C</td>
<td>Study of tracheal and central bronchial aerodynamics using bronchoscopy and CFD using anatomical realistic model. Pressure and flow pattern were computed.</td>
</tr>
<tr>
<td>2006</td>
<td>Taylor, A. B. et al.</td>
<td>Study the three dimensional simulation of reactive gas uptake in single airway bifurcation</td>
</tr>
<tr>
<td>2006</td>
<td>Jin, H. H. et al.</td>
<td>Used LES for inhaled particle deposition within the human upper respiratory tract considering breathing intensity of $Q=30,60,90$ L/min.</td>
</tr>
<tr>
<td>2005</td>
<td>Yang, X. L. et al.</td>
<td>Study of inflammation of COPD lead to narrowing and obstructing airway using 3D incompressible laminar Navier-Stokes equation. 5th until 7th generations were considered.</td>
</tr>
<tr>
<td>n.a</td>
<td>Koombua, K. et al.</td>
<td>Study on inhalation induced stresses and flow characteristics in human airways through fluid-structure interaction analysis</td>
</tr>
</tbody>
</table>
Table 1.2: Summary of the human airways flow studies

<table>
<thead>
<tr>
<th>Researcher</th>
<th>Flow pattern/flow structure</th>
<th>Diseases (COPD)</th>
<th>Fluid structure interaction</th>
<th>Particles study/Aerosol</th>
<th>CFD</th>
<th>Experimental</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calay R. K et al (2002)</td>
<td>😊</td>
<td></td>
<td></td>
<td></td>
<td>😊</td>
<td></td>
</tr>
<tr>
<td>Zhang, Z. et al. (2002)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>😊</td>
<td></td>
</tr>
<tr>
<td>Yang, X. L. Et al.(2005)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>😊</td>
<td></td>
</tr>
<tr>
<td>Wall W. Et al.( 2007)</td>
<td>😊</td>
<td></td>
<td></td>
<td></td>
<td>😊</td>
<td></td>
</tr>
<tr>
<td>Heistracher T. Et al (1994)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>😊</td>
<td></td>
</tr>
</tbody>
</table>
1.4 Objectives

The objectives of the thesis are to get the correlations of the human airway flow parameters with the present of COPD diseases. The influenced parameters such as pressure redistribution, airflow velocity and fluid shear stress and fluid structure behaviours in human airways during inhalation for both normal and Chronic Obstructive Pulmonary Diseases (COPD) airways geometry are being studied. Computational Fluid Dynamic software used to study the effect of various stages of (COPD) along the first generation (G1) and second generation (G2) of human airway.

1.5 Scopes of the project

The scopes of the projects are:

a. The focus anatomy is 1st to 2nd generation of human airways known as trachea-bronchial bifurcation.

b. Flow studies with COPD will be investigated.

c. Three-dimensional flow analysis using commercial software (COSMOS FLOWORKS 2007).

d. Fluid flow behaviours and structure will be investigated.

e. Incompressible fluid for both laminar and turbulent flow will be considered.
1.6 Project flow chart

The flow chart describes the sequences of the project process from airway geometry modelling until the results data evaluation.

![Project Flow Chart](image-url)

**Figure 1.3** Project Flow Chart
REFERENCES


[20] [http://www.oandp.org/Assets/Images/resident/7b.jpg](http://www.oandp.org/Assets/Images/resident/7b.jpg)


