Computational Fluid Dynamics Study in Biomedical Applications: A Review

Ernie Illyani Basri1, Adi Azriff Basri2, Vizy Nazira Riazuddin3, Siti Farhana Shahwir4, Mohammad Zuber5, Kamarul Arifin Ahmad6

123456 Department of Aerospace Engineering, Faculty of Engineering, Universiti Putra Malaysia
43400 Serdang, Selangor Darul Ehsan, Malaysia.

Received [18th March 2016]; Revised [15th April 2016]; Accepted [9th June 2016]

Abstract: Computational Fluid Dynamics (CFD) is a widely adopted methodology of computer-based simulation in order to solve complex problems in many modern engineering fields as well as biomedical field. CFD is becoming a key component in developing updated designs and optimization through computational simulations, resulting in lower operating costs with enhanced efficiency. Even though biomedical application is pertaining to the complexity of human anatomy and human body fluid behaviour, the recent CFD in biomedical application is more accessible and practicable due to the availability of high performance hardware and software with advances in computer sciences. Many simulations and clinical results have been used to study the analyses in biomedical applications, particularly in blood flow and nasal airflow. The study of blood flow analysis includes the circulation of blood of ventricle function, coronary artery and heart valves. Meanwhile, the nasal airflow analysis consists of the basic airflow in human nose, drug delivery improvement and virtual surgery. Therefore, this review discusses the essential methodology of CFD as a reliable tool for researchers and medical scientist in understanding the physiology and pathophysiology of cardiovascular system and respiratory system through simulation. CFD plays a major role as a decision support prior to undertaking a real commitment to execute any medical design alterations and provide the direction to develop medical interventions.

Keywords: Computational Fluid Dynamics (CFD), biomedical applications, blood flow, nasal air flow
I. INTRODUCTION

Significant advances in the computer-based simulation are growing rapidly in accordance with its importance use and rapid acceptance for various applications. Computational Fluid Dynamics (CFD) is one of the widely adopted methodologies of computer-based simulation which defined as a branch of fluid dynamic that uses numerical solutions of the governing equations for simulating real fluid flows [1]. The applications of CFD received tremendous attention and widely adopted for solving complex problems in various modern engineering fields including electronics packaging, chemical engineering, turbines, external and internal environmental architectural design, marine and environmental engineering, metrology, hydrology and biomedical engineering [2,3]. CFD is becoming a key component in developing updated designs and optimization through computational simulations. However, the recent CFD is still emerging in biomedical application due to the complexity of human anatomy and human body fluid behaviour. Nevertheless, it is becoming more accessible and practicable by virtue of the advent of digital computer with high performance hardware and software[2]. Since the importance of knowledge of body fluids and system components are expected to perform and bio-fluid physiology study has been growing over the last several years, the advancement of biomedical practices and technology has been stimulated. The biomedical research with the aid of CFD software is still emerging which incorporated the physiology and pathophysiology of cardiovascular system and respiratory system through simulation.

Various researches of simulation and clinical results had been studied, particularly the analyses of blood flow and nasal airflow. In most researches, the blood flow analysis studied the circulation of blood of ventricle function, coronary artery and heart valves. Meanwhile, the nasal airflow analysis studied the basic airflow in human nose, drug delivery improvement and virtual surgery. The examples of CFD simulations applied in cardiovascular and respiratory systems are depicted as in Figure 1 and Figure 2, respectively.
To date, CFD is increasingly applied in a wide range of critical engineering system, which incorporating an expert area of mathematics and a branch of fluid mechanics. CFD modelling has already received tremendous attention from biomedical researches along with the development of medical devices. Furthermore, detailed characterization of complex physiology and the measurement of computation metrics can be determined by incorporating both
imaging procedure and CFD simulation [4]. CFD models are continuously being interpreted into clinical tools for physicians to apply across the spectrum of various diseases of cardiovascular and respiratory systems. Therefore, this paper explores the CFD study using the state-of-the-art in clinical area, highlighting the biomedical applications.

II. RESEARCHES OF BIOMEDICAL IN CFD APPLICATIONS

CFD plays an important role by offering chances for simulation prior to undertaking real commitment to develop medical interventions in the correct direction and to execute any medical design alteration [7]. The researches of biomedical CFD applications received tremendous attention in the past few years due to the importance of computational medical simulations of circulatory functions. The biomedical CFD applications for cardiovascular and respiratory systems are discussed in the subsequent sub-section.

A. Cardiovascular systems

The adoption of CFD directed towards theoretically highly beneficial within cardiovascular medicine, clinical trials, improving diagnostic assessment and device design in order to predict physiological responses to intervention and compute the prior hemodynamics parameters that unable to be measured [4]. Researches of CFD applications in regards to cardiovascular system are addressed the associated methodological, analytical assessment and result of three main physiologies of heart functions namely valves, arteries and ventricle.

Cardiovascular is pertaining to the heart disease which is the major cause of death around the world [8]. Heart valve disease is the common disease which causes by the narrowing of aortic valve or leaking of blood flow on the valve leaflet. Recent study, Basri et al. [5] studied the hemodynamic properties of the effect different valve opening for 45, 62.5˚ and fully opening by using the combination of magnetic resonance imaging (MRI) and CFD simulation. The authors investigated the hemodynamic properties in terms of pressure, velocity and wall shear stress to determine blood behaviour of severed aortic stenosis. The result shows the significant decrease of blood pressure on the small valve opening, which caused the obstruction of blood ejection due to narrowing of valve. Hence, the study found that the lower leaflet opening shown detrimental effect on blood flow and induced higher stress on the leaflets. Besides that, Basri et al. [9] compared the normal aortic valve (fully open) and stenosed aortic valve (62.5 opening) through the study of hemodynamic properties. The authors used CFD simulation on a 3D aortic valve which imported from MRI data scan. The study observed an increased velocity by 13.7% and a reduced of 2.9% in the mass of blood entering at the aortic branches of stenosed aortic valve compared to normal aortic valve. Thus, the study proved a significant reduction of blood supply to provide blood to head, neck and arm of human body.

Meanwhile, Tan et al. [10] studied on a patient specific assessment of stenosed aortic valve and compared the aortic flow pattern before and after deploying transcatheter aortic valve. The authors carried out CFD simulation that
incorporated MRI data scan to investigate the flow patterns of thoracic aortas in terms of velocity profile and wall shear stress. The result of flow patterns shows a reduction of 20% of jet flow at an instantaneous velocity streamlines and a lower time-averaged wall shear stress after implantation. Hence, the combination of imaging and simulation approach in this study led to an individual evaluation of the disturbed blood flow patterns and wall shear stress on the aorta before and after undergoing the implantation procedure. Jamuna and Abnurajan [8] measured velocity and pressure of the blood flow through a patient-specific aorta in different conditions. Those conditions are normal aorta, aorta with plaque at the valve sinus side and aorta with needed bi-leaflet valve implant. The authors incorporated a computed tomography (CT) image of a patient-specific and analysed by using CFD simulation. It is observed that the blood pattern after implanting a valve is similar to normal aorta, where an increased percentage of velocity and blood pressure are shown to be 58.5% and 81.8% respectively. Sirois et al. [11] also studied the implantation of aortic valve on a patient-specific by using CT images and CFD simulations. The authors performed a quantitative analysis of the hemodynamic in terms of blood flow patterns before and after implantation procedure. A reduction of pressure drop at 25.27 mmHg and increased of effective orifice area from 0.53 to 1.595 cm$^2$ shown a significant result following the valve implantation.

On the other hand, Sun and Xu [12] reviewed on the applications of CFD simulations based on 3D luminal reconstructions in regards to coronary artery disease. The study is carried out to analyse the behaviour of circulatory blood flow mainly the local flow fields and flow profiling by virtue of changes of coronary artery geometry. The assessment of hemodynamic properties are carried out considering parameters of wall shear stress, oscillatory shear index and average wall shear stress gradient of 30 patients. Gao et al. [13] also studied the coronary artery disease of stent implantation as an interventional procedure for the disease treatment. The authors compared the blood flow before and after stent implantation and analysed the parameters in terms of wall shear stress and blood velocity. From the study, the wall shear stress and blood velocity are greater at the region of stenosis prior to implanting the stent of by which the result shows a reduction of maximum flow rate in coronary artery and an increment value of wall shear stress after the implantation procedure. Chaichana et al. [14] studied the hemodynamic effects of simulated plaque in left coronary artery models of patient-specific coronary stenosis. Three parameters are measured namely wall shear stress, pressure gradient and flow velocities by using CFD analysis and compared between the presence and absence of plaques in the left coronary models during cardiac cycle. It is observed that highest pressure gradient in stenotic regions caused by the plaques and lower flow velocity areas found at postplaque locations but wall shear stress is similar at the stenotic regions.

B. Respiratory systems

Despite the importance of cardiovascular system for blood circulation and nutrients transportation throughout the human body, respiratory system also plays an essential role for human lung function primarily for nasal breathing. A CFD-based analysis provides better understanding of airflow characteristic incorporated with fluid
dynamics in nasal cavity to obtain functional and anatomical data. Researches of CFD applications in regards to respiratory system received attentions concerning the basic airflow studies on the physiology of nose, drug deposition and virtual surgery of surgical intervention. Recent studies conducted by combining computational analysis with imaging to gain significant realistic numerical simulations of respiratory system.

Segal et al. [15] studied the differences in respiratory flow patterns of four difference human nasal cavities by using MRI scans and CFD simulations. The study is conducted by performing numerical simulation of steady state inspiratory laminar airflow for flow rate of 15 L/min and compared the measurements in terms of streamline patterns, velocities and helicity values. The authors observed that the majority of flow passed through the middle and ventral regions of nasal passages; however it is found that there is variance of the amount and location of swirling flow among subjects. Wen et al.[16] also simulated steady laminar nasal airflow for flow rate of 7.5 to 15 L/min to present flow patterns between the left and right nasal cavities by adopting CFD simulation software (FLUENT) and CT scan images of human nasal cavity models. The authors measured the flow patterns features included high velocities in the constrictive nasal valve area region, vortex formations posterior to the nasal valve regions and high flow close to the septum walls. The results shows the nasal resistance value within the first 2-3 cm contribute up to 50% of the total airway resistance and vortices were found at the upper olfactory region and posterior to the nasal valve region. Croce et al.[17]also simulated steady state laminar airflow for flow rate of 353 ml/s in left and right nostrils using CFD simulation software (FLUENT) from CT scan images of a plastinated head using a commercial software package AMIRA (Mercury Computer System, Berlin). The authors described the flow patterns in a physiologically realistic binasal model considering the pressure drop. The results found that the major total pressure drop in the nasal valve region and predominant airflow in the inferior median part of nasal cavities. Vortices also are observed downstream from the nasal valve and towards the olfactory region.

Other than the basic airflow studies on the physiological function of the nose, drug deposition is of fundamental importance in the treatment of different lung disease and allergies. The recent study of CFD in relations to drug deposition received great interest in order to characterize the local deposition patterns and optimize drug delivery in the respiratory system. Bahmanzadeh et al.[18] studied the effect of endoscopic sphenoidotomy surgery on the flow patterns and deposition of micro-particles in the human nasal passage and sphenoid sinus. The authors presented transient airflow patterns of pre- and post- surgery during a full breathing cycle under cyclic flow condition. The transport and deposition of inhaled micro-particles are evaluated by using Lagrangian approach to determine the unsteady particle which entering the nasal airway for inhalation phase of breathing cycle. The study found that the increased airflow due to sphenoidotomy and increased deposition of micro-particles in the sphenoid region. In the post-operative case, 25µm particle size is observed to be able to penetrate into the sphenoid region and highest deposition for 10µm particles at about 1.5% occurred during resting breathing.
Dastan et al. [19] studied the deposition of fibrous particle in different human nasal passages by using CFD simulations. The authors developed an in-house code to solve the combined equations of translational and rotational of motion of ellipsoids for fiber transport and deposition in the nasal airways. The result shows a significant effect of deposition fraction by virtue of variation of nasal airways. The deposition fraction is highly affected by the nasal geometry and of airflow rate in the nasal valve and main airway regions. Hence, it is proven that the aerodynamic diameter based on the Stokes equivalent diameter employed in the impaction parameter could collapse the simulation data of spherical and fibrous particles to a single curve.

Abouali et al. [20] studied the airflow distribution and particle deposition in the nasal airway, maxillary and frontal sinuses on the developed virtual uncinectomy and middle meatal antrostomy. The study considered the inhalation of macro and nano-particles to determine the penetration of airflow into the sinus cavity. The micro-particle consists of the evaluation of the path and deposition of particles in the nasal passages and maxillary sinuses by using a Langrangian trajectory analysis approach. Meanwhile, the nano-particles included the transport and deposition analysis by using a diffusion model. The rate of particle deposition in the maxillary and frontal sinuses are analyzed and compared between pre and post-surgery conditions. The result shows that almost no particles entered the sinuses in the pre-operative condition. However, the inhaled nano and microparticles easily entered the sinuses due to the increase of airflow penetration into the sinus cavity after surgery.

Despite that, virtual surgery in relation to CFD simulation also received great interest as to determine the best possible surgical treatment in a constricted airway [21]. In most studies, the virtual surgery consists of removing one or both of the obstruction in different proportions in order to enhance the nasal airway comparing to its baseline condition.

The recent study by Moghadas et al. [22] studied the effect of septal deviation on the flow patterns and deposition of micro/nanoparticles in the realistic human nasal airways before and after septoplasty. The authors simulated the steady airflows through the nasal passage by using Eulerian and Lagrangian approaches for nano- and micro-particles. From the simulation, the results shows the flow field and particle deposition depending on the passage geometry. For micro-particles, the deposition rate with septal deviation is higher compared to the normal and post-operative passage. Meanwhile, the deposition of nano-particles shows similar trends for both normal and post-operative passage. Hence, the aid of simulation provides a suitable tool for predicting the airflow and particle deposition patterns in the nasal passages that specific surgical interventions would produce.

Xiong et al. [23] compared nasal airflow after two different surgical interventions involving three facets such as opening the paranasal sinuses, excising the ethmoid sinuses, and excising or preserving the uncinate process, in a cadaveric head model through CFD simulations. The study found a significant large nasal cavity airflow velocity changes are apparent during the procedure of uncinate process and similar nasal cavity airflow when preserving the uncinate process. The uncinate process excising procedure shows a greater increase in airflow volume compared
with the uncinate process preserving procedure. Previously, Xiong et al. [24] carried out a numerical simulation of nasal cavity airflow pre and post virtual functional endoscopic surgery (FESS) with the aid of CFD simulations (FLUENT). The authors aim to investigate and numerically visualize the airflow trace, distribution, velocity, air pressure and airflow exchange between the nasal cavity and paranasal sinus on a normal adult subject. The result shows an increased airflow distribution in the maxillary, ethmoid and sphenoid sinuses, and the increment of 13% through the area connecting the middle meatus and the surgically opened ethmoid.

On the other hand, Garcia et al. [25] used CFD simulations of medical imaging software (MIMICs, Materialise) to study the airflow, water transport, and heat transfer in the nose of an Atrophic Rhinitis (AR). The subject is a patient that received a treatment of a nasal cavity-narrowing procedure which implanted a rib cartilage under the mucosa along the floor of the nose and removed septum spur. The reconstructed nose is simulated and the nasal airflow was assumed as laminar with 15 L/min corresponding to resting breathing rate. The simulation shows that the anthropic nose lead to unconditioned inspired air as effectively as the healthy geometries.

III. CFD MODEL CONSTRUCTION

According to Lee [7], all the details of flow-related information and overall performance assessments of CFD are classified into three main facets, namely pre-processor, solver (processor) and post-processor. The procedure of constructing CFD modelling is described as in Figure 3.
Referring to Figure 3, the pre-processor is the input of modelling element that includes problem thinking, discretisation (meshing) and generation of a computational model. The solver is the processing element, where it involves numerical solution methods by virtue of governing equations and algebraic solution. The post-processor is the output element where the computational results are visualized by achieving an acceptable convergence of the equations of state that had been solved for each cell.

IV. MERITS AND LIMITATIONS OF BIOMEDICAL APPLICATIONS IN CFD

CFD has received increasing interest from mathematical curiosity to become an important technique to study complex physiological flows pattern and demonstrating their potential especially in cardiovascular and respiratory systems. To date, CFD has been adopted by medical researchers to facilitate in predicting the characteristic of
circulatory blood flow inside the human body and airflow in human nasal breathing. Hence, it offer benefits such as lower the chances of post-operatives complications, facilitate in developing better surgical treatment, high efficiency with less destructive medical equipment and convey a good understanding of biological procedures [7]. From theoretical point of view, CFD provides benefits by concentrating on the construction and solution of governing equations and the study of numerous approximations to these equations. Meanwhile, the experimental and numerical approaches highlighted the merit of CFD as an alternative cost-effective means of simulating real fluid flow, particularly involving human body systems. Hence, it provides detailed visual and comprehensive information when comparing the fluid dynamics of analytical and experimental approaches.

Despite the merits of CFD, there are also some limitations of applying CFD. CFD is limited to describe physical models and quality of input data of real world processes in order to determine the accurate CFD solutions such as turbulence, multiphase flow, and compressibility. Thus, numerical results must be thoroughly analysed and examined in order to properly make critical judgements about the computed results. Furthermore, numerical errors may occur when solving equations on a computer invariable such as round-off error and truncation error. This is due to the practicability of CFD depending on several factors such as specific materials and process, accurate algorithm for the governing equations, powerful CFD packages, as well as high speed and large computers.

V. FUTURE OF CFD IN BIOMEDICAL ENGINEERING

Rapid developing of an outstanding major computational modelling and technological challenges, which directed towards the evolution of CFD has recognised by regulatory authorities. Excellent creative models and the development of novel applications for simulating complex fluid mechanics challenges in regards to human anatomy of cardiovascular and respiratory systems are now being progressively applied with the ability of CFD simulation programs. Therefore, it is important to demonstrate the effectiveness of simulations results relative to invasive measurement through observational trials, particularly in multicentre clinical studies. Clearly, that these methods will direct towards high potential to change clinical practice that benefits to patients, health providers and clinicians.
REFERENCES


