Journal of Emerging Technologies in Mech. Engineering Research" (ISSN:0976 – 2558), Volume 4, 2013

Understanding Nasal Airflow in a 2-D Geometry

Mohammad Faisal Khan

Department of Aerospace engineering,

University of Petroleum & Energy Studies, Dehradun, Uttarakhand, India

Faisal.khan@stu.upes.ac.in, faisalmechster@gmail.com

ABSTRACT

Computational fluid dynamics has been very popular tool in medical field since last two decades, in order to study the functioning and causes of obstacle in functioning of various organs inside human anatomy. Developments in the field of imaging techniques, computational methods and computer aided design now enable the prediction of flow behaviour and flow properties in cardiovascular organs, blood cells, nasal cavity, lungs, blood pumping of heart etc. Experimenting human to study flow behaviour of the fluid (air, blood or any liquid drug) inside the body is a big challenge as compared to experimental techniques available in medical field. Computational Fluid Dynamics plays a vital and significant role in this field to provide accurate scenario. Computational modelling and simulation gives accurate predictions of fluid flow properties, based on fundamental physical governing laws. The CFD technique can also applied to air breathing in nasal cavity which otherwise cannot be captured by standard diagnostic methods.

The process of preparing 3D geometry of a nasal cavity is based on the images obtained from computer tomography (CT) scans, MRI or acoustic rhinomanometry. This procedure is complex, time and cost involving and access to these images is very much limited. In this paper, a 2D geometry of nasal airway has been used to study the flow property inside the airway during respiration to provide the basic fundamentals of computational fluid dynamics, characterization of flow properties to understand the physiology and patho-physiology and limitations of CFD application in nasal airflow in a 2D geometry. The Navier-Stokes and continuity equations were solved numerically to determine the turbulence airflow pattern inside the nasal cavity. The geometry is prepared in Gambit and solved in simulation software Fluent.

Keywords— Nasal airflow, CFD simulation, biomedical engineering, nasal cavity, Computational Fluid Dynamics.

I. INTRODUCTION

The nose is the major portal of air exchange between the internal and external environment. The nose participates in the three vital functions i.e. conditioning inspired air towards a temperature of 37˚C and 100% relative humidity, olfaction and providing local defence by filtering inhaled particulate matter and gases [1]. The nasal cavity also filters the minute particles of size 10 micron and these particles will be removed from the nasal cavity by mucociliary clearance [2].

The normal nose is capable of sustaining 20 to 30LPM, of airflow, if larger volumes are required, oral breathing must supplement nasal breathing. However, the net humidity loss increased by 42% when breathing is oral [3]. Inspiring warm air leads to a decrease in nasal resistance whereas inspiring cold air leads to increase the resistance [4]. The flow pattern of heat, gas and minute particles greatly depend upon how the air is flowing inside the cavity. Therefore the knowledge of airflow field is basic approach to understand the nasal physiology [5].

In a healthy adult, total nasal airway resistance remains constant, but the airflow of each nasal cavity varies in a reverse order. This alteration in airflow is called nasal cycle and reflects in vascular engorgement of the turbinates and septal tuberculum [1][6][7]. Often anatomical or pathological disorders of nasal functioning results disturbance and obstruction to nasal airways and increase the resistance of flow, hence leads to functional rhino-surgery. Currently available techniques provides only limited specific information i.e. narrowest cross section area or overall resistance and peak flow but it is not much informative and helpful to contribute the management of nasal airflow disorders. It is difficult to position the measuring device within the cavity to measure the narrow calibre of the nasal passage, moreover the device obstruct the airflow which is primary consideration to measure inside the nasal cavity. Under such circumstances where incomplete information is available, ENT surgeons have to be dependent on the clinical judgements merely to decide whether a patient with the symptoms of nasal obstruction would be benefited from the surgery or not [8].

Computational fluid dynamics is a very popular and trusted technique, being used in all fields of engineering including medical field. It is capable to provide a computer model of all kind of flows, exhibiting the fluid properties under various conditions and complex geometrical areas. It predicts all unknown flow properties of particular interest like pressure, temperature, velocity and the flow visualization by simulation presents more clear picture to rhinologists to understand and remedial approach of airflow obstruction due to anatomical, pathological or traumatic disorder inside the nasal cavity.

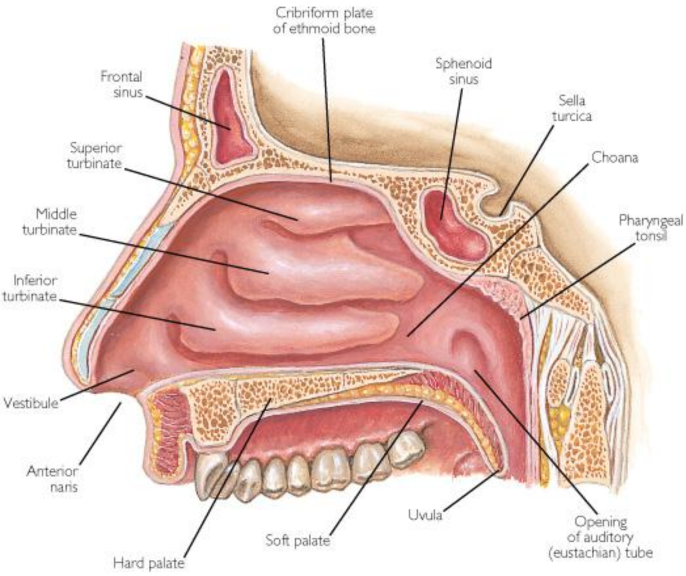


FIGURE-1 ANATOMICAL STRUCTURE OF NASAL AIRWAY

http://www.hyperbaric-oxygen-info.com/respiratory-system-diagram.html

II. LITERATURE REVIEW

Since the beginning of the 20th century, numerical methods have been developed that facilitates to solve the complicated Navier-Stokes equations which are applicable to all kind of flows whether compressible or incompressible. Numerical methods are the techniques of solving the discretised equations, by converting partial differential or integral form of the problem into algebraic form, which are obtained from fundamental physical governing law. The Navier-Stokes equations are solved at each point in the flow field in order to obtain the solution of unknown properties for prediction of flow behaviour. The volume of the flow is therefore divided into many smaller volumes or elements, to form a mesh or grid. Initially, the equations are solved for an estimated value of variables for one of the element and the answer is used as a starting point, called boundary to solve all the other equations in the mesh. The conditions at the boundaries are stated; called boundary conditions. These boundaries usually consist of an inlet for the flow to enter, an outlet, and some form of restraining walls to contain the flow. It is specified that the flow cannot cross the containing boundaries and a real value is given for a characteristic on either the inlet or outlet flow (e.g. flow rate, flow speed, or pressure). In order to validate, the solution must be converged in each step of solving the algebraic equations. The process continues to iterate until the answers converges to those specified at the boundaries. The final answer will be the solution to the Navier-Stokes equations plus or minus a small truncation error. The corresponding approximation of the flow using numerical methods is called a numerical model. At the beginning of the 20th century, there was only one way to solve all these arithmetic equations, i.e. manually at the expense of time and hard efforts. The trend continued until the modern computational techniques provided the ability to perform complex and huge amount of calculations efficiently that CFD became a practical engineering tool [8].

III. METHODOLOGY

To study the characteristics such as heat and mass transfer in the upper human respiratory tract (HRT) various models were developed. These models were based on the images obtained from computed tomography (CT) scans, MRI and or acoustic rhinomanometry (AR). This procedure is complex and access to these images is limited. The cost involved with the use of these data too is high. Hence simplified airway geometry of the HRT consisting of nasal airway is developed in Gambit, based on the CT images of a 30 years old male shown in Figure-2, obtained from All India Institute of Medical Science (AIIMS), New Delhi. The geometry of the respiratory tract is complex and hugely varying for each individual. Human tissue is highly flexible; it contracts and expands based on the amount of pressure applied on the human body. The pressure exerted by body posture also leads to the expansion or contraction of the internal organs including the HRT. For example when a person is lying backwards, there is expansion in the human body and while bending inwards or forward there is a contraction of the muscles. The above factors are complicated to address and contribute to the complexity in defining the geometry of the respiratory tract. Hence simplified models which could be standardized for the purpose of CFD simulation were developed and used in this study.

FIGURE-2 SAGITTED SLICE OF HIGH RESOLUTION CT SCAN DATA

The fact that the nasal hair do not appear in the scan and geometry, inspite the specific study considering the influence of nasal hair on the nasal airflow, shows that there is no significant effect of the absence on the flow within the cavity [8].

Meshing of the geometry is also done in Gambit. It is an unstructured triangular meshing having 2,11,522 cells, 3,18,468 faces and 1,06,947 nodes. The geometry is exported in Gambit in .msh format and finally imported to fluent for the simulation.

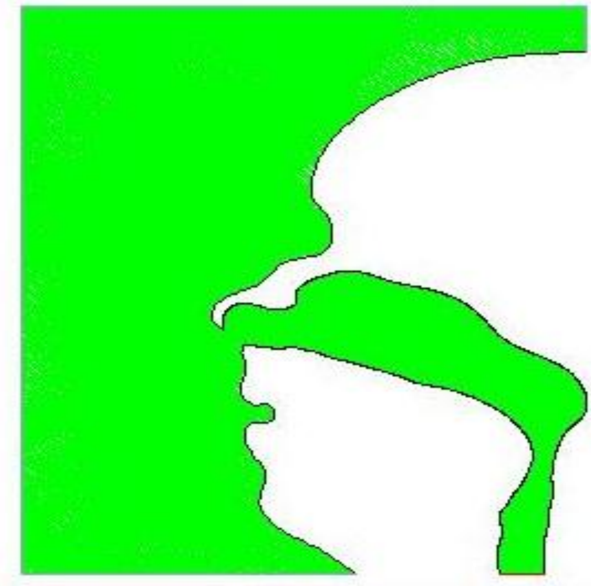


FIGURE-3 GAMBIT MODEL OF NASAL CAVITY

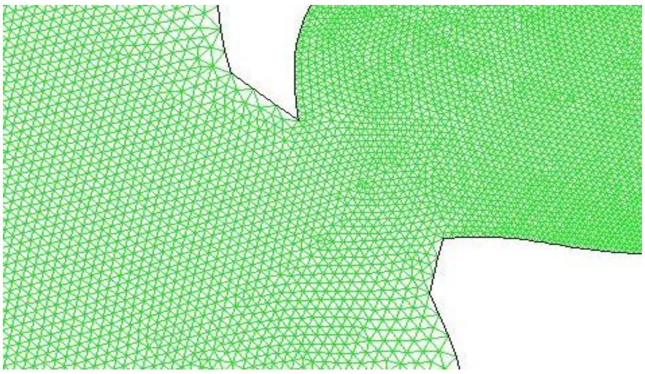


FIGURE-4 TRIANGULAR MESH

IV. BOUNDARY CONDITIONS & CFD SIMULATION

The CFD simulation was performed using the commercial CFD solver Fluent. Air behaves a Newtonian fluid and as the Mach number (The ratio of fluid velocity to the velocity of sound) is less than 0.3, it is incompressible. Continuity equation or conservation of mass (equation – 1) and Navier Stokes equations (Conservation of Momentum, equation - 2) were used to describe the motion of fluid flow in this study. This equation is governed by Newton's second law which states that the rate of change of momentum of a fluid particle is equal to the sum of body forces (work at volumetric level) and surface forces (pressure and shear force). The continuity equation and Navier-Stokes equations are given below.

$$\frac{D\rho}{Dt} = \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} \rightarrow (1)$$

$$\rho\frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + f_x \rightarrow (2)$$

$$\rho\frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + f_y$$

Where ρ is the density, u, v are the 2 velocity components, p is pressure, f is the body force and τ the shear stress. The Realizable k-e turbulence model, a two equation model was applied in adaptation for turbulent flow of low Reynolds number. Density and dynamic viscosity of the air has been taken as 1.225 kg/m3 and 1.7894e-05 kg/m-s respectively.

The following flow boundary conditions were assumed for CFD simulation.

1. Homogenous, isothermal, unsteady – state, incompressible airflow.
2. Nasal wall assumed to be rigid.
3. No slip condition at the cavity wall.
4. Ambient conditions 25˚C and 101325 Pascal.
5. A pressure difference of 250 Pascal between inlet and outlet (lungs) is applied [12].

V. RESULTS AND DISCUSSION

The simulation is done for a transient flow representing 5 breathing cycles within approximately 35 seconds [12]. Total time steps 325 (i.e. 32.5 sec) has been performed for the computation, with 0.1 time step size and 20 iterations per step. Figure 5 shows the pressure variation during each inspiration (ascending peaks, where pressure gradient is varying from -250 to +250 Pascal) and expiration (descending peaks, where pressure gradient is varying from +250 to -250 Pascal). At every 700 iterations i.e. 3.5 sec one inspiration or expiration process is completed. The static pressure profile for an expiration cycle is shown in Figure-7 (B).

The maximum velocity varies from 3.47 m/s to 6.12 m/s during normal breathing which is agreed with the result obtained with a 3-D model by Leung in [10]. The airflow in the nasal cavity is along the main nasal passage & middle turbinate and velocity increases slightly in the vestibules (Figure-9). Due to the effect of increased sphenoid sinus, the throat portion may have a narrow passage which may cause less airflow in the lungs. Region of turbulence during expiration cycle are shown in Figure-10 which occurs in anterior nans (maximum), vestibules, inferior turbinate and sella turcica. The turbulence kinetic energy (TKE) is the mean kinetic energy per unit mass associated with eddies in turbulent flow which is maximum near the nostril when the air is breathing from the lungs to ouside.

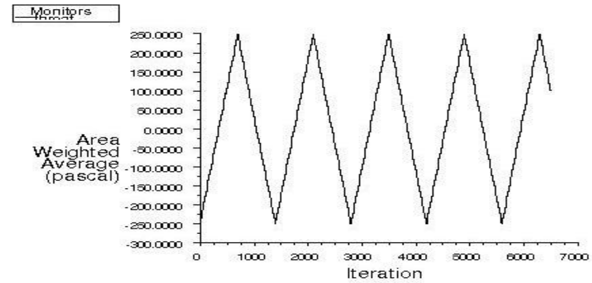


FIGURE-5 PRESSURE GRADIENT VARIATION IN 5 BREATHING CYCLES

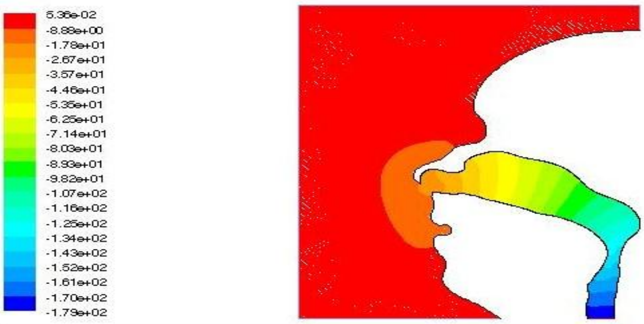
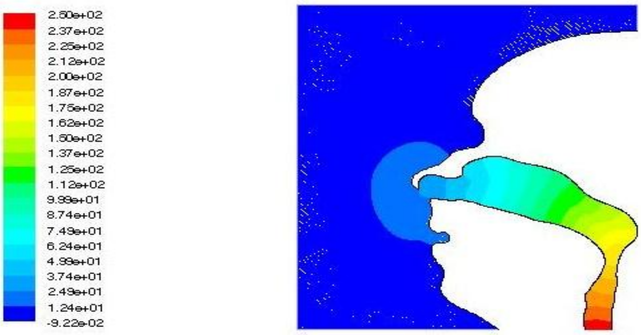
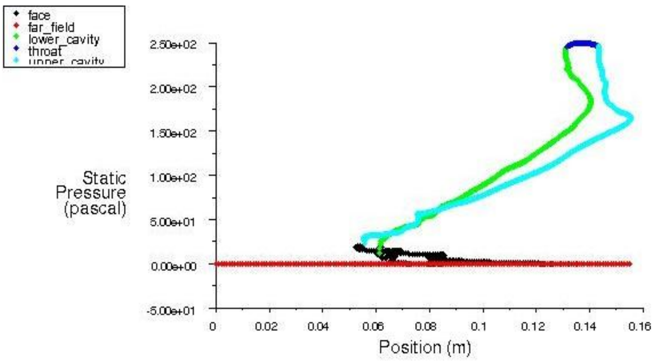


FIGURE-6 PRESSURE CONTOUR AT TIME = 0.5 SEC (INSPIRATION JUST BEGINS)



(A)



(B)

FIGURE-7 PRESSURE CONTOUR AT TIME = 3.5 SEC. (EXPIRATION IS STARTING)

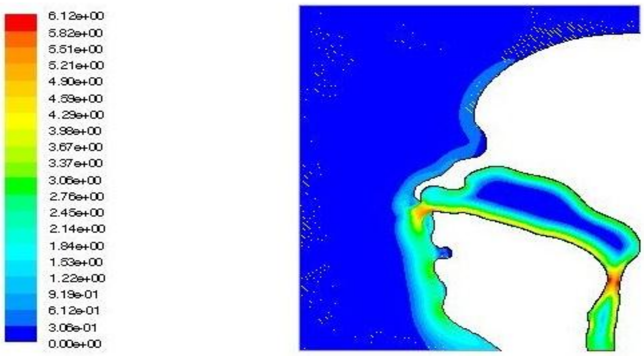


FIGURE-8 VELOCITY PROFILE (M/S) AT TIME = 5 SEC. (EXPIRATION)

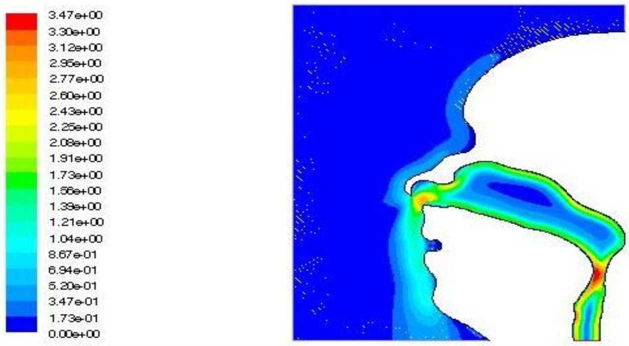


FIGURE-9 VELOCITY PROFILE (M/S) AT TIME = 9 SEC. (INSPIRATION)

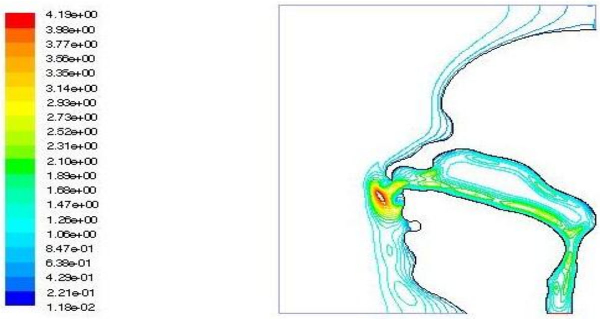


FIGURE-10 TURBULENCE K. E. (M2/S2) AT TIME = 5 SEC. (EXPIRATION)

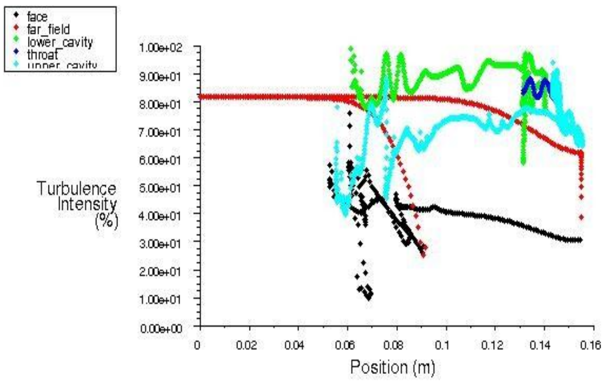


FIGURE-11 TURBULENCE INTENSITY AT TIME =3. 5 SEC. (INSPIRATION JUST FINISHED)

The value of TKE directly represents the 'strength' of the turbulence in the flow which is caused by the upper and lower cavity wall and sudden change in the air passage. When the air has been breathed inside the nose after 3.5 sec (Figure-11), the fluctuation in the turbulence in the vestibule and throat region is more than the middle air passage. The disturbance or fluctuation in the turbulence is measured by turbulence intensity. The results vary from model to model because of the uniqueness of the human anatomy.

VI. CONCLUSION

In this study, a 2-D model approach for the simulation of nasal airflow was presented. It gives a basic understanding of the breathing cycle which is a complex phenomenon. Although a 3-D model will provide more realistic results nevertheless this approach is able to predict the flow behaviour under normal breathing conditions. The results can be improved by improving the geometric model to more realistic to reduce error. There might be variations in the result with different nasal cavity model as the human anatomy is unique and greatly varies with age, gender and physique. In further studies, liquid drug spray delivery with different angles considering a multiphase flow, humidification and temperature effects in mucosa of the upper cavity for heat transfer, comparison between nasal and oral breathing will be considered.

REFERENCES

[1] Asli Sahin-Yilmaz and Robert M. Naclerio, "Anatomy and Physiology of the Upper Airway", Proc Am Thorac Soc Vol 8.pp 31-39, 2011

[2] Hilberg O, "Objective measurement of nasal airway dimensions using acoustic rhinometry : methodological and clinical aspects.", Allergy, 57(2), pp. 5-39veering SA

[3] Svensson S, Olin AC, Hellgren J. "Increase net water loss by oral compared to nasal expiration in healthy subjects", Rhinology 2006, 44: 74-77

[4] Salman SD, Proctor DF, Swift DL, Eveering SA. "Nasal resistance: description of a method and effect of temperature and humidity changes", Ann Otol Rhino Laryngol 1971, 80 : 736-743

[5] K. Keyhani, P. W. Scherer, and M.M. Mozell. "Numerical simulation of airflow in the human nasal cavity", J. Biomed. Eng., vol. 117, pp. 429-441, 1995

[6] Lang C, Grutzenmacher S, Mlynski B, Plontke S, Mlynski G, "Investigating the nasal cycle using endoscopy, rhinoresistometry, and acoustic rhinometry ", Laryngoscope 2003, 113:284-289

[7] Lindman J, Leiacker R, Rettinger G, Keck T, "The relationship between water vapour saturation of inhaled air and nasal patency", Eur Respir J 2003, 21:313-316

[8] Neil Bailie, Brednan Hanna, John Watterson, Geraldine Gallagher, "An overview of numerical modeling of nasal airflow", Rhinology, 44, 53-57, 2006.

[9] K. Hemitiwakorn, N. Phoocharoen, S. Tungjitkusolmun, C. Pintavirooj, "Nasal airflow simulation in human using a computational fluid dynamics (CFD)", (BMEiCON 2009)

[10] A.Y.T. Leung, W.S. Tsui, J.N. Xu and J. Lo. "Nasal airflow simulations in models derived from cone beam and spiral CT scans using CFD", Int.J. of Appl. Math and Mech, vol 3(3), pp.49-61,2007

[11] Ivo Weinhold, Gunter Mlynski, "Numerical simulation of airflow in the human nose", Eur Arch Otorhinolaryngol (2004) 261:452-455

[12] Stefan Zachow, Alexander Stenmann, Thomas Hildebrandt, Werner Heppt, "Understanding nasal airflow via CFD simulation and visualization", Proc. Computer Aided Surgery around the Head, Pro business Verlag, pp, 173-176, 2007

[13] Joerg Lindemann, Tilman Keck, Kerstin M. Wiesmiller, Gerhard Rettinger, Hans-Juergen Brambs, Daniela Pless, "Numerical simulation of intranasal air flow and temperature after resection of turbinates ", Rhinology, 43, 24-28, 2005

[14] Raghavan Srinivasan, Kambiz Farahmad, Mohsen Hamidi, "CFD Heat TransferSimulation of the Human upper Respiratory Tract for Oronasal Breathing condition", Proceedings of the 2011 IAJC-ASEE International Conference ISBN 978-1-60643-379-9

[15] Phd Thesis by Amrita Mahtani, Guilly Mendoza, Weilin Yang, Robin Zhou, "The way to spray : Modeling nasal spray deposition", April 30, 2009

[16] Phd Thesis by Michelle Chau, Pimkhuan Hannanta - Anan, Oleh Krupa, Jack Yi, "Spray away allergies – Modelling intranasal drug delivery", May 3rd, 2012