An investigation of computational fluid dynamics of human nasal airflow using LS-DYNA®

V N Riazuddin¹, C C Chen¹, Ahmad Faridzul¹, M A Jasni¹ and J Chen¹
¹Dyna Forming Engineering & Technology Sdn. Bhd., 534A, Jalan Seremban Tiga 15, 70300 Seremban, Negeri Sembilan, Malaysia.

E-mail: vizynazira@dfe-tech.com

Abstract. Understanding the properties of airflow in the nasal cavity is essential in determining the nasal physiology and in diagnosis of various anomalies associated with the nose. This numerical study presents the characteristics of flow features inside a human nasal cavity. A three dimensional nasal cavity model is constructed from computed tomographic images of a healthy adult nose. Pressure inlet of 0 Pa is defined at the left and right nostril inlets representing the atmospheric condition. Negative pressure value is defined at the pharynx outlet to model the airflow suction during inhalation. This study shows the flow characteristics and distribution in the nasal cavity is dependent on the complex geometry of the nasal airway. This simulation of breathing airflow enables us to visualize the flow features inside the complicated human nasal cavity which is otherwise impossible to be determined by any other method.

1. Introduction
Nasal cavity is the main pathway for air containing oxygen to flow into the lung when inspired. Furthermore, the physiological function of the human nasal is highly dependent on the fluid flow characteristics in the nasal airway. However, the complicated structure, small and narrowed cross section area inside the nasal cavity has proven to be a significant obstacles understanding the flow phenomenon through the nasal airway[1]–[3]. Numerical simulation of airflow based on Computational Fluid Dynamics (CFD) approaches can be very useful to enhance our knowledge and understanding of nasal function and abnormal anatomical deformities. The non-invasive numerical simulation approach has the advantages of providing information such as velocity field throughout the nasal passage, allowing reconfiguration of the geometry in future studies to model various anatomical deformities[3], [4].

A number of researchers have shown the validity and potential use of CFD in evaluating the flow conditions inside the nasal cavity. Early work regarding this topic was performed by Elad et al., (1993) who conducted numerical simulations of steady laminar flow through a simplified nose-like model which resemble the complex anatomy of human nasal cavity using the finite element software package FIDAP (Fluid Dynamics International). The number of mesh created for this nasal model is approximately <3000 elements. They found that he turbinate is an obstacle in the airway that increases the resistance to airflow [5]. Segal et al., (2008) performed numerical simulation of steady state inspiratory laminar airflow for flow rate of 15 L/min. In their study, three dimensional computational models of four different human nasal cavities which constructed from coronal MRI scans were used. In their study, they found that in all four nasal models, the majority of flow passed through the middle...
and ventral regions of the nasal passages. The amount and the location of swirling flow differed among the subjects [6]. Wen et al., (2008) also simulated steady laminar nasal airflow for flow rates of 7.5 to 15 L/min using computational fluid dynamics software FLUENT. An anatomically correct three-dimensional human nasal cavity computed from CT scan images were used. The solution was found to be mesh-independent at approximately 950,000 cells. Results shows that the nasal resistance value within the first 2-3 cm contribute up to 50% of the total airway resistance. Vortices were observed in the upper olfactory region and just after the nasal valve region [7].

Computational fluid dynamics can be solved by using various numerical methods such as Finite Volume (FV) and Finite Element (FE). Finite Volume Method (FVM) is a commonly used in fluid dynamics analysis, whereas Finite Element Method (FEM) is commonly used for structural analysis. However, FEM is also applicable of solving engineering problems involving fluid flow. The LS-DYNA® Incompressible Computational Fluid Dynamics (ICFD) solver [4] developed by Livermore Software Technology Corporation (LSTC) is based on the finite element approach applied to fluid mechanics. The ICFD solver is developed to solve low speed fluid flow where the Mach number is less than 0.3 with the fluid flow can be assumed as incompressible. The ICFD solver can run as a stand-alone solver to study fluid dynamics problems or coupled to solid mechanics solver to study fluid-structure interaction (FSI) problems. One of the main advantages of ICFD solver is the automatic volume mesher feature. The automatic volume mesher can greatly simplify and speed up the mesh generation process particularly for complicated geometry, during the pre-processing stage. Recently, the ICFD solver has been used by many researchers to solve several fluid flow problems with good success [8]–[13].

Hence, in this study, the author intends to use the ICFD solver to investigate the natural physiological breathing conditions through a normal human nasal cavity. A pressure inlet of 0 Pa is defined at both left and right nostril inlets representing the atmospheric condition, while negative pressure value is defined at the pharynx outlet to model the airflow suction during inhalation. The steady airflow numerical simulations are carried out using three dimensional (3-D) nasal cavity model derived from tomography scan images of a normal nasal passage.

2. Methodology
The 3-D nasal model is developed from a series of two dimensional (2-D) CT scan images by using an image processing software named Materialise Mimics Innovation Suite. Figure 1 illustrates the CT scan images obtained from axial, coronal and sagittal plane, as well as the developed 3-D model of the human nasal cavity.
Figure 1. CT scan images from axial coronal and sagittal plane and 3-D model of the human nasal cavity.

The developed 3-D nasal model is imported into finite element pre- and post- software PreSys™ by using the .stp file format for surface mesh generation process. The aspect ratio of the surface mesh is maintained below a value of 3. The mesh model is then exported into .k LS-DYNA® keyword file format for subsequent ICFD simulation setup in LS-PrePost. The volume mesh is generated by using the automatic volume mesher available in ICFD solver. The volume mesh generated is based on the surface mesh bounding the computational model. It is very important to make sure that there are no duplicate nodes and elements or open gaps on the surface mesh. The surface mesh must consist of 2-D triangle elements. Figure 2 illustrates the surface mesh and cross-section area of the volume mesh of the nasal model.

Figure 2. The surface mesh and cross section area of the volume mesh of the nasal model.
The ICFD simulation is based on the numerical solution of the Reynolds Averaged Navier-Stokes equation representing the general equation for 3-D flow of incompressible and viscous fluids. The pressure inlet boundary condition is imposed at the nostril inlet whereas a pressure outlet boundary condition is defined at the pharynx outlet. A no slip boundary condition is defined at the nasal wall. Pressure inlet of 0 Pa is defined at both left and right nostril inlets representing the atmospheric condition, while negative pressure value is defined at the pharynx outlet to model the airflow suction during inhalation. The results are considered converged when the residual value reduced to a value of 10e-4.

3. Results and Discussions

In this section, the grid independence test performed on the nasal cavity model and airflow characteristics obtained from the simulation are presented.

3.1 Grid Independence Test

![Figure 3. Grid independence test based on the fluid velocity.](image)

The nasal cavity model with 196,037 unstructured tetrahedral mesh is initially developed and used to solve the inspiratory airflow with -2.5 Pa suction pressure at the pharynx outlet. Grid independence test has been conducted in the same nasal cavity model with different size of mesh. The grid independence test depicted in figure 3 shown that the fluid velocity obtained at the nasal valve region converged as the mesh resolution approached 1,042,208 cells. Hence, in this study, the simulation results obtained from the mesh resolution of 1,042,208 tetrahedral elements is discussed. This is considered sufficient, taking into account the computational time and system memory. The quality of the volume mesh generated by LS-DYNA® ICFD solver can be checked by referring to the icfd_mstats.dat file. This file is generated when the simulation starts. The file contains the overall mesh quality information. The George Quality (Q) is used to evaluate the mesh quality. Higher Q value means poor mesh quality. Generally, a Q value of less than 10 is considered acceptable with nice tetrahedral volume element. In this case study, the worst Q value obtained for the generated tetrahedral volume element is around 0.86. The equation used to calculate the George Quality, Q can be found in the LS-DYNA® ICFD Theory Manual [14].
3.2. Velocity

Figure 4 shows the velocity streamlines through the nasal cavity during inhalation. It is observed that the flow velocity increases as the flow passes through the nasal valve area. At the nasal valve area, as the cross-section area decreases, the velocity increases. Similarly, the airflow velocity is also observed increasing as the flow enters the posterior part of the nasal cavity where the cross-section area decreases. The average velocity obtained in the nasopharynx section is 3537.16 mm/s. As can be observed in figure 4, most of the flow passes through the middle meatus. This flow pattern was similarly observed by Segal et al., 2008 in their research study [6]. Velocity streamlines, shown in figure 4, also illustrate low velocity recirculating flow reaches the olfactory region during inhalation. This recirculating flow was similarly observed in other research findings [7][2].

![Figure 4. Velocity streamlines during inhalation.](image1)

![Figure 5. Velocity vector of inspiratory flow.](image2)

Figure 5 shows the velocity vector of the inspiratory flow through the nasal cavity. It is observed that the flow enters the nostril at normal direction through the inlet. After that the airflow changes direction to near 90° to enter the nasal valve region. As the airflow leaves the nasal valve region, it is distributed laterally into the inferior, middle and superior turbinate regions. At the posterior part of the nasal cavity, the airflow bends down to near 90° again to exit the outlet.
3.3. **Pressure and Airflow Resistance**

Pressure distribution through the nasal passage can be used to evaluate the impact of the morphology of the nasal airway on breathing airflow resistance. The nasal airflow resistance is defined as the ratio of pressure drop to the volume flow rate, \( R = \frac{\Delta p}{Q} \), where \( \Delta p \) is pressure drop across the airway passage and \( Q \) is airflow rate.

| Case study          | Pressure drop (Pa) | Airflow resistance (Pa-min/L) |
|---------------------|--------------------|-------------------------------|
| Segal et al., 2008  | 4.72-14.266        | 0.472-1.427                   |

The pressure drop obtained from this case study is compared with the pressure drop obtained by Segal et al., 2008. Segal et al., 2008 in their research study has conducted experiment to calculate pressure drop through four differences nasal cavities derived from healthy individuals. In their research study, they obtained pressure drop values in a range of 4.72-14.266 Pa[6]. As shown in table 1, in this study, the pressure drop obtained for volume flow rate of 10L/min is 11.42Pa. Hence, it can be deemed that the simulation results have a good agreement with experimental results. However, the variation of the pressure drop value obtained could be due to the variation of nasal anatomy between each individual subject.

![Figure 6. Pressure contour during inhalation.](image)

Figure 6 illustrates the average pressure distribution through the nasal cavity during inhalation. The pressure is observed decrease posteriorly. The highest average pressure is observed at the anterior part of the nasal cavity, while the lowest average pressure is observed at the posterior part of the nasal cavity. During inhalation, the lung expands and sucks in the air containing oxygen from the atmosphere. Hence, negative pressure is observed at the posterior part of the nasal cavity.
4. Conclusion
In this work, Computational Fluid Dynamics is carried out to study the steady state inspiratory airflow through the human nasal cavity. The airflow field of human nasal cavity is analyzed. The pressure drop obtained from this study was compared with other published data. The pressure drop value obtained from the simulation results shows good agreement with experimental results. The results obtained shows that the airflow characteristics and distribution in the nasal cavity is dependent on the geometry of the nasal airway. The flow velocity increases as the cross section area decreases. The small cross-sectional area of the nasal valve plays vital role to the distribution of airflow laterally into the inferior, middle and superior meatus. The low recirculating flow could help improve the sense of smell, by delivery air to the superior part of the nasal cavity where the olfactory nerve is located. The nasal meatus increase the surface area of the nasal cavity. Hence increase the amount of inhaled air to contact with the cavity wall which could help humidify the air we breathe.

Acknowledgement
The author would like to acknowledge Livermore Software Technology Corporation for providing the technical support.

References
[1] G. J. M. Garcia, N. Bailie, D. A. Martins, and J. S. Kimbell, “Atrophic rhinitis: a CFD study of air conditioning in the nasal cavity.,” J. Appl. Physiol., vol. 103, no. 3, pp. 1082–1092, 2007.
[2] V. N. Riazuddin et al., “Numerical study of inspiratory and expiratory flow in a human nasal cavity,” J. Med. Biol. Eng., vol. 31, no. 3, pp. 201–206, 2011.
[3] V. N. Riazuddin, K. A. Ahmad, M. Tamagawa, M. Zubair, N. A. Rashid, and N. Mazlan, “Numerical simulation of airflow in diseased human nasal airway,” in Proceeding of Mechanical Engineering Research Day 2017, 2017, no. March, pp. 90–91.
[4] V. N. Riazuddin et al., “Computational Fluid Dynamics Study of Airflow and Microparticle Deposition in a Constricted Pharyngeal Section Representing Obstructive Sleep Apnea Disease,” J. Med. Imaging Heal. Informatics, vol. 6, no. 3, pp. 1507–1512, 2016.
[5] D. Elad, R. Liebenthal, B. . Wenig, and S. Einav, “Analysis of airflow patterns in the human nose,” Med. Biomed. Eng. Comput., no. 11, pp. 585–592, 1993.
[6] R. A. Segal, G. M. Kepler, and J. S. Kimbell, “Effects of differences in nasal anatomy on airflow distribution: A comparison of four individuals at rest,” Ann. Biomed. Eng., vol. 36, no. 11, pp. 1870–1882, 2008.
[7] J. Wen, K. Inthavong, J. Tu, and S. Wang, “Numerical simulations for detailed airflow dynamics in a human nasal cavity.,” Respir. Physiol. Neurobiol., vol. 161, no. 2, pp. 125–135, 2008.
[8] I. Caldichoury, F. Del Pin, and V. Lapoujade, “Incompressible CFD Results Using LS-DYNA . For High Reynolds Number Flow Around Bluff Bodies,” in 8th European LS-DYNA Users Conference, 2011, pp. 1–9.
[9] E. Yreux, “Fluid Flow Modeling with SPH in LS-DYNA ®,” in 15th International LS-DYNA Users Conference, 2018, pp. 1–6.
[10] F. Del Pin, I. Caldichoury, R. R. Paz, and C. Huang, “ICFD : Summary of Recent and Future Developments,” in 15th International LS-DYNA Users Conference, 2018, pp. 1–7.
[11] C. Huang, I. Caldichoury, F. Del Pin, and R. R. Paz, “CFD Validations with FDA Benchmarks of Medical Devices Flows,” in 15th International LS-DYNA Users Conference, 2018, pp. 1–9.
[12] X. Gao, “Assessment of simulating underwater impact using the ICFD solver in LS-DYNA,” Memorial University of Newfoundland, Canada, 2016.
[13] M. Le Garrec, M. Seulin, and V. Lapoujade, “Airdrop Sequence Simulation using LS-DYNA ® ICFD Solver and FSI Coupling,” in 15th International LS-DYNA Users Conference, 2018, pp. 1–17.
[14] I. Caldichoury, R. R. Paz, and F. Del Pin, *ICFD THEORY MANUAL Incompressible fluid solver in LS-DYNA*. Livermore Software Technology Corp, 2014.