Computational fluid dynamics simulations of the airflow in the human nasal cavity.

P. Castro Ruiz, F. Castro Ruiz, A. Costas López, C. Cenjor Español
1Fundación Pública Hospital do Salnés, Vilagarcía de Arousa, Pontevedra. 2Escuela Técnica Superior de Ingenieros Industriales de Valladolid. 3Centro Médico POVISA, Vigo, Pontevedra. 4Departamento de Cirugía de la Facultad de Medicina de la Universidad Autónoma de Madrid.

Abstract: Objective: The aim of this study is to visualize and analyze the complex three-dimensional airflow pattern in the human nasal passageways using a computer simulation. Materials and methods: Computational Fluid Dynamics (CFD) was used. This technique consists of simulating how airflow moves in certain environments. CFD solved the equations governing flow numerically. CFD has proved to be a very efficient tool for studying nasal flow. Results: By means of a CT scan, an anatomically accurate, three-dimensional representation of the human nasal cavity was obtained. The airflow in both breathing phases was visualized and the current lines, the velocity profiles and pressure and turbulence intensity fields were observed. Conclusions: The flow patterns show the channeling effect of turbinates, the laminar distribution and the influence of the breathing phase.

Key words: Nasal airflow. Nasal cavity. CFD. Flow simulation.

INTRODUCTION

The nose is considered to be a complicated and physiologically active conduct. The airflow that circulates inside it is conditioned by laws of physics that govern the movement of fluids. Research into the interrelation between the anatomy of the nose and the flow pattern of inhaled and exhaled air from the nostril to the choana was begun years ago and it has still yet to be described sufficiently. In the 1950s, authors such as Proetz and Stuiver began their experiments by passing fluid through a model of the nasal cavity. Later, Masing studied the trajectory of fluid through the nostrils using cinematographic techniques. Since then, many studies of an experimental nature have been conducted to investigate what the flow through the nostrils is like and the factors that influence it. All these studies have had to resolve problems associated with the complex internal structure of the nostrils and their small size.

There have been fewer analytical studies of nasal flow. The first studies considered the airways to be a succession of conducts of more or less regular geometry (cylinders or prisms). Elemental flow patterns were assigned to each one of these, (such as flow in an expansion, flow after a step or in a bend), and from these elemental flow patterns, flow in the nasal cavity was established. Due to the low resolution of the geometries used, and to the failure to take into account the interaction of the flow with adjacent elements, the results obtained were inevitably somewhat imprecise. In this study, an alternative and novel approach called Computational Fluid Dynamics was used to study flow through the nasal cavity. Computational Fluid Dynamics (CDF) has proved to be a valuable tool in the study of this field. This technique consists of resolving the equations that govern the movement of a fluid numerically and therefore enables the simulation of the behavior of a fluid.

The objective of this study is to simulate airflow through the nasal cavity using Computational Fluid Dynamics. For this purpose, an aerodynamic model of the nose was developed based on the physiology of the respiratory system and by applying the laws of hydrodynamics. Once the aerodynamic model had been built, a numeric model of the nasal cavity was developed. The numerical assays allowed the airflow trajectory through the nasal cavity to be observed using a general-purpose CFD code.

MATERIAL AND METHODS

From the fluid-dynamics point of view, material can only be in two states: solid or fluid. Fluid is a substance that continually changes shape under the...
action of cutting force. A fluid can be a liquid or a gas. In the last century, two scientists; the French engineer Claude Navier and the Irish mathematician George Stokes, established the differential equations that govern the movement of a fluid. These equations define, in any point in space, the speed and pressure of a fluid. They are differential equations that in most situations do not possess an analytical solution. The Navier-Stokes equations are a representation of the principals of the conservation of mass, quantity, movement and energy; they are a system of five equations in derived nonlinear partials. From the resolution of these conservation equations, the movement of a fluid particle under certain environmental conditions can be determined.

Computational fluid dynamics is the methodology to find the numerical solution to the Navier-Stokes equations that govern the flow of a fluid in a space and time domain. It enables us to obtain a complete description of the fluid field. The main tasks required to obtain a simulation of flow can be grouped into three stages. Preprocess is the stage when the geometric domain is discretized (the geometric problem is specified, the computational volume is created and the mesh is generated), the properties of the fluid are specified and the initial and environmental conditions are imposed. These conditions are those that, together with the geometry, characterize a fluid-dynamic problem. In the resolution stage, the equations to be solved and the resolution algorithms and their parameters are specified, and the solution to the system of equations that govern the process is generated iteratively. Finally, in the post-process stage, the results obtained are stored and processed for their later analysis; this is the stage of visualization and the analysis of the results, with the objective of validating the behavior of the fluid and drawing conclusions regarding its reliability or identifying possible errors. One of the main advantages of CFD is that, when a specific problem is being studied, the cost of the flow simulation is a lot lower compared to the experimental cost. Furthermore, it offers the possibility of verifying the theoretical results whose conditions are impossible to reproduce in experimental installations, for example that of the ideal fluid. A further advantage is that the solution arrived at provides complete and detailed information of the velocity, pressure, temperature, etc. fields, in space and time. Among the disadvantages, we must mention that the reliability of the results is linked to the correct mathematical formulation of the process to be simulated. That is to say, the more realistic the environmental conditions imposed, and the greater the degree of exactness of the numerical resolution algorithm of the system of equations, the more precise will be the solutions arrived at. The role of CFD in the prediction of fluids in the field of engineering has reached its height in the present day thanks to the three-dimensional view of fluid dynamic problems over the traditional two-dimensional analysis of fluid dynamics, theoretical or experimental.

CFD as a science is made up of many different disciplines such as the physics of fluids, applied mathematics and IT. Its fields of application are very varied; it is possible to find applications as different as aerodynamics (flow around cars, planes, ships and buildings), the environment (smoke, dispersion of pollutants in the atmosphere), air-conditioning (heating, the ventilation of closed spaces) and medicine (blood flow, artificial organs, etc.). CFD uses computers as a work tool to numerically solve the equations that govern the movement of fluids. In the existing software packages, the user must specify the conditions of the problem he/she wishes to solve, as well as provide the computer with certain resolution parameters so that the program can find the correct solution to the problem.

RESULTS

A numerical model was developed that was implemented in two general-purpose CFD codes: Fluent and Power Flow.

The generation of the computational mesh is a complex and very laborious process on account of the anatomical complexity of the nasal cavity. It is necessary, first of all, to precisely define the geometry. Then, a closed surface must be generated (as if it were a shell) representing the internal surface of the nostrils, and finally, a mesh within this computational volume must be generated in which, despite the sharp variations in geometry, the number of cells should be as low as possible and must not be too distorted. To reproduce the internal anatomy of the nasal cavity and the nasopharynx with precision, CT images were obtained from a 30-year-old woman. Coronal cuts of the nasal cavity and nasopharynx were made every 1.5 mm with a resolution of 0.4 mm. A detailed representation of each of the internal sections of the nasal cavity was obtained from each of these images using Excel, simply by moving the points onto it. A computational volume was made using the CAD CATIA program from the detailed series that describe each one of the 81 sections and the separation between them. It should be remembered that this volume corresponds to the interior volume of the nostrils and therefore, its exterior surface matches the inner surface of the nasal cavity. The reference system used has its coordinate origin located in the nasopharynx. The “z” axis runs practically parallel to the nasal floor and passes from the nasopharynx towards the nostril. The “x” axis runs from the septum towards the external wall and the “y” axis from the floor to the roof (Figure 1).

The generation of the mesh is a decisive stage in achieving a model that produces good results. It is
necessary to reach a compromise between high-resolution meshes, with a high number of cells, but which require a longer calculation time, and low-resolution meshes with little calculation time, but that do not produce reliable results. It is therefore necessary to conduct a sensitivity study that enables us to establish the minimum resolution of the meshing, from which the results are independent of the number of cells. The difference between the results from the 140000-cell model and those of 1600000 cells is less than 2%, while the calculation time can be 50 times greater for the latter. Therefore, it was decided to use the 140000-cell model. To construct this meshing it is necessary to use an auxiliary program with which a tetrahedral cell meshing was made; this task is complicated and laborious. Once the mesh has been made it is imported into the code, the environmental and functional conditions are imposed and the case is executed (Figure 2).

To establish the initial and environmental conditions, it is necessary to make a series of approximations about the nature of the flow. A unanimous position is not found in the literature regarding whether nasal flow is laminar or turbulent. Although the critical Reynolds’ number that separates the laminar regime from the turbulent one for conduits of such a particular geometry such as the nasal cavity is unknown, the flat shape of the velocity profiles in the nasal cavity, measured experimentally, is more similar to the characteristic shape of the turbulent flows than the parabolic shape of the velocity profiles in laminar flows. It was therefore decided to assume that the flow in the nasal cavity is mainly turbulent. Nasal flow is not stationary, but given that the Strouhal Number is lower than the unit (Strouhal = 0.23) the quasistationary hypothesis is acceptable. The Strouhal Number is a parameter used in fluid dynamics; if it is greater than the unit, the non-stationary effects are significant and cannot be ignored. Although nasal flow is air and this is a gas, due to the low speeds of the air inside the nose, the Mach number is less than 0.3 (Mach = 0.06), which clearly indicates that the flow is incompressible. The Mach number is a parameter used in fluid dynamics that quantifies the effects of compressibility. Other approximations of the numeric model were to consider isothermal flow and smooth aerodynamic walls, as although in the real nose there is an exchange of mass and heat between the airflow and the walls that can produce effects of thermal buoyancy these are negligible over the velocity profiles. Neither has the flow of mucus in the nose been considered; given its minimal thickness and low velocity, its effects on the velocity profiles can be considered negligible. Furthermore, a rigid wall model, not susceptible to valvular deformation, was considered, given that the maximum flow at which the nasal valve collapses is far above the flows considered in this study. Neither has the deformation that the turbinates suffer because of the nasal cycle been considered given that the time scale of the model is much less than that of the nasal cycle.

Finally, the nasal cavities have not been considered either. Throughout the previous simulations it was observed that small changes in the geometry barely modify the flow pattern through the nose, although they do modify the local value of the speed. It was decided not to include these cavities, as their simulation would further complicate the model.

The environmental conditions imposed on the flow entrance and exit sections were: on inhaling, negligible gage pressure on the nostril and static negative gage (vacuum) pressure on the oropharynx. On exhaling, the condition of negligible static gage pressure was imposed on the nostril and the condition of static positive gage pressure on the oropharynx.

Figure 1. Computational volume of the nasal cavity.
Any numerical simulation should be valid. For this purpose, Hahn's experimental results were used, that is to say, simulations conducted in identical conditions to Hahn's experimental conditions were executed, except that the nasal geometries were not completely equal. The results were later compared. Hahn expounded the measurements of the velocity field within the nasal cavity, obtained experimentally using hot wire anemometry on a 20:1 scale nasal cavity model, in which, by using a ventilator placed in the nasopharynx, a stationary airflow through the model was achieved. For the comparison it was necessary to consider two important sources of error. The first being due, as already mentioned, to the geometry of the nasal cavity used in this study being different to that used by Hahn. Although both corresponded to healthy nasal cavities, one was a man's and the other a woman's, with certain differences existing between the two: the computational model of the nasal cavity is slightly shorter and more permeable than the experimental model (Figure 3). The second source of error is connected to the measurement of the smallest velocities. As Hahn mentions in his paper, the convective effects around the hot wire probe can cause errors of around 2% in the range of small velocities. Furthermore, the positioning error of the probe is around 5%.

To carry out this validation, the four coronal sections where Hahn made measurements and four equivalent sections in the computational model were chosen. The comparison between the profiles of experimental velocities and those obtained using numerical simulation was made in 8 lines. Due to the geometric differences and in order to facilitate the comparison, velocity was represented adimensionalized in each point with the maximum velocity along the line and the distance in regard to the adimensionalized wall and the length of the line. It can be observed that the agreement between the experimental results and those of the simulation is quite good in 7 of the 8 profiles.

There are notable discrepancies for line 5 that could be due to the geometry's sharp variation in proximity (Figure 4).

With the numerical model, six cases were executed in three situations. A situation of breathing calmly, inhaling (case 1) and exhaling (case 2); a situation of deep breathing, on inhaling (case 3) and on exhaling (case 4); and a situation of forced breathing, equally for inhalation (case 5) and exhalation (case 6). In calm inhaling the flow pattern has the following characteristics: the current that enters through the anterior zone of the nostril progresses towards the superior zone of the nasal cavity, with a velocity of less than 0.8 m/s. The rest of the flow that reaches the vestibule through the posterior and central zone is distributed throughout the inferior region of the nasal cavity at velocities of around 0.6 m/s, and through the mid-region of the nasal cavity at a velocity of 2 m/s. The higher velocities, those of more than 2 m/s, are observed in the mid-region, with areas of flow re-circulation in the margins of the nasopharynx, with a velocity of 0.1 m/s. In the mid-region the velocities are greater than in the lateral region at 2 m/s and 0.6 m/s respectively. The static pressure descends as the flow passes through the nasal cavity. The greatest differences in pressure are produced in the narrow nasal vestibule. The greatest turbulence is observed in the vestibule with turbulence intensities of 80%. In the nasal cavity, the turbulence intensity is 30%. Minimum turbulence is found in the regions of the inferior meatus and the olfactory groove (Figure 5).

When calmly exhaling, the lines of distribution of the flow are more uniform. The flow that enters through the nasopharynx is similarly distributed in the mid and inferior regions, with velocities of around 1.8 m/s. Much lower velocities are observed in the superior region, of around 0.3 m/s. The greatest pressure losses take place in the turbinal area. Of the velocity environments, it was deduced that most of the flow circulates through the mid and inferior region with a velocity of more than 1 m/s. There are few areas of flow...
Figure 4. Comparison between experimental velocity profiles (pink) and those of the numerical simulation (blue).

Recirculation in the nasal tip that have a velocity of 0.2 m/s. Most turbulence is observed in the choana and in the nasopharynx, with a turbulence intensity of 70%. In the flow's exit through the nasal orifice, the turbulence intensity is 40%. The minimum turbulence is observed in regions of the inferior meatus and roof regions (Figure 6).

Figure 5. Flow pattern of calm inhalation
Figure 6. Flow pattern during calm exhalation

If one observes the current lines that pass through the inferior region of the nasal cavity, they leave through the anterior part of the nostril on exhaling; on inhaling, however, this behavior is not as pronounced. On exhaling, the flow mainly travels parallel to the floor due to the guiding effect of the turbinates. On inhaling, the flow is more chaotic, especially in the vestibule, until the guidance of the turbinal zone begins. This shows the channeling effect of the turbinal zone (Figure 7).

The current lines that flow through the mid-region, both on inhaling and exhaling, enter and leave through the whole surface of the nostril. However, the flow on exhalation crosses a very specific area of the narrow nasal vestibule. The flow on inhalation is much more

Figure 7. Current lines through the inferior region
disperse in this last section. In both cases, the flow is quicker than in other regions (Figure 8).

The flow that passes through the roof of the nasal cavity comes from the anterior zone of the nostril on inhaling. The current lines that cross the region next to the olfactory cleft indicate that the flow that comes into contact with this region is greater during inhalation. In both phases there is a re-circulating flow in the olfactory area of the nasal roof (Figure 9).

If the flow pattern in the nasopharynx is analyzed, it can be observed that it is quite similar during inhalation and exhalation. However, if the transversal section is carefully analyzed, it can be seen that there is a helicoidal movement during inhalation that does not exist during exhalation. This helicoidal movement encourages the inhaled air to mix, and therefore its properties become more uniform (Figure 10).

**DISCUSSION**

Simulation techniques are relatively recent; the papers published have been written over the last 15 years. Tarabichi and Fanous in 1993 carried out a numerical simulation of flow in the nasal valve; the numerical model developed was limited only to the area of the valve. Kimbell in 1993 developed a model of the nasal cavity, but of a rat. Kepler in 1998, developed a numerical model of the nasal cavity of the Rhesus monkey. Zhao, Brunskill and Lieber in 1997 carried out an analysis of stationary flow, in inhalation and exhalation, in a numeric model of symmetrical bifurcation of the airway. Elad in 1993, Keyhani in 1995 and Subramaniam in 1998 built anatomically correct numerical models of the nasal cavity, but in these numerical studies the authors only considered
laminar and stationary flows. The numerical studies of Elad and Keyhani reveal that the highest velocities of flow appear close to the floor of the nasal cavity. However, they also observed a second region of high velocities: the mid-region of the nasal cavity. In this study, as in Subramaniam’s, it was found that the highest velocity of flow appears in the mid-region of the nasal cavity.

In this work, the utility of the numerical model in the study of flow through the nasal cavity was shown. The model developed has been validated with the experimental results of other authors. The results of the simulation provide much more detailed information than the experimental results. From the simulations conducted, flow through the nasal cavity can be visualized in the following way: the air accelerates from the environment to the narrow nasal vestibule, from which point, the flow quickly decelerates before entering the pre-turbinal area. The expansion that the flow suffers as it crosses the nostril causes a disturbance of the flow in the vestibule, which calms as it crosses the narrow nasal vestibule. This flow heads towards the nasal cavity where the narrow passages created by the turbinates channel the flow. The majority of the flow passes through the mid-region. Only a small part of the flow heads towards the superior region. The flow leaving the turbinal area acquires a helicoidal component that is maintained as far as the nasopharynx. During the exhalation, the flow that comes from the nasopharynx distributes itself uniformly throughout the nasal cavity. In the pre-turbinal region, this exhalation flow accelerates, reaching its maximum velocity in the narrow nasal vestibule. On its way towards the nostril, a whirlwind begins in the tip of the nose.

The visualization of the flow using computational fluid dynamics shows the guiding effect the turbinates have over airflow in the nasal cavity, both when inhaling and exhaling, and the laminar distribution of the airflow in the nasal cavity. It is observed that the flow that comes into contact with the olfactory region is a flow re-circulating at low velocity and of greater magnitude during inhalation. It is also shown that the respiration phase influences the flow pattern of the vestibule and of the choana, distributing itself mainly through the mid-region on inhalation and through the inferior region on exhalation. It is confirmed that, although the pressure distribution patterns, velocity and turbulence are independent on the type of respiration, the type of respiration influences the values of the velocity of the flow and the intensity of the turbulence, being greater with forced respiration than with deep breathing and these both greater than with calm breathing.

The first physiological concepts of the nose, the same as all primitive physiological concepts, were based on intuition, but with time it was recognized that intuition was not enough and that rigorous scientific validation should be depended on in order to be able to discern between reality and fantasy. Mathematics and physics are essential in order to study and comprehend pressure, flow velocity and turbulence. This study enables one to go into nasal physiology in depth using more modern study techniques from the movement of fluids. Therefore, it can be considered that this study falls into the area of nasal bioengineering research that poses a real challenge. It opens up fields of work in two different disciplines; among future jobs of a medical nature, the influence of anatomical modifications in
nasal flow, such as septal perforations or adenoidal hypertrophy, could be mentioned. Jobs of a technical nature would be directed towards the simulation of biophysical aspects, such as the elasticity of walls, the mucosa and the exchange of heat between the air and the walls of the nasal cavity.

References