An overview of numerical modelling of nasal airflow*

Neil Bailie[1], Brendan Hanna[1], John Watterson[2], Geraldine Gallagher[1]

[1] Department of ENT Surgery, Antrim Area Hospital, Northern Ireland
[2] School of Aeronautical Engineering, Queen's University Belfast, Belfast, Northern Ireland

SUMMARY

Computer modelling of fluid flows is a mature technology used widely in engineering. The process, known as computational fluid dynamics (CFD), allows accurate prediction of fluid flow and associated phenomena based on the mathematical laws governing fluid behaviour. A fluid may be defined as any substance that can flow and thus both liquids and gases behave as fluids. The mathematical predictions of CFD can therefore be applied to nasal airflow.

In current clinical practice, it is only possible to perform a few limited measurements of nasal airflow, and the clinical relevance of these measurements is questionable. Computer models are not limited by the anatomical inaccessibility of the nasal cavities, and a detailed objective characterisation of airflow can therefore be provided in all areas of an individual nose. In addition, the ability to remodel computer simulations offers a potential predictive tool for planning nasal surgery.

This article provides an overview of the basic concepts of computational fluid dynamics, and a summary of the current capabilities of this technology in the characterisation of nasal airflow. The objective is to give otorhinolaryngologists a basic understanding of the computer modelling of nasal airflow, and the background information with which to evaluate CFD-based rhinology literature.

Key words: airflow, computer simulation, human, nasal obstruction, nasal surgery

INTRODUCTION

The complex anatomy and physiology of the nasal airway has proven a significant obstacle in the understanding and treatment of nasal obstructive disorders. Much of our current concepts of nasal airflow characteristics are based upon experiments on a relatively small number of nasal cavities [1-6]. Many of these were cadaver studies, where the septum was often replaced with a glass plate and coloured smoke or dyes were then used to visualise nasal airflow. In these studies, the effect of post mortem changes and the introduction of an artificial septum must raise doubts about the validity of the conclusions reached.

In addition to an incomplete basic understanding of nasal airflow, currently available methods for objective assessment of nasal patency provide only global characterisation of nasal cavities (overall resistance, peak flow) or limited specific information (narrowest cross-sectional area) and their contribution to the management of nasal airflow disorders has been disappointing. The narrow calibre of the nasal passages means that not only is it difficult to position measuring devices within the nose, but that any such device will then obstruct the airflow that it was intended to measure. Thus, ENT surgeons tend to rely solely on clinical judgement when deciding whether a patient with symptoms of nasal obstruction would benefit from surgery, and in deciding what type of procedure should be performed. Unfortunately, the results of surgical intervention are not always favourable. The ability to predict detailed airflow characteristics for a specific patient using computer simulation may be of considerable benefit and may help to determine pre-operatively the value of potential surgery.

Computer modelling of fluid flow, using a technique known as computational fluid dynamics (CFD), is a mature and trusted technology that is widely used in the engineering industry. It is capable of making accurate predictions about fluid behaviour in a computer model of any flow system being investigated. CFD predicts all the flow variables within the flow system; for example, pressure and velocity. It can offer the rhinologist the ability to visualise airflow characteristics in three dimensions, assess pressure-flow relationships; and investigate flow phenomena not readily obtainable by other means, e.g. nasal wall friction or the uptake of inhaled toxins. In recent years CFD predictions of nasal airflow have begun to appear in the literature [7-15]. However, this remains a highly specialised field and little is known about it within otorhinolaryngology. This article is intended to

*Received for publication: April 5, 2005; accepted: August 4, 2005

introduce the basic concepts, terminology and capabilities of CFD to help otorhinolaryngologists understand and evaluate publications pertaining to the use of CFD in rhinology.

BACKGROUND

Over the centuries, by observation and experiment, scientists have developed and refined mathematical laws that describe the world around us. For example, Blaise Pascal determined that pressure is equal to force over area (the Pascal unit of pressure is named after him) and Isaac Newton determined that rate of change in momentum is proportional to the force applied (the Newton unit of force is named after him). Gradually it was realised that some of these laws had almost universal application such as the conservation of energy (energy in a system can change from one form to another but total energy always remains the same) and conservation of mass (the amount of substance going into a system must equal the amount coming out). This meant that new equations could now be formulated, based on equations already known, to describe other aspects of the physical world. The discipline of theoretical physics grew from such beginnings.

In the nineteenth century, a set of equations was sought that could describe all the parameters of fluid flow (fluids encompass both liquids and gases). These equations would allow any aspect of fluid behaviour (e.g. speed, pressure, temperature) to be calculated at any time or at any place in a fluid flow. Furthermore, calculation of the fluid flow at every point in the flow for the duration of the flow would mean that the entire flow could be simulated. One hundred and fifty years ago such an equation set, commonly referred to as the Navier-Stokes equations, was elucidated. They are expressions of the conservation laws of mass, momentum and energy and how these quantities are distributed through the fluid flow by forces acting on and within the fluid. These forces would have effect over time and distance: integration with respect to time and distance would therefore yield the solution to fluid flow. However, integration is a complicated mathematical operation that only has exact solutions for limited circumstances. Therefore, the Navier-Stokes equations could not be solved except for very simple flows.

Since the beginning of the 20^{th} century, mathematical techniques (numerical methods) have been developed that allow calculations to be performed on increasingly complicated flows using the Navier-Stokes equations. Numerical methods are ways of formulating mathematical problems in such a way that they can be analysed by arithmetic means (addition, multiplication, division, etc.). To predict the characteristics of a fluid flow, a solution to the Navier-Stokes equation is required at every point in the flow. The volume of the flow is therefore divided into many smaller volumes (or *cells*), to form a *mesh* or *grid*. For each cell, the integral equations are replaced with arithmetic equations. Initially, an estimate is made of what each of the variables in one of the equations might be for one of the cells. That equation is solved and the answer is used as a starting point to solve all the other equations in the mesh. The edges of the mesh represent the external boundaries of the flow. The conditions at the boundaries are stated; hence they are 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). The answers to the arithmetic equations in the cells adjacent to the boundaries are compared to these specifications. If they are more or less than the specifications, greater or lesser numbers are used to begin the calculations all over again. The process continues over and over (*iterations*) until the answers get closer and closer to those specified at the boundaries. The final answer will be the solution to the Navier-Stokes equations plus or minus a small error. The resultant approximation of the flow using numerical methods is termed a *numerical model*.

At the beginning of the 20^{th} century, there was only one way to solve all these arithmetic equations – manually! Examination halls full of students, each solving hundreds of equations (in return for a penny), proved that the numerical technique did work, but too much manual effort (and expense!) was required to use the method frequently. It was not until the modern computer provided the ability to perform vast numbers of calculations efficiently that CFD became a practical engineering tool.

COMPUTATIONAL FLUID DYNAMICS

Computers have allowed the rapid automation of the numerical methods used to approximate the solution to the Navier-Stokes equations. Hence the terms computer model and computer simulation are often used interchangeably with numerical model and numerical simulation. Rapid advances were made in numerical techniques and computer programming for CFD throughout the second half of the twentieth century. This was largely driven by aerospace programmes and the need to design both better aircraft and better engines, particularly turbine systems. Successful engineering applications, including the space programme, have made CFD a highly developed, mature and trusted technology in industry. Despite this, CFD has only recently begun to be used to investigate nasal airflow. There are several reasons for this:

Firstly, to begin the CFD process a geometric model of the flow system is required (Figure 1). For the nasal cavities this means obtaining a CT or MRI scan, in its electronic format so that it can be loaded straight into a computer. In order to obtain a realistic CFD flow calculation, the scan must be sufficiently detailed to allow the shape of the nasal cavities to be accurately represented in the geometric model. This requires closely spaced scan images (less than 2 mm) and in practice, prior to the introduction of helical CT scanning, this was not possible without exposing the patient to a large radiation dose.

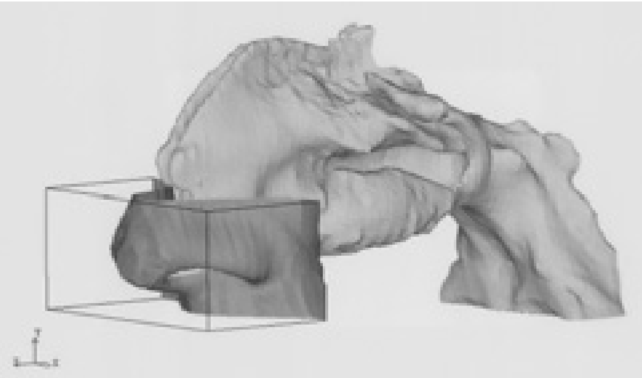

Figure 1. Three-dimensional computer model of the air contained within the nasal passages. A box of air surrounding the external nose is used as the inflow boundaries.

Secondly, the geometric model of the flow system has to be divided into many smaller volumes to create a mesh (Figure 2). The cells within a mesh are classified as structured or unstructured. A structured mesh has a repeating geometric structure and is usually formed of 6-sided bricks (hexahedra). An unstructured mesh is commonly formed from triangular-based pyramids (tetrahedra). Until recently, CFD calculations were reliant upon having a structured mesh. Structured meshes are, however, extremely difficult to create within a complex geometry such as the nose: Kimbell reported an investment of three to six person-months to produce a single new mesh [16], essentially precluding this technique as a practical tool in rhinology. Fortunately, the last few years have seen the introduction of numerous high-quality flow solvers for unstructured meshes and more 'user-friendly' unstructured mesh generation packages.

Thirdly is computing power itself. Computing capacity has been growing at an exponential rate. Whereas a PC today can perform a CFD analysis on an unstructured mesh of the nasal cavities in less than a day, a super computer ten years ago would have struggled to complete this task at all.

Thus, in today's world of high speed personal computers, advanced numerical and meshing techniques, and state of the art multi-slice low-dose CT scanners, computational fluid dynamics is fast becoming the engine of rhinological research. Many more publications in this field can be expected over the coming years.

HOW TO EVALUATE A RHINOLOGY PAPER USING CFD AS THE RESEARCH METHOD

As previously stated, CFD models the airflow inside the nose; i.e. it provides a close approximation to the solution of the Navier-stokes equation. The research paper should provide sufficient detail on the CFD technique, with reference to experimentally derived information, to allow the reader to establish the accuracy of the results.

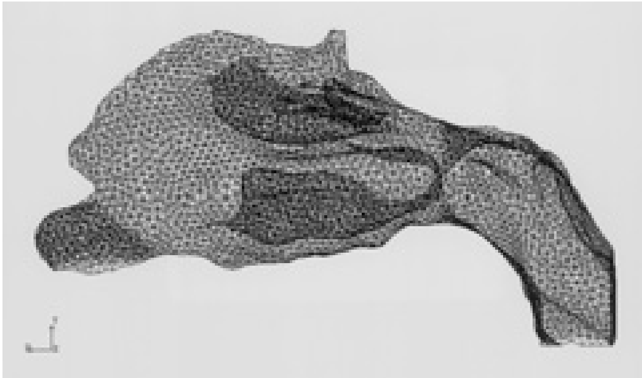

Figure 2. An example of a nasal airway mesh.

1. Assessing the CFD technique

a) Nasal Geometry

The technique begins with acquiring a geometrical representation of the nasal cavities from a CT or MRI scan. The scan should consist of the entire nose from the nasal tip (not part of a routine diagnostic nasal scan) to the posterior wall of the nasopharynx. If these extremes are not included, then the effect of the nostrils on drawing in airflow and the effect of the nasopharynx directing the airflow down to the pharynx will not be modelled leading to a potentially unrealistic representation of airflow in the nasal cavities. The distance between the scan slices should also be sufficiently small. When making a geometrical model of the nose the computer will join one slice to another with straight lines. A slice width of more than 2mm tends to produce stair-step artefact (a phenomenon whereby a smooth curve is represented by a series of jagged steps) in the outline of the nasal cavities.

A potential problem is that nasal hairs do not appear on the scans. However, the only study to specifically consider the influence of the nasal hairs on airflow found that their absence had no significant effect on the flow within the nasal cavity [17]. Another potential difficulty arises because the nasal cavity is treated as a rigid, static, structure. The nasal alae are capable of movement at very high inspiratory flow rates ($>$ 45L/min). However, during normal nasal breathing these tissues resist movement by virtue of their own impedance to deformation, aided by the action of dilator muscles contracting synchronously with respiration [18]. Thus, for a resting breathing cycle it is acceptable to model the nasal cavity as a rigid structure. Vascular engorgement of the turbinates and septal body can produce changes in the airway calibre over time. In some individuals this process has been described as occurring reciprocally in each nasal cavity, i.e. as a nasal cycle. The ability to remodel computer geometry allows for the correction of the effects of asymmetrical vascular engorgement if necessary. However, except for the instant at which the CT scan was performed, further conjecture about the influence of a possible nasal cycle on a patient's airflow cannot be made using CFD.

b) The mesh

The reader should be satisfied that a good mesh has been created. This means that the geometry should have been divided into enough small volumes (cells) so that the airflow pattern is properly represented and not averaged across large regions to the extent that small but important features of the airflow fail to appear in the final simulation. However, too many cells will result in long computation times to model the airflow. Thus, in practice, a balance must be achieved between having a sufficient mesh density to correctly predict the flow behaviour without unnecessarily increasing the computation time. The number of cells within the mesh will depend on the meshing technique and the particular nasal geometry being investigated. A statement by the authors that a grid-independence study, or grid adaptation, has been performed should satisfy the reader that the mesh density is adequate. Grid-independence studies are comparative studies using finer and finer meshes (i.e. meshes consisting of smaller and smaller volumes, amounting to increased mesh density). Increasing the mesh density beyond a critical number of tiny volumes will incur little change in the CFD prediction. Beyond this point the results become independent of mesh density, i.e. grid independence. Grid independence studies do not need to be performed on all nasal geometries; it is sufficient that they are performed on one reference geometry provided changes are not made to any of the CFD techniques. Alternatively, grid adaptation can be utilised. It is a process whereby only parts of the nasal cavities are divided into a finer mesh. Usually these are areas where the flow appears complex.

The exact size of the cells within a mesh can vary from one volume to the next to allow them to be fitted around the nasal geometry, but should not differ by more than 30% from neighbour to neighbour to prevent errors in the calculation (referred to as mesh smoothness). The volumes may sometimes be skewed trying to fit them into the nose, again with possible errors in calculations. The mesh should contain as few skewed elements as possible. Reporting that a mesh has been subject to a quality assessment should reassure the reader that smoothness and skewness have been examined.

The mesh must begin and end at definite points: the inlet and outlet in the case of the nasal airflow simulations. The mesh can extend beyond the limits of the nasal cavity: part of the air in front of the face can be modelled to give a realistic inflow of air, or tubes can be modelled to simulate devices such as nasal inhalers. Computer models where the mesh begins at or inside the nostril, or finishes before the nasopharynx is reached, may not provide realistic simulations of airflow, by failing to account for the influence of upstream or downstream geometry on the flow within the nasal cavity.

c) The mathematical model

Simplifications to the Navier-stokes equations are usually made when modelling nasal airflow to speed up the process. This is referred to as mathematical model selection or approximation level. Basically, if a characteristic of a flow is already known or the effect of a particular entity is thought to be negligible, then there is no point calculating this. The reader should be satisfied that they agree with these assumptions. Nasal airflow is usually simplified by modelling it as an incompressible flow. This is acceptable below a Mach number of 0.3 (Mach number is defined as the ratio of the flow speed to the local speed of sound). A typical peak Mach number in the nasal valve is about 0.03, which easily justifies the choice of incompressible flow.

The movement of nasal airflow is usually modelled as if it were a steady flow (a continuous inhalation or exhalation) rather than cyclical, as it is during the respiratory cycle. It is possible to model the entire nasal respiratory cycle using CFD, and indeed this has been done by our research group. However, both theoretical calculations [9] and the experimental findings of Proetz [1] suggest that nasal airflow may be considered quasi-steady, in that the acceleration and deceleration phases would not substantially alter the flow patterns observed during the constant flow rate phase. Moreover, time dependent calculations are significantly more expensive in terms of computer resources.

Nasal airflow should be treated as a viscous fluid (subject to forces of friction). The presence of viscous effects raises the question whether the flow is laminar or turbulent. Most people have a concept of laminar flow being a straight streamlined flow and turbulence being an unpredictable haphazard mixing of a fluid. Somewhere in between there is a transition between laminar and turbulence. A more precise definition allows us to see that a degree of mixing in a flow still allows it to be described as laminar. A laminar flow is one in which the mixing processes are molecular, i.e. they arise from interactions at the molecular scale. Such a flow includes small perturbations. In contrast, the mixing in a turbulent flow occurs over a wide range of length scales producing an unstable flow with the most vigorous mixing occurring at the largest scales. Reports of eddies observed at low nasal airflow velocities are to be explained as the result of flow separations rather than necessarily indicating that turbulence is present. The numerical modelling of turbulence requires the solution of additional equations to those used for laminar flow. Previous difficulties in turbulence modelling have been cited as a preclusion to the use of CFD for nasal airflow prediction [6]. Recent advances in computer processing power, and the introduction of robust numerical turbulence models, now permits modelling of nasal airflow as a turbulent flow. However, experimental evidence suggests that nasal airflow is predominantly laminar at resting flow rates [5,17,19,20]. Thus for airflow during resting (e.g. sitting) conditions the less computationally-expensive modelling of laminar flow can be applied.

Modelling of heat and moisture transfer from the nose to the nasal airflow is readily achievable using CFD and is an important capability for nasal airflow research. Again, this increases computation time and theoretical calculations [9] have shown that omission of heat and moisture transport from the CFD model should not affect the overall flow structures during normal respiration.

2. Experimental Validation

Confidence in the results of computer simulated airflows will ultimately come from experimental validation of the results. This requires the creation of physical models of the nasal cavity and the building of flow rigs to simulate fluid flows. It is both time consuming and expensive. Two robust methods of experimental validation have been published in the engineering literature. One is to make a large scale physical model of the nose so that small probes can be inserted to measure the airflow without causing a significant disturbance to the flow [17]. The second method is to create transparent physical models. Small particles can be added to the fluid flow through the model and pictured with lasers and cameras. This method (particle image velocimetry) permits non-invasive measurement of flow in the physical model [3].

It is not necessary to experimentally validate all nasal geometries, but a change in CFD technique (or software) should be validated on a physical model.

CLINICAL RELEVANCE

At present, numerical simulation of nasal airflow requires the patient to undergo a nasal CT or MRI scan followed by processing through a number of separate computer programmes. It is now possible to produce a numerical simulation following a CT scan in approximately one person/day. However, even with optimisation of the airflow analysis via a single automated computer programme, the reliance on nasal scanning is likely to preclude its routine clinical use in all patients. There are two main potential applications of numerical modelling of nasal airflow that are of relevance to clinical practice: Firstly, the technique offers a powerful research tool to assist in improving our understanding of basic nasal airflow physiology in both health and disease. Secondly, it offers the potential for planning surgery aimed at relieving nasal obstruction. It is relatively simple to alter the geometry of computer-generated models of the nasal airway; thus, it is possible to simulate surgery by correcting the perceived anatomical abnormalities on the computer model – so-called 'virtual surgery' – and compare flow predictions between the native and virtually operated geometries. Whilst use of this technique is clearly not warranted in the case of a gross obstructing lesion, it may be of benefit in more complex obstructing lesions or where the surgeon is unsure of the exact relevance of the obstruction. This may allow corrective surgery to be tailored to an individual nose and may be of particular use in helping the surgeon decide whether a more minor obstructing lesion warrants corrective surgery at all.

ACKNOWLEDGEMENTS

Research into nasal airflow using CFD in Queen's University Belfast by the authors Brendan Hanna and Neil Bailie has been supported by Fellowship funding from Antrim Area Hospital, Northern Ireland.

REFERENCES

1. Proetz AW (1951) Air currents in the upper respiratory tract and their clinical importance. Ann Otol Rhinol Laryngol 60: 439-467.
2. Swift DL, Proctor DF (1977) Access of air to the respiratory tract. In: Brain JD, Proctor DF, and Reid LF eds. Respiratory Defence Mechanisms Part 1. Marcel Dekker Inc: 63-93.
3. Girardin M, Bilgen E, Arbour P (1983) experimental study of velocity fields in a human nasal fossa by laser anemometry. Ann Otol Rhinol Laryngol 92: 231-236.
4. Horung DE, Leopold DA, Youngentob SL, Sheebe PR, Gagne GM, Deaver T, Mozell M (1987) Airflow patterns in a human nasal model. Arch Otolaryngol Head Neck Surg 113: 169-172.
5. Schreck S, Sullivan KJ, Ho CM, Chang HK (1993) Correlations between flow resistance and geometry in a model of the human nose. J App Physiol 75: 1767-1775.
6. Simmen D, Scherrer J, Moe K, Heinz B (1999) A dynamic and direct visualization model for the study of nasal airflow. Arch Otolaryngol Head Neck Surg 125: 1015-1021.
7. Tarabichi M, Fanos N (1993) Finite element analysis of airflow in the nasal valve. Acta Otolaryngol. Head Neck Surg 19: 638-642.
8. Elad D, Liebenhal R, Wenig BL, Einav S (1993) Analysis of airflow patters in the human nose. Med & Biol Eng & Comput 31: 585-592.
9. Keyhani K, Scherer PW, Mozell M (1995) Numerical simulation of airflow in the human nasal cavity. J Biomech Eng 117: 429-441.
10. Subramaniam RP, Richardson RP, Morgan KT, Kimbell JS (1998) Computational fluid dynamics simulations of inspiratory airflow in the human nose and nasopharynx. Inhal Toxicol 10: 473-502.
11. Chometon F, Ebbo D, Gillieron P, Koïfman P, Lecomte F, Sorrel-Dejerine N (2000) Simulation numérique de l'écoulement de l'air dans les fosses nasals. Ann Otolaryngol Chir Cervicofac 117: 98-104.
12. Bockholt U, Mlynski G, Muller W, Voss G (2000) Rhinosurgical therapy planning via endonasal airflow simulation. Comput Aided Surg 5: 175-179.
13. Hörschler I, Meinke M, Schroder W (2003) Numerical simulation of the velocity field in a model of the nasal cavity. Comp Fluids 32: 39-45.
14. Lindemann J, Keck T, Wiesmiller K, Sander B, Brambs HJ, Rettinger G, Pless D (2004) A numerical simulation of intranasal air temperature during inspiration. Laryngoscope 114: 1037-1041.
15. Pless D, Keck T, Wiesmiller K, Rettinger G, Aschoff AJ, Fleiter TR, Lindemann J (2004) Numerical simulation of air temperature and airflow patterns in the human nose during expiration. Clin Otolaryngol Allied Sci 29: 642-647.
16. Kimbell J (2001) Computational fluid dynamics of the extrathoracic airways. In: TB Martonen ed. Medical Application of Computer Modelling. The Respiratory System. Southampton: Wit Press: 12-45.
17. Hahn I, Scherer P, Mozell M (1993) Velocity profiles measured for a large-scale model of the human nasal cavity. J App Physiol 75: 2273-2287.
18. Cole P (2000) Biophysics of Nasal Airflow: A review. Am J Rhinol 14: 245-249.
19. Bailie N (2005) Modelling nasal airflows. Belfast, Northern Ireland: Queen's University of Belfast. PhD Thesis.
20. Grant O, Watterson J, Bailie N, Cole J, Currie T, Thornhill D (2005) Validation of a numerical method for assessment of human nasal airflow.
AIAA: 5000.

Mr Neil Bailie FRCS
51 Earlsfort
Moira BT67 0LY
Northern Ireland