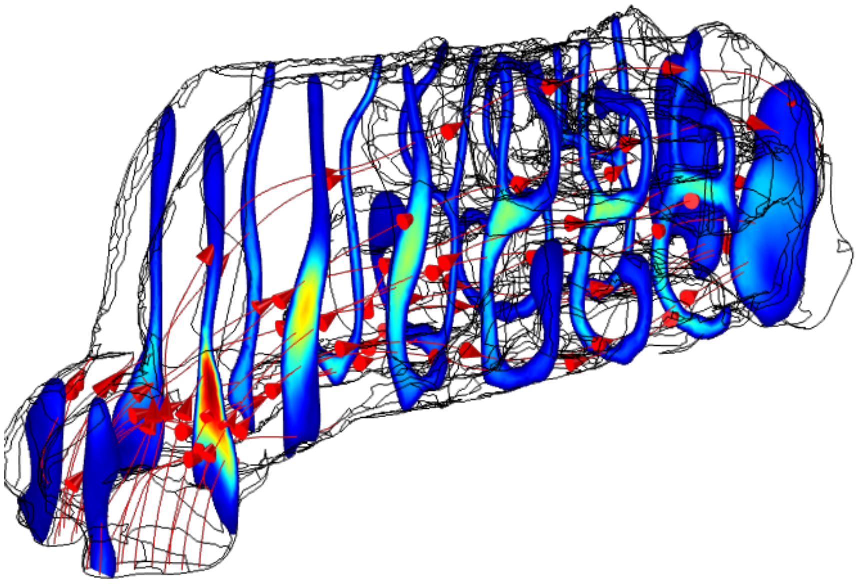
CHALMERS
UNIVERSITY OF TECHNOLOGY



Nasal Obstruction - Diagnosis and Prediction using Computational Fluid Dynamics

Using Computational Fluid Dynamics (CFD) to Study Nasal Obstructions and Studying Digital Demonstrations in Fluid Dynamics

Master's thesis in Learning and Leadership

Johan Ronnås and Jakob Widebrant

Mathematical Sciences
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2020

Master's thesis 2020

Nasal Obstruction - Diagnosis and Prediction using Computational Fluid Dynamics

Using Computational Fluid Dynamics (CFD) to Study Nasal Obstructions and Studying Digital Demonstrations in Fluid Dynamics

Johan Ronnås and Jakob Widebrant

Mathematical Sciences
Chalmers University of Technology
Gothenburg, Sweden 2020

Nasal Obstruction - Diagnosis and Prediction using Computational Fluid Dynamics

Using Computational Fluid Dynamics (CFD) to Study Nasal Obstructions and Studying Digital Demonstrations in Fluid Dynamics

Johan Ronnås and Jakob Widebrant

Supervisor: Torbjörn Lundh, Mathematical Sciences
Examiner: Samuel Bengmark, Mathematical Sciences

Master's Thesis 2020
Mathematical Sciences
Chalmers University of Technology
SE-412 96 Gothenburg
Telephone +46 31 772 1000

Cover: Visualisation of flow during inhalation in the nasal passage.

Typeset in LaTeX
Gothenburg, Sweden 2020

Nasal Obstruction - Diagnostic and Prediction using Computational Fluid Dynamics

Using Computational Fluid Dynamics (CFD) to Study Nasal Obstructions and Studying Digital Demonstrations in Fluid Dynamics

Johan Ronnås and Jakob Widebrant
Mathematical Sciences
Chalmers University of Technology

Abstract

When a patient visits the doctor and expresses trouble breathing through the nose, the diagnostic method and whether nasal surgery is needed comes with a lot of uncertainties. Only around 60-80 % [27] of the patients are satisfied with the initial procedure. In order to better understand the airflow in the nose and what consequence the standard operations have, the airflow the nose have been simulated and studied. Using CT-scans and 3D models CFD (Computational Fluid Dynamics) simulations have been done. In order to study what effect an operation would have, these models have been altered according to common surgical procedures. The CFD simulations for the 3D models resulted in reconstructed rhinomanometry curves (a common measurement on patients with nasal obstruction) which have some similarities with real rhinomanometry curves.
A lecture on the subject of fluid dynamics have also been performed for medical students. This was used for studying how different formats of online teaching effects learning. No significant difference in test results was found for different formats. However, a majority of the participants enjoyed the lecture and see future possibilities for collaboration with engineers.

Keywords: Nasal obstruction, Rhinomanometry, CFD, k-ϵ model, online lecture.

Acknowledgements

We would like to especially thank Torbjörn Lundh for all the help and guidance during this project. Johan Hellgren has also been an essential part in making this possible. Ebrahim Ghahramani for lending a helping hand when everything seemed lost (hope you the best in continuing this work). Last but not least Coffee, without it all those nights spending in trying to get the simulations to converge would not have been possible.

Johan Ronnås, Gothenburg, May 2020

Jakob Widebrant, Gothenburg, May 2020

Contents

List of Figures

List of Tables

1
Introduction

Nasal obstruction is a common problem among patients of all ages [2]. This problem can lead to difficulties for the respiratory system and be the cause of discomfort for patients. As such it is of importance to be able to accurately diagnose and treat nasal obstruction.
Computational fluid dynamics, CFD, is an area of physics involving the use of numerical techniques to solve fluid dynamic problems. Applying this to the airflow in the nasal passage can provide a better understanding of the characteristics of this system as well as provide useful tools for diagnosis and treatment. It is an interesting area to explore due to the combination of medicine, physics and numerical analysis. When working in such an interdisciplinary field it is also interesting to consider the communication and exchange of knowledge between individuals of different academic backgrounds. With an increasing proportion of such learning happening online it can be important to know what works well and if traditional ideas about teaching still apply. These are the areas that will be explored in this thesis.

1.1 Background

There are many causes for nasal obstruction, but they can generally be divided into two categories: anatomical and non-anatomical [7]. In other words, obstructions are either caused by the structural properties of the nose, such as deformations of the physical walls of the nasal cavities or by other issues such as inflammatory swelling of the nasal mucosa. It can therefore be difficult to differentiate between the different origins of the nasal obstruction and to diagnose the patient. Furthermore, nasal obstruction can be perceived differently by different patients, which causes more problems in identifying the source of the obstruction. In fact, only around 60-80 % of patients are satisfied after surgical procedure [27].
The tools available to doctors today includes external and internal inspection of the nasal cavity as well as measurements of airflow and pressure. These methods can however be quite limited in explaining the full picture of the situation due to the nose being a complex system of cavities in which the air flows. One way in which medical doctors could get a better idea of how the air flows within a particular patients nose is with the help of computer generated 3D models and CFD.
CFD is a tool based on solving partial differential equations numerically to calculate velocities and pressures of a fluid in a given geometry. COMSOL Multiphysics is one set of computer software that provide the functionality to perform such calculations. Given therefore a model of a patient's nose, it is possible to get an approximation

for the dynamics of the air in all parts of the model, something that is difficult to do using only physical measuring tools.
Building on this method of using computational techniques for diagnosis, one can also imagine that treatments including surgical procedures or changes of the nasal cavities can be simulated. This simulation can then be used to predict the outcome and effectiveness of potential treatments.
One of the first steps in the process of using CFD for diagnosis of nasal obstruction is to recreate what the current tools can provide doctors. One of the main tools today is rhinomanometry, which is a screening measurement the airflow and pressure in one nostril at a time, and yields a curve ("rhinomanomatry curve"). This tells the doctor how much resistance there is in each nostril as well as the over all obstruction in the nose.
When developing new techniques or developing technical improvements in medicine that will be used by doctors, it is important that the new information is presented in a comprehensive way. As engineers and doctors come from different background and have limited insight into each other's fields, the communication process can become time consuming and there is risk for misunderstandings. Furthermore, digital communication in the form of video calls and online lessons is getting increasingly common and often is necessary because of long distances. During time of writing this report, the COVID-19 pandemic putting restraints on physical meetings. Therefore, a part of this project is to explore the effects of different teaching methods of fluid dynamics in a digital environment to medical students. The aim is to provide insight from a didactical perspective into what works well in this particular setting and with this particular audience. The overall aim is to further improve communication between engineers and doctors.

1.2 Purpose

The overall purpose of this project is to explore the possibilities in using computational fluid dynamics in diagnosis and treatment of nasal dysfunction by recreating rhinomanometry curves. Furthermore this project will investigate how fluid dynamics can be taught to a relevant audience in a digital environment. This purpose can be divided into the following questions:

1. How can nasal airflow be translated into a fluid dynamics problem and what are the consequences of different choices?
2. Can rhinomanometry, an existing diagnostic tool, be reproduced using CFD?
3. Can a surgical procedure be virtually reproduced and what consequences will be predicted using CFD?
4. What is the effect of varying formats in online teaching of fluid dynamics on learning?
5. What are important factors to consider in online teaching from a practical and didactical perspective?

The first two questions aim to create an understanding of what the key components are in building a reliable CFD-model of the nasal passage. By investigating different alternatives and examining how they affect the recreation of a well established measuring method, we can demonstrate the possibilities and limitation of the model.

Question three is an example of the use-case of a CFD-model where investigation is made virtually. Finally, the goal question four and five is to build a better understanding of teaching and communication between doctors and engineers in a digital environment.

1.3 Structure of Report

Because of the two areas that are studied in this thesis the structure of the report are somewhat different from a regular report. The CFD simulations and the study of the nose comes first (theory, method, results and discussion) then follows the workshop (theory, method, results and discussion). Last of all is a combined conclusion chapter in the end.

1.4 Ethical Considerations

This project will hopefully aid the diagnosis and treatment of nasal dysfunctions and it therefore has the potential to benefit society. The project may also prove to expand the understanding of how nasal air flow works and could be of value to further studies in this area. However, it is also important to consider the limitations of the results before applying any methods discussed in this report in practice.
Another ethical consideration in the work of this project is the use of personal data such as CT-scans. These images and other such data have been anonymised as to ensure the integrity of the individuals who are the sources of the data. The data collection in the lesson test was also anonymous in order to not share any personal information or opinions.

2

Theoretical Background for CFD Analysis

The use of computational fluid dynamics in the study of nasal airflow is an interdisciplinary topic. First of all it concerns the anatomy of the nose and medical considerations of the respiratory system. This includes how the nose is structured and the function of different parts. Secondly, a large part of this study is fluid dynamics and the equations that govern the way air behaves. Finally, there is an aspect of numerical analysis in using computer based methods in finding solutions to the fluid dynamics problems. As such this chapter will present a broad overview of some of these topics and their relevance to the project.

2.1 Nasal Anatomy

The anatomy of the nasal passage is, like the rest of the human body, complex with multiple different elements. Figure 2.1 shows the different areas of the nasal cavities, excluding the paranasal sinuses. The main focus of this project is the anatomy where the air flow during inhalation and exhalation, hence this figure. Key features that have been studied are the inferior turbinate (also called the lower concha) and the nasal vestibule (the part coloured green in the figure). In the area of the nasal vestibule the nasal valve - sometimes referred to as nasal isthmus - is the one of interest (at the break point between the green and the blue part in the figure). The nasal valve is a small opening approximately 1 cm from the nares beyond which the nasal cavity gets wider. Because of the small size, it is thought to be responsible for creating a majority of the resistance in flow in the nasal passage [24].

The anatomy of the whole body are in many aspect symmetrical on the right and left side. In an ideal case the anatomy of the nose are the same for right and left side, but there often are a lot of differences between them. Examples of those variations could be the amount of mucus, the nasal septum could be irregular (e.a septum deviation) and the conchae could be of different sizes and shapes etc. [7, 11].

On the grand scale the nose starts with two nostrils and the flow is separated by a wall. Down in the throat the two airways converge into one and is later joined by the mouth. The pathway of the air in the nose could overall be thought of like the letter “n”.

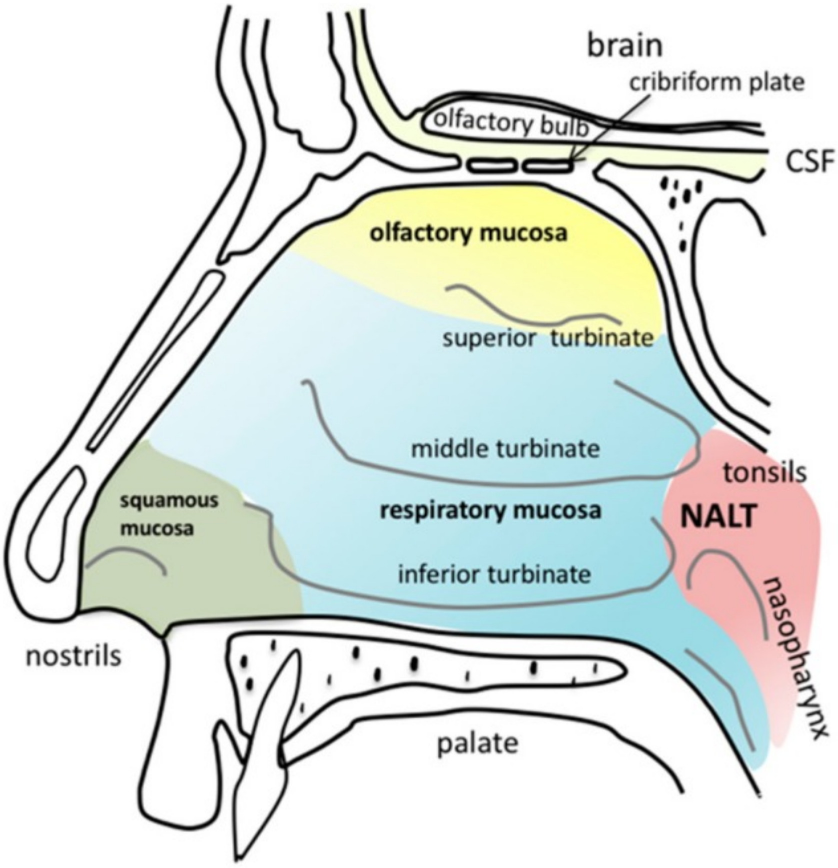


Figure 2.1: Note that in this report the inferior-, middle- and superior turbinate are referred to as conchae. The one of interest is the biggest concha (inferior tubinate). CC: Attribution, Gänger S, Schindowski K. doi: `https://dx.doi.org/10.3390%2Fpharmaceutics10030116`. `https://commons.wikimedia.org/wiki/File:Anatomy_of_the_human_nasal_cavity.png`. No changes where made to the picture.

2.2 Computational Fluid Dynamics

Fluid dynamics is mainly governed by one set of differential equations, namely the Navier-Stokes equations. These can be solved numerically using the Finite Element Method to describe the velocity and pressure of a fluid for a given set of conditions. These conditions include a given geometry with set properties for the each boundary, called boundary conditions, as well as properties of the fluid. One software for solving fluid dynamics problems is COMSOL Multiphysics which is used for this project.

2.2.1 Boundary Conditions

In the case of nasal airflow, there are some quite natural assumptions to be made for the geometry and boundary conditions. The nasal cavity can be modelled as a continuous volume from the nostrils to the nasopharynx. At each of these respective

"ends" there is an open boundary which the air can cross. In the case of inhalation, the nostril is acting as an inlet and the nasopharynx is acting as an outlet. All other boundaries are fixed walls.
When simulating the process of breathing, the walls are considered "no-slip" meaning that the air along the boundary has zero velocity. For the inlet we want to specify a given flow rate corresponding to the rate of breathing. This can be done in several ways. One approach is setting a fixed velocity (m/s) at this boundary which can be calculated as $\frac{Q}{A}$ where Q is the desired flow rate and A is the area of the inlet. Another way is to specify only the flow (ml/s), and letting COMSOL derive a fully developed flow profile for the inlet. Finally, a fixed pressure (Pa) can be set at the inlet, creating a pressure difference which will drive the flow. These different approaches imply subtle differences in the modelling of nasal airflow, but can lead to differences in the results. Finally, we want to specify a condition involving the pressure in the system. This is done at the outlet at a set relative pressure of 0 Pa to atmospheric pressure. It is useful working with relative pressures as the driving force of breathing is only the difference in pressure, not the absolute values.
In the eyes of a mathematician these three boundary conditions (pressure (Pa), velocity (m/s) and flow (ml/s)) should be equivalent. It should also be equivalent if the nostril or the throat is set as inlet/outlet and the other one is set to "open boundary". This is not necessary the case when using CFD in COMSOL. To summarise all the choices are listed below:

- Location of source: COMSOL wants to know where the source of flow originate, this can either be an inward flow (that is at the nostril when inhaling) or an outward flow (that is in the throat/nasopharynx when inhaling).
- Reason of flow: what physicals phenomena is the reason that the flow occurs. In the case of fluid dynamics the main reasons are: pressure, velocity and/or flow.[1]
- Laminar or turbulent: COMSOL can simulate both laminar and turbulent flow. This needs to be specified in order to compute the simulation. The turbulent flow also needs to specify which model, for this project the k-ϵ model was mainly used, see 2.3.3.
- Wall: how the wall of the nose behave, if it's considered to move and if no-slip is applied.

2.2.2 Fluid Properties

In regards to the fluid properties of the system, the properties of air is used, taken from COMSOL library of common materials[2]. These are presented in table 2.1.

2.3 Turbulence

One of the main difficulties in modelling fluid dynamics is the phenomenon of turbulence. Turbulence, or turbulent flow, is defined as the opposite of laminar flow.

[1]These three are in a direct correlations and if certain aspects of the geometry (for example cross sectional area) are known, these should be equivalent.

[2]`https://www.comsol.se/material-library`

Table 2.1: Values of fluid (air) properties used for simulation.

Dynamic viscosity	1.8140×10^{-5} Pas
Kinematic viscosity	1.5062×10^{-5} m^2/s
Density	1.2043 kg/m^3
Reference temperature	293.15 K

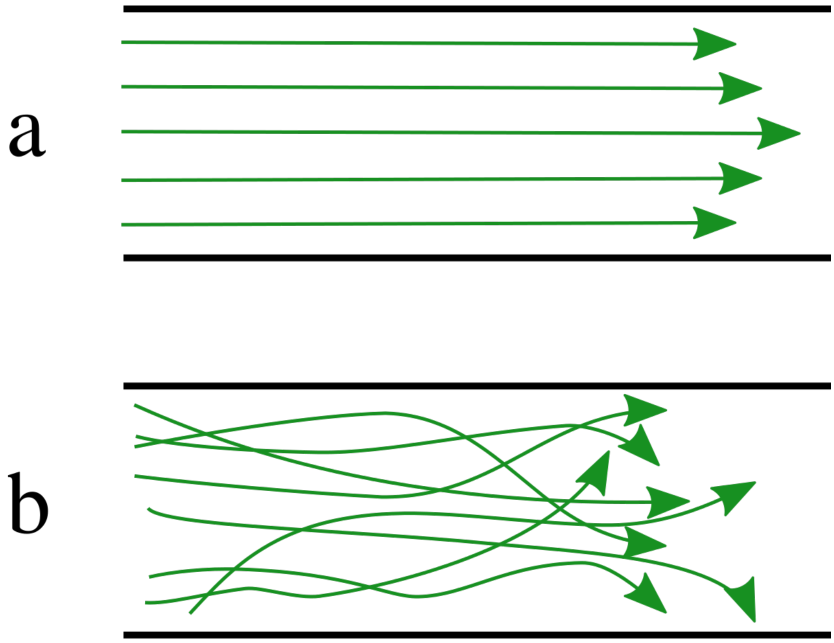


Figure 2.2: An illustration of laminar and turbulent flow. At the top (a) we see laminar flow that is smooth and have no mixing, at the bottom (b) we see turbulent flow with mixing and swirls.

Laminar flow is when fluid particles are moving in such away that there is little to no mixing [13]. As such, laminar flow can be understood as "smooth" while turbulence is more of a chaotic nature, see figure 2.2. This implies we can use a simpler model based on Navier-Stokes equation as opposed to turbulent flow which requires further assumptions in order to find useful solutions for[12]. One approach to this problem is by using the so called *Reynolds number, (R_e)* and the *Reynolds-averaged Navier-Stokes equations, (RANS)*[3].

2.3.1 Reynolds Number

The Reynolds number describes the ratio between viscous and inertial forces within a fluid and is dependent on the viscosity of the fluid, the size of the relevant geometry and the speed of the flow. If the viscous forces are dominating which means a lower Reynolds number, such as in a highly viscous fluid, the flow will be characterised as laminar. If inertial forces are instead the predominant, the Reynolds number will be larger and there is a greater tendency for turbulence. It is difficult to determine an

[3]Read more about the RANS equations on Wikipedia's web page about the subject `https://en.wikipedia.org/wiki/Reynolds-averaged_Navier\T1\textendashStokes_equations`

Table 2.2: Calculations of the Reynolds number.

Flow Rate [ml/s]	Corresponding Velocity [ml/s]	R_e
10	0.26	122
100	2.6	1220
150	3.9	1834
250	6.5	3056
500	12.992	6112

exact value of the Reynolds number for which the flow will be laminar or turbulent, but in a consensus report [3] concerning methods in rhinology stated that around 2300 is a critical point at which turbulent flow is likely to occur.
The Reynolds number for a given situation can be calculated as

$$R_e = \frac{uL}{v}$$

where u is the flow velocity (m/s), L is the characteristic dimension (m), and v is the kinematic viscosity (m^2/s) of the fluid. For the situation of nasal airflow we can use the kinematic viscosity of air at room temperature which is 1.5062×10^{-5} m^2/s. The characteristic length is not as straight-forward to find for a complex geometry such as the nasal passage. In a study by Segal et al. [25] they calculated a hydraulic diameter of the nasal passage to be between 5.8 mm and 8.4 mm. As such we can use an estimated value of 7 mm to get an idea of the range of the Reynolds number. R_e is calculated for different velocities and presented in table 2.2 indicating that the transition between laminar and turbulent occurs for flow rates between 150 and 250 ml/s

2.3.2 Other Studies

We can also consider previous studies and their assumptions on the nature of nasal airflow. Generally, there can be seen a large variation in the velocities used in simulations and therefore the the view on whether the airflow is turbulent or laminar. For instance, Keusterman et al. [17] and Zhao et al. [34] used a turbulence model for a flows greater than 150 ml/s while for lower flow rates they argue that a laminar model can be used with sufficient accuracy. Wen et al. [30] presented a list of different studies and their corresponding assumptions for given flow rates and they found that laminar models were used for flow rates up to 250 ml/s, agreeing with the estimated Reynolds numbers.

2.3.3 Kappa-epsilon Model

The most commonly used method for modelling turbulence in CFD is the *kappa-epsilon (k-ϵ) method*. It is based on using two variables and corresponding differential equations representing the turbulent kinetic energy within the fluid (k) and the dissipation rate of the turbulent kinetic energy (ϵ). According to COMSOL's web page [8] the k-ϵ model is good for convergence rate, low memory usage and simulation flow around complex geometries. Due to its high convergence rate it's often used as a "pre-simulation" to provide other models (such as k-ω, SST, v2-f, etc) with an initial value[4].

2.4 Rhinomanometry

There are several different methods of studying and diagnosing difficulties in nasal breathing among patients. In a consensus report [3] which aimed to describe aspects of different methods currently used, they present several variations on rhinomanometry including different types of pressure-measuring techniques as well as rhinoresistometry.
The majority of rhinomanometry is based on measuring the pressure and flow in the nose of a patient. The most common type is active anterior rhinomanometry [3, 5] in which the flow is measured in one nostril and the pressure is measured in the other, while the patient is breathing normally. This can be compared to posterior rhinomanometry where the pressure is instead measured at the pharynx using a tube. In either case the flow is plotted against the pressure creating a curve which can indicate the resistance within the nasal passage. The convention is to invert the values of flow in one of the nasal cavities to fit them in the same plot. Figure 2.3 shows an example of a rhinomanometry curve and figure 2.4 shows the idealised shape of the curve.
The nasal airway resistance (R) is calculated as $R = \Delta P/Q$ where ΔP is the difference in pressure and Q is the flow. As this resistance can vary with the pressure, it is standard practice [3, 5] to report the resistance at a pressure of 150 Pa.
Other and more sophisticated methods exist of using the measurements gained in rhinomanometry. When the air is moving at low speeds and is of a laminar characteristic, the pressure is directly proportional to the flow [9]. However, as the flow increases, this is no longer the case, and instead the relationship follows the quadratic Rhohrer's equation [9]

$$\Delta P = k_1 Q + k_2 Q^2,$$

which means the resistance is linearly related to the flow as

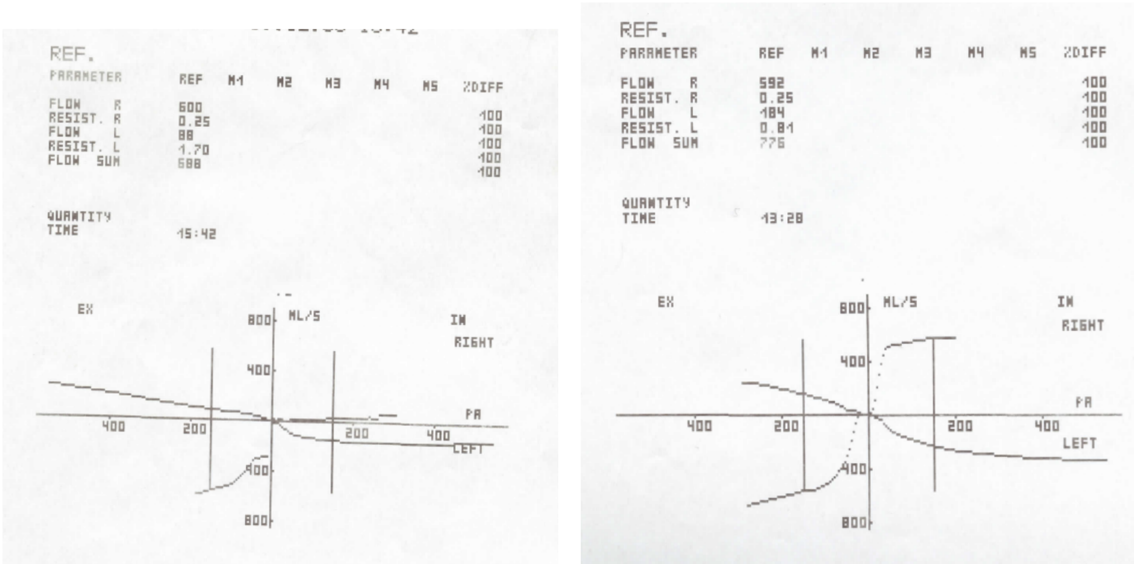
$$R = \frac{\Delta P}{Q} = k_1 + k_2 Q.$$

In the case of the nasal cavity, the flow can transition between laminar and turbulent during normal breathing. As such the consensus report [3] suggest an equation that combines the linear and quadratic relationship in Rhohrer's model:

$$\Delta P = m(Q) \cdot k_1 Q + n(Q) \cdot k_2 Q^2$$

where $m(Q) = 1$ and $n(Q) = 0$ during laminar conditions and $n(Q) = 1$ and $m(Q) = 0$ during turbulent conditions. This idea is the basis of rhinoresistometry

[4] Read more about the k-ϵ model on COMSOL's website: `https://www.comsol.com/blogs/which-turbulence-model-should-choose-cfd-application/`.



(a) Rhinomanometry curve from a patient. The patient has likely some obstruction in the nasal passage, as indicated by the low flow rate and the flat curve. Figure provided by Sahlgrenska University Hospital.

(b) Rhinomanometry curve from patients.The patients is experiencing increased flow and the data is from after an application of decongestant have been preformed. Figure provided by Sahlgrenska University Hospital.

Figure 2.3: The figures show the data from a rhinomanometry measurement. On the x-axis we have pressure in Pa and on the y-axis we have flow in ml/s. The line at 150 Pa indicate where reported values are taken. These values can be seen at the top of each picture.

[5] which works by plotting the airway resistance against the flow instead of flow against pressure. This creates a graph that during some parts is constant, indicating laminar flow, and other parts linear, indicting turbulent flow. This information combined with the actual values of the resistance can indicate the aerodynamics of a particular patients nasal cavity.

Finally, an interesting aspect of rhinomanometric methods is the problems in the usefulness of the results. While the measurements and derived values are an objective indicator of nasal obstruction, a report from Jones el al. [16] showed that there is no correlation to the sensation of nasal resistance. As such, it is possible that many patients complain of feeling obstructed while rhinomanometry fails to indicate any problems. Jones el al. [16] argues that airway resistance is mainly dependent of a small part of the nasal passage, namely the anterior region, while the receptors responsible for the sensation of obstruction is more evenly distributed. This also implies that procedures which can decrease airway resistance can have little to no effect on the patient's nasal sensation and patient satisfaction.

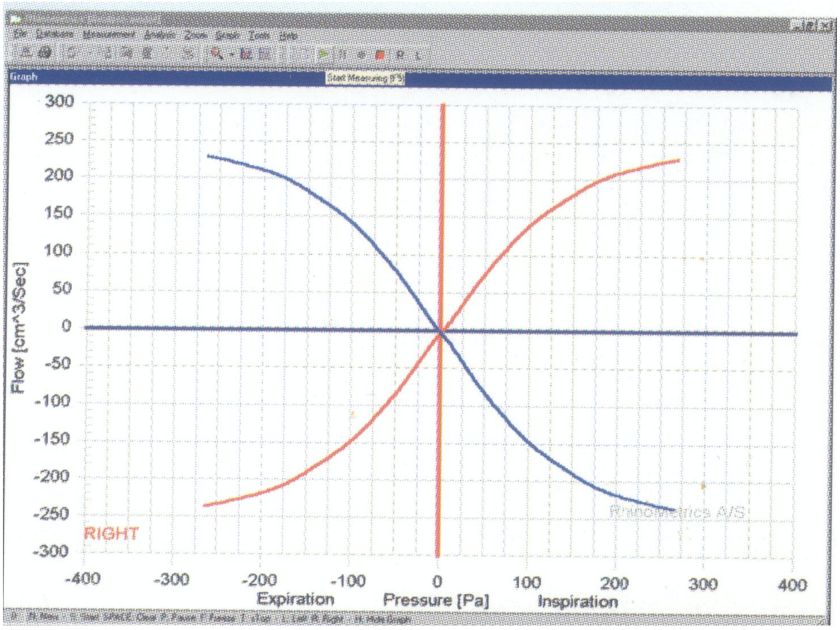


Figure 2.4: An "textbook example" of an rhinomanometry curve. If compared with figure 2.3a and 2.3b we see that this example differs a lot to the measurements done on patients. Figure provided by Sahlgrenska University Hospital.

2.5 Segmentation of CT-Scans

An important first step of using CFD tools for examining patients nasal passage is the segmentation of CT-scans and the generation of 3D models. The critical part of this process is identifying the parts in CT-scan that are part of the nasal cavity and relevant to the simulation. This identification process is no trivial task as it requires understanding of anatomy and is prone to human mistakes. The main difficulties lies in identifying the correct border between empty space and tissue, as well as in isolating the region of the nasal passage from other empty spaces, such as the paranasal sinuses. However, there are plenty of tools of automating some steps of this process as well as attempts at creating a fully attempted system.

One attempt of automatic segmentation of the nasal cavity was done by Huang et al. [14]. They developed a framework based on using statistical shape models which achieved a relatively high level of accuracy, however still requiring some level of manual processing before being ready for CFD simulations. Similar work was done by Keustermans et al. [17] in which they also used statistical shape modelling in preparation for CFD studies, but involving manual selection of the boundary regions in the CT-scans. Another approach was taken by Liu et al. [22] who developed through the use of image processing a "standard nose" by averaging dimensions of a set of patients' CT-scans. This approach of creating a standard geometry could also be helpful in dealing with the problem of having to manually segment CT images for CFD simulations. Finally, Zhang [33] attempted to create much simplified geometric model consisting of flat planes and building it based on characteristic dimensions of a patient's nose and found that it could partly replace a full reconstruction. In other words, there are several ways of automating or replacing the segmentation step of CT-scans in preparation of CFD analysis.

3

Methods for CFD Simulation

In order to investigate the use of CFD in analysis of nasal airflow, a number of different types of models have been developed using different types of geometry, numerical methods and processing of resulting data. All models use COMSOL Multiphysics for numerically solving the equations of fluid dynamics, i. e. Navier-Stokes equations. There where a total 6 different models (pipe, nose model from online source and four from ct-scans) and some of these models where altered in different ways. This chapter presents each model and their corresponding properties.

3.1 Breathing Flow Rate

We can determine an approximate range of values for the flow rate of air through the nasal passage by considering the volume inhaled and breathing rate. If we approximate the breathing process as a sine function, the maximum amount of flow can be calculated by:

$$\text{Vol}_{inhale} = \int_0^{\frac{t_0}{2}} A \sin \frac{x \cdot 2\pi}{t_0} \text{dx}$$

$$\text{Vol}_{inhale} = \frac{A \cdot t_0}{\pi}$$

where Vol_{inhale} is the total amount inhaled, t_0 is the total time for breathing (exhale and inhale) and A is a constant with the dimensions ml/s.
According to Gordon et al. [15] the total volume inhaled during so called quiet breathing is 500 ml and a adult takes about 12 to 18 breaths per minute. Which means the duration of a inhale and exhale is around 4 second. This yields A to be approximate

$$A = \frac{\text{Vol}_{inhale} \cdot \pi}{t_0} \approx 400 \text{ ml}/s$$

Because A is the amplitude of the sinus curve, A is equal to the maximum amount of flow and we can consider up to 400 ml/s to be a reasonable range for normal breathing. However, we see higher values in the rhinomanometry curves from patients in figure 2.3. Therefore a large range of values was used in simulating for different boundary conditions and different models (pipe and models from CT scans). The aim was to induce a flow of a value around 600-800 ml/s which is over the estimated value.

3.2 Pipe Simulations

In order to investigate the role of different settings in the setup of the fluid dynamics problem a simple model consisting of a straight pipe was made. This pipe had similar dimensions to the nasal passage with a diameter of 14 mm and a length of 100 mm. Six different scenarios were simulated:

- Laminar model, specified velocity at inlet.
- Laminar model, specified pressure at inlet.
- Laminar model, fully developed flow at inlet.
- Turbulent model, specified velocity at inlet.
- Turbulent model, specified pressure at inlet.
- Turbulent model, fully developed flow at inlet.

Measurements of flow and average pressure were then taken both at the inlet and for the whole volume.

3.3 Variation on Nasal Valve Size

One interesting part of nasal anatomy is the nasal valve. In order to investigate the effect on the relative size of this area on airflow, a simple model was created. This model is based on a CT-scan[1] but consisting only of the most anterior part of the left cavity, isolating the nasal valve region. Figure 3.1 shows this simplified model. For each model a CFD simulation was performed using a turbulence model and with a range of flow rates used as the inlet boundary condition.
In addition, four similar models were created with varying size of the nasal valve. In the first one the size of nasal valve was decreased by 40% and in the other three, the size was increased by incrementing steps of 10% percent. This yields a total of five variations which are displayed in figure 3.2.

3.4 CT-scans Models

Four CT-scans (called A, B, C and D) were provided by Sahlgrenska University Hospital and were anonymized. One of CT-scans were of a healthy nose, while the remaining three had varying degrees of obstruction. This section describes the process from CT-scan to simulation on a 3D model.

3.4.1 CT-scan to 3D model file

To convert CT-scans to 3D models a program called Slicer[2] was used. As the CT-scans depicted the whole head and part of the neck they had a lot more structure and details than needed for the simulation. Slicer could import CT-scans and then - either manually or automatic - convert certain areas to a 3D model (exported as

[1] found at *Embodi 3D*, `https://www.embodi3d.com/files/file/115-file-pack-for-3d-printing-with-osirix-tutorial/` a website dedicated to medical imaging and modelling.

[2] `https://www.slicer.org`

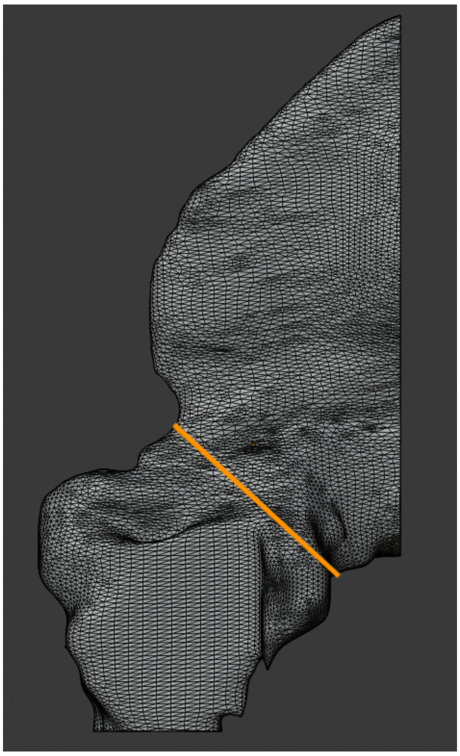


Figure 3.1: Simplified model for testing the effect of the size on the nasal valve. In the figure we see the nose from the side. The nostril (inlet) is at the bottom and to the right in the figure is approximately where the conchae begin (see figure 2.1).

an .stl file). Due to all of different areas in the nose with air and mucus, this made the process hard to automate. In other words most of the work was done manually by going through the CT-slices and marking the areas of interests, that is where the most of the air flows (see nose anatomy 2.1). Due to many irregularities, such as mucus, a lot of approximations where necessary. These approximations were discussed with a doctor to get as much accuracy as possible.
In Slicer a setting can be applied where you set an interval (between -1024 and 2048) of what you want to be a part of you structure. This interval have no physical representation but the lower numbers corresponds to lower density. For the most part an interval between -1024 and -300 was suitable to represent air, but to get a solid structure and remove for example mucus the upper bound was set to as much as 200 sometimes. The interval was found by trial and error.
Before the structure was exported as a .stl file, Slices own “smoothing” function was used to remove any big spikes or small holes in the structure.

3.4.2 Mesh Generation

In order to prepare the 3D model generated from a CT-scan there are a number of steps to be taken. The goal of this process is to produce a three dimensional mesh that can be used within COMSOL Multiphysics to simulate fluid flow. This mesh has a number of requirements, namely:

- A clearly defined surface for the inlet and outlet of airflow.

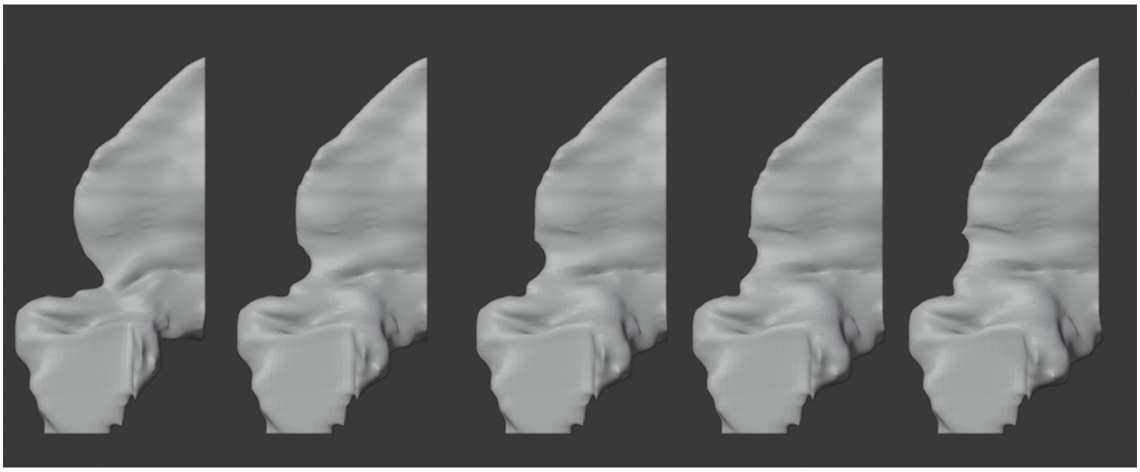


Figure 3.2: Surgery in silico of the nasal valve in a simplified model. From left to right: 60%, 100% (base model), 110%, 120%, and 130% of original size respectively.

- It is ”water-proof”, i.e. the surface is continuous with no holes or intersecting parts.
- The elements, both volume and surface types, are of good quality, i.e. they are of similar size and isotropic.

The first step in fulfilling these requirement was done by importing the 3D model into Blender[3], a 3D modelling software. Here one horizontal and one vertical cut was made producing clean surfaces for the inlets and outlet respectively. This could sometimes cause the mesh-triangles near the cut to deform creating holes and intersections, violating the second requirement. In each particular case, these problems were fixed by manually editing the mesh.
The model is then imported into COMSOL in which several different functions were used to create a surface mesh of high quality. These functions include an ”adapt” tool which given an absolute size attempts to reform the triangles of the mesh to the given size. This was done twice with increasing sizes given. In the second iteration, the algorithm mainly increase the sizes of the triangles which leads to a more isotropic mesh. Finally the mesh was refined by splitting each element a number of times to generate the desired number of elements before the free tetrahedral (three dimensional) mesh was generated.

3.4.3 Mesh Refinement Study

A simple mesh refinement study was performed on the first model A based on a CT-scan. Four different meshes were generated with approximately 200 thousand elements, 1.8 million elements, 2.8 million elements, and 4 million elements respectively. Using these meshes a simulation was performed using a laminar flow model with the boundary conditions of a fixed velocity corresponding to a flow of 250 ml/s at the inlet and a fixed relative pressure of 0 Pa out the outlet. For each simulation five different measures were taken in addition to the computation time: the average pressure at the inlet, the average and max pressure in the whole volume, and the

[3]`https://www.blender.org/`

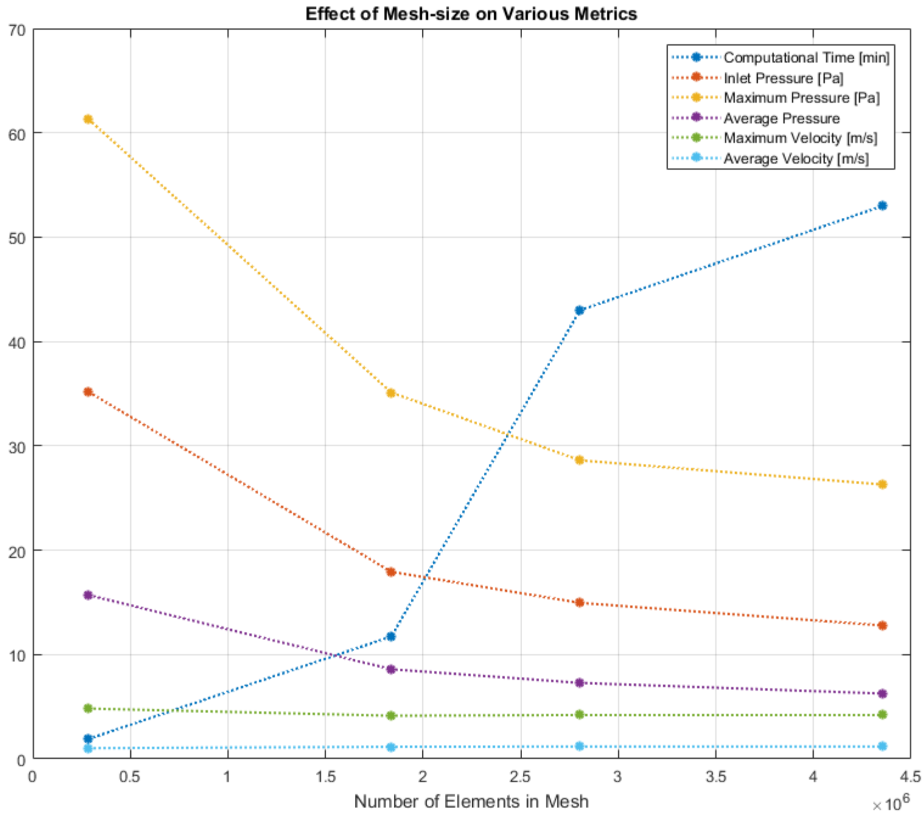


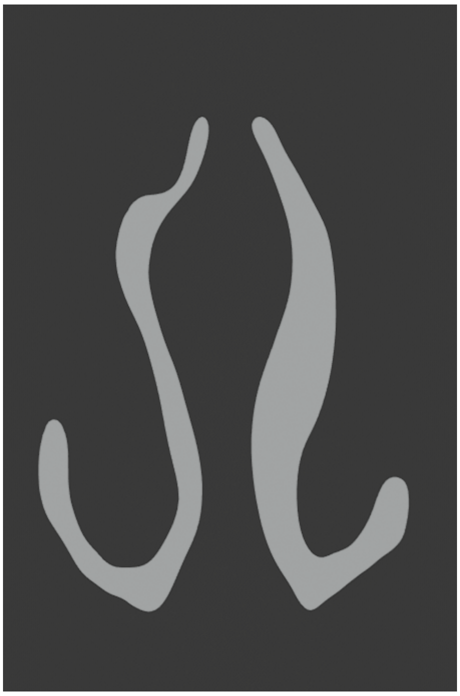
Figure 3.3: Result of mesh refinement study. Indicates the values of different metrics in the nasal cavity for different mesh sizes. These metrics include maximum and average velocity and pressure for the whole volume and for the inlet. We see the computation time sharply increase above 2 million elements but relatively smaller changes in the metrics.

average and max velocity in the whole volume. Figure 3.3 display the resulting values. Based on this data a mesh size of around 1.8 million elements was chosen to be used in the simulations using segmented 3D models. For finer meshes we see some difference in the resulting values but not enough to justify the sharp increase in computational time.

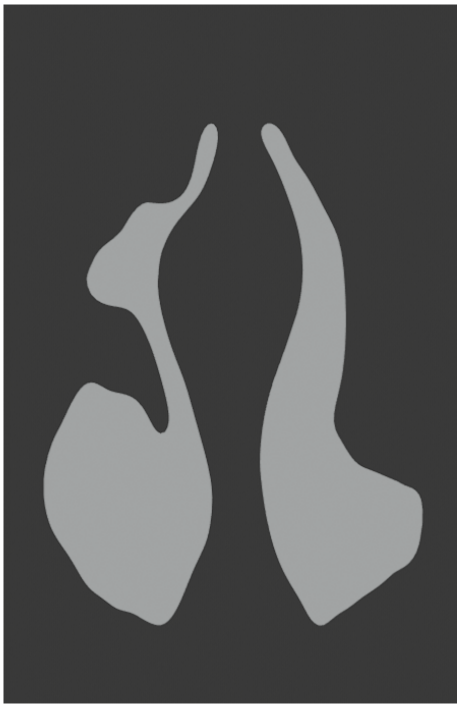
We can compare this number of elements to what has been used by some other similar studies. Keusterman et al. [17] performed a grid independence study and concluded that a mesh size of between 2 and 3.6 million elements was reasonable. They state that this is comparable to other studies, but a large variation can be found in the detail of geometric models. For instance, Segal et al. [25] used between 170 000 to 350 000 elements for their simulations while Kumar et al. [20] used a model consisting of up to 10 million elements.

3.4.4 Simulating a Common Procedure

One procedure which is sometimes used to treat nasal obstruction is to reduce the size of the conchae [32]. An interesting question is if it is possible to predict the result of such a procedure using CFD. Based on the model of the healthy patient this procedure was emulated by simply allowing more of the volume around the conchae to belong to the 3D model in the segmentation of the CT-scan. The resulting change can be seen in figure 3.4 which displays a coronal cross-section at 60 mm from the tip of the nostril.



(a) Before operation



(b) After operation

Figure 3.4: Cross section of the virtually operated nose. On the left is the nose before the virtual operation on the right is after the operation when the conchae has been reduced in size The light grey area is where the air could flow.

3.5 Recreating Rhinomanometry Curves

In order to analyse the results of all CFD simulations and provide comparison to real-life measurements, rhinomanometry curves for all models were recreated. This was done by letting one of the nostrils act as an inlet, while the other one was set as a wall. The conditions were then reversed After running the simulation, the average pressure and flow was measured at the inlet of the open nostril. This mimics how a rhinomanometry procedure is done in real-life where only one side is studied at a time. The flow for the right side is then inverted and is plotted together with the flow from the left side against pressure.

For the simplified pipe-model and the model of the nasal valve additional measurements were taken, including average and maximum pressure and velocity for the whole volume.

4

Results from CFD Simulations

The results are separated in to three different sections. First of all are the results from a CFD simulation on a pipe with roughly the same size as a nose, for different sets of boundary conditions. Then follows the simplified model of the nasal vestibule. The last section contains the results from models based on patient CT-scans, including the simulated surgical procedure.

4.1 Pipe Model

The simulations on the pipe model indicate there can occur large differences when using different boundary conditions. The pipe were in the same order of magnitude as a nose but there are obviously a lot of geometric differences between a pipe and a nose. The values and the recreated rhinomanometry curve differ substantially when comparing them to a standard rhinomanometry curve.

4.1.1 Laminar Flow with Different Boundary Conditions

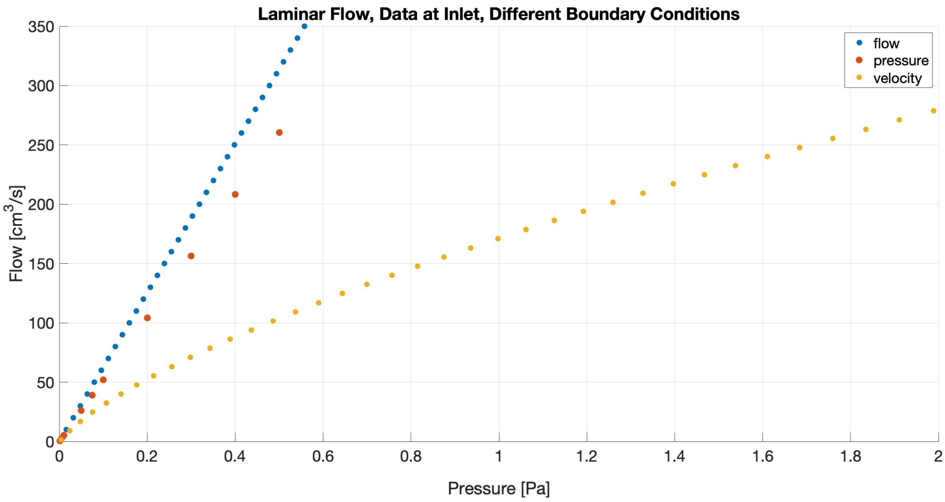
The results from the laminar flow simulations in the pipe are shown in figure 4.1. We see that using velocity as a boundary condition is less linear than for flow and pressure and generates higher resistance. There is also a larger variation in values from the other two cases. Using flow as a boundary condition yields the lowest resistance.

4.1.2 Turbulent Flow

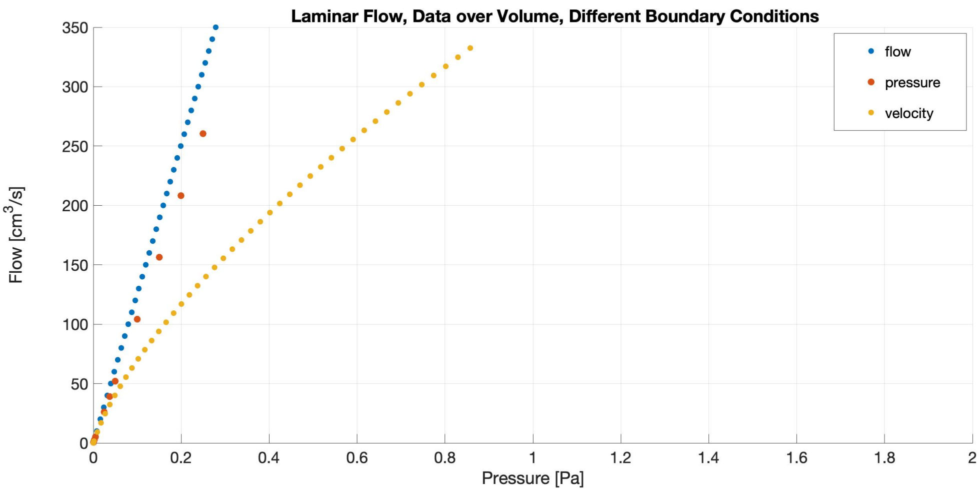
When using velocity and flow as boundary conditions the turbulent flow follows a non linear curve compared to laminar flow, as seen in figure 4.2. We notice in the figure that the pressure values are not in the same order of magnitude as the patients data.Using pressure as boundary condition seem to differ the most, since it part ways from the others. The maximum pressure that was able to converge was a pressure of 0.6 Pa. When increasing the pressure above that the simulation diverge[1].

There is a difference between laminar and turbulent in the simulated rhinomanometry curve as seen in figure 4.3. For relatively small values of flow the two data sets are almost the same (as expected) but as the flow gets higher the two curves departs form each other.

[1] An example of how the data looks when it diverge can be found in appendix A.2.



(a) Different boundary conditions at inlet. Values of flow and pressure are average taken at the inlet.



(b) Different boundary conditions at inlet. Values of flow and pressure are calculated by Comsol and are the average over the entire volume.

Figure 4.1: The figures display the pressure required to induce a given flow with given boundary conditions in a pipe and are measured in two different ways. Figure a) shows the values at the inlet for example when using velocity as boundary conditions and the pressure average at the inlet is 1 Pa, the flow is average 175 ml/s. In figure b) we see the values when using the same boundary conditions but the average over the entire volume, for example when using velocity as boundary condition and the average pressure is 0.3 Pa, the flow is 150 ml/s.

Due to the amount of time one simulation take on even such a simple geometry as a pipe the amount of data points have been reduced compared to the laminar flow simulations.

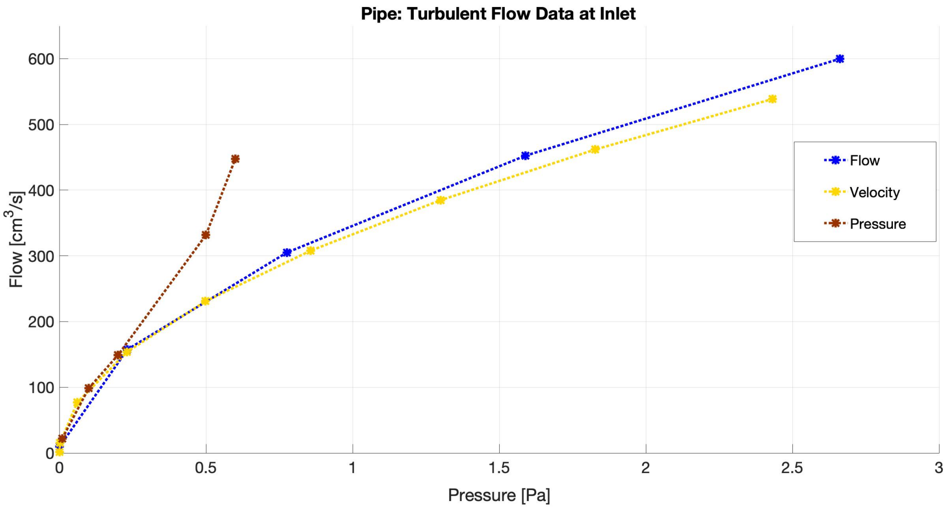


Figure 4.2: The figure displays the pressure required to induce a given flow with given boundary conditions.

4.2 Varying Nasal Valve Size

It was possible to simulate rhinomanometry curve for values of flow[2] between 0 ml/s and 181 ml/s. Some of these altered models where able to converge at 250 ml/s. Note that none of these rhinomanometry curves pressure values are in the same order of magnitude as the patient's data (see figure 2.3) or the textbook example (see figure 2.4). To study more similarities and differences between the original model and the altered size models, certain data where examined at a flow rate of 161 ml/s and at 250 ml/s.

4.2.1 Rhinomanometry Curves

The results from the simulations on the models with different size on the nasal valve can be found i figure 4.4. Notably all of the models that have been modified to have a bigger nasal valve follows roughly the same curvature as each other and as the original. The one rhinomanometry curve that differs from the other is when the nasal valve is a lot smaller then the original. For this case we see a curvature that looks more like a nose with obstruction.

[2]The one boundary conditions that was able to converge to a solution was when flow (ml/s) was set at the nostril.

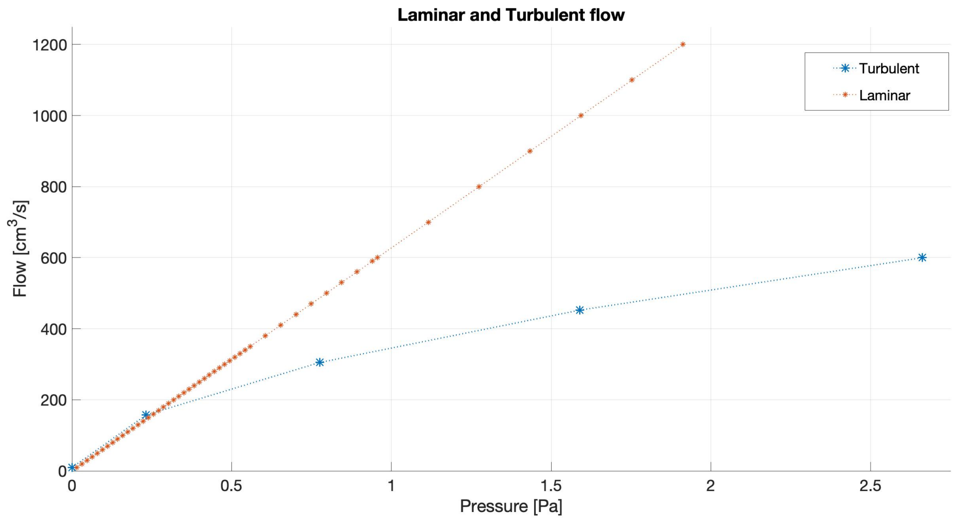


Figure 4.3: Comparison of laminar and turbulent flow in the pipe model. Both simulations where done with flow (ml/s) as boundary condition, there are a lot of fewer data points for turbulent flow due to the amount of time it takes to do one simulation on turbulent flow.

Table 4.1: Different derived values on the noses with different sizes on the nasal valve. The input was flow at 161 ml/s. More data can be found in appendix A.1, such as values of kappa (k), epsilon (ϵ), rotation rate and shear rate.

Flow: 161 ml/s

Scale of nose	0.6	1.0	1.1	1.2	1.3	1.4
Average						
Velocity	2.7	2.4	2.3	2.2	2.2	2.3
Pressure	6.17	1.46	1.48	1.43	1.41	1.55
Maximum						
Velocity	6.6	6.3	5.8	5.7	5.8	6.0
Pressure	22	89	12	16	12	12

4.2.2 Flow at 161 ml/s

For a flow of 161 ml/s we see in table 4.1 that velocity and pressure - on both average and maximum - there is little to no difference between the bigger nasal valve and the original model, with one exception: the maximum pressure on the original model differs a lot from the other values. As with the rhinomanometry curve the smaller one's values deviate from the other ones but the average velocity is noticeably close to the others.

4.2.3 Flow at 250 ml/s

Here follows the results from a flow rate of 250 ml/s. The original model didn't converge at the flow rate of 250 ml/s.
As with 161 ml/s and the rhinomanometry curve we see in table 4.2 that the larger

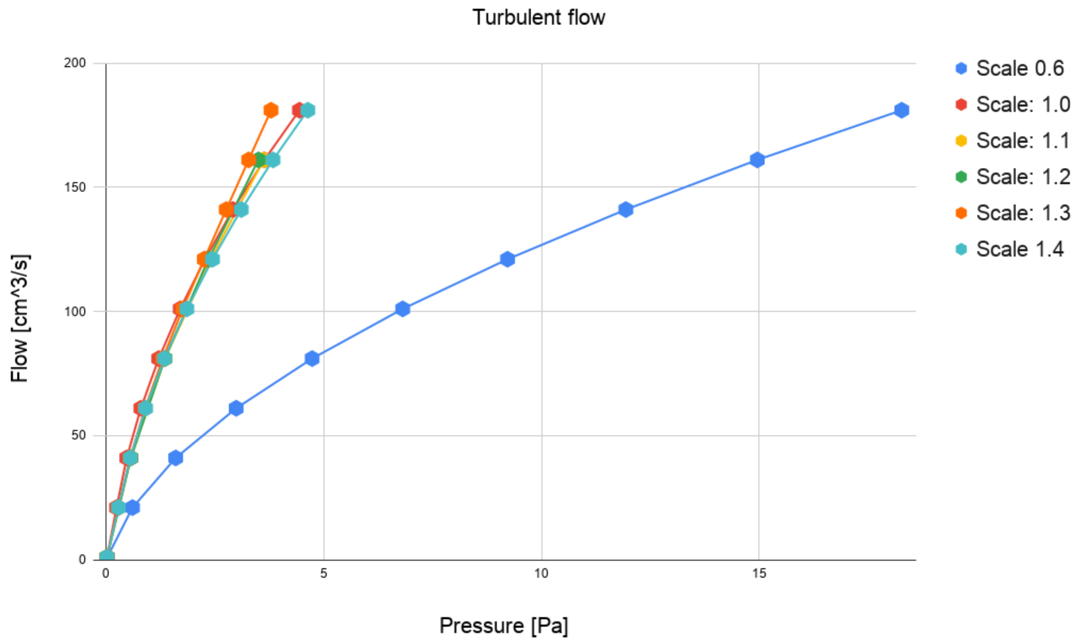


Figure 4.4: Rhinomanometry curves for different sizes of the nasal valve (the green area in figure 2.1). The scale is approximately how much the cross sectional area of the nasal valve have been altered. Data are taken at the inlet. All curves are similar except for scale 0.6.

once have over all roughly the same values. The smaller one differ mostly on the data of the pressure, both on average and maximum.

4.3 CFD simulation on CAD-models from CT-scans

This section presents rhinomanometry curves which were successfully recreated through CFD. It was possible to simulate airflow on all four different patients (called A, B, C and D). Rhinomanometry curves are also shown for before and after in silico surgical procedure on one patient, model A.

4.3.1 Creating CAD-models from CT-scans

The making of CAD files on all four patients was successful, but with varying degree of accuracy. As an example one nose (D) had an almost total blockage on one side. This nose was only simulated on the one side where it was possible to create a structure which could be converted in to a .stl-file.[3]

[3]The CAD-files can be accessed on the following google drive folder until 2025. `https://drive.google.com/open?id=10R7mA6cuDbQyGmo9fUlZnNzlRZzKV2FN`

Table 4.2: Different derived values on noses with different sizes on nasal valve. The input was flow at 250 ml/s, the simulation on original model (scale 1.0) didn't converge. More data can be found in appendix A.1, such as values of kappa (k), epsilon (ϵ), rotation rate and shear rate.

Flow: 250 ml/s					
Scale of nose	0.6	1.1	1.2	1.3	1.4
Average					
Velocity	4.2	3.7	3.6	3.4	3.8
Pressure	13.0	3.1	2.9	2.5	3.5
Maximum					
Velocity	10.1	9.1	9.1	9.1	9.4
Pressure	49.2	26.0	37.2	26.4	28.0

Table 4.3: Resistance of each nasal cavity for four models. Resistance is measured at 150 Pa.

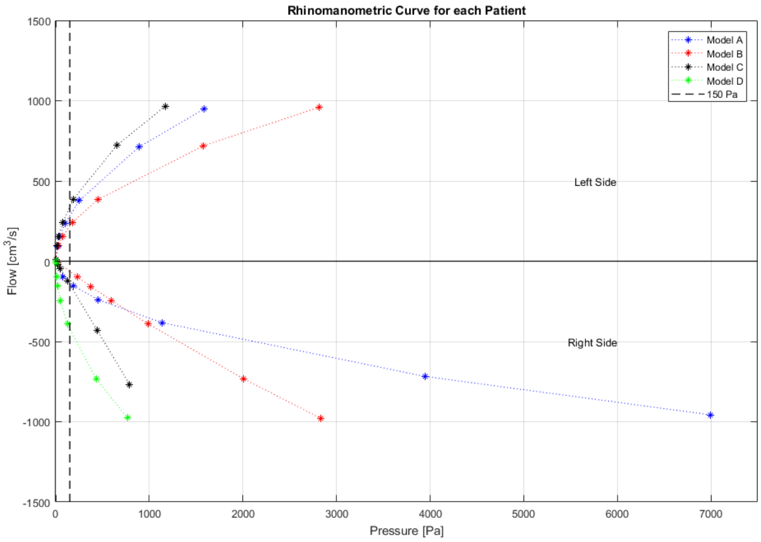
	Left	Right
Model A	0.5281	1.1237
Model B	0.6854	2.3450
Model C	0.4428	1.0289
Model D	-	0.3563

4.3.2 Rhinomanometry Curves on Four CT-scan

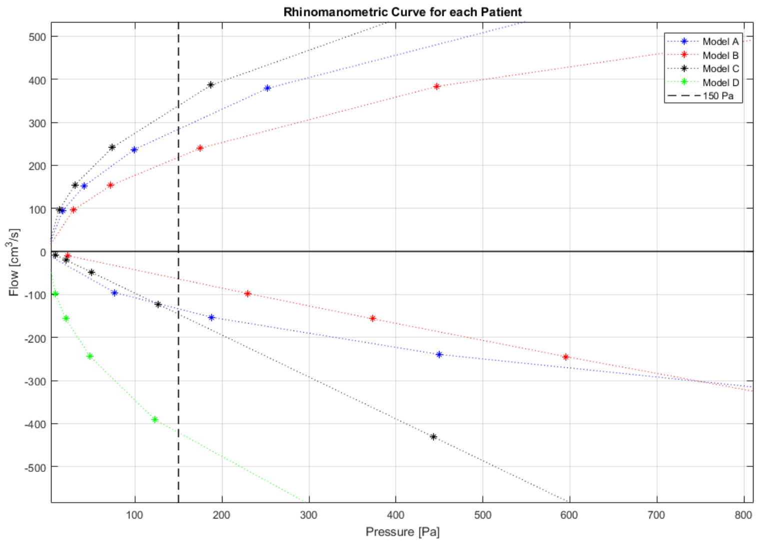
Figure 4.5a shows recreated rhinomanometry curves for each patient assuming laminar flow. Figure 4.5b shows the same curves in the region around the 150 Pa. The corresponding resistance at 150 Pa for each model is presented in table 4.3. We can see that in model B there is a much higher resistance in the right side. For model D, the left side is completely obstructed and as such no value can be reported, on contrary the flow on the right side is the one of the four models with the lowest resistance. Model C's right side is almost a perfect linear curve. If we compare to the simulations on the pipe we similar features in the laminar model. This would probably mean that the curve would look different in a turbulence model.

4.3.3 Rhinomanometry Curves of Simulated Procedure

The simulated rhinomanometry curves for the model with the simulated surgery can be found in figure 4.6. Where figure 4.6a is modelled with turbulent flow and figure 4.6b is modelled with laminar flow. Both simulations used a velocity boundary condition at the nostril and one nostril were simulated one at a time.
Figure 4.7 display all rhinomanometry curves but in a smaller region around the critical value of 150 Pa. The resistances of each case estimated at this point is given in table 4.4. Note that for the turbulent case of the operated left side, the pressure never reached 150 Pa.
It is interesting to see that for the laminar model the enlarged cavity reduces resistance while for the turbulent model it is the opposite. The turbulent model reports

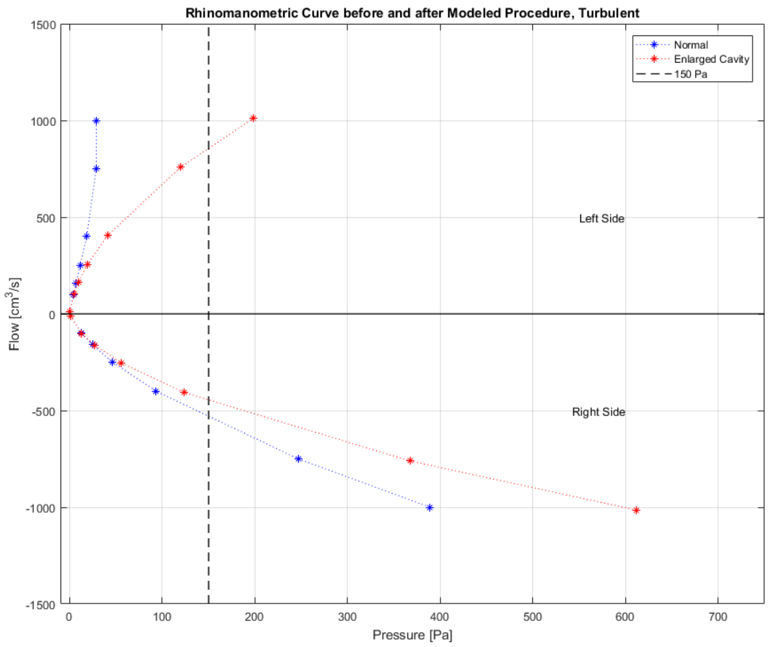


(a) Rhinomanometry curves for pressures up to 7000 Pa

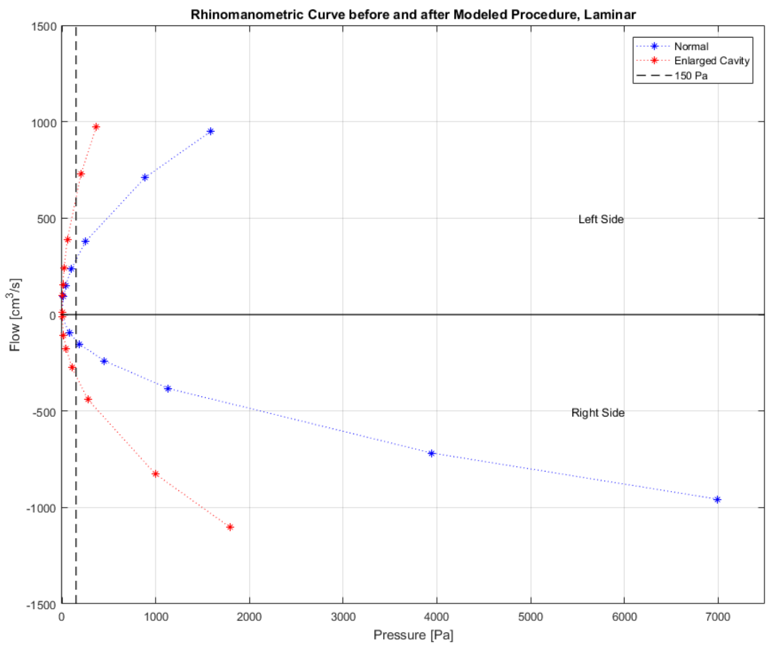


(b) Rhinomanometry curves centred around lower pressures, up to 800 Pa

Figure 4.5: Recreated rhinomanometry curves for different patients displaying the required pressure to induce a given flow using a laminar model. The two figures show the same data but with different scale on the x-axis. We see a large variation for different patients and for different sides of the nasal passage. The line at 150 Pa indicates the point at which resistance is measured



(a) Turbulent flow.



(b) Laminar flow.

Figure 4.6: Rhinomanometry Curves from a CFD simulation. The x-axis is to show all the data obtained by the simulation. Boundary condition was velocity (m/s) at one nostril at the time. The other nostril was set to open boundary. The line at 150 Pa indicates the point at which resistance is measured. By comparing the the two figures we see that the laminar model can simulate higher values of flow but the accuracy of these values are questionable.

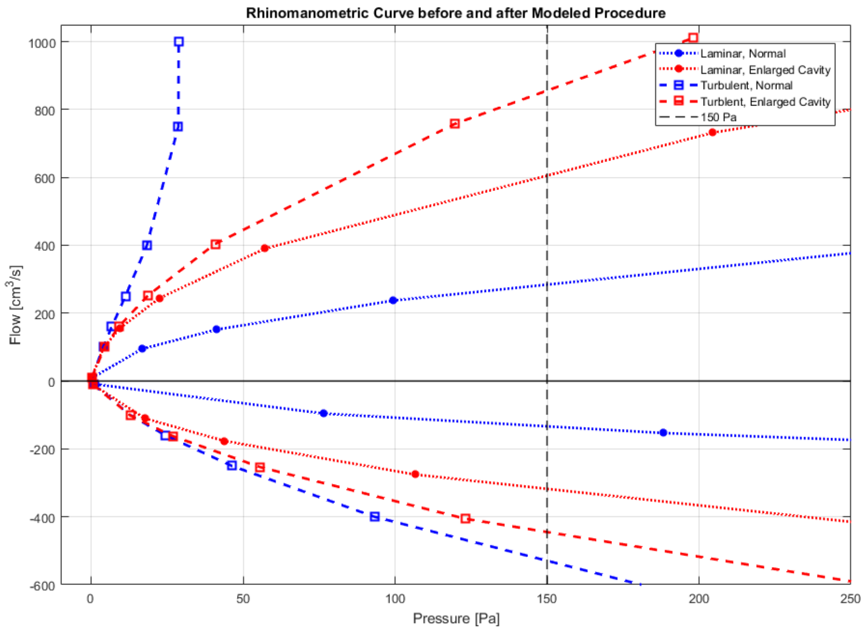


Figure 4.7: Rhinomanometry Curves from a CFD simulation. The x-axis span is reduced to better compare with the rhinomanometry curve. The boundary condition was a given velocity (m/s) at one nostril at the time. The other nostril was set as closed. The line at 150 Pa indicates the point at which resistance is measured.

Table 4.4: Resistance of each nasal cavity before and after operation with laminar and turbulent conditions. Resistance is measured at 150 Pa.

	Left	Right
Normal, Laminar	0.5281	1.1237
Enlarged Cavity, Laminar	0.2478	0.4723
Normal, Turbulent	-	0.2833
Enlarged Cavity, Turbulent	0.1753	0.3372

a lower resistance than the laminar model.

5
Discussion on CFD Simulations

Here follows the discussion on the results that the simulations gave. First the simulated rhinomanometry curves are discussed, then the properties of the simulations, then the process of create 3D models and last but not least future work.

5.1 Rhinomanometry Curves

Even though some rhinomanometry curves were able to be reproduced, these results needs to be studied more. The process is time consuming and more models and simulations should be studied to be able to draw conclusions with certainty. But with that said two things could be argued for through this study.
First of all it's possible to get a plot of pressure and flow that have similarities with rhinomanometry curves. Arguably it is feasible that a CFD simulation of the nasal passage could be more useful and provide more data to the doctor. A program that's specifically made for CFD simulations from CT-scans (or 3D models) on the nose would probably be more accurate and have a shorter computational time, this of course is no easy job and would require a lot of money and special competence to make possible.
Secondly it is worth noting that the pressure that's needed to induce flow is consistently lower in all the simulations than the rhinomanometry curves from patients, even in the best cases. This could be because of approximation done in the model and CFD simulation, for example hair, mucus and different humidity in the air where not simulated, also the walls/skin where approximated not to move. But the differences in values between the simulation and the measurements on patients could be explained by how these values are measured on patients. The setup used in the simulation might not exactly correspond to the real-life scenario and it would be interesting to further investigate this difference.
The one thing that differs the most in the simulated rhinomanometry curves is that it does not flatten as much as the curves from patient data. This is maybe because there is no upper limit on the simulation boundary condition (as long as it doesn't diverge) but a human is only capable of muster a certain amount of muscle power and thereby certain amount of pressure in the lungs.

5.1.1 Pipe Model

The curves of flow and pressure on the pipe model is very different from the rhinomanometry curves. This is reasonable and the purpose of the pipe model was not to obtain curves that were similar to the rhinomanometry curves. With this in mind we see that when using turbulent flow the curve does flatten out which is similar to the rhinomanometry curve. Even though the resistance is not calculated we see that it's substantially lower then for every nose model. This indicate that it's the complex geometry in the nose that causes the resistance.

5.2 Alteration of the Nasal Valve

Increasing the size of the nasal valve does not seem to change the rhinomanometry curve indicating that the original nasal valve is not the cause of the resistance in this model. The largest difference in the rhinomanometry curves for the different sizes is seen for the smaller valve. This difference could mean that this area is important for nasal airflow and the overall resistance in the nose. We see that the values for the pressure is much lower than in the full model of the nose. This can be explained by the fact that the air only travels through a small part of the nose.

5.3 Surgical Procedure on the Conchae

The operation on the 3D models was successful in that it was possible to alter the geometry and perform CFD simulations. We get inconsistent results for the effect of operation when using a laminar and a turbulent model. In the laminar model the resistance is decreased after operation, as expected, and in the turbulent model the resistance is increased. Perhaps the turbulent model correctly simulates what is called *empty nose syndrome* which is described by Kuan et al. [19] where the patient report increased obstruction after turbinate surgery. Why this is only indicated in the turbulent model could be explained by how a larger cavity could increase the tendency for turbulent flow which could be the cause of increased resistance. However, we also see that the left side of the nasal passage in the turbulent model for before operation is approaching a resistance of zero. This does not seem reasonable, and one could question the validity of the results. Further comparisons between different turbulence models could be done to explain these results.

5.4 Conditions for Simulation

The boundary conditions where one of the main sources of problems in the project. A lot of hours where spent waiting for simulations to be done, only for the simulation to diverge on the last hour.

5.4.1 Laminar or Turbulent

As discussed in the theory, for low velocities and in turn low flow rates, laminar flow is an accurate approximation. For instance, considering the rhinomanometry curves for the models of a surgical operation, figure 4.7, we see that the geometry with an enlarged cavity generate similar curves for both the turbulent and laminar

model for low flow rates, below 200 ml/s. This agrees with the theoretical values of laminar flow being predominant up to around 200 ml/s. However, this is not seen in the unedited nose, raising the question why.
In the world of medicine, accuracy is very important. Except for the long simulation time and the risk of divergence the turbulent model would be preferred over the laminar, as it should be more accurate. As seen in many of the figures the laminar flow and turbulent flow yields different values for higher velocities which means that a laminar flow model is not applicable in the turbulent flow region[1].
As some other CFD studies, for example Leong et al. [21] conclude, a major limitation in using CFD and simulating turbulent flow in the nose is the required computational power. With a personal computer the computational time for one rhinomanometry curve could take days to simulate even with low accuracy and few data points. In our case using a laminar model can on the other hand take about 10-20 minutes on average. Due to the time taken for turbulent simulations some laminar simulations were done and included in the report, also because they are applicable for low velocities.

5.4.2 Choice of Boundary Condition

As stated previously pressure, velocity and flow as boundary conditions have only minor differences. The study on the pipe model showed a substantial difference in outcome with these conditions. It shows how sensitive the results are to different assumptions in fluid dynamic problems.
It's difficult to say to say which one of the boundary conditions that is most realistic but it is reasonable to argue that it is either velocity or flow as demonstrated by figure 4.2. The pressure starts to diverge at much lower values then flow and velocity indicated by the curve changing direction between 0.5 Pa and 0.6 Pa, and the fact that the simulation diverge at 0.75 Pa.
There were inconsistencies in which boundary conditions that where able to converge, for the most part flow (ml/s) where the most reliable but there where times where flow didn't work and other boundary conditions where able to converge.

5.5 3D Models from CT-scans

The process of generating a 3D mesh from a CT-scan is difficult to automate. Due to a lot of areas in the nose having the same density (air) the automated algorithms considered all these regions as areas of interest, hence the need for manual selection. Manual work has both benefits and disadvantages. On the positive side it eliminated many problems with automated computer process, such as certain traits in the CT-scans could be spotted and corrected. For example sometimes mucus could look as skin and bone which in the automated process would yield a 3D model that would have been inaccurate. This can of course be found and corrected by a human. On the other hand the automated process would be free from human error and more

[1]It is very likely that the turbulent flow model that's the one that's more realistic if the two models yields different results.

consistent in its results.

5.6 Future Work

There are a number of steps involved going from patient to a the presentation of CFD results. In order for this method to be a viable tool for doctors, these steps need to both be practical and reliable. This project has explored some of the considerations involved in use of CFD, but there are several more areas to be studied.

5.6.1 Automation of Diagnosis

As discussed in the theoretical background there have been several attempts at automating the process of building a 3D model of the nasal cavity. Expanding on these methods one can imagine the whole process, from CT-scan to a rhinomanometry curve, to be fully automated. This would involve the development of specialised software, combining algorithms from different existing programs. This would be of great practical use given that it works for a wide range of patient data.

5.6.2 Recreating other Rhinomanometry Methods

This project has focused on recreating the curves produced by anterior rhinomanometry as this is the most common form. However, the limitations of other methods could also be explored. Recreating rhinoresistometry would simply involve further manipulation of the data and could perhaps provide further insight. Moreover could CFD be the basis of data analysis not possible in a real-life scenario. Based on data for the velocity and pressure of the whole volume one could extract values for particular regions of interest. This approach could perhaps give answers to why rhinomanometry does not always correlate with patients' sensation of obstruction, as discussed in the theoretical background, and provide alternative metrics.

5.6.3 Simulating Surgical Operation

This project made a small test of simulating a surgical procedure and estimating the outcome in the form of a rhinomanometry curve. This idea could be further expanded upon by creating models of other surgical operations as well as comparing to real-life situations. One could imagine a specialised software in which surgery and treatments easily could be replicated, comparable to CAD software, and then relevant simulations could be performed.

6

Theoretical Background for Lesson Test

Classrooms demonstrations have been studied before, one thing these have in common in the conclusion that only watching the theory that the teacher have presented is not an efficient tool for enhance understanding [4] though demonstration is a good tool improve students engagement.

6.1 5E Teaching Model

When teaching a subject, a lot of thought is often put into the order of the different topics within that subject. Furthermore, one often tries to vary the types of activities the teaching session consists of. For instance, a class could start with a lecture and followed by students solving problems, or a discussion. Tanner [28] makes the case that not only is the order of topics important for learning, but also the order of the types of activities. She explores Bybee's model that identifies five key elements of a successful lesson and proposes that the order of these five elements is critical for how effective the teaching is. The five elements is abbreviated *5E* and consists of:

- Engage
- Explore
- Explain
- Elaborate
- Evaluate

The first step of this structure, *engage*, consists of activities that will increase the students interest in the topic and engage them with the material. Tanner [28] points to research that shows that engaged students are more likely to learn in a given situation and argues therefore that this should be the first step of a lesson. The goal of the second step, *explore*, is to get students to ask questions which will prepare them to receive more information about the topic which is presented in the third step, *explain*. The fourth step *elaborate* provides the students an opportunity to apply their newly gained knowledge in new situations. The final step *evaluate*, where the student's knowledge is evaluated, is a way both for the instructor and the student to gain insight into and reflect upon what they have learnt. While the order of these steps is important according to Tanner [28] she also states that they don't necessarily have to fit into a single lesson constrained by time. Instead this structure could be applied to a series of lessons, or some steps could be performed outside of class.

In a learning scenario consisting of a laboratory activity, the 5E model can provide a guide to structuring the lesson in order to increase effectiveness. The actual experiment can be seen as an *explore*-activity in which the students are encouraged to try to predict outcomes and see potential relationship between the components involved. This is true whether the activity is led by the teacher or by the students, as long as the students are actively participating in understanding the situation at hand. With the laboratory activity as the second step of a lesson, it is important that it is preceded by some sort of engagement in which the students understand why the topic is interesting and important to them. The experiment should also be followed by explanation of what was observed and clarify any misunderstanding. Finally the lesson should finish with an opportunity for elaboration and evaluation.

6.2 Experiments as a Didactical Tool

As this study aims to explore the variations in online teaching with an experiment as the main learning activity, it is interesting to consider similar variations in an ordinary classroom. Crouch et al. [4] performed a study comparing different methods in using a scientific experiment in a classroom. They found that having students observe an experiment yields no better understanding than simply explaining the phenomena without the demonstration. However, when the students were engaged in the activity by being asked to predict the outcome as well as discussing the observations, an increase in learning was seen. This indicates that the student involvement is the key factor in learning and not the fact that there is a physical demonstration present. This idea is further supported by McComas [23] who suggests that involving the students further in the design of experiments is beneficial, as this engages the student and makes them ask critical questions about the topic being studied.

6.3 Demonstrations in Fluid Dynamics

A commonly used model within teaching is Kolb's experiential learning theory [18] which is based on the idea that learning happens when experience is transformed into knowledge. The theory splits up different ways of doing this into categories, or learning styles, and presents the sort of activities that caters to the different types of learning. What is common between these learning styles is that the learner is engaged with the topic at hand, whether that be in the form of active experimentation or reflective observation. As such Garrison and Garrison [10] argues for the importance of ways to engage the learner and create an interest in the topic. They further argue that in a topic such as fluid dynamics which can be quite abstract, one way of increasing interest and engagement is through the use of classroom demonstrations. Therefore they created a collection of demonstration and experiments involving fluid dynamics, designed to clarify abstract concepts.

6.4 Online Teaching

Online teaching is an area that has become increasingly relevant with the continuous improvement and use of digital communication. While many models and ideas concerning how to create effective learning still apply in a digital environment, there are some additional challenges as the nature of communication is changed. Steele et al. [26] argues that there is a lack of a single best strategy for teaching online and that there is a strong need for one. It is therefore an interesting topic to explore and to find out what are some critical aspects that needs to be considered.
Steele et al. [26] discusses the social role of a teacher in a classroom and how that aspect changes in an online course. They argue that the teacher must establish a strong and clear presence in a digital classroom as well as build social relationship with the students, something that might be more difficult without physical presence. They further discuss how this aspect can differ in different scenarios and is dependent on factors such as the age of the student. Older, more experienced students will have more confidence and not need the same guidance and presence of a teacher as younger students.
Another interesting aspect of online teaching that differs from traditional teaching is that the effective learning can be highly dependent on the students technical abilities. In a study by Wei and Chou [29] they found that students' internet self-efficacy, i.e. how comfortable they are using internet and technology, had a major impact on their learning, as well as their perception of online teaching. This can mean that there can be a large variation in how much individuals learn based on their experience with computers and technology. This idea is further supported by Abdous [1] who found that an introductory *online learning orientation* can be helpful in increasing students self-efficacy and their satisfaction with online teaching.
Finally one particular form of teaching that changes when switching to a digital environment is group discussions. Woods and Bliss [31] discusses how the nature of a discussion is different in a physical setting and in an online setting. They found that in a digital environment and in what they call an asynchronous discussion, all students get the opportunity to reflect upon the question and answer, while it is common in a physical setting that extroverted students dominate the conversation. Further, they discuss how in very large groups which can be common in online courses, are difficult to facilitate and create a valuable discussion. They argue that an online discussion can be successfully facilitated by designing well-thought-out questions that are relevant to the students and stimulates higher level of thinking. Furthermore they emphasise the effect of class size and how this is an important factor to be considered when planning online discussions.

7
Methods in Lesson Test

The content of the learning session was chosen based on the aspects of fluid dynamics that are relevant for the human body in general, and this project in particular. As such the central topic is *frictional head-loss in a pipe*[1] and *Reynolds equation, see section 2.3.1* including the difference between laminar and turbulent flow. These were demonstrated using two simple experiments involving airflow and water-flow through a drinking straw. Details of the demonstration is presented by an instruction document in appendix B.1.

7.1 Lesson-plan

When teaching in a digital environment there are many different tools and methods available. In order to study the effect of variation in teaching, only the format in which the experiment was presented was varied between the groups. These variations differed mainly in the degree of interaction and activity between teacher and student, while the content remained the same. The three different types of activities were:

- Group 1. A live demonstration similar to how a teacher would perform an experiment in a ordinary classroom.
- Group 2. A recorded video of the demonstration.
- Group 3. A home-experiment where the students performed the experiment by themselves at home.

The demonstration consisted of two parts, the first about airflow and frictional head-loss, the second about laminar and turbulent flow in water. The first part consisted of blowing through a drinking straw and feeling the resistance in airflow. The speed of the air through the straw as well as the length and diameter of the straw was then varied to observe any difference in the feeling of resistance. The second part consisted of pouring water through drinking straws of different diameters and lengths while also varying the speed of the incoming flow. The character of the outgoing flow was then observed to be either clear and smooth, indicating laminar flow, or noisy, indicating turbulent flow.

Each of the three sessions was planned to take an hour to complete and were performed during the same day. The remaining part of the lesson was kept as similar as possible and designed according to the 5E model. The overall lesson-plan for all three variants is presented in table 7.1 and the slides used for the lesson can be found in appendix B.2

[1]This equation and it's properties can be read about on the Wikipedia page `https://en.wikipedia.org/wiki/Friction_loss`

Table 7.1: The rough features of lesson plan. The lesson plan is based on the concept of 5E.

Title	Content
Introduction	Presentation of the study, what it will involve and the general context of the project.
Background	Introduction to fluid dynamics and why it is important for medical students.
Experiment	The demonstration, varied between the groups.
Theory	Presentation of the equations for frictional head-loss and the Reynolds number. Emphasis on which parameters are involved.
Questions and discussions	The students answer some open-ended questions about the content and its application in the medical field. These answers are then discusses together.
Test	The students take the test for data collection purposes.

Table 7.2: Number of completed test in each group. Note that there could have been more students on the lecture that choose not to answer the survey.

Group	Number of completed tests
1	5
2	10
3a	5
3b	5

7.2 Participants in the Lesson

The participants were all medical students at Sahlgrenska University Hospital in a course taken toward the end of their education. The lesson was given as an optional part of the course. This group was chosen as it is part of the audience which this study targets. Furthermore, this is a relatively homogeneous group, coming from similar backgrounds in terms of education and previous knowledge, providing opportunity for comparison of different teaching methods.

Because of the use of digital communication it was difficult to get an exact number of participants in each group as people could join the meeting late or leave early. This also meant that not all who participated answered the test in the end of the session. Furthermore, in the last group which were to perform the experiment themselves, not all participants had the necessary material. As such, these individuals got to watch the pre-recorded demonstration instead. This means that group 3 can be split into group 3a, consisting of those who actually performed the experiment, and group 3b, consisting of those who watched the video instead. The number of test results collected from each group is presented in table 7.2.

7.3 Testing

After each session data was collected in the form of a short test consisting of 17 questions in an online survey which the students were given around 10 minutes to complete but with the option to continue after the session had ended. This method of testing the students' learning was chosen because of its simplicity. Alternative methods using additional testing before the session as well as after several weeks would probably provide better understanding of how much the students have learned, but as the main aim was to compare the different variants, this was deemed unnecessary.

The questions were chosen based primarily on what was deemed to be the central aspects of the content in the lesson as well as to adhere to a suitable level of learning outcomes as according to the six levels of Bloom's taxonomy [6]. In this case the questions mainly target the lower levels, namely remembering, understanding and applying. These levels were deemed sufficient because the overall goal for the lesson was to give the participants a basic intuition for some of the properties of fluid dynamics.

For instance questions concerning the two formulas (Reynolds equation and head-loss equation), were constructed as to test the student's understanding of the properties involved as opposed to calculation based questions which would instead test the students ability to perform procedures. One question asked about the properties of laminar flow is an example of remembering facts and basic concepts, while another one concerned with mixing fluid is an example of applying knowledge to a new situation. Overall this test constitutes a range of questions testing knowledge on different levels on the topic of frictional head-loss and laminar and turbulent flow. All questions can be found in appendix B.3.

There were also three questions to determine if the students liked the lesson and if they thought the lesson was on a reasonable difficulty level. Finally a question was asked what the participants thought a future collaboration between doctors and engineers could look like.

7.4 Analysing result

For each student who answered the test a score was given between 0 and 17 equal to the number of correct answers. The test results were then divided into sets corresponding to each group. Group 2 and 3b were combined as they experienced the same variation in the demonstration.

An average value $\hat{\mu}_i$ was calculated for each group as well as a total average, $\hat{\mu}$. This was then used to perform hypothesis testing using the F-statistic. The null hypothesis was stated to be that the variation in the demonstration has no effect on learning which translates to the true mean u_i being equal for each group. Significance was considered to be observed for a p-value less than 5%. The sum of squares between the groups, SSB, was calculated with 2 degrees of freedom and the sum of squares within the groups, SSW, was calculated with 17 degrees of freedom. This means that the F-statistic is given by

$$F_{stat} = \frac{17 \cdot \mathrm{SSB}}{2 \cdot \mathrm{SSW}}$$

where $F_{stat} \sim \mathrm{F}(2, 17)$ from which p can be calculated.

7.5 Observations

In addition to the quantitative data collection in the form of test results, some observations were made concerning the preparation and execution of each type of demonstration. An open data collection where anything of interest was noted during both the preparation and the actual lessons. This method was chosen because the interest was to get an insight into the broad aspects of planning and performing demonstrations online. These aspects were not necessarily known beforehand and as such a protocol could not be developed that would necessarily capture everything of interest.

8

Results from Lesson Test

The results from the learning study are the test scores of all participants in combination with the observations made performing the lessons. These are presented in this chapter in combination with the statistical analysis of the test scores.

8.1 Test Results

Table 8.1 displays the number of correct answers to each question from each group. The table also shows the proportion of correct answers. We can see that group 1 (live demonstration) has the lowest proportion at around 54 % correct answers, while group 3a (home experiment) has the highest proportion at almost 70 % correct answers. For question 9 concerning the impact of length of straw on the Reynolds number, no participant in group 1 answered correctly. In contrast almost all participants in the study answered question 10 correctly which was about where one can find turbulent flow within the human heart.

8.2 Statistical Analysis

It is interesting to consider whether the results are due to variations within each group or because of variation between the groups, which would indicate that the choice of teaching method has an effect on learning. Figure 8.1 displays a histogram of the test scores for each participant, divided into the three different types of demonstrations. This indicates quite a large variation both within the groups and between the groups. The F-statistic was calculated to be 1.477, which corresponds to $p = 0.218 \approx 20\%$ (which is obviously greater than 5 %). As such, we fail to reject the null hypothesis, and this data is not sufficient evidence that the choice of teaching method in digital environment impacts learning.

8.3 Observations

Because this investigation involves human subjects it is interesting to consider qualitative aspects of the result as well. In the following sections observations from the preparation and the teaching each of the groups is described.

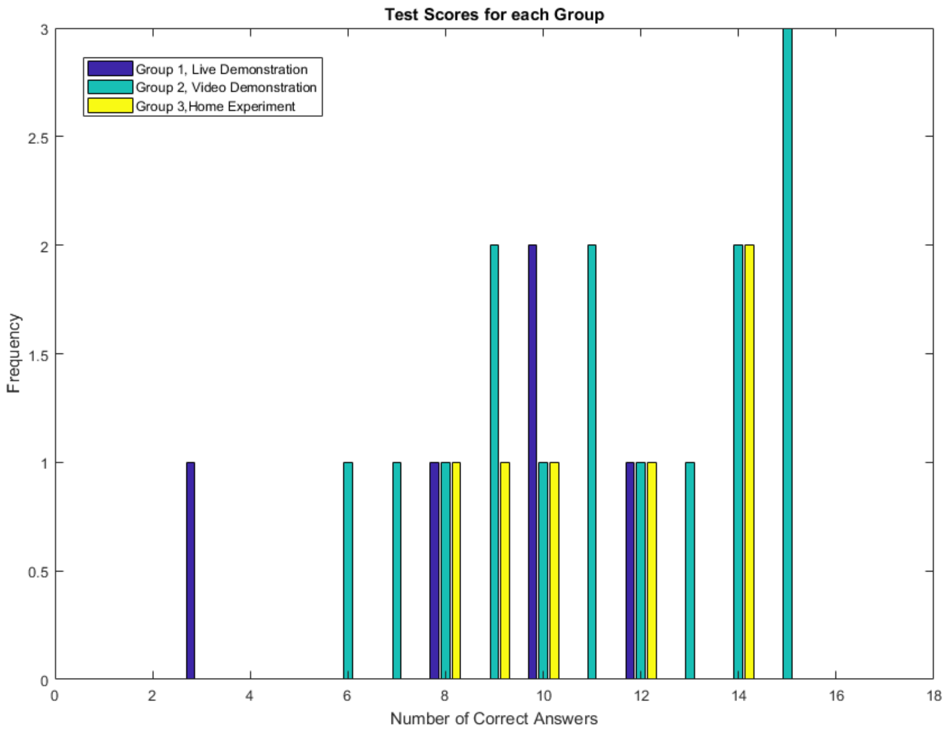


Figure 8.1: Histogram of test scores for each of the three groups. While the first group seem to score slightly lower, there is no clear distinction between the three groups.

Table 8.1: Number of correct answers to each question from each group.

Question	Group 1	Group 2	Group 3a	Group 3b
1	5	7	5	3
2	4	7	2	3
3	4	7	3	5
4	3	6	3	2
5	3	3	1	2
6	3	8	4	4
7	1	7	4	3
8	3	9	4	4
9	0	5	3	3
10	5	10	5	4
11	4	8	4	4
12	1	7	4	2
13	2	6	4	3
14	2	5	4	3
15	2	8	3	4
16	0	6	2	4
17	4	7	4	3
Participants	5	10	5	5
Proportion	0.54	0.68	0.69	0.66

8.3.1 Preparations

When preparing the lessons many aspects were shared between the three variations. For instance, a document with instructions were made for the group performing the experiment at home, but this was also used for recording and performing the demonstration. As such the time taken to prepare these instructions can be considered shared between all variations. One aspect that isn't shared is the recording and editing of the video made. This took about half a day and means that using a pre-recorded video takes the most preparation time.

8.3.2 During the Lessons

The first group experienced some technical difficulties with setting up the online meeting. This resulted in a 10-15 minute late start making the rest of the presentation slightly stressed. One example is that in the live demonstration the effect of the length of the pipe on the turbulent flow was not mentioned. This loss of time was compensated for by letting the students take the test after presentation was over and the time was up.
Those who took the test in the first group answered all questions within 10 minutes after the given time. This can be compared to the second group who got time for the test within the given hour and where all but one finished the test in 10 minutes as planned. The participant who exceeded the estimated time handed in 40 minutes after the test was given. In the last group all participants completed the test in 10 minutes.

The sink used for the live demonstration was smaller than the one used in the pre-recorded demonstration meaning it was more difficult to show the audience what was going on.
During the live demonstration questions were asked to the audience to predict the outcome of the experiments, but they did not answer. The questions were asked to the whole group and in such a way that did not force anyone to answer.

8.4 Engagement

20 out of the total 25 students answered that they experienced the lecture as both "useful as a doctor" and "fun and interesting". 16 out of 25 thought that the lesson was on an adequate difficulty level with the rest of the answers spread even amongst "too easy" and students who didn't wish to answer. As with the other questions there is no statistical difference between the three groups, but noticeably most of the students thought it was an interesting lesson on a good difficulty level.
For the final question concerning future work with engineers all answers are presented in appendix B.4. Among the answers we generally see high expectations for the future and that collaboration will increase. The examples given can be loosely split into two categories. The first is that engineers will collaborate with doctors in day to day work, such as in radiology and with the technical machines already in use today. The other category is that engineers will work in collaboration with doctors to develop new medical techniques and treatments. The answers also mentions that the interactions with engineers will be highly dependent on which field one works in.

9

Discussion on Lesson Test

The test scores indicate a relatively high rate of learning and the participants found the session interesting, which can be considered a success. It is difficult to distinguish any differences between the three groups, which raises the question why that is. There are also other aspects of this study that are interesting and worthy of discussion, such as the practicality of the different formats and student engagement.

9.1 Number of Participants

The main reason why the results cannot be considered strong evidence that one format of teaching is more effective than another is probably the small sample size. Increasing the number of participants would decrease the internal variance within each group, making it easier to distinguish the cause of variance within the whole group. We can consider group sizes of 5-10 individuals too small for this type of study and perhaps a sample size of around 20-30 individuals in a future study will provide better answers. This size is not unreasonably considering that it is not uncommon that classes consist of 30 or more individuals.

9.2 Design of Experiment

There are some aspects of the quantitative results that stand out and are worthy of discussion. Question 10 concerns turbulence in the heart which almost all participants answered correctly. This could be because the topic had been taught in another part of the education and it could therefore be argued that it is not relevant for the results of this study. Another explanation could be how the question is formulated.

> *If we consider the overall structures of the heart, can we find both laminar and turbulent flow there or just one of them? Where?*

The question is slightly open-ended and there are several possible correct answers as the student is asked to give examples. Allowing several variants to be marked as correct could mean that it was a too easy question. In order to attribute this result to the effect of the lesson, a pre-test could be used to determine what the students knew beforehand. The question could be developed by asking where they obtained the knowledge and to motivate their answer.

Question 9 concerns the effect of length on the Reynolds number, to which none in the first group (live demonstration), answered correctly. This is likely because this aspect was not mentioned during the demonstration due to limited time. This

result could be a large part of why this group seems to perform worse than the other groups.
One student in the first group did not answer any of the questions 3 through 9 and 13 through 17. This test score can clearly be seen as an outlier in figure 8.1 and could be an additional explanation to why this group performed worse than the other two. If we were to remove this outlier it would be even more difficult to see any significant difference between the three groups. Once again a larger number of participants would be beneficial as such outliers would not have such a significant effect on the results.

9.3 Student Engagement

As discussed in the theoretical background, the role of a demonstration or experiment is primarily to engage the learner by creating interest as well as making the learner ask critical questions about the topic. Based on this it is interesting to consider how the different formats of demonstration succeeds in achieving this goal. One might think that the group performing the experiment themselves will experience the highest level of engagement. But the fact that the students are physically involved does not necessarily mean that they are mentally engaged. This is suggested by Crouch et al. [4] as well as McComas [23] who both argued for the importance of engaging students in the discussion and questions revolving around the experiment. The fact that this group of learners performed the experiment by themselves at home instead hindered the interaction between student and teacher as the teacher could not monitor or guide the student in what they were doing. This however a problem that could be solved by for example having the students film their experiments and maintaining communication throughout the lesson.
This can be compared to a situation where the instructor is performing the experiment. In such a case, the level of student interaction can vary with the way the demonstrations is structured. For instance, as Crouch et al. [4] showed, simply letting the students predict the outcome of the experiment can have substantial effects. This type of student engagement might be easier to achieve in a live demonstration, as compared to a pre-recorded one, because of the ability to react to student interactions. However, one can imagine that a video demonstration could be used in conjunction with a teacher-led discussion, in which the teacher pauses the video at specific times to let the students reflect upon what they see.

9.4 Practicality of Different Formats

While the test scores fail to indicate which form of demonstration has the biggest effect on learning in a digital environment there are other aspects one might consider, such as the practicality of the format. This includes the required effort and time needed in preparation, but also the ease of use during an online seminar.
As discussed in the results, all three formats have much of the preparation in common, such as planning and preparing instructions. The one variation that needed more time was the pre-recorded video. The video can however be reused or one

could find relevant material online. During the actual lesson the video was convenient compared to the other formats and gives the instructor the ability to focus on leading a group discussion. A pre-recorded video will also give the opportunity for students missing the class to take part of the material afterwards. One must however remember the importance of the teacher's presence in the online classroom, as discussed by Steele et al. [26], and the social relationships with students. Woods and Bliss [31] also emphasise how material made by others can feel less relevant to students hindering group discussions and individual reflection. Therefore pre-made videos might seem very promising in their usefulness, but might not be as effective if they replace the teacher-led lessons.

Letting the students perform the experiment at home meant little interaction by the teacher. While this might seem practical, it also means it is difficult to know what the students have done and how successful they were. Furthermore there is the issue of required materials for an experiment. Despite providing information well in advance to the students that would be needing material, only a few were actually prepared. This creates the issue of how to handle such a situation as well as the inequality between those who can easily perform various experiments and those who cannot. One benefit of a traditional classroom situation is the equal access to educational material.

There could be ways to combat the problems in home-experiments. Perhaps a solution is to work in small groups were the instructor can communicate with individual students through video and voice chat during the session. This would be similar to how a teacher can walk around a classroom and have individual conversations with students, however, the digital setting would be an extra barrier of communication. To deal with the problem of providing equal conditions for all participants one could either choose and design experiments that require little to no material, or find a way to distribute the required material. In either case, it is a limitation of experiments in digital teaching as compared to in a traditional environment.

10
Conclusion

There is a broad range of aspects to consider in the application of computational fluid dynamics on nasal airflow. This approach has may uses in both diagnosis and treatment of nasal dysfunctions and there is a lot of potential in the collaboration between the medical and engineering field.
Rhinomanometry curves can be reproduced by imitating the the setup using CFD. Some differences can be seen from real-life examples, for instance how the simulated version can take on extreme values for high flow rates. These differences can be further investigated, and the method could be further developed to provide more reliable results.
The simulated surgical procedure of the conchae and the edited size of the nasal valve indicates how this technology can be used for prediction of different treatments.
This project has found that there the many choices involved in the development of a CFD-model which might not be obvious and have a substantial effect on the results. These choices include whether to consider the flow turbulent or laminar, which boundary conditions to use, and how the geometric mesh is generated. Any recommendation on which choices to make can not be concluded.
Finally the study on teaching fluid dynamics in a digital environment could be further developed using a larger sample size as it failed to provide evidence on which format of demonstration causes the most effective learning. Nevertheless, based on the observations from the lesson, traditional ideas about teaching probably still apply in a digital environment. There are several practical aspects to consider and challenges to overcome. While letting students perform experiments themselves might be an effective way of teaching in a traditional classroom situation, this might not be the case in online teaching due to difficulties in communication and monitoring progress digitally.

Bibliography

[1] M'hammed Abdous. Well Begun is Half Done: Using Online Orientation to Foster Online Students' Academic Self-Efficacy. *Online Learning*, 23(3):161–187, 2019.

[2] Michael J. Bradshaw, Shamik Bhattacharyya, Nagagopal Venna, and Jonathan F. Cahill. Neurologic Manifestations of Systemic Rheumatologic Diseases. In Syed A. Rizvi, , Jonathan F. Cahill, , and Patricia K Coyle, editors, *Current Clinical Neurology*, pages 321–342. Springer International Publishing, 2020.

[3] P. A.R. Clement and Frans Gordts. Consensus report on acoustic rhinometry and rhinomanometry. *Rhinology*, 43(3):169–179, 2005.

[4] Catherine Crouch, Adam P. Fagen, J. Paul Callan, and Eric Mazur. Classroom demonstrations: Learning tools or entertainment? *American Journal of Physics*, 72(6):835–838, 2004.

[5] Duygu Demirbas, Cemal Cingi, Hamdi Çakli, and Ercan Kaya. Use of rhinomanometry in common rhinologic disorders. *Expert Review of Medical Devices*, 8(6):769–777, 2011.

[6] Patricia A. Eber and Trent S. Parker. Assessing student learning: Applying bloom's taxonomy. In *Assessing Student Learning: Applying Bloom's Taxonomy*, 2007.

[7] Aaron Esmaili and Aanand Acharya. Clinical assessment, diagnosis and management of nasal obstruction. *Australian Family Physician*, 46(7):499–503, 2017.

[8] Walter Frei. Which turbulence model should i choose for my cfd application? `https://www.comsol.com/blogs/which-turbulence-model-should-choose-cfd-application/`, 2017. [Online; accessed 26-May-2020].

[9] Eduardo Gaio and César Melo. A pattern to evaluate airway resistive phenomenon using Rohrer's equation. *Advances in physiology education*, 31(1):121, 2007.

[10] Laura A. Garrison and Timothy J. Garrison. A demo every day: Bringing fluid mechanics to life. *ASEE Annual Conference and Exposition, Conference Proceedings*, 122nd ASEE(122nd ASEE Annual Conference and Exposition: Making Value for Society), 2015.

[11] Johan Hellgren. Private Communication, 2020.

[12] Johan Hoffman and Claes Johnson. *Computational Turbulence*, pages 57–63. Springer Berlin Heidelberg, Berlin, Heidelberg, 2007.

[13] Johan Hoffman and Claes Johnson. *Laminar and Turbulent Flow*, pages 51–55. Springer Berlin Heidelberg, Berlin, Heidelberg, 2007.

[14] Robin Huang, Anthony Nedanoski, David F. Fletcher, Narinder Singh, Jerome Schmid, Paul M. Young, Nicholas Stow, Lei Bi, Daniela Traini, Eugene Wong, Craig L. Phillips, Ronald R. Grunstein, and Jinman Kim. An automated segmentation framework for nasal computational fluid dynamics analysis in computed tomography. *Computers in Biology and Medicine*, 115(October):103505, 2019.

[15] James A. Wise Eddie Johnson Brandon Poe Dean H. Kruse Oksana Korol Jody E. Johnson Mark Womble Peter DeSaix J. Gordon Betts, Kelly A. Young. *Anatomy and Physiology*. OpenStax, Houston, Texas, 2013.

[16] A. S. Jones, D. J. Willatt, and L. M. Durham. Nasal airflow: Resistance and sensation. *The Journal of Laryngology & Otology*, 103(10):909–911, 1989.

[17] William Keustermans, Toon Huysmans, Bert Schmelzer, Jan Sijbers, and Joris JJ Dirckx. The effect of nasal shape on the thermal conditioning of inhaled air: Using clinical tomographic data to build a large-scale statistical shape model. *Computers in Biology and Medicine*, 117(January):103600, 2020.

[18] David Kolb, Richard Boyatzis, and Charalampos Mainemelis. *Experiential Learning Theory: Previous Research and New Directions, in in Perspectives on Thinking, Learning and Cognitive Styles*, chapter 9, pages 227–252. Case Western Reserve University, 01 2001.

[19] Edward C. Kuan, Jeffrey D. Suh, and Marilene B. Wang. Empty Nose Syndrome. *Current Allergy and Asthma Reports*, 15(1):1–5, 2015.

[20] Haribalan Kumar, Ravi Jain, Richard G. Douglas, and Merryn H. Tawhai. Airflow in the human nasal passage and sinuses of chronic rhinosinusitis subjects. *PLoS ONE*, 11(6):1–14, 2016.

[21] S.C. Leong, X.B Chen, H.P Lee, and D.Y Wang. A review of the implications of computational fluid dynamic studies on nasal airflow and physiology. *Rhinology Journal*, 48(2), 6 2010.

[22] Y. Liu, M. R. Johnson, E. A. Matida, S. Kherani, and J. Marsan. Creation of a standardized geometry of the human nasal cavity. *Journal of Applied Physiology*, 106(3):784–795, 2009.

[23] William McComas. LABORATORY INSTRUCTION IN THE SERVICE OF SCIENCE TEACHING AND LEARNING, 2005.

[24] D.F. Proctor and I.H.P. Andersen. *The Nose, Upper Airway Physiology and the Atmospheric Environment*. Elsevier Biomedical Press, 1982.

[25] Rebecca A. Segal, Grace M. Kepler, and Julia S. Kimbell. Effects of differences in nasal anatomy on airflow distribution: A comparison of four individuals at rest. *Annals of Biomedical Engineering*, 36(11):1870–1882, 2008.

[26] John Steele, Rick Holbeck, and Jean Mandernach. Defining effective online pedagogy. *Journal of Instructional Research*, 8(2):5–8, 2019.

[27] Carolina Sundh and Ola Sunnergren. Long-term symptom relief after septoplasty. *European archives of oto-rhino-laryngology : official journal of the European Federation of Oto-Rhino-Laryngological Societies (EUFOS) : affiliated with the German Society for Oto-Rhino-Laryngology - Head and Neck Surgery*, 272(10):2871–2875, 10 2015.

[28] Kimberly D. Tanner. Order matters: Using the 5e model to align teaching with how people learn. *CBE—Life Sciences Education*, 9(3):159–164, 2010. PMID: 20810945.

[29] Huei-Chuan Wei and Chien Chou. Online learning performance and satisfaction: do perceptions and readiness matter? *Distance Education*, 41(1):48–69, 2020.

[30] J. Wen, K. Inthavong, Z. F. Tian, J. Y. Tu, C. L. Xue, and C. G. Li. Airflow patterns in both sides of a realistic human nasal cavity for laminar and turbulent conditions. *Proceedings of the 16th Australasian Fluid Mechanics Conference, 16AFMC*, 13(December):68–74, 2007.

[31] Kathryn Woods and Kadi Bliss. Facilitating Successful Online Discussions. *Journal of Effective Teaching*, 16(2):76–92, 2016.

[32] Ting Ye and Bing Zhou. Update on surgical management of adult inferior turbinate hypertrophy. *CURR. OPIN. OTOLARYNGOL. HEAD NECK SURG*, 23(1):29–33, 2015.

[33] Jun Zhang. Characteristic size research of human nasal cavity and the respiratory airflow CFD analysis. *Journal of Biosciences and Medicines*, 01(02):23–27, 2013.

[34] Kai Zhao, Pamela Dalton, Geoffery C. Yang, and Peter W. Scherer. Numerical modeling of turbulent and laminar airflow and odorant transport during sniffing in the human and rat nose. *Chemical Senses*, 31(2):107–118, 2006.

A

CFD simulations.

Flow: 250 ml/s

Scale of nose	0.6	1.1	1.2	1.3	1.4
Average					
Scale of nose	0.6	1.1	1.2	1.3	1.4
Velocity	4.2	3.7	3.6	3.4	3.8
Pressure	13.0	3.1	2.9	2.5	3.5
Reynolds number	18.5	19.8	18.9	18.0	17.6
ϵ [m^2/s^3] (Turbulent dissipation rate)	830	490	490	520	600
K [m^2/s^2] (Turbulent kinetic energy)	0.92	0.73	0.73	0.76	0.82
Rotation Rate	1860	1550	1570	1540	1660
Shear rate	2120	1730	1730	1690	1850
Maximum					
Velocity	10.1	9.1	9.1	9.1	9.4
Pressure	49.2	26.0	37.2	26.4	28.0
Reynolds number	92	94	89	85	81
ϵ [m^2/s^3] (Turbulent dissipation rate)	38000	11000	11000	21000	24000
K [m^2/s^2] (Turbulent kinetic energy)	5.1	2.8	2.8	3.8	4.1
Rotation Rate	44000	26000	81000	40000	43000
Shear rate	41000	31000	84000	41000	44000

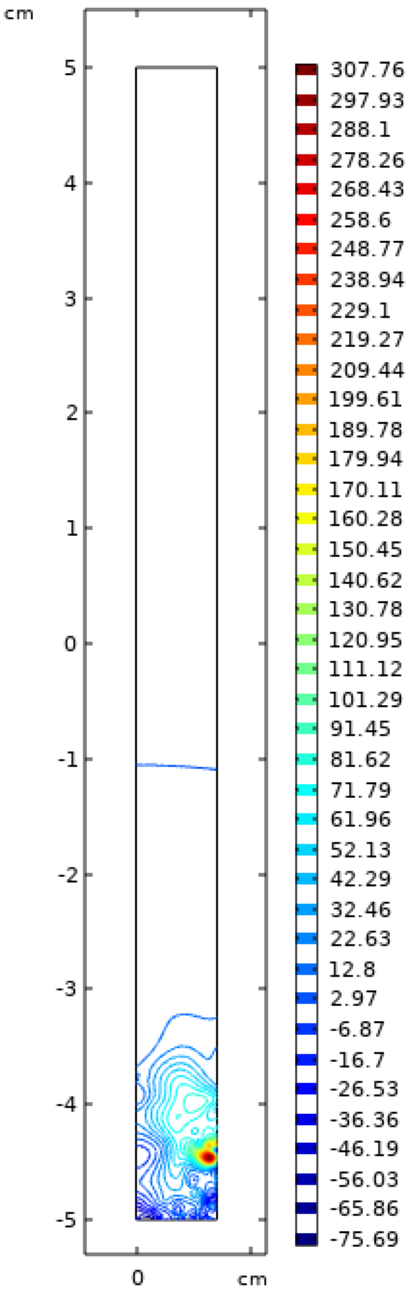
Table A.1: Different derived values when the input was flow at 250 ml/s, due to unknown reasons the simulation on original model (scale 1.0) didn't want to converge at this flow rate.

A.1 Values on the base-case nose

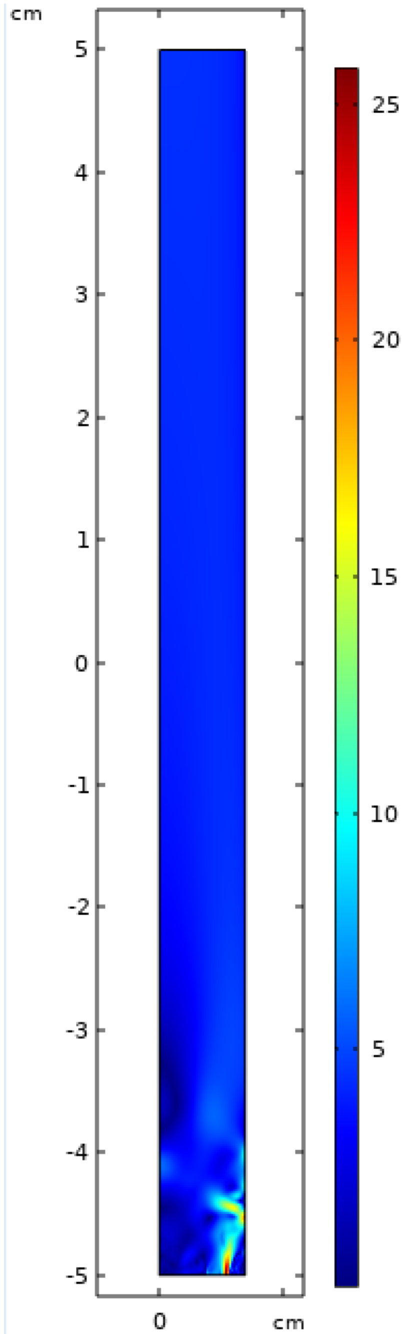
In table A.2 and table A.1 are more values derived from the simulation on the nose found on the internet.

A.2 Example of a divergence

In figure A.1 a simulation that diverge is visualised. The divergence should be obvious even for the untrained eye. The simulation in figure A.1 didn't diverge "much".



(a) The straight pipe when divergence occurred, figure shows the pressure contour.



(b) The straight pipe when divergence occurred, figure shows the velocity (m/s).

Figure A.1: The figure shows an example of occurrence of divergence, the input on this model was pressure at the inlet and the pressure was 0.75 Pa. The numbers are in this scenario not to unreasonable but as seen in the picture the pressure contours are not realistic.

Flow: 161 ml/s

Scale of nose	0.6	1.0	1.1	1.2	1.3	1.4
Average						
Velocity	2.7	2.4	2.3	2.2	2.2	2.3
Pressure	6.17	1.46	1.48	1.43	1.41	1.55
Reynolds number	11.6	7.6	12.4	11.9	11.3	10.9
ϵ [m^2/s^3] (Turbulent dissipation rate)	280	239	175	173	180	202
K [m^2/s^2] (Turbulent kinetic energy)	0.49	0.48	0.41	0.40	0.42	0.44
Rotation Rate	1170	1090	990	990	980	1040
Shear rate	1330	1200	1090	1080	1060	1140
Maximum						
Velocity	6.6	6.3	5.8	5.7	5.8	6.0
Pressure	22	89	12	16	12	12
Reynolds number	60	41	60	57	54	50
ϵ [m^2/s^3] (Turbulent dissipation rate)	13000	316’000	3800	4000	7100	7500
K [m^2/s^2] (Turbulent kinetic energy)	3.0	4.7	1.6	1.7	2.2	2.3
Rotation Rate	26000	670’000	17000	42000	25000	28000
Shear rate	26000	1’030’000	21000	44100	27000	30000

Table A.2: Different derived values when the input was flow at 161 ml/s.

B

Fluid dynamics lecture material.

B.1 Lab-pm

The lab-pm that where used during both demonstrations and the home-lab are to be found below.

Inledning

Detta experiment handlar om friktionsförluster vid luftflöde. Friktionsförluster har vi överallt i vår vardag och ni kommer stöta på det indirekt i erat arbetsliv, exempelvis har vi friktionsförluster när luften strömmar i näsan och blodet som färdas i våra blodådror. Målet med övningen är att ni ska få en grundläggande förståelse för vad som påverkar hur stora förlusterna blir i olika situationer.

Material

- 2-3 sugrör, olika storlek
- A4 papper
- Sax
- Vattenkran
- Penna och papper för anteckningar

Instruktioner

Del 1. Luftflöde

Blås i sugrören av olika storlek och försök svara på följande frågor:

- Vad gör dina lungor för att orsaka luftflöde genom sugröret?
- Vilket sugrör är lättare att blåsa igenom?
- Testa att klippa sugrören på olika sätt för att göra det lättare att blåsa igenom. Vilka samband ser du?

Del 2. Vattenflöde

Spola vatten och håll sugröret precis under kranen så att vatten rinner genom sugröret. Målet är att hela sugröret skall vara vattenfyllt, detta kan vara lite svårt. Om man dels klipper sugröret med en vinkel så öppningen blir större kan det hjälpa att få in mer vatten i sugröret, även kan det hjälpa om man använder sina händer som en tratt.

Tanken är att ni skall lyckas både få klart flöde (laminärt flöde) och brusigt flöde (turbulent flöde) ut ur sugröret (givetvis inte samtidigt). Detta går att uppnå genom att variera diverse olika saker, man kan ändra:

- Styrkan på strålen ut ur vattenkranen
- Lutningen på sugröret
- Färgen på sugröret
- Längd på sugröret
- Diameter på sugröret
- Material (plast/papper/metall/glas) på sugröret
- Temperatur på vattnet

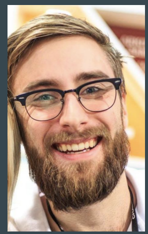
Vilken/vilka faktorer som spelar stor roll för om det blir laminärt eller turbulent flöde? Är det någon faktor som inte spelar någon roll?

B.2 PowerPoint presentation

A PowerPoint presentation was created for the lecture. This presentation was also a big part of the how the lesson was planned. This PowerPoint presentation can be found below.

Vilka är vi?

- Vi är inte läkare, vi kanske använder termer i vissa lägen som inte är rätt. Säg till om ni inte förstår.
- Vi förutsätter att ni har koll på gymnasiefysiken. Om ni har glömt bort något: fråga!
  - Kommer handla mest om tryck.



Även: ställ mer än gärna frågor under föreläsningen.

Sahlgrenska och Chalmers

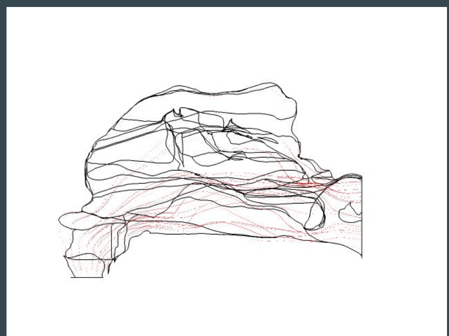
Vad händer?

Vad görs?

Hur påvekar detta er?

Vårt examensjobb

Vårt syfte med examensjobb: Modellera luftflöden i näsan för att kunna hjälpa till med diagnostik samt prediktion av behandling.



Tillvägagångssätt: CT-scan -> 3D-modell -> CFD simulering

Varför strömningsmekanik?

- Strömningsmekanik: vätska eller gas som är i rörelse
- All transport av blod i kroppen
- Allt flöde av luft till lungorna
- Samma mekanik i näsan som i blodådror
- Komplext problem

Komplext problem

- Strömningsmekanik utgår från en krånglig ekvation, Navier-Stokes ekvation.
- Kul trivia: Om ni hittar en lösning på ekvationen får ni ett pris på 1 miljon dollar.

Navier-Stokes Equations:

Cartesian Coordinates

$$x: \rho\left(\frac{\partial v_x}{\partial t} + v_x\frac{\partial v_x}{\partial x} + v_y\frac{\partial v_x}{\partial y} + v_z\frac{\partial v_x}{\partial z}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 v_x}{\partial x^2} + \frac{\partial^2 v_x}{\partial y^2} + \frac{\partial^2 v_x}{\partial z^2}\right) + \rho g_x$$

$$y: \rho\left(\frac{\partial v_y}{\partial t} + v_x\frac{\partial v_y}{\partial x} + v_y\frac{\partial v_y}{\partial y} + v_z\frac{\partial v_y}{\partial z}\right) = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v_y}{\partial x^2} + \frac{\partial^2 v_y}{\partial y^2} + \frac{\partial^2 v_y}{\partial z^2}\right) + \rho g_y$$

$$z: \rho\left(\frac{\partial v_z}{\partial t} + v_x\frac{\partial v_z}{\partial x} + v_y\frac{\partial v_z}{\partial y} + v_z\frac{\partial v_z}{\partial z}\right) = -\frac{\partial p}{\partial z} + \mu\left(\frac{\partial^2 v_z}{\partial x^2} + \frac{\partial^2 v_z}{\partial y^2} + \frac{\partial^2 v_z}{\partial z^2}\right) + \rho g_z$$

Bild av: Ryan Toomey

Från: wikicommons

Experiment

Vad hittar vi för förklaringar i fysiken?

Friktionsförluster

- Friktion ger upphov till energiförluster, detta märks genom en förlust i trycket.
- Tryckförlusten i ett rör ökar med:
  - Större flöde
  - Längre rör
  - Smalare rör

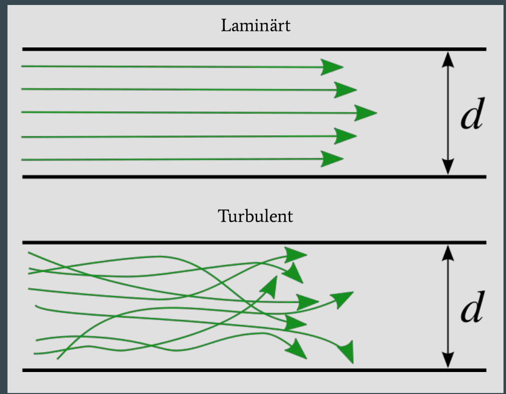
$$\triangle p = f_D \cdot \frac{\varrho}{2} \cdot \frac{v^2}{D} \cdot L$$

Hur märkte vi detta när vi blåste i sugröret/sugrören?

Turbulent och laminärt flöde

- Laminärt
  - Strömlinjeformat
  - Inga virvlar
  - Energieffektivt
- Turbulent
  - Virvlar
  - Svårberäknat
  - Oförutsägbart

Hur såg vi detta i sugrörsexperimentet?

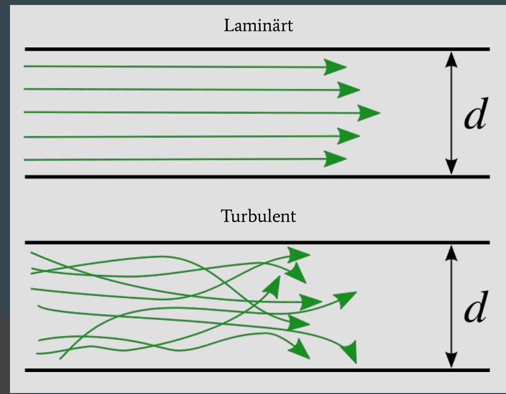


Reynoldstalet - Turbulent och laminärt flöde

Benägenhet till turbulens beskrivs med Reynoldstalet:

- Högre tal -> större risk för turbulent flöde
- Ökar med
  - Högre flödeshastighet
  - Större diameter
  - Mindre viskositet

$$Re = \frac{\varrho \cdot u \cdot D}{\upsilon} = \frac{u \cdot D}{\nu}$$



Frågor på google enkät del 1

Gör bara del 1 och klicka sedan in på del 2 där det står stop/paus och vänta där när du blir färdig.

Länk finns i zoom-chatten

Frågor på google enkät del 1

Era svar och diskussion

Modelltänk

- Prediktionskraft vs begriplighet
- Olika modeller användbara i olika sammanhang

Test

- I syfte att vi ska se hur effektivt det här momentet var
- Anonymt

Tack för oss!

Har ni frågor, feedback och/eller kommentarer är ni mer än välkomna att kontakta oss på:
jakob.widebrant@gmail.com eller johanronnas@gmail.com

B.3 Questionnaire

Here follows the questionnaire that the doctors students where asked to answered at the end of the lecture about fluid dynamics. These are in Swedish since the lecture and the course was in Swedish.

Frågor workshop del 1

*Obligatorisk

1. Framförallt kunde vi ändra hastigheten på vattnet för att få turbulent eller laminärt flöde, hu skulle det bli om vi kunde ändra vätska. T.ex om vi hade honung som rann i sugröret, hade c blivit mer turbulent eller mer laminärt?

   Markera endast en oval.

   - Mer trubulent
   - Mer laminärt
   - Opåverkat

2. Var är det större sannolikhet att det är turbulent flöde: i näsan eller blodådrorna? Förklara gärna hur du tänker, vad talar för och emot ditt påstående.

   ____
   ____
   ____
   ____
   ____

3. Hur kan sugröret fungera som modell för olika situationer i kroppen? Sugröret innebär en kraftig förenkling, vad får det för konsekvenser?

   ____
   ____
   ____
   ____
   ____

4. Finns det situationen när du pratar med en patient där denna sugrörsmodell kan vara användbar? Förklara hur du tänker.

PAUS/STOP

Vänta här så går Jakob och Johan igenom svaren och diskuterar föregående frågor innan du/vi går vidare till nästa del.

Avslutande frågor

5. Hade du sugrör hemma? *

Markera endast en oval.

- Ja
- Nej, tittade på filmen
- Nej, försökte med pappersrör istället

6. Hur är trycket i lungorna jämfört med atmosfären när vi andas in?

Markera endast en oval.

- Lägre
- Högre
- Samma

7. Vad beskriver Reynoldstalet?

8. Vilken påverkan skulle en ökning av följande storheter ha på Reynoldstalet? Skulle Reynoldstalet bli högre eller lägre eller inte ändras?

Markera endast en oval per rad.

	Högre	Lägre	Påverkar inte
Hastighet	⬭	⬭	⬭
Densitet	⬭	⬭	⬭
Tryck	⬭	⬭	⬭
Tvärsnittsarea	⬭	⬭	⬭
Temperaturen	⬭	⬭	⬭
Viskositet	⬭	⬭	⬭
Längd	⬭	⬭	⬭

9. Om vi betraktar de övergripande strukturerna i hjärtat, kan vi finna både laminärt och turbulent flöde där eller bara en av dem? Vart i såna fall?

10. Du har en vätska t.ex blod som du önskar blanda så mycket som möjligt, vill du ha turbule flöde eller laminärt flöde?

Markera endast en oval.

- ⬭ Turbulent
- ⬭ Laminärt
- ⬭ Spelar ingen roll

11. Nämn en egenskap som finns hos laminärt flöde som inte finns hos turbulent flöde

12. Vilken påverkan skulle en ökning av följande storheter ha på tryckförlusten i t.ex ett blodkärl? Skulle tryckförlusten bli högre eller lägre eller inte ändras?

Markera endast en oval per rad.

	Högre	Lägre	Påverkar inte
Hastighet	◯	◯	◯
Densitet	◯	◯	◯
Tvärsnittsarea	◯	◯	◯
Temperatur	◯	◯	◯
Blodkärlets längd	◯	◯	◯

13. Hur tror du att du som framtida läkare kommer samarbeta med ingenjörer?

14. Ranka hur väl du håller med följande påståenden med avseende på innehållet i denna föreläsning

Markera endast en oval per rad.

	Stämmer inte	Stämmer	Vet inte
Användbart som läkare	◯	◯	◯
Roligt och intressant	◯	◯	◯

15. Hur var svårighetsgraden på denna föreläsning?

Markera endast en oval.

- ◯ Den var alldeles för lätt, jag blev jätteuttråkad
- ◯ Den var för svår, jag förstod ingenting
- ◯ Precis lagom
- ◯ Inget av de andra alternativen passar in på mig

B.4 Regarding Collaboration between Doctors and Engineers

B.4.1 Question

Hur tror du att du som framtida läkare kommer samarbeta med ingenjörer?
How do you think, as a future doctor, you will collaborate with engineers?

B.4.2 Answers

- Med tekniska saker
- Beror på vilken specialitet.
- Tillsammas ibland
- Förhoppningsvis med än idag, då "medicinen och fysiken" mer och mer kan hänga samman/lyfta varandra
- "AI!
- har inte funderat på detta. men tror absolut att man kommer jobba i team med ingenjörer. t.ex med olika tekniska redskap
- Nytänkande forskning med ny teknologi.
- ingenjörer tar fram ny teknik som vi får användning för. Ingenjörer rattar många av de maskiner som finns på ett sjukhus.
- Oj, lång fråga att besvara men teknik är ju redan instrumentellt för sjukvården och mer framträdande lär det ju bli.
- Mycket inom radiologi för utveckling av exempelvis utrustning samt i tolkningen av bilder. Kirurgin kommer att samarbeta mycket i utvecklingen av operationsmetoder.
- Vet inte riktigt!
- Beroende på var man arbetar så tror jag det finns stora möjligheter att man kommer träffa på/arbeta med ingenjörer, tex. gällande nya tekniker/metoder, läkemedel, m.m. Det kommer ju garanterat hända mycket på forskningsfronten i framtiden och samarbetet kan säkerligen bli mer än vad det är idag.
- För att arbeta fram tekniska lösningar som kan effektivisera och förbättra diagnostiska verktyg eller hjälpmedel vid kirurgi eller träning inför kirurgiska ingrepp exempelvis
- Ingenjörer är den del av tekniken på sjukhuset redan nu, så jag gissar på samma sätt. Det sker en stor utveckling hela tiden så varför inte i större utsträckning?!
- kanske
- ABSOLUT! Massa spännande lösningar som vi kan komma fram till tillsammans
- Kalibrering av instrument, support av maskiner och instruemtnt
- Beror på vilken specialitet. Maskiner byggs ju av ingejörer med sårt att se hur t.ex. en allmänläkare eller psykiater skulle ha för nytta av det, just nu
- Vet ej.
- Förstå/utveckla nya produkter inom hälsovård.
- Det kommer bli mer och mer.

- Att dom kommer in som patienter :) Antagligen inom forskningen med.
- Mer och mer