Norwegian University of
Science and Technology

Energy Efficient Climatization for Rooms with Cooling Demand

-based on a Concept with Displacement Ventilation and Low Supply Air Temperature

Anne Kristine Amble

Master of Science in Energy and Environment
Submission date: September 2008
Supervisor: Per Olaf Tjelflaat, EPT
Co-supervisor: Tor Arvid Vik, Rambøll Norge AS

Norwegian University of Science and Technology
Department of Energy and Process Engineering

Problem Description

1. An overview should be made of the princples of the buildings and climatization solutions used in classrooms in Norway the last 15 years.
2. Relevant scenarios for displacement flow ventilation with low supply air temperature used in classrooms should be analyzed using empirical calculation methods.
3. A CFD tool should be used, and simulations made for different scenarios of classroom ventilation with displacement flow and low supply air temperature with the aim of analyzing thermal conditions and air quality. The results should be compared to the conditions obtain with ideal mixing flow ventilation.
4. The energy balance of a typical classroom, with a ventilation aggregate, should be analyzed at different outdoor climate and use. This should be done by numerical energy simulations over the year for Oslo climate.
5. Guidelines for design and sizing of climatization of this type of classroom and similar rooms with cooling need should be developed.

Original problem description in Norwegian:

1.Det skal lages en kort oversikt over bygnings- og klimatiseringsprinsippene i de ventilasjonsløsningene som har blitt benyttet for klasserom her i landet i de siste 15 årene.
2.Aktuelle scenarier for fortrengningsventilasjon med lav tilluftstemperatur som benyttes for i klasserom skal analyseres ved bruk av empiriske beregningsmetoder.
3.Et CFD-verktøy skal tas i bruk, og det skal gjennomføres beregninger for ulike scenaria av klasseromsventilasjon med fortrengningsventilasjon med lav tilluftstemperatur for å analysere termiske forhold og luftkvalitet. Resultatene bør sammenlignes med forholdene som oppnås ved ideell omrøringsventilasjon.
4.Energibalansen for et typisk klasserom, med tilhørende ventilasjonsaggregat, skal vurderes ved ulike uteklima og bruk. Dette bør gjøres ved numeriske energisimuleringer over året for Oslo-klima. Det bør sammenlignes med et scenario med ideell omrøringsventilasjon.
5.Design- og dimensjoneringsveiledning for klimatisering av denne type klasserom og liknende rom med kjølebehov skal utvikles.

Assignment given: 18. April 2008
Supervisor: Per Olaf Tjelflaat, EPT

Acknowledgements

This thesis aims to look at alternative classroom ventilation. There is much focus on reducing the use of electricity use. At the same time, pupils and teachers in the Norwegian schools suffer from poor indoor environment. This report is written with the wish of finding solutions to both problems.

I would like to thank my supervisor Per Olaf Tjelflaat for his guidance. Furthermore there are several people at Rambøll who have been most helpful with big and small problems: Tor Arvid Vik as co-supervisor, Sven Egil Nørsett for help with Ansys CFX, Bjørn Terje Pettersen for enabling the master thesis and giving me a warm welcome at Rambøll. I have had great time there thanks to all the friendly people at the VVS/FDVU division. I would also like to thank Lana Saracevic for discussions and constructive criticism along the way as well as Daniel Liljar, Kristine Raaen and Catherine Churchill.

Trondheim, September 2008

Anne Kristine Amble

Abstract

In relation to new building directives and regulations there is much focus on energy efficiency and reducing the use of electricity. There is therefore a need for ventilation systems with low Specific fan power (SFP) and to limit or avoid the need for mechanical cooling.

At the same time questions are raised concerning whether increasingly complicated ventilation systems are the best way to achieve energy efficiency. They are more vulnerable for poor design and maintenance which especially school buildings have suffered from.
This has led to the development of several natural and hybrid ventilation concepts. One of these which has had success in school buildings the last 15 years in Sweden and Norway is the so-called Swedish model. Characteristic of this solution is that sub terrain culverts are used for passive cooling and heating of the ventilation air which passes through double inner walls and is supplied to the room high on the inner wall. The supply air falls towards the floor mixing in the surrounding air and spreads along the floor as displacement flow with an outlet in the ceiling. This works particularly well in classrooms where the high occupation density creates a cooling demand large parts of the year. These schools therefore have a very low SFP factor and no mechanical cooling.

In the following report a classroom model with the Swedish ventilation concept is studied with the aim of developing a method of dimensioning such air flow. CFD simulations are used to confirm expected behaviour and the resulting thermal comfort and air quality at typical winter, spring and summer scenarios. Finally a Guide for preliminary design and sizing for climatization of rooms with cooling needs using the Swedish model is presented.

The most important results were that there was a good correspondence between the empirical calculations and CFD simulations. Adequate carbon dioxide levels and temperature can be achieved without use of primary energy for heating at for outdoor temperatures above 5 ºC. There was a substantial risk of draft unless a near zone of 2-3 m is used. However the CFD model was limited to a slice of the classroom and the accuracy of the results suffer somewhat from this.

Guidelines are developed for preliminary design and sizing of supply air based on empirical expressions of air plumes by Eimund Skåret(2000) and can be found in the appendix.

Contents

Figures

Tables

1 Introduction

How can the need for energy efficiency be combined with good indoor air quality?
There are great challenges in finding good solutions for ventilation in schools. Schools are large work places and also have the largest number of sufferers from asthma and allergies. There is much focus on reducing energy use in buildings in the Technical Regulations under the Planning and Building Act 2007(TEK)[1]. At the same time indoor air quality (IAQ) in schools is a highly neglected area, as is often portrayed in media. A survey performed by the Norwegian Institute for Air Research (NILU, 2003) showed that 35% had unsatisfactory, and 7% unacceptable indoor climate conditions. There is an ongoing debate as to whether the air quality demands in schools can be met with increasingly complicated ventilation systems. Studies have shown the negative effect air filters have on the perceived IAQ(Mysen, 2005). Mechanical balanced ventilation uses much electricity for fan power. At the same time purely naturally ventilated schools do not meet the requirements in the TEK guidelines (REN)[2] concerning necessary air volumes and heat recovery. Alternative solutions need to be found, such as hybrid solutions profiting from both natural and mechanical forces. The challenge lies in documenting that alternative solutions meet the requirements. This is a cost demanding process, which in turn means that conventional solutions are often chosen. Another problem arising with increased demands for improved insulation is an increasing cooling demand. This is particularly a problem in classrooms where the density of people is high.

This master thesis seeks is to find an optimal concept for room ventilation with a relatively large heating load such as a classroom or a conference room. Preliminary work by Halla, (2007) will be further developed. Ideally no mechanical cooling should be necessary, and the pressure drop and fan power should be low. The goal is to attain satisfactory IAQ with the lowest possible use of primary energy and cooling.

One possible solution to be examined in this report is a hybrid ventilation type inspired by the so-called Swedish model. Schools that have been built in Norway and Sweden during the last 15 years using this model have low specific fan power (SFP) and use no mechanical cooling. In this solution air is supplied with low impulse and low temperature through a vent placed high on the inner wall. The air then falls towards the floor while mixing in with the room air. It then spreads over the floor where thermal forces will cause it to rise up and exit through the exhaust vent placed in the ceiling. This form of displacement flow is very efficient for cooling purposes. The air flow is controlled primarily by temperature sensors, but CO_2- sensors are also an option. Good insulation of the façade, solar shading, the use of daylight and thermal mass are essential keys to success.

---

[1] Technical Regulations under the Planning and Building Act 2007. Renewed as a result of the (EU) European Energy Performance of Buildings Directive (EPBD). National Office of Building Technology and Administration (2007).

[2] Guidelines for the (TEK) Technical Regulations under the Planning and Building act 2007 4th ed. by the National Office of Building Technology and Administration (2007).

The simulation program ANSYS CFX is used for computational fluid dynamics (CFD) simulations of thermal conditions and air quality. Typical scenarios for winter, spring/autumn and summer are analysed for the above mentioned classroom. Possibly this concept can be adapted to other rooms with cooling demands and even for mechanical low pressure ventilation plants.

This report requires that the reader has a technical background, and is somewhat familiar with certain ventilation and energy expressions. However, important ventilations principles will be explained in the first part. An overview of classroom ventilation practice the last 15 years in Norway will be given, followed by a detailed description of the Swedish model as well as an introduction to ANSYS CFX. The next part includes empirical calculation methods to evaluate scenarios for supplying air where different plume models are compared. Following this is the part where the above mentioned CFD simulations will be found. The heat balance of a typical classroom will be evaluated for Oslo climate. A discussion and conclusion will be based upon this. In Appendix 5 guidelines are presented for design and technical specifications for climatization in this type of classrooms and rooms with similar cooling needs.

2 Theory

2.1 Air flow patterns

In this chapter the theory behind convection plumes, vertical plumes with buoyancy and cold drafts is explained as well as the principles of displacement flow and mixing flow ventilation.

2.1.1 Thermal plume/convection plume

A thermal convection plume (figure 1) is created when air next to a warm surface is heated thus reducing its density and causing it to rise. This is called the thermal force of buoyancy. The angle from the centre line to the boundary is approximately 12,5 º. The thermal plume has several similarities to the air jet. Because of mass and energy conservation the convective heat effect must equal the amount of heat at any cross section of the plume. This is expressed in equation (2.1) (Skåret, 2000 p65):

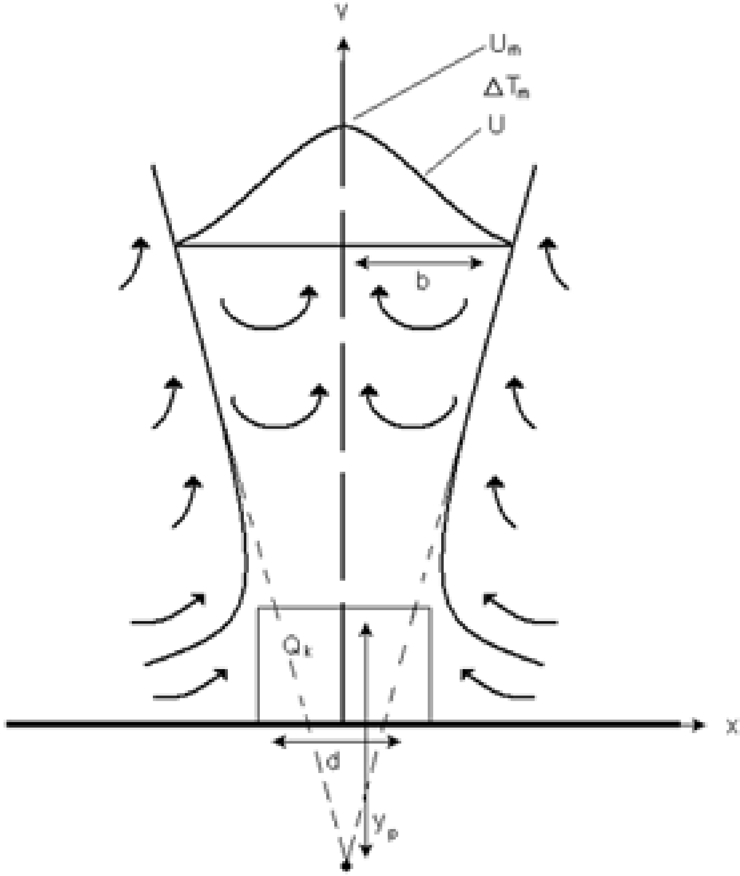

Figure 1 Convection plume

$$P_k = \rho \; c_p U_m \Delta T_m A_m \int_0^1 \frac{U\Delta T}{U_m \Delta T_m} \frac{dA}{A_s} \qquad (2.1)$$

P_k - convective heat effect [kW]
ρ - density [kg/m3]
U_m - maximum speed of plume/central speed [m/s]
ΔT_m – maximum temperature difference between plume and surrounding air. [K]
As - cross-sectional area [m^2]

The integral can be denoted as I3 as for a free air jet and has the value 0,1785 (Skåret, 2000 p 21). This simplifies the equation to:

$$P_k = c_p U_m \Delta T_m A_s I_3 \qquad (2.2)$$

For a point source (axisymmetrical) the governing equations are:

Central/maximum velocity:

$$U_m = 1{,}28\left(\frac{P_k}{y+y_p}\right)^{1/3} \quad \text{[m/s]} \qquad (2.3)$$

Maximum temperature difference:

$$\Delta T_m = 20{,}9\, P_k^{\;2/3}(y+y_p)^{-5/3} \quad \text{[K]} \qquad (2.4)$$

Volume flow:

$$q_{Vk} = 0{,}055\, P_k^{1/3}(y+y_p)^{5/3} \quad \text{[m3/s]} \qquad (2.5)$$

where y_p is the distance from an imaginary source pole on the central axis to the top of the hot surface(Skåret, 2000: 67-68). yp is approximately 0,7 diameters.

These simplifications of Navier Stokes equations presume conditions with small temperature differences, such as is the case in an occupied room. Furthermore certain coefficients are assumed constant: Thermal expansion coefficient, β=1/300 K, density, ρ=1,2 kg/m3, heat capacity, cp= 1 kJ/kgK.

The theory of convection plumes can be applied to human as heat sources. The volume flow is greatly reduced with vertical temperature differences (Nilsson, 2003 p 225)

2.1.2 Vertical plumes with buoyancy

When a vertical plume with a lower temperature has a downward direction into a room, it will develop the same profile as a thermal plume. But at some point the direction vector will turn negative when the plume temperature assimilates the temperature of the surrounding air bringing the plume to a halt. The driving force of the plume is more the temperature difference than the initial velocity as can be seen from the Archimedes number in eq. (2.6)

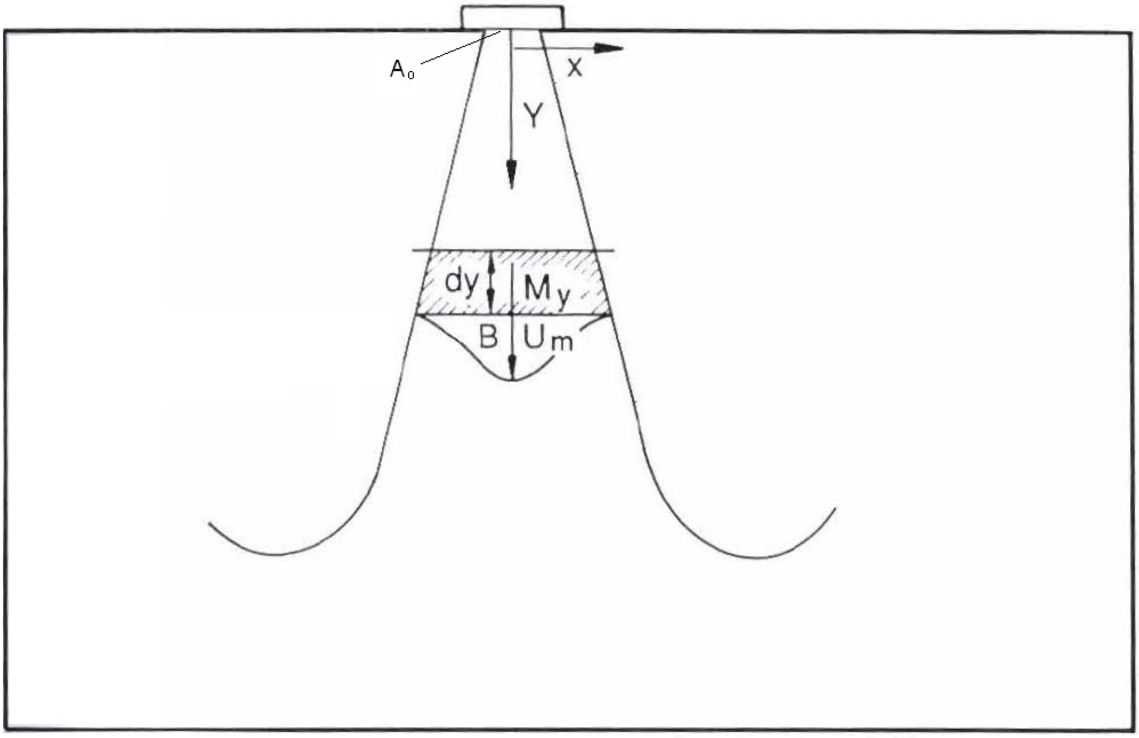

Figure 2 Vertical plume with buoyancy (Skåret, 2000, p 57)

The velocity of the plume will create a sub pressure which will induce surrounding air increasing the volume flow. The central velocity *Um* at a distance *y* is described with the following equation for axisymmetric round plumes (Skåret 2000, p57-64):

$$U_m = U_0 \left(Ar_0 \frac{\rho_0}{\rho_r} \frac{4{,}3}{C_b^2} \frac{\sqrt{A_0}}{y + y_p} + \left(\sqrt{\frac{\rho_0 i}{\rho_r \varepsilon}} \frac{1{,}54}{C_b} \frac{\sqrt{A_0}}{y + y_p} \right)^3 \right)^{\frac{1}{3}} \quad \text{[m3/s]} \qquad (2.6)$$

U_0 - Initial velocity [m/s]

A_0 - Inlet opening [m^2]

Ar_0 - Archimedes number in the opening $Ar_0 = \frac{g\sqrt{A_0}\beta\Delta T_0}{U_0^2}$

i- Impulse factor approximately 1.

ΔT_0 - Temperature difference between the plume and surrounding air.

$\rho_0 \approx \rho_r$ - Air density difference assumed minor

$C_b = tg(12{,}5^o)$

y_p - Pole distance, approximately 0,7 times the diameter of the opening.

ε - Contraction coefficient for calculation purposes approximated 1 (Skåret, 2000, p19)

2.1.3 General for both plume models

If the plume is close to a wall the sub pressure the velocity creates will make it cling to the wall. This is called the Coanda effect. According to theory, the volume flow in such a plume can be calculated as half of a free plume with twice the inlet area, A_0, or in the case of a convection plume twice the area of the convection source.

The calculations assume zero vertical temperature difference, to a certain extent this can be accounted for by using the average vertical temperature.

It is important to note that these expressions are valid from 6-10 diameters from the inlet opening i.e. up to y=1m. As these are functions of $y+y_p$, Um and ΔT approaches "infinite"/large as y→ y_p which is obviously not the case in a real scenario.

The velocity profile of an air jet as described by Abramovich in 1963 (Skåret, 2000 p 17) :

$$U(r) = U_m\left(1-\left(\frac{r}{b}\right)^{1,5}\right)^2 \quad \text{[m/s]} \qquad (2.7)$$

$b = C_b\left(y + y_p\right)$ is the distance from the central axis to the boundary layer
r is the distance normal to the central axis of the plume to b.
Um is the central velocity of the plume

2.1.4 Cold draft

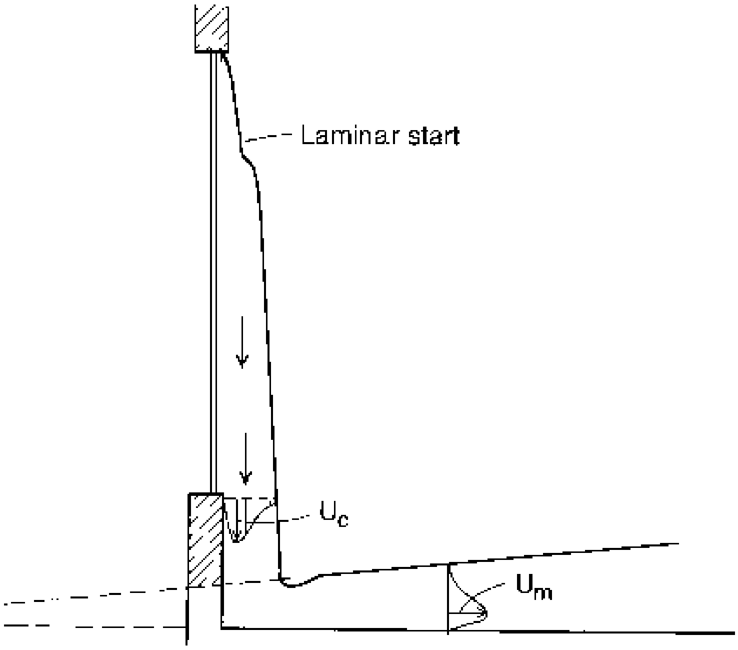

Figure 3 Cold draft created by window (Skåret, 2000 p 203)

Cold surfaces such as windows cool the air causing a downward draft.
This has some major disadvantages:

- The cold surface causes asymmetric radiation which can lead to thermal discomfort.
- Risk of cold draft for people close to it.
- Layer of cold air at floor level as seen in figure 3 creating large temperature gradient.
- In cases of displacement flow, contaminated air from the upper zone can be transported down to the breathing zone.

To determine the maximum velocity of the cold draft plume the temperature of the surface needs to be known. In the case of a window, indoor and outdoor temperature can be easily decided. The temperature difference between the cold surface and the room is expressed as:

$$\Delta T_f = \Delta T_{tot} \frac{U}{\alpha_s + \alpha_k} \qquad (2.8)$$

ΔT_{tot} - indoor-outdoor temperature difference [K]
U - thermal transmission coefficient [W/m^{2}K]
α_s - radiative heat exchange factor, 4,5 [W/m^{2}K] for a radiationwise grey surface
α_k – convective heat exchange, 1,31ΔT1/3 = 2,5 [W/m^{2}K] for approximate calculations.

The stretch of laminar flow, y_l, is determined for a critical value of the Grashof number:

$$Gr = \frac{g\beta\Delta T_f}{\nu^2} = 10^9 \quad (2.9)$$

):
$$y_l = \frac{\nu^2 10^9}{g\beta\Delta T_f} \quad (2.10)$$

The starting point for turbulent flow, y_{pt}, is:

$$y_{pt} = y_l \left(1 - \frac{0.144^9}{\left(g\beta\Delta T_f\right)^{0.21} y_l^{0,64}}\right) \quad (2.11)$$

Thus the maximum velocity at lower edge of cold surface:

$$U_{\max} = 0{,}53\sqrt{g\beta\Delta T_f \left(y - y_{pt}\right)} \quad (2.12)$$

where y in this case denotes the height of the cold surface.

The air flow at the lower edge of the cold surface per meter width of surface:

$$q_v = 0{,}116\left(g\beta\Delta T_f\right)^{0,4}\left(y - y_{pt}\right)^{1,2} \quad (2.13)$$

2.1.5 Mixing flow ventilation

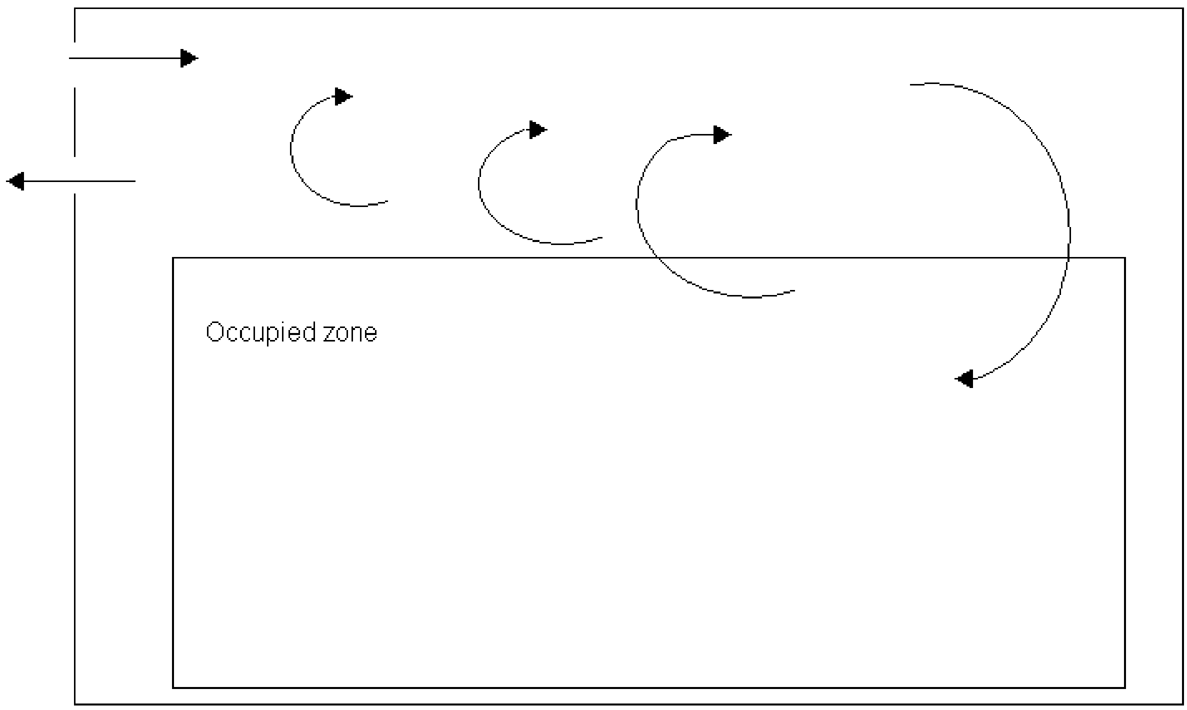

Figure 4 Mixing flow ventilation

In mixing flow ventilated rooms air is supplied at high velocity 1-2 degrees below desired room temperature. The air jet clings to the ceiling, the so-called Coanda effect, and the surrounding air is mixed in until the velocity decreases and the air jet falls. Ideally this causes a homogenous concentration of pollutants in the room. There is a number of different supply and exhaust combinations which can not all be described in this report. In a typical concept the inlet is placed high on the wall and air is supplied as a high velocity air jet along the ceiling. There is however a risk of short circuit ventilation where the fresh air flows directly to the outlet, leaving the occupied zone increasingly polluted.

2.1.6 Displacement flow ventilation

Displacement flow ventilation, as the name indicates, displaces the air. This way pollution and surplus heat can be removed from the breathing zone effectively. This can be done passively by taking advantage of heat sources and the fact that warm air rises. The inlet is usually placed at floor level, or even through the floor, where air is supplied through diffusers at low speed a few degrees below desired room temperature. The air flows along the floor, gradually warming and the heat and breath from people makes the air rise taking the pollution with it (see chapter 2.1.1 on thermal plumes). This requires that the near zone of the air supply is kept unobstructed and unoccupied as this will be a draughty area. Alternatively, the fresh air can be supplied through perforated/textile channels in the ceiling or diffusers hanging from the ceiling (Airson, 2008) where ideally the cool air sinks to the breathing zone before rising again as conventional displacement ventilation. This is called active displacement flow and can effectively remove either cold and warm pollutants by placing exhaust vents at floor level and in the ceiling (Ståbi,2002 p154).

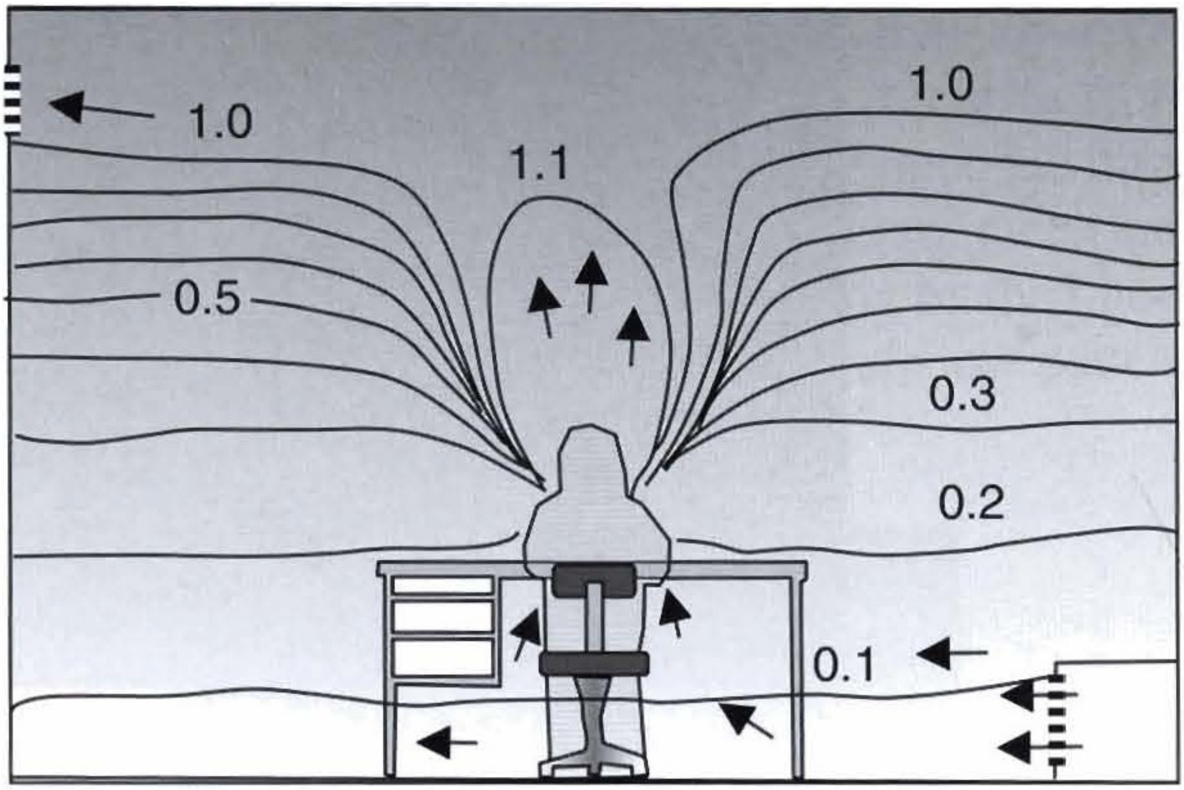

Figure 5 Displacement flow with lines of constant contamination (Nilsson, 2003 p 228)

A special case which will be further explained in the following chapter is the Swedish natural ventilation model which is much used in schools. In this type cool air is supplied through low pressure, low velocity vents placed high on the wall. This falls as a downdraft towards the floor, mixing in surrounding air. It spreads along the floor before rising towards the ceiling where the outlet is. This is the type of displacement flow this report will address. Displacement flow is used in both mechanical and natural ventilation systems.

2.1.7 Ventilation efficiency /degree of contamination

Ventilation efficiency is a measure of the efficiency of the removal of pollutants from the occupied zone. It does not take into consideration energy efficiency, as excessive air volumes can give good ventilation efficiency.

$$\varepsilon_v = \frac{C_e - C_s}{C_p - C_s} \qquad (2.14a)$$

C_e –concentration of pollutants in exhaust air

C_s - concentration of pollutants in supply air

C_p - concentration of pollutants in measuring point i.e. breathing zone

If simply considering contamination levels above supply air levels the equation reduces to:

$$\varepsilon_v = \frac{C_e}{C_p} \qquad (2.14b)$$

For ideal mixing ventilation the efficiency is 1 since the concentration is homogeneous. A much lower level than 1 indicates short circuit ventilation. For displacement flow, values can be higher than 1. Therefore one can achieve the same air quality in the breathing zone with less air volumes than for mixing ventilation. For displacement flow one often operates with a so-called *two zone model* where the room is divided vertically in an occupied zone and an upper zone. The concentration of contaminants in each zone is homogenous.

It is important to note that although it is quite common to use the term ventilation efficiency, it is more precise to call it *degree of contamination.* The degree of contamination, μ, as defined by Nilsson (2003 p 227) can be denoted as:

$$\mu = \frac{c_l - c_s}{c_e - c_s} \qquad (2.15)$$

c_l - local contamination (occupied zone/measuring point)

c_s - contamination in supply air.

c_e – contamination in exhaust air.

2.2 Climatization principles in classrooms

In this chapter an overview is given of the principles used for ventilation of classrooms in Norway the last 15 years.

2.2.1 Overview

The most common way of ventilating school buildings is by mechanical balanced ventilation. However several schools in Norway and Sweden have been built with alternative solutions. In this chapter the different categories of classroom ventilation will be presented. As this report focuses on the so-called Swedish model, a detailed description will be given of this.

Byggforsk (2005) presents the following concepts as alternatives for choosing ventilation system in schools:

- Balanced ventilation. Supply and exhaust air are equal. This category includes both mechanical and low pressure hybrid solutions.
- Exhaust ventilation system. Predominantly in older schools and is not recommended.
- Decentralized ventilation units placed on the outdoor wall of the ventilation zone.

In Norway the following concepts are in use according to the Direcorate for Primary and Secondary Education (2004):

- Mechanical ventilation.
- Culvert ventilation solution <u>with</u> preheating of air.
- Culvert ventilation solution <u>without</u> preheating of air.
- Direct façade ventilation.

The bottom three solutions profits from natural driving forces such as wind and buoyancy. These are often referred to as *natural* or *hybrid* ventilation varying on the degree of mechanical assistance. As for the hybrid solutions, these can be subcategorized in Swedish, Danish, German, Norwegian and Finnish models where the Swedish model is almost fully natural and the Finnish almost mechanical. These are thoroughly described by Dokka et al. (2003) in a report connected to the *Økobygg*[3]- program. The report presents the principles and experiences one has had with building integrated ventilation and is meant as a guide to architects and engineers. The following description on natural and hybrid ventilation types is a rendering from this report.

2.2.2 Mechanical ventilation

The early form of mechanical ventilation was purely exhaust fan driven. In many of today's older schools, pure exhaust systems are the only means of ventilation. As buildings got more and more air-tight, supply air was needed in order to achieve air change and not just low pressure. When supply and exhaust air volumes are equal, it is called *mechanically balanced ventilation (MBV)*. Fresh air is actively treated (filtered, heated etc), transported through channels and reheated if needed before being supplied as mixing flow (high velocity) or displacement flow(low velocity) (see figure 6 and 7). In rooms with a high density of people, such as a classroom ore a conference room, displacement flow is often more effective.

[3] Økobygg was a 5-year program to promote environmental sustainability in building and construction industry. The program ended in 2002. More info at www.grip.no.

However this method requires cooler supply air, which might require mechanical pre cooling of air. It also comes in conflict with area use, as displacement flow diffusers are placed near the floor.

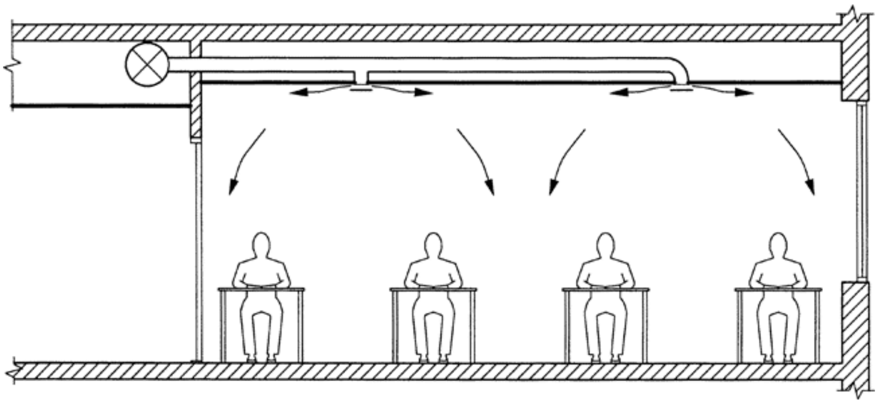

Figure 6 Mixing flow in classroom (Byggforsk, 2005)

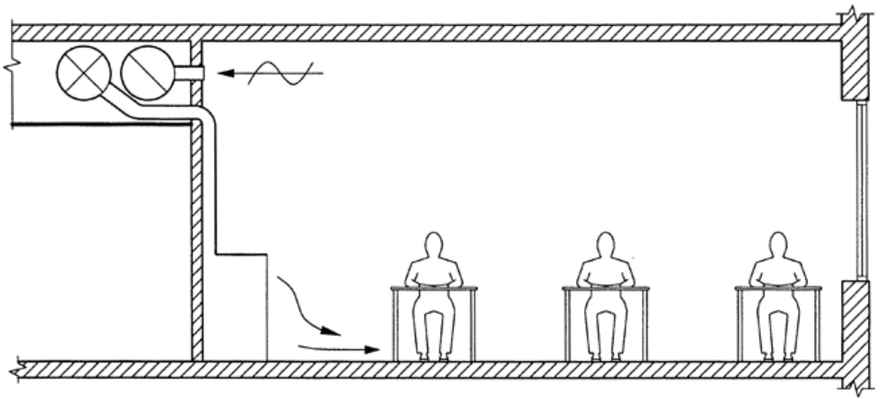

Figure 7 Displacement flow in classroom (Byggforsk, 1996)

The system usually has a substantial use of electricity for fan power due to high pressure drop in ducts and air handling units, but a good degree of heat recovery as the fresh air intake channels can easily be placed near the exhaust. A rotating heat exchanger can recover up to 80% of the heat from the exhaust air. When considering that a very large proportion of heat loss in modern buildings is through ventilation, this is a considerable contribution.

As opposed to natural and hybrid ventilation, mechanical ventilation allows various ways of placing inlet and outlet vents, creating a range of different air flow patterns, creating both mixing and displacement flow, whereas natural ventilation is synonymous with displacement flow ventilation. MBV is recognized for providing the REN recommended air volumes and heat recovery, which makes it the "easy choice" amongst engineers. A disadvantage with MBV is difficult access for maintenance and cleaning and thus the risk of microbial growth. The high degree of technical components increases the risk of malfunctioning. Unfortunately many systems are badly designed and lack maintenance causing poor indoor environment. According to Rustad (2008), an engineer in Undervisningsbygg[4], mechanically balanced mixing flow with diffusers or back edge vents has been used recent years in schools.

[4] Undervisningsbygg Oslo KF is one of Norways biggest contractors and renovators of school buildings.

Displacement flow has also been used but to a lesser extent because of conflict with area use and near zone requirements.

2.2.3 Finnish building integrated ventilation model

The *Finnish model* is a building integrated low pressure drop ventilation solution (SFP<1.5 $kW/m^3/s$) that does not necessarily profit much from natural ventilation forces. It is a balanced ventilation system with a compact aggregate, and might be categorised as mechanical ventilation. It is characterized by extremely large ventilation ducts to keep the pressure drop at a low. This is obviously space demanding but easy access makes cleaning and maintenance easy. Corridors can be used as exhaust ducts keeping the pressure drop at a low also here. Air distribution is a combination of variable air volumes (VAV) and displacement flow. Air volumes are controlled by CO_2-sensors. Façades face north and west to avoid solar heat gain and the need for cooling.

2.2.4 Façade ventilation

Traditionally schools were naturally ventilated with the possibility of opening windows, where only buoyancy forces and wind drove the air flow. This does not meet today's requirements for thermal comfort and necessary air volumes as these natural ventilation forces vary with outdoor temperature and wind conditions. Opening of windows is the simplest form of facade ventilation which in many schools in poor condition is the only means of achieving good indoor air quality. In schools with high outdoor air quality, this can be a fool proof solution. A great part of indoor environment satisfaction is related to being able to affect the indoor environment. The main disadvantages are lack of filtration of pollen, thermal discomfort and noise. Direct façade ventilation has been further developed to what is called the ***Danish model***. Air is supplied directly through the façade, either through windows or openings placed near the ceiling, or near the floor where the air passes a heater. Exhaust vents are placed centrally in the building to profit from overflow ventilation, equipped with a fan when the natural forces prove too feeble, or for night time cooling. Air flow is controlled either by temperature or CO_2-levels. Heat is usually recovered by a heat pump in the exhaust air. Air filters are possible to install, but pose a maintenance challenge.

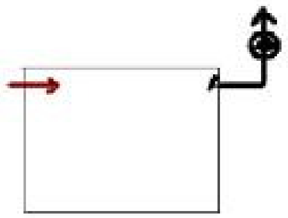

Figure 8 Principle of direct air intake in façade (Direcorate for Primary and Secondary Education, 2004)

A variant of this is often referred to as the ***German model***, where the supply air is preheated in an atrium or double glass façade.

2.2.5 Culvert solutions: Norwegian and Swedish Models

There are two kinds of building integrated ventilation systems that include a supply air culvert: *The Norwegian and Swedish models*. Each has a different way of achieving low energy use: heat recovery and seasonal adaptation. In both cases air is supplied through a culvert as shown in figure 9 and 10. The culvert passively treats the supply air by a slight preheating effect and a substantial cooling effect as well as particles and pollen being deposited. The exhaust is placed in the ceiling as so-called lanternines creating a stack effect. Normally the buoyancy forces and wind provide sufficient air change but assisting fans can be started if necessary. Whereas the Norwegian model uses traditional displacement flow diffusers, the Swedish model transports air from the culvert through double brick walls to the inlet placed high on the wall. The Norwegian solution has means of preheating the air in the culvert as well as a hydronic heat exchanger. These represent a pressure drop, which requires extra fan power. The Swedish solution with its seasonal adaptation uses smaller air volumes in wintertime to reduce the ventilation heat loss.

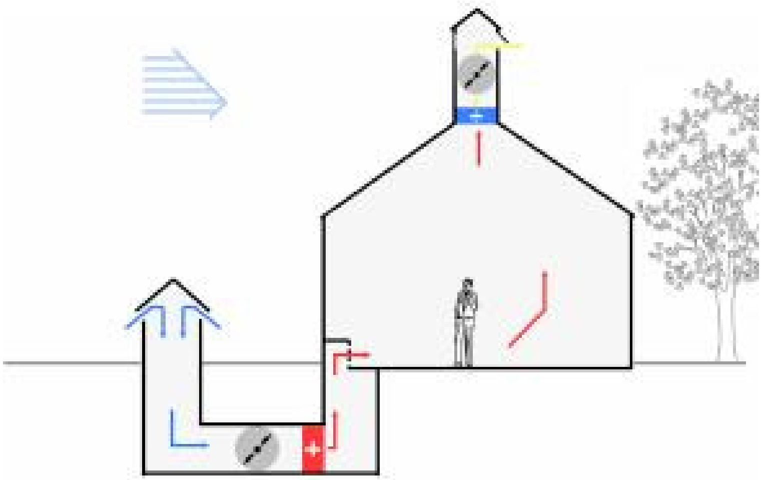

Figure 9 Norwegian culvert solution (Dokka et al, 2003)

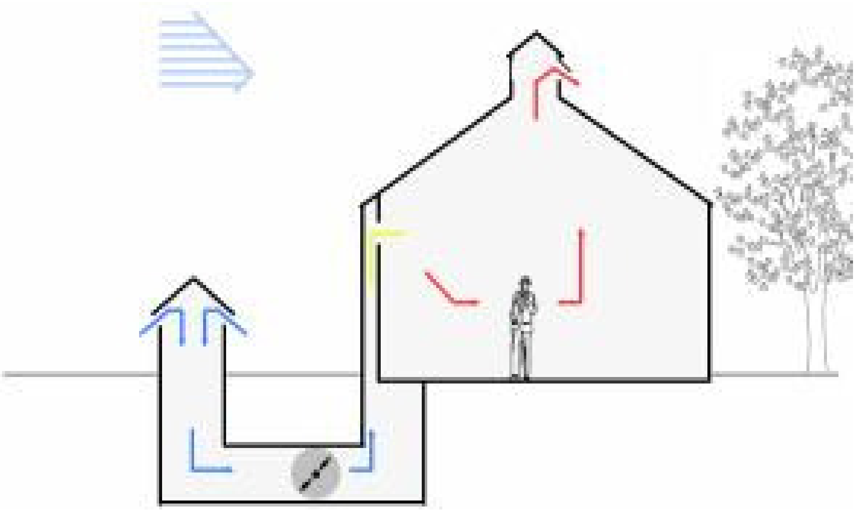

Figure 10 Swedish culvert solution (Dokka et al, 2003)

2.3 Swedish model in detail

The Swedish model of building integrated ventilation was introduced in the mid nineties with Torkel Anderson and Håkan Gilbro as central figures. Several Swedish schools have been built with this ventilation principle and according to the same Økobygg report by Dokka et al.(2003) 10 schools and 3 other buildings are built following the Swedish model in Norway.

2.3.1 Principle

The model profits from natural buoyancy forces and wind and has extremely low pressure drop throughout the system. Assisting fans are installed for hot and calm days when natural forces are not sufficient. However most parts of the year they will be. In addition windows can be opened during breaks. The intake tower is sheltered from the wind The air is treated passively through large concrete culverts were particles deposit, air is cooled in summer and moderately heated in winter. This is where the assisting fans are placed. The culvert area is so large that the pressure drop is minimal and it is accessible for cleaning and maintenance of technical installations often kept there. The air is then transported from the culvert through double walls in central parts of the building getting preheated and enters the room through inlets placed 2-2.5 m upon walls. The supply air is either distributed as horizontal jets through small inlet openings (mixing flow ventilation), or as vertical plumes flowing along the wall. The vertical plume will induce surrounding air heating it before it reaches the floor. This results in a form of active displacement flow ventilation. As explained in chapter 2. this gives better ventilation efficiency.

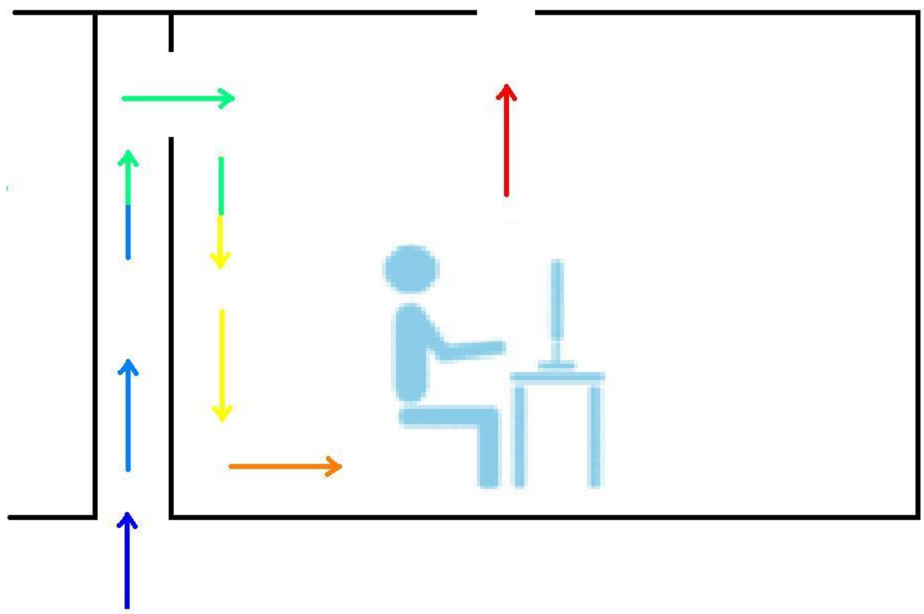

Figure 11 Air flow pattern in Swedish model

The outlet openings often consist of a "chimney" with controllable windows that also provide daylight to the room. Air flow is controlled by opening or closing outlet openings either automatically or by users (they have "valve authority"). They should always be opened on the leeward side to the wind so as to profit from it and avoid draft from wind blowing directly in through the opening (controlled by a wind vane). If automatically controlled, a principle of seasonal adapted air volumes is used with lower air volumes in wintertime, and full opening and larger air volumes in summertime. The outlet openings are then controlled by indoor temperature sensors or possibly humidity sensors. Alternatively CO_2 sensors are used, although these are more expensive, or by registering presence (not recommended when number of people varies). In addition to sustaining adequate air flows on hot and calm days, the assisting fans in the culvert makes sure the humid air from the classroom does not flow back in the culvert where there is a risk of condensation. These should be sequentially controlled after the outlet openings so that these are fully opened before fans are initiated.

2.3.2 Energy use

Heat recovery poses extra pressure loss and thereby higher electricity use for fan power and is not used. The argumentation is that the air volumes in wintertime are kept so low, that there is not much to profit from exhaust air heat recovery. There has been mixed experiences with the controllability, leading to higher heating loads than designed. However the fan power i.e. electricity need, is in many cases negligible and in schools where heating can be provided from a low exergy source, this is a major benefit.

2.3.3 Indoor environment

To avoid draft issues during winter, different strategies are used:
- Supply air placed high on wall to make sure temperature and velocity are at acceptable levels before reaching occupied zone.
- Air volumes are throttled during cold periods.
- increasing the near zone. This requires a large classroom where desks can be moved away from the inlet wall.

Experience has shown that the Swedish model achieves very good thermal comfort in summertime. Even with outdoor temperatures above 30 ºC, the indoor temperature seldom exceeds 25 ºC. The reason is the substantial cooling effect of the culvert. It cools the supply air temperature by 4-8 ºC on hot days. It also represents a large thermal mass together with for example brick walls and concrete floor. Thermal mass stabilizes the indoor temperature throughout the day by being cooled in night time and absorbing the heat in daytime. Besides, many of these schools are designed to avoid solar gain.

The controversy in this method are the low air volumes in wintertime (down to 1 air change per hour) which can at times give CO_2 concentrations up to 2000 ppm (see chapter 2.4). This is justified because the humidity content in outdoor air is very low in wintertime causing dry indoor air: Air with 100% RH at 0 ºC reduces to 25% RH when heated to 20 ºC. Keeping relative humidity levels between 40-60% has been shown to reduce the risk of irritation of skin and eyes, respiratory illnesses, allergies etc. (Gjerstad et. al, 2007 p116). It is equally important that RH levels are not too high so that no condensation forms on cold surfaces such as windows and window frames. If the temperature of these surfaces is lower than the dew point temperature of the indoor air condensation will occur. This can also be a problem in the culvert, but mostly in summertime if warm and humid outdoor air meets the cool culvert surfaces. Porous concrete together with deposited particles provide good conditions for microbiological growth. Although there have been no reports of this in any of the Norwegian schools, it has occurred in Sweden. It is possible to treat the culvert walls to make them smooth, incline and drain them, provide good cleaning routines etc.

Material choice is important, not only for thermal storage, but also for the indoor environment. Only well documented low emitting materials should be used, so that lower ventilation rates (0.7 l/s/m^2) can be used to ventilate gases and odours from materials.

2.3.4 Ventilation efficiency

The active displacement flow requires a high ceiling, but it also has greater ventilation efficiency than mixing flow ventilation. This partially compensates for the low air volumes in wintertime as long as it keeps running during break time to remove the accumulated pollution. The possibility of quickly ventilating the whole room by opening the windows should also be there (but preferably used only in summertime).

2.3.5 Control strategy

In the schools where the control of air flow is exclusively manual (opening and closing of outlet openings), indoor climate parameters vary a lot, which at times leads to poor indoor climate, and thus poor learning conditions. However once the users have a sense of control over their indoor environment, they tend to tolerate variations more. There is also a risk of over ventilation which will be an energy drain. The users' influence should therefore be limited to opening of windows.

Outdoor air quality should be high as there is only passive filtration in the culvert. The culvert should be large enough for easy access, and be cleaned approximately twice a year (especially after the pollen season).The concept is expected to have a long lifetime, as the solutions are simple and robust. It is also easy to maintain.

2.3.6 Limitations

- Not suitable in buildings where strict demands are made for air volumes and CO_2-levels.
- Depends on large internal heat loads.
- Problematic in buildings with several floors as each room will be difficult to control. (Schools are often on ground level because of accessibility.)
- Suitable only in mild coastal climates, and in the southern part of Norway. With only passive preheating of supply air the concept does not handle extremely cold temperatures, or there will be a serious draft problem.
- Difficulties proving the system will fulfil TEK/REN- requirements concerning energy use(heat recovery).
- Area requirements. Will need a larger area than conventional classrooms as there is a large near zone below the inlets.
- Will need a culvert to achieve passive cooling.
- Knowledge of the model is essential as well as early cooperation between architects and engineers. Using traditional well known solutions even though a worse alternative should not be underestimated as the knowledge, infrastructure, retailers, products etc. are easily available.

2.4 Indoor environment in classrooms

In this chapter the factors that influence the air quality and thermal comfort are presented as well as the requirements and recommendations to achieve good indoor environment.

2.4.1 Interior

The traditional form of classrooms is with the teacher in front at the blackboard, and a maximum of 28 pupils. Today's modern school requires area flexibility for group work and special needs education. As shown in figure 12, a classroom may even be partially divided to meet these requirements. This represents a challenge for the HVAC engineer. This report will concentrate on the traditional classroom form which has many similarities to a conference room.

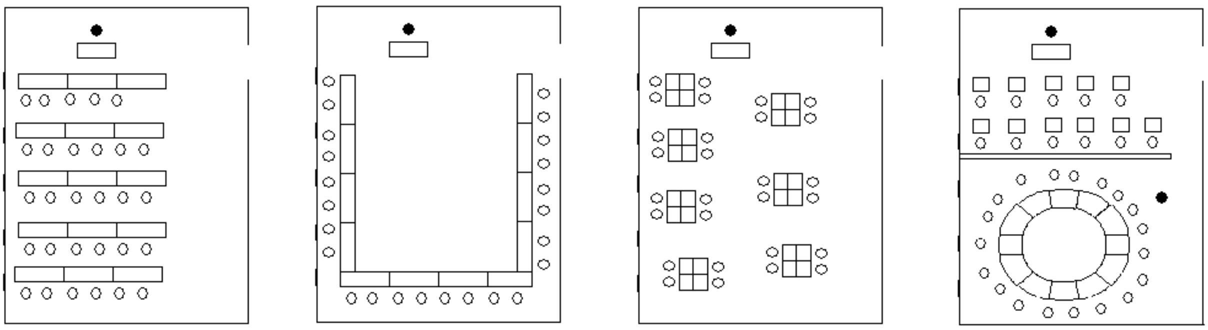

Figure 12 Interior of classrooms

The Norwegian board of Health Supervision (1998) recommends a minimum of 2-2.5 m^2 per pupil plus area for employees. This is a high intensity of people with an equal stress on the indoor climate. In addition to this, the distance from the pupil's desk to heating sources, windows and supply air inlet should be at least 80 cm.

2.4.2 Indoor environment parameters and comfort criteria

About 30% of the population have asthma and allergy related afflictions. This is particularly a problem amongst children, as half the afflicted grow out of these with age (Byggforsk, 2005). Measures to meet these needs include moisture prevention, good hygiene practices as well as filtration of pollen.

The Norwegian board of Health Supervision further states that the indoor environment norms for schools are as following5:

- Operative temperature: 19-26 °C, with no higher than 22 °C in the cold season.
- Floor temperature: 19-26 °C.
- Vertical temperature gradient: T1,1m – T0,1m < 3 °C   per vertical meter.
- Air velocity (average speed over 3 minutes): maximum 0,15 m/s in the occupied zone.
- CO2 –level: maximum 1800 mg/m3= 1000 ppm absolute level.

[5] Acoustic and actinic environment will not be treated in this report.

- Relative humidity level should be kept below 40 % in winter and 70 % in summer. Extremely dry air happens rarely in schools, and is therefore of no concern. (Byggforsk, 2005)

In addition to these norms it is generally recommended that the temperature difference from vertical surfaces should be less than 10 °C (Nilsson, p107) In summer time average air velocities for the occupied zone can be allowed up to 0,25 m/s (Ståbi, 2001 p75).

Outdoor CO2 concentrations vary from 350ppm in rural areas with no significant pollution sources to over 400 ppm in city centres. The IDA classification from EN 13779(2004) states low indoor air quality above 1000 ppm above outdoor levels, which means approximately 1400 ppm absolute value.

The predicted percentage of dissatisfied (PD) in a room in relation to the CO2 concentration is 20% at 600 ppm above outdoors (1000 ppm absolute). And vice versa; to keep the amount of dissatisfied below 40%, CO2 concentration should be kept below 2000 ppm (or approximately 2400 ppm absolute (Nilsson, 2003 p 146).

Although CO2 is far from harmful at these levels, they are a good indicator of the perceived air quality. The reason is that the concentration of CO2 varies at the same rate as other bio effluents such as body odours. CO2 is relatively easy and affordable to measure and is therefore the easiest way to measure pollution generated by humans (Nilsson, 2003 p142)).

2.4.3 Measuring points for thermal comfort

For a location where people are seated, temperature and velocity should be measured at 0,1 m, 0,6 m and 1,1 m above floor level. This corresponds to the height of a person's ankle, centre of gravity and head (Byggforsk, 1999).

2.4.4 Near zone

The near zone is often defined as the area within a distance from the inlet diffuser where the isovent is 0.2 m/s and is denoted as *L0,2* and *B0,2* for the length and width. An example of such values for a displacement flow diffuser can be found on Auranor's home page (Auranor, 2008): For supply air at 3 °C under room temperature L0,2 varies from 0,49-2,7 m and for 6°C L0,2 varies from 0,65-2,70 depending on the model dimension. Norwegian Building Research Institute(Byggforsk, 2005). recommend that the near zone is kept small as it is practically difficult to restrain the use of the area in front of the inlet. For naturally ventilated classrooms following the Swedish model, the near zone is quite extensive, meaning that the area requirements are larger.

2.4.5 Heat production

The metabolic rate of an adult at sedentary work is 1.2 met, or 70 W/m^2 body surface. The body surface for an average person is 1,77 m^2 (Nilsson, 2003 p 93)which for a 14-16 year old would be slightly less. The relative importance of the mechanisms of heat loss from the human body is estimated by Fanger (Nilsson, 2003 p 95) for 1 met, 1clo, 23 ºC, 50 % RH: 102 W distributed on 32 % convection, 37 % radiation, 2 % convection through airways and the remaining 29 % evaporation from sweat and breath. The evaporative share increases with activity level and temperature. The heat used for evaporating sweat does not contribute to room heating although increasing the enthalpy. For calculation purposes of internal heat production, only about 70 % of the heat created from a human can be taken into account. This is no exact numbers as it is dependant of many factors. Using the numbers above, would mean that the dry heat from an adult is approximately 87W which is used for calculation of internal heat generation when considering effect and energy balance of a room.

2.4.6 Indoor air pollution sources

According to Vik (2008) the Danish Institute of Building Research operates with a CO2 production from an adult being 17 times the metabolic rate. At sedentary work an adult produces 1.2 met (100 W) with a resulting CO2-production of 20,4 l/h. Furthermore a child's CO2 and heat production is 60% of an adults. This means that a child produces 70 W and 12 l CO2 per hour. However these numbers vary a lot, much because people come in all shapes and sizes and it is difficult to predict the level of activity. For example Nilsson (2003 p144) and prNS3563 (2002) operates with 19 l CO2 per hour for a schoolchild aged 14-16, at 1.2 met activity level.

2.4.7 Relative humidity and temperature

Studies have shown that there is a clear correlation between relative humidity (RH) and perceived air quality. Air at a low temperature and low relative humidity gives a lower predicted percentage dissatisfied (PPD) than air with high temperature and high RH. This seems to be the case down to RH levels at 15% at which eyes and skin are dehydrated (Nilsson et al, 2003 p 156). Other sources (Gjerstad et al.p 116) claim that Keeping RH levels between 40-60 % reduces the risk of infections, respiratory illnesses, allergies etc. An average person produces 50 g water/h through breath and evaporation from the skin (sweat)(Byggforsk,1999).

2.4.8 REN Recommended air volumes

The recommended ventilation rates as found in REN (National Office of Building technology and Administration, 2007) are based on emissions from people, materials, activities and processes. The latter is not an issue in most classrooms.

Ventilation due to personnel load:	7 l/s per person with light activity level.
Ventilation due to material emissions:	0.7 l/s/m^2 for well documented low emitting materials.
	2 l/s/m^2 for undocumented materials
	1 l/s/m^2 otherwise

This results in an air flow of 9 l/s per person with the area demand of at least 2m^2/person.

2.5 Computational fluid dynamics

In this chapter the theory behind computational fluid dynamics will be explained as the CFD program Ansys CFX is used for simulating the air flow.

2.5.1 Introduction to CFD

Computational fluid dynamics (CFD) is a computer based tool for solving fluid mechanical problems. It numerically solves equations of fluid flow, namely the equation of continuity, three momentum equations, the energy equation, a transport equation and two turbulence model equations for a fluid domain with applied boundary and initial conditions (Nilsson, 2003 p. 271). CFD software can be used to simulate temperature distributions, CO2-levels, air flow patterns, heat transfer etc in rooms by giving solutions in 2D or 3D, transient and steady state.

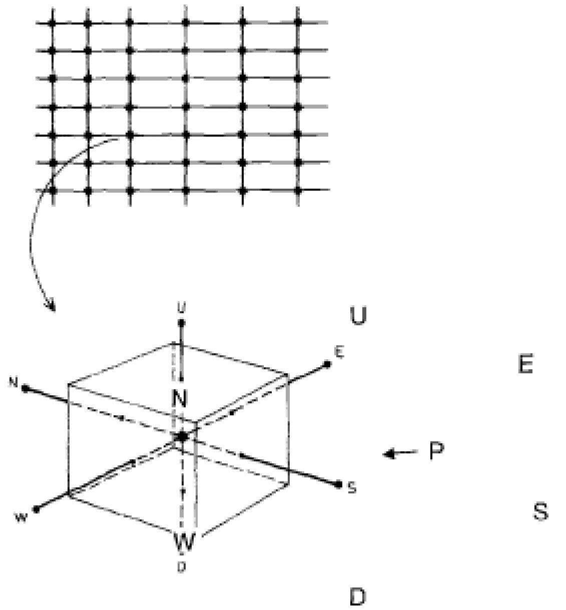

Figure 13 Grid point distribution and a control volume around a grid point P (Nilsson, 2003 p 270)

After creating a geometry the volume one wishes to analyse is divided into control volumes by creating a mesh. This is a discretisation of a continuous fluid for the computer to handle. Within each mesh is a node in a certain state which is affected by the neighbouring nodes. The above mentioned equations are resolved for each node. To find an accurate solution, thousands of iterations are required depending of the size of the mesh. Even with high speed computers a simulation can take many hours, even days and still only result in an approximate solution. The technology is however constantly improving.

2.5.2 RANS/Modelling of Turbulent flow

Turbulence modelling is very complex as turbulent movement consists of both eddies of different length scales and very brief fluctuations. So-called *Direct Numerical Simulation* (DNS) resolves this very accurately. However as this requires almost infinitesimal time steps, the computation time is too long for most practical cases. Therefore turbulence models are used. The ANSYS CFX (2007a) help states the following on this matter:

"In general, turbulence models seek to modify the original unsteady Navier-Stokes equations by the introduction of averaged and fluctuating quantities to produce the Reynolds Averaged Navier-Stokes (RANS) equations. These equations represent the mean flow quantities only, while modeling turbulence effects without a need for the resolution of the turbulent fluctuations. All scales of the turbulence field are being modeled. Turbulence models based on the RANS equations are known as Statistical Turbulence Models due to the statistical averaging procedure employed to obtain the equations."

Although the theory is too extensive to be summarized here, Reynolds' decomposition is the basic tool for deriving the RANS equations from Navier-Stokes equations. This consists of decomposing the turbulent flow, for example the velocity of a fluid, u(x,y,z,t) in an averaged component $\bar{u}(x,y,z,t)$ and a time varying fluctuating component, $u'(x,y,z,t)$ so that $u(x,y,z,t)=\bar{u}(x,y,z,t)+u'(x,y,z,t)$

2.5.3 K-epsilon turbulence model

ANSYS CFX offers a *Best Practices Guide for HVAC(2007b)* which recommends use of a K-ε turbulence model. The *k* [m/s2] stands for turbulence kinetic energy which is the variance of the fluctuations in velocity. The ε [m^2/s^3] stands for turbulence eddy dissipation which means the rate at which the fluctuations dissipate. Its greatest weakness lies in the fact that with low velocities, it is difficult to achieve convergence (Schild, 1997).

2.5.4 y+ value

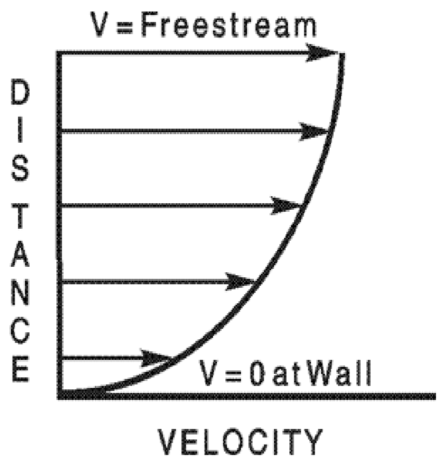

Figure 14 Fluid flow over no-slip wall (CFD online, 2007)

The mesh resolution is extremely important as this determines whether the solution is realistic. To avoid too many nodes, and save on simulation time, the mesh can be refined in certain areas of the model. Creating an inflation layer yields very fine mesh length scales next to the surface. To determine if the mesh is fine enough, the y+ value should be examined. An example is flow over a wall with a no-slip boundary condition, where the velocity gradient is large: from zero to free stream at a short distance from the wall. If the mesh is not fine enough next to the wall, one will not achieve the characteristic boundary layer profile of flow over a plane as seen on the figure 14. The y+ value should be smaller or equal to 11 as this indicates that at least one node is in the laminar zone (ANSYS CFX , 2007c).

$$y^+ \equiv \frac{u_* y}{\nu} \qquad (2.16)$$

Where u* is the friction velocity at the nearest wall, y is the distance to the nearest wall and ν is the kinematic viscosity of the fluid.

Previously, to save on computational time, it was common to use wall functions in CFD simulations providing near wall treatment without accurately resolving the boundary layer.

This required great care in selecting mesh length scales that gave a y+ value that was neither too big nor small. ANSYS CFX has developed an automatic wall treatment which allows a switch between wall functions and a fine mesh resolution (low-Reynolds number grid) (ANSYS CFX , 2007d) .

3 Method

3.1 Development of model and application of theory

In this chapter the description of the type of classroom ventilation chosen is described and empirical calculations based on chapter 2.1 (theory) are applied to assist an optimal dimensioning of a model to be used in the CFD simulations. A similar work is performed by a master student Halla (2007) with the objective of analysing a typical Swedish naturally ventilated classroom for a winter scenario. Much of the same method is therefore applied here.

3.1.1 Adaptation of Swedish model to empirical expressions

The inspiration for the type of classroom ventilation, originates from the Swedish model described in chapter 2.4 The principle is shown on figure 24 where the air is supplied high on the wall marked *A*. This is the ventilation used winter time. As the outdoor temperature rises, the outlet opening is opened further (can be opened on both sides) which allows fresh air to enter also here, marked *B*. In warm weather, the windows can be opened, and this ventilation mode is marked *C*. If outdoor temperatures surpass the comfort limit and cooling is required, the only ventilation mode is again *A*. In which case the culvert provides the cooling and the assisting supply air fan is taken into use. In the CFD simulations to be performed the focus will be on the classroom itself, not concerning (upstream or downstream) how air is supplied or exhausted. There will be no simulations for summer temperatures where windows have to be opened. To be able to use the theoretical expressions concerning air plumes, some simplifications of the air supply need to be done.

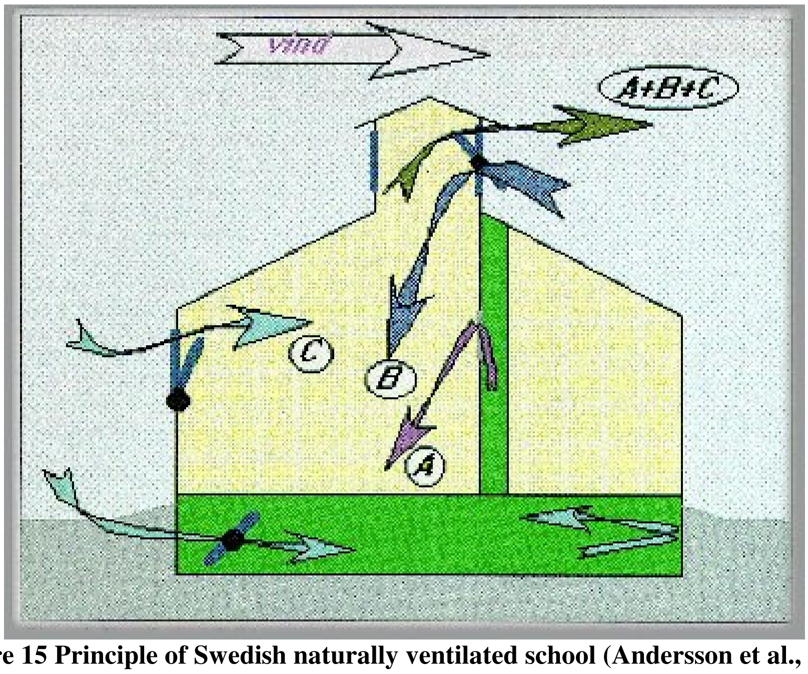

Figure 15 Principle of Swedish naturally ventilated school (Andersson et al., 1996)

There are two ways of making approximations of the flow from a highly placed inlet vent: Assuming an upside down convection plume from a cold source and a vertical plume with buoyancy originating in the ceiling. In both cases the plume originates in the ceiling rather than the wall. These are shown in figure 16 and 17 and will hereby be referred to as the vertical plume and convection plume model. In both cases the plume will go alongside the wall. The implication of the air flow in wall plume as opposed to a free plume is also described in chapter 2.1.

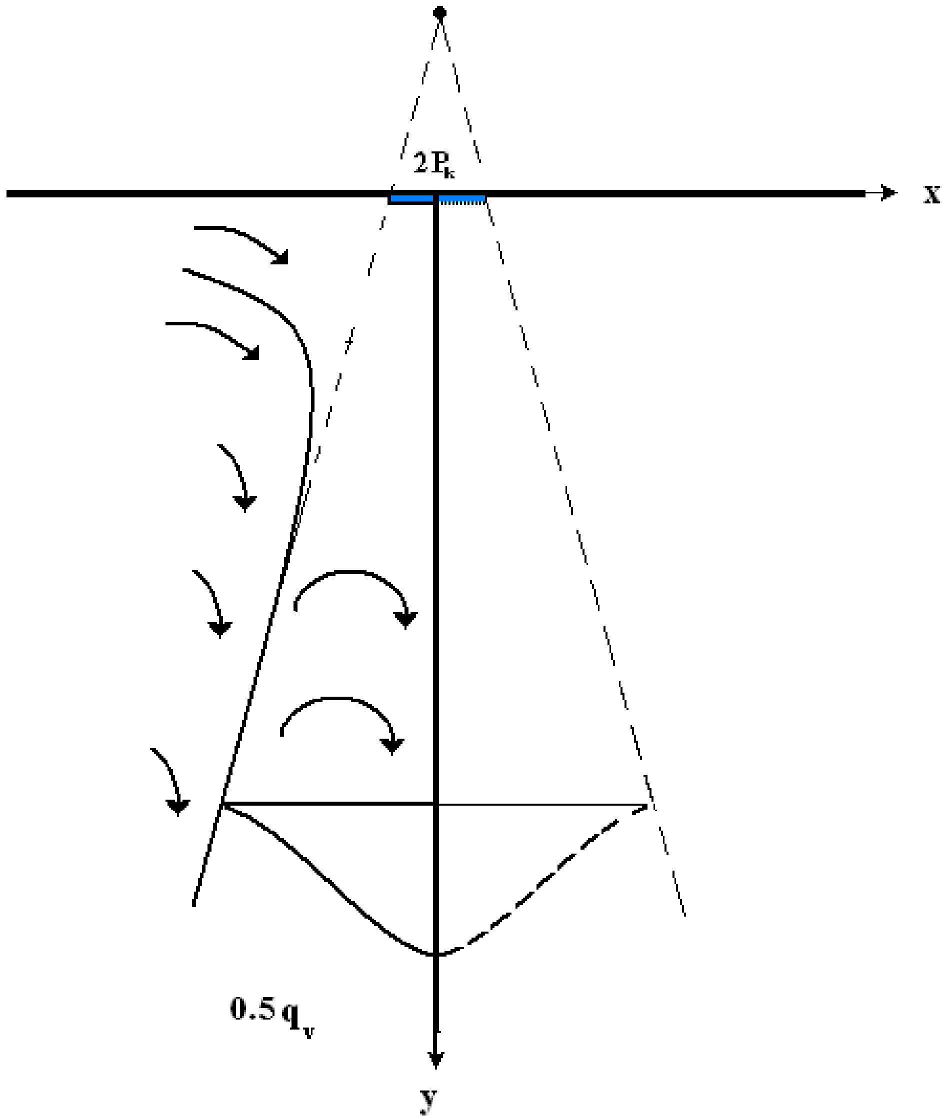

Figure 16 Inlet modelled as convection plume from cold source in ceiling

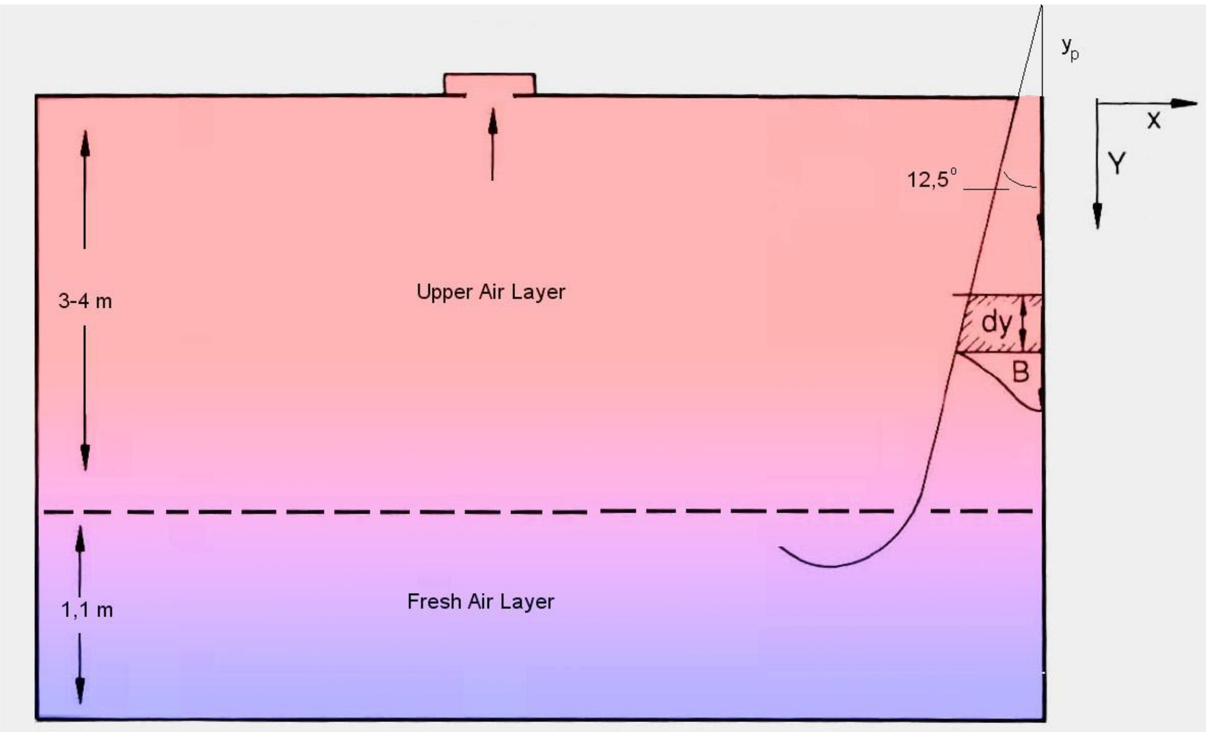

Figure 17 Inlet modelled as vertical plume with buoyancy

The vertical plume is simply a vertical air jet with lower temperature than that of the room it enters. For the convection plume, an imaginary cold convection source is placed in the ceiling, so that it is exactly the same as a convection plume driven by a hot source turned upside down. The heat removed by the cold "source", P_k, is equal to the heat removed with the same amount of under temperate air as by the vertical plume.

$$P_k = qc_p\rho\Delta T \qquad (3.1)$$

(not to be mistaken for eq 2.2)

ΔT is the temperature difference of the room temperature and the plume initial temperature.

Here too one uses the symmetry plane and assumes that the wall plume air flow is approximately the half of a free plume with twice the convection source. The fresh air supply is imagined placed below the fresh air layer i.e. directly in the occupied zone (although this is irrelevant for other than CO_2 calculations). The central velocity, central temperature and air flow of the plumes will be calculated for different supply air temperatures. The details of the calculations can be found in a Matlab script in appendix 1. In chapter3.4 the behaviour of the two plume models are presented. To avoid tedious repetition, they are presented alongside the equivalent plume models from CFD simulations. Figure 18 shows the velocity profile of a plume using eq. (2.7). This will be similar for both plume models.

Assumptions:

-Indoor temperature 20 ºC
-Inlet temperature10ºC
-Cp = 1.005 kJ/kgK
-Rho=1.205 kJ/kgK
-Radius=0.1 m
-Uo= 0.2 m/s
-Infinite space
-No vertical temperature
difference

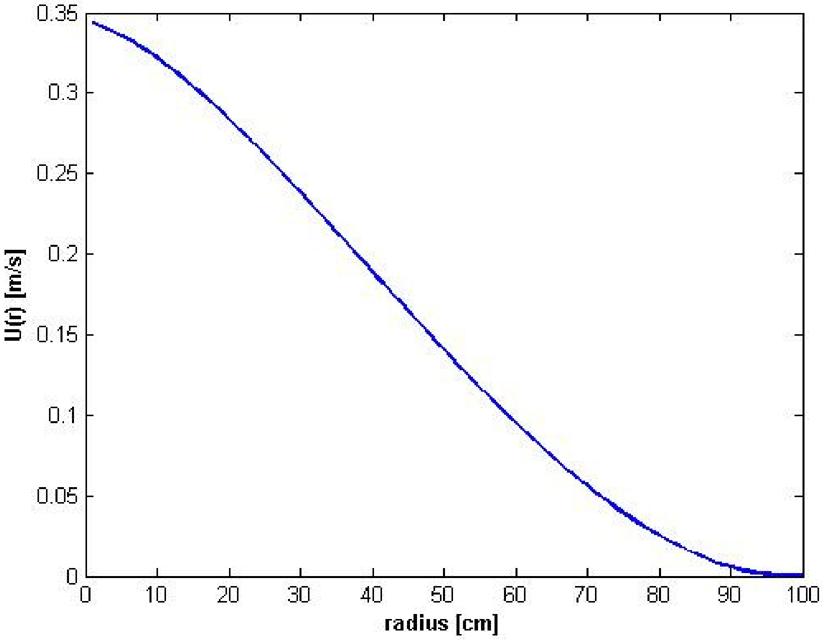

Figure 18 Velocity profile of plume

3.2 Comparison of empirical plume models and CFD simulation

For design purposes, it is not realistic or practical to always use CFD simulations. Often enough, empirical expressions can give a good enough answer. In this report, comparison of empirical and CFD simulations is important in order to validate the empirical methods.

Before using the empirical expressions for dimensioning i.e the vertical and convection plume models, they should be compared to the correspondent CFD simulations. This is to strengthen the validity of the dimensioning, and help predict problem areas, and what needs to be taken into account. A model of both the convection plume and the vertical plume was therefore made in Ansys CFX. It was also interesting to examine both a free plume and a wall plume. For all cases, the surrounding temperature is constant at 20 ºC, and the inlet radius is 0,1 m. The CFD model is a 4x4x4m room with adiabatic, no-slip boundary condition walls and a k-ε turbulence model. The supply air temperature is 10 ºC for the vertical plume, and a cold source equivalent for this for the convection plume. The floor is slightly heated to keep the energy balance of the room and thus the surrounding air at a steady state level of 20 ºC as the calculations assume a constant surrounding temperature. Overall the CFD results close to the floor will not correspond to theory as the plume will bend away from centre.

3.2.1 Comparison of wall plume and free plume

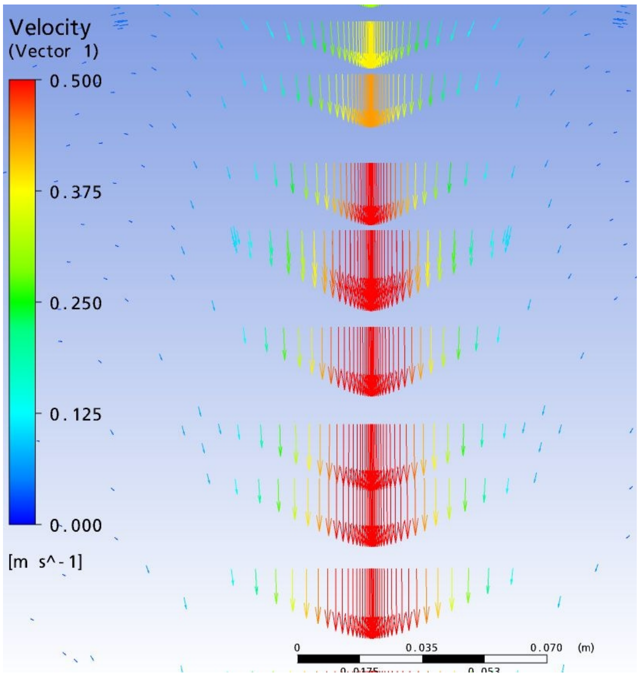

Figure 19 Velocity profile of a free plume

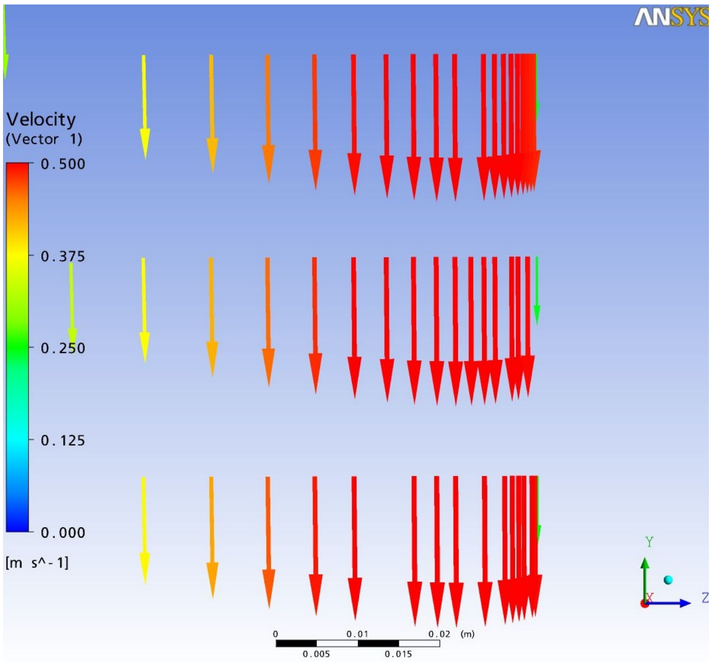

Figure 20 Velocity profile of a wall plume

The empirical expressions assume that the volume flow of a wall plume can be treated as the half of a free plume with the double convection source or inlet area (See chapter 3.1 and 2.1.3). Figure 19 shows the velocity profile of a free plume. In reality, a wall will decelerate the velocity. A zoom of the wall plume (figure 20) shows that the velocity profile of the wall plume is not simply half of the free plume. Because of the no-slip wall boundary condition, the velocity next to the wall is smaller.

3.2.2 Isotherm Plume

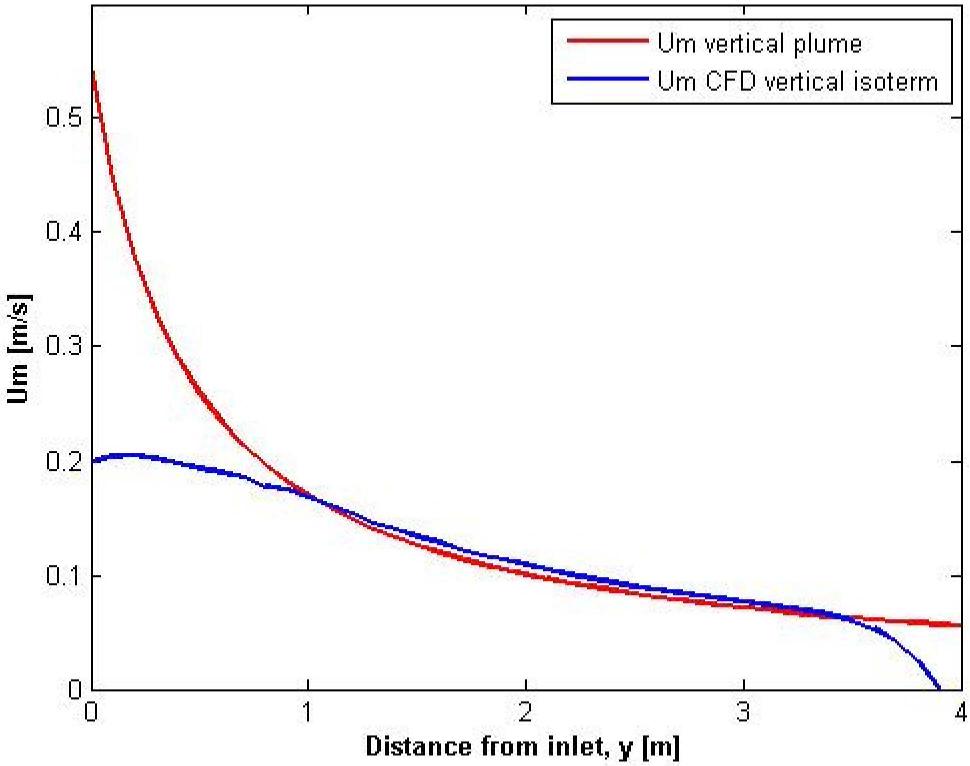

Figure 21 Central velocity of isotherm vertical plume

Figure 21 shows the central velocity, *Um*, of a vertical plume where air is supplied at 20 ºC to a room at 20 ºC. The inlet air velocity is 0,2 m/s, which clearly indicates that the left part of the empirical graph is not realistic. As mentioned in chapter 2.1.3, the expressions are not valid for y≤ 6-10 diameters e.g. 0,6-1 m from the inlet. The abrupt fall of the blue curve at the right, is due to the constraints of the CFD model room whereas the theoretical expression is valid for an infinite space. The maximum velocity is bent away from the centre. However, were it not for the floor, it might seem as though the CFD plume velocity would not decrease as quickly as the theoretical plume.

3.2.3 Central velocity of plume models

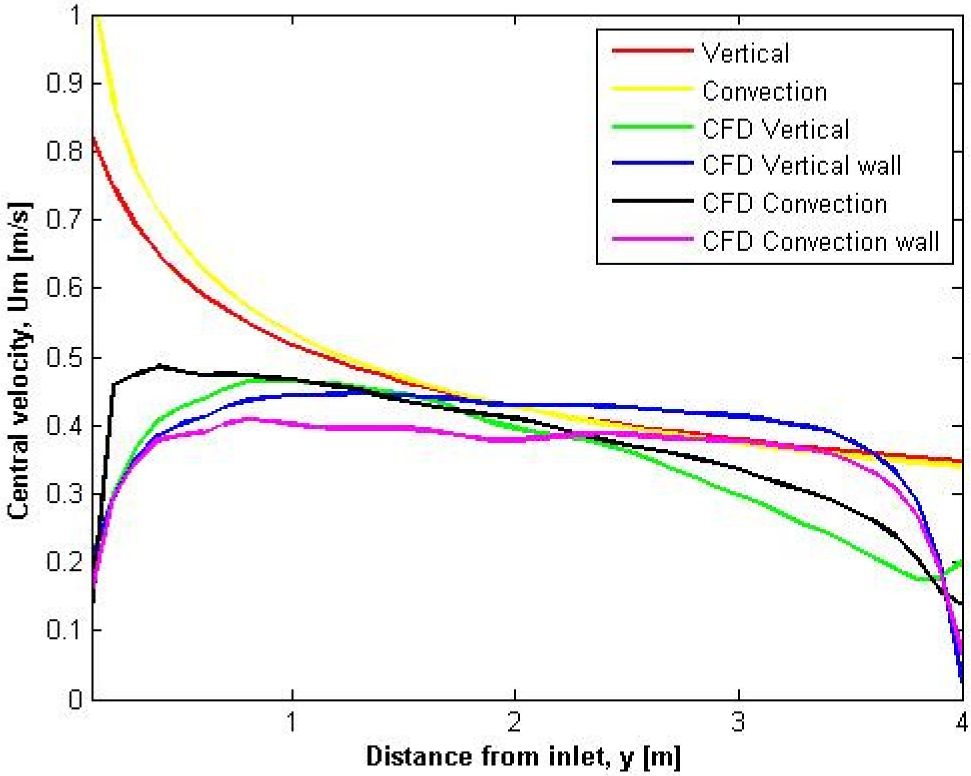

Figure 22 Central velocity of plumes

Figure 22 shows the central velocity, *Um*, of the theoretical, and simulated plumes (denoted CFD). The simulated velocities are samples from the central axis for the free plumes. For a wall plume, with a no-slip wall condition, the sample axis is placed 1,5 cm from the wall. Therefore the results of the wall plume are more uncertain. As was the case for the isotherm plume, the area of interest is also here at 0, 6 m ≤y ≤ 3,5 m. There is a great similarity between the two theoretical central velocities, where the convection curve in yellow is slightly steeper than the vertical curve in red. Also for the CFD simulations this is the case. One explanation might be that the temperature difference is much greater for the convection plume which causes an initial large acceleration just after the convection source (inlet) where the velocity is zero. This will induce a lot of the surrounding air which in turn slows the maximum velocity shortly after.

The maximum velocity of the wall plumes is more constant than the free plumes until they approach the floor where they quickly decelerate. The velocity at about 1m above the floor i.e. 3 m from the inlet, indicates which draft issues there might be. Here, the CFD wall plumes and empirical velocity correspond with a deviation of less than 0,05 m/s.

3.2.4 Central temperature of plume models

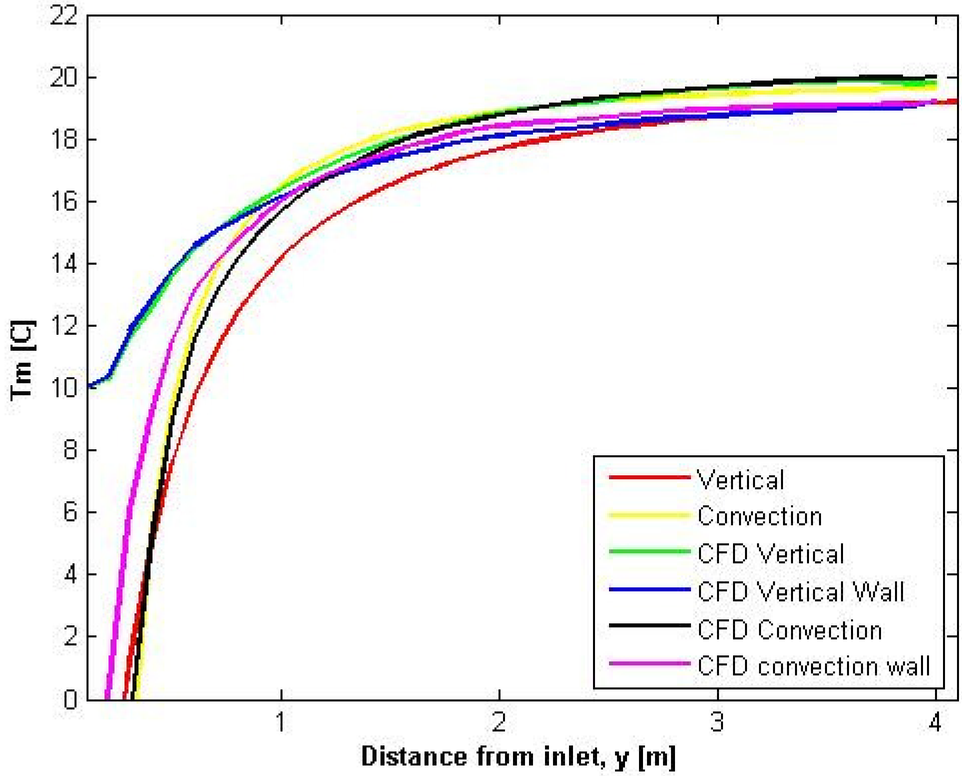

Figure 23 Central Temperature of plumes

Figure 23 shows the central temperature, *Tm*, of the various plumes. The theoretical convection model indicates a very quick assimilation to the room temperature. The other plumes have lower temperatures as they move down into the room, with the wall plumes at the lowest. They do not induce as much air as the free plumes which make the heat transfer slower. As the inlet temperature is 10 ºC, there is no reason why *Tm* at any point should be lower than that. As for the theoretical expressions, this is already explained. For the CFD convection plume the incredibly low temperature just after the inlet is because a very large amount of heat is removed from the air at a very small area. At 3m from the inlet, which is approximately where the occupied zone is, the temperature has reached 18.5 ºC. This may cause draft problems. It is important to note that the CFD temperatures are slightly lower than the empirical.

3.2.5 Air flow of plume models

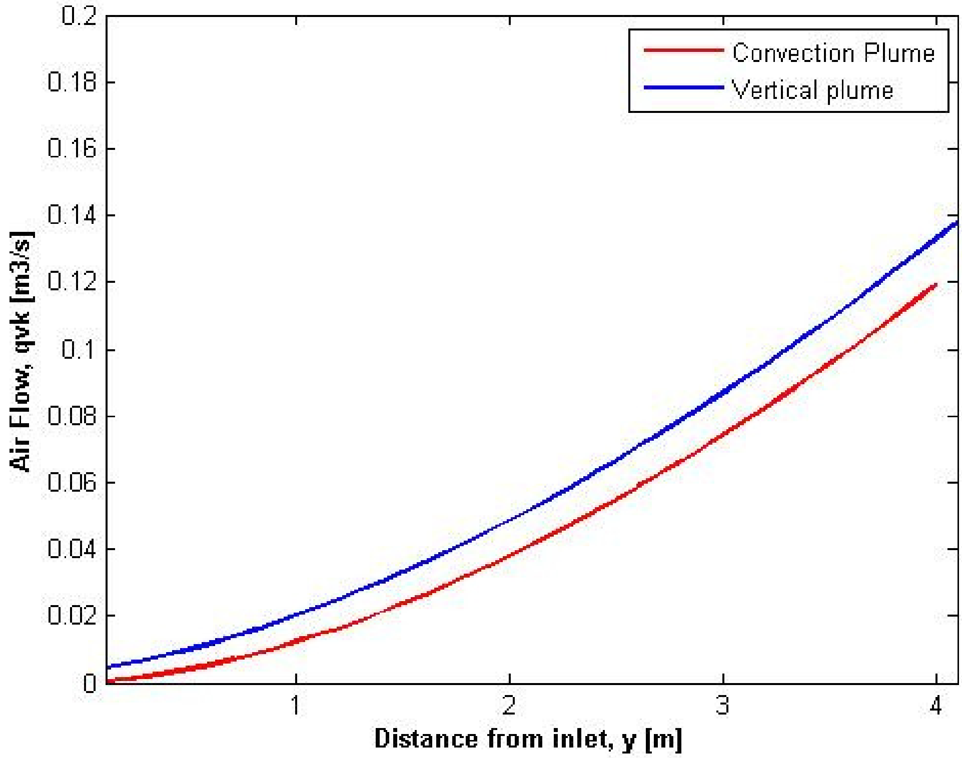

Figure 24 Air flow of plumes

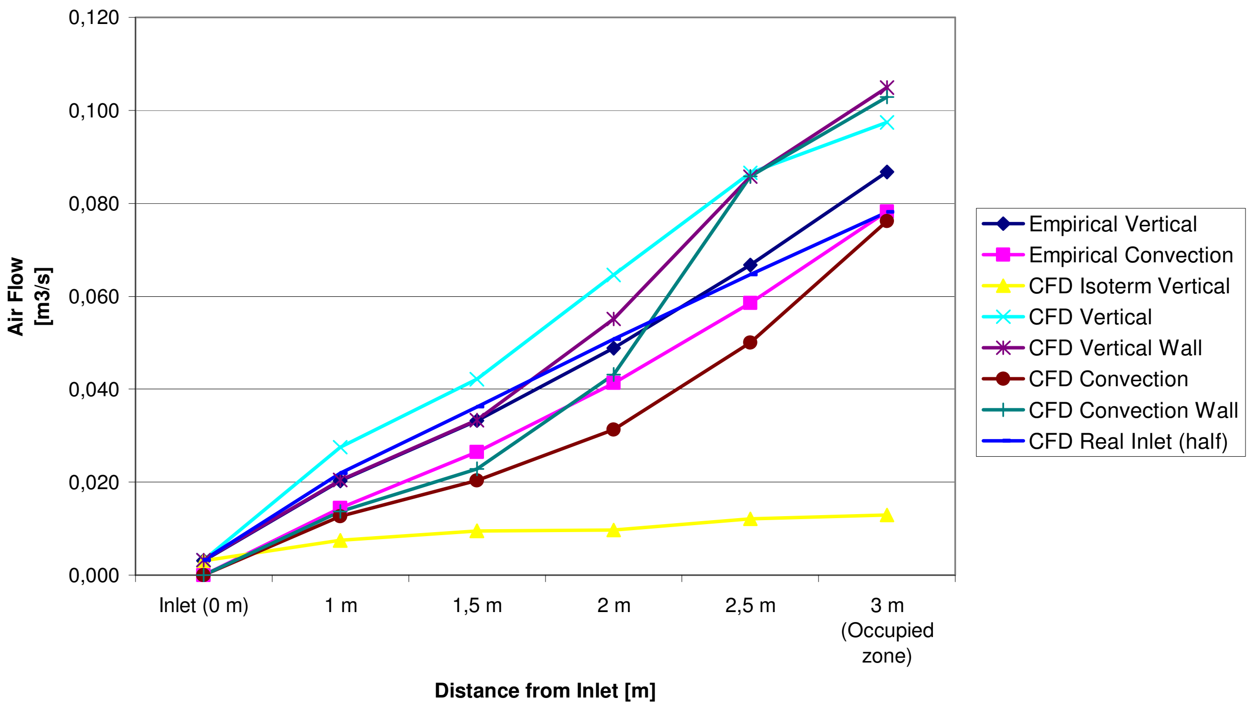

Figure 25 Air flow through cross sections at chosen heights

Figure 24 is the plot of the air flow for the two models varying with the distance *y* from the inlet. The air flow is half of that of a free plume with a double inlet area. As the plume moves into the room, surrounding air is induced thus increasing the air flow.

Figure 25 shows the air flow through cross sections at different heights. The method of finding this in Ansys CFX is to create a sample plane and ask for the air flow through it. This is a point of great uncertainty as the velocity outside the plume is not zero as in the theoretical expressions. The CFD plumes have a greater reach of influencing the surrounding air, although the total air volumes might not be larger. The area of the cross section sample was chosen to have approximately the same width, *b*, as the theoretical plume would have. The size of the sample plane had greater influence close to the floor. Therefore any conclusions should be drawn with care. For the free plume, the only the half volume flow is plotted. The yellow line with the lowest values represents the isotherm vertical flow. This is of course much lower, as there is no temperature gradient to drive the plume. This shows the importance of under temperate air, lest the inlet air does not even reach the occupied zone. It seems as though the vertical plumes, both theoretical and CFD, have a larger air flow than the convection plumes; although the convection wall plume is an exception when approaching the occupied zone. All the CFD lines have a similar path as in Figure 24.

Both models give a similar picture of the plume behaviour. As the CFD air flows are within a confined space, this will obviously affect the air flow pattern.

3.2.6 Impacts on the air flow that the theory does not account for

In the empirical calculations from chapter 2.1, the vertical temperature difference is ignored. In reality this is not the case: With a vertical temperature difference of 2 K/m the buoyancy driven air flow rate caused by a seated person 1,5 m above floor level is reduced from 150 m^3/h at 0 K/m to less than $50m^3/h$, which shows what impact this has on a convection plume and probably also the vertical plume. In any case the theoretical approach results in an even larger air flow rate (Nilsson, 2003 p229).

The influence of radiation on the air flow is not so apparent, but nevertheless there. As mentioned, there is a vertical temperature difference in a room with displacement ventilation. Therefore the ceiling is heated by convection, and at the same time emits heat by radiation to the floor. This means that the floor is warmer than the air supply layer, causing this to rise. So in fact, the upward pull is not only due to the heat generated by people. However, the theoretical upward pull, as mentioned above, is greater than the actual.

3.3 Dimensioning of air flow

In this chapter a study is made of the impact the air flow has on heat balance/temperature, the carbon dioxide level and the relative humidity levels. These calculations will serve as input for the CFD simulations. The simulations will then show whether or not the dimensioning based on the empirical theory is reasonable/proves valid. Oslo is used as a reference climate.

3.3.1 Ventilation for temperature regulation

In rooms with high occupancy, removal of heat is an important task of the ventilation system. A heat balance is useful for finding out the necessary air volumes to remove heat surplus.

The heat loss due to ventilation can be expressed as:

$$P_v = q\rho c_p \Delta T \quad [\text{W}] \qquad (3.2)$$

The heat loss due to infiltration can be expressed as:

$$P_i = nV\rho c_p \Delta T \quad [\text{W}] \qquad (3.3)$$

The heat loss due to transmission of heat through exterior walls can be expressed as:

$$P_t = \sum UA\Delta T \quad [\text{W}] \qquad (3.4)$$

Where
q - Air flow [m^3/s]
n - Air change rate [h^{-1}]
ρ - Density of air [kg/m^3]
c_p - Heat capacity of air [k/kgK]
ΔT - Indoor design temperature- Outdoor temperature [K]

(Nilsson, 2003 p321)

The total effect balance of the building is thus:

$$Ph + Pp + Pl = Pv + Pi + Pt$$

P_p- internal generation of heat due to people

P_l- internal generation of heat due to lighting
P_h- Heating required to keep temperature at lowest acceptable level 20ºC.

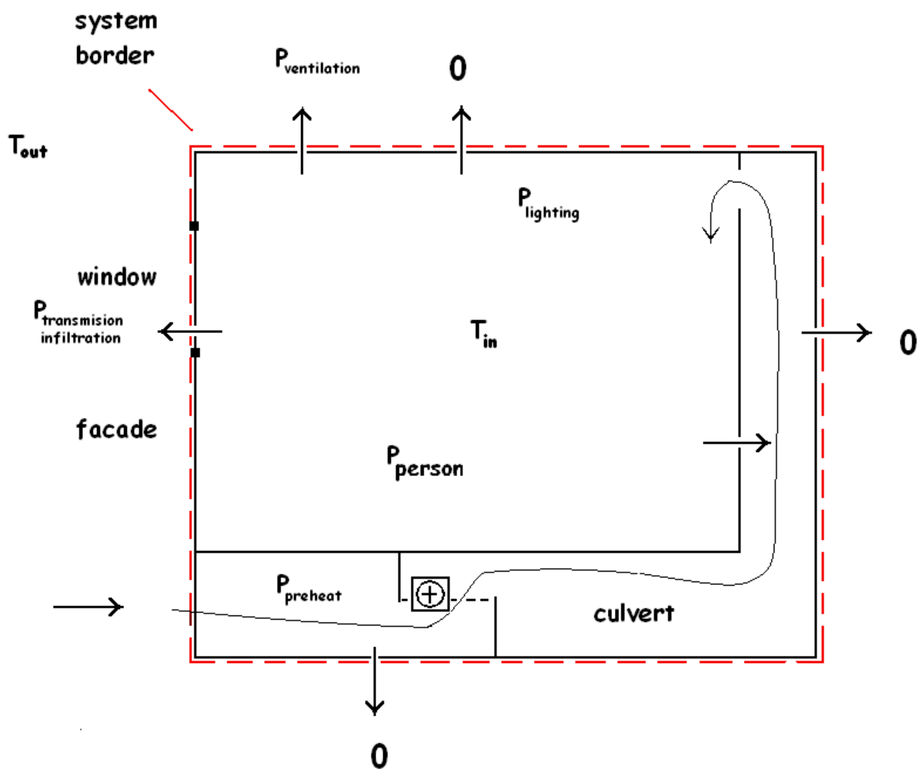

Figure 26 Heat balance of classroom (cross section)

Furthermore the heat required, P_h, to compensate for the cooling effect of supply air and heat loss through building envelope can be divided in space heating and preheating of supply air. The supply air should be preheated to approximately 0ºC. Some of the space heating will also contribute to preheating the supply air through the floor and walls to the supply air culvert and channel so that the supply air is in fact higher when it enters the room. This does not affect the heat balance, but improves the thermal quality as the risk of draft decreases.

Assumptions:
- Steady state/momentary (effect)
- All other surfaces than façade facing similar zone (adiabatic)
- Thermal bridges ignored as transmission loss is not the main focus here
- Infiltration losses neglected as only one façade faces external.
- Solar radiation heat gain and other equipment not included
- No heat recovery (depends on system solution and will be discussed later)
- Electricity for fan power not included
- Assuming heat production of 14-16 year olds, 80W dry heat per person. (see chapter 2.4)
- Heat emitted from lighting is 10 W/m^2 floor area (NS3031, 2007)
- Recommended air flow: 100 % REN[6] = 7l/s/person and 1 l/s/m^2 floor area (for comparison)
- Thermal mass neglected.

[6] Most Schools built after the Swedish model have strict demands on using low emitting materials which means the area specific ventilation can be reduced to 0,7 l/m^2.

Table 1 Input values for heat balance calculations

Calculation input	Value	Unit
Indoor Temperature, Ti	20	ºC
Floor area	70	m^2
Volume=7*10*4	280	m^3
Façade =10m*4m	29,5	m^2
Window (15 % of floor area)	10,5	m^2
U value	0,2	W/m2K
U value window	1	W/m2K
Infiltration, n	0	air change/h
Heat capacity air, cp	1,005	kW/kgK
Density air , rho	1,205	kg/m^3
Recommended air volumes/m2	**0,001**	**$m^3/s/m^2$**
Recommended air volumes /person	**0,0093**	**m^3/s/person**
Heat from lighting , Pl	0,7	kW
Heat emission per person , Pp	**0,080**	kW

Table 2 Air flow used in calculations

Air flow: l/s/person			
Percentage of REN	Number of people		
	30	20	10
100 %	**9,3**	**10,5**	**14**
80 %	**7,5**	**8,4**	**11,2**
60 %	**5,6**	**6,3**	**8,4**
40 %	**3,7**	**4,2**	**5,6**
20 %	**1,9**	**2,1**	**2,8**

Note that these calculations are simplified and do not apply for any real case. They merely serve as an example of how such an effect balance evaluation can be made.

The following figures portray a potential heat balance in order to evaluate the heat gains and losses of a classroom. The full details of these spreadsheets, as well as for other typical Oslo temperatures and can be found in a table in Appendix 2.

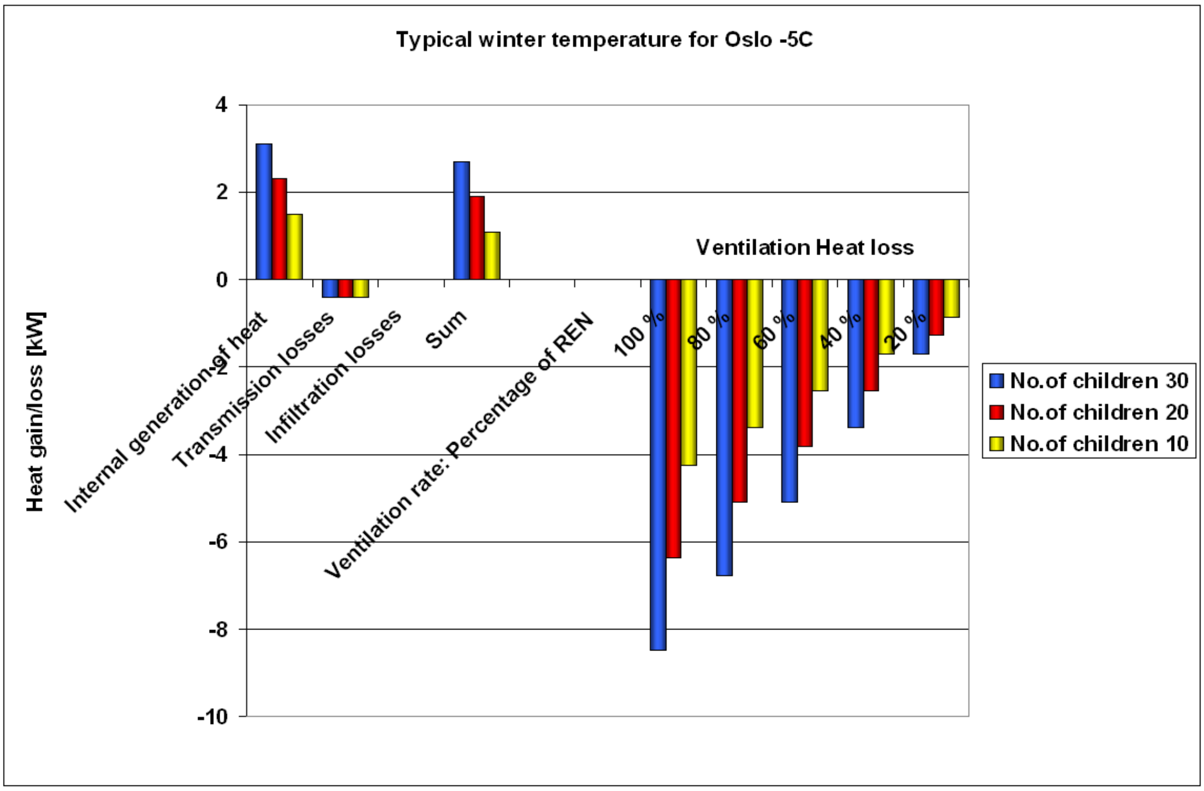

Figure 27 Heat Balance of Classroom at -5°C outdoor temperature

In Figure 27 the heat gain are the positive columns and the heat loss the negative. The different coloured columns show the heat gain from 10, 20 or 30 children in the classroom. The transmission losses are independent of these and are of a small order of magnitude as only one wall faces the external. The infiltration losses are in this case neglected. The sum is simply the heat balance excluded ventilation heat losses. To the right are the ventilation heat losses for various percentages of recommended air volumes. It shows the magnitude and importance of using the appropriate air volumes as a great amount of heat will be lost with excessive air volumes. By comparing the sum of the internal heat generation and transmission losses to the ventilation heat losses one can determine that air volumes between 20 and 40 % will give a heat balance of zero and no space heating will be needed.

In figure 28 a similar balance is shown at a warmer exterior. In comparison the ventilation heat losses are very much inferior to those in figure 27 and the transmission loss almost negligible. In this scenario about 60-80 % of recommended air volumes should be used in order to achieve a heat balance and 20 ºC. Any lower and the room temperature will rise, any higher and the room temperature will sink. This shows that it is possible to achieve heat balance at relatively low temperatures in a well insulated room with a high internal heat production.

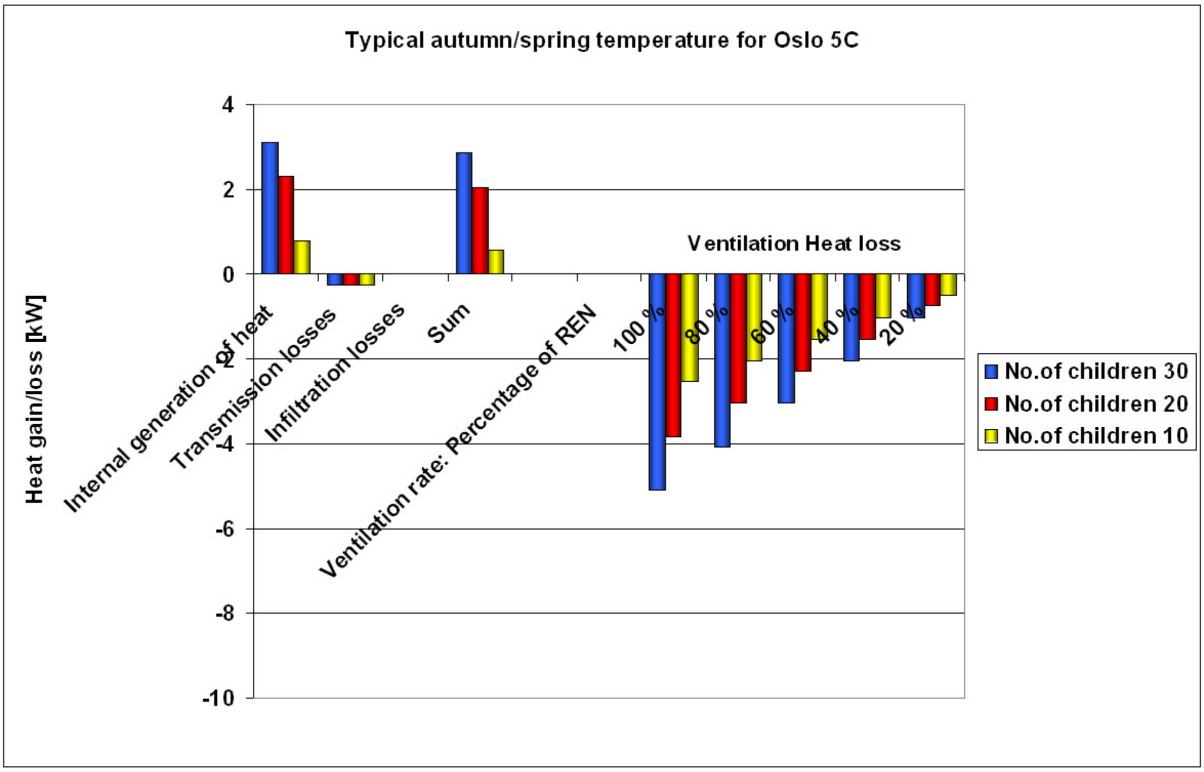

Figure 28 Heat Balance of a classroom at 5ºC outdoor temperature

The number of children might vary a lot. Hardly ever will there be 30 people in a classroom, and most days there is someone absent. The transmission losses which are independent of the number of people are relatively small. Although the internal heat generation of heat is greatly reduced with fewer people, so are the air volumes and the resulting ventilation heat losses. The area specific ventilation rate stays constant regardless of the occupancy in these calculations for comparative purposes, which works in disfavour of the heat balance.

Figure 29 and 30 show the heat balance with 15ºC outdoor temperature and respectively 20 ºC and 25 ºC designed indoor temperature for different air volumes classified as percentages of recommended air volumes. A positive heat balance indicates that there is a heat surplus, and a negative that space heating will be necessary.

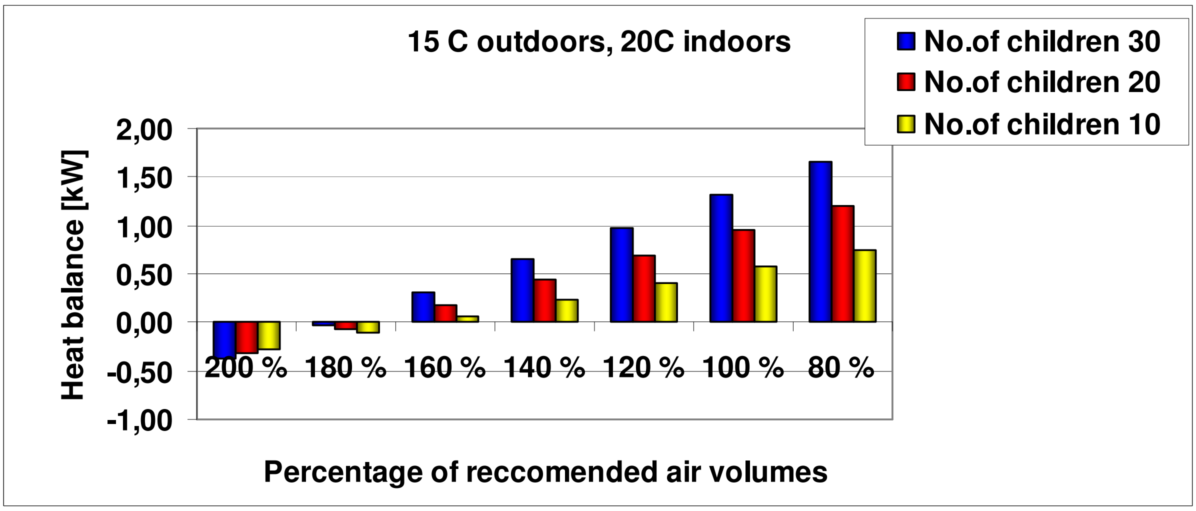

Figure 29 Heat balance at 15ºC outdoors, 20ºC indoors

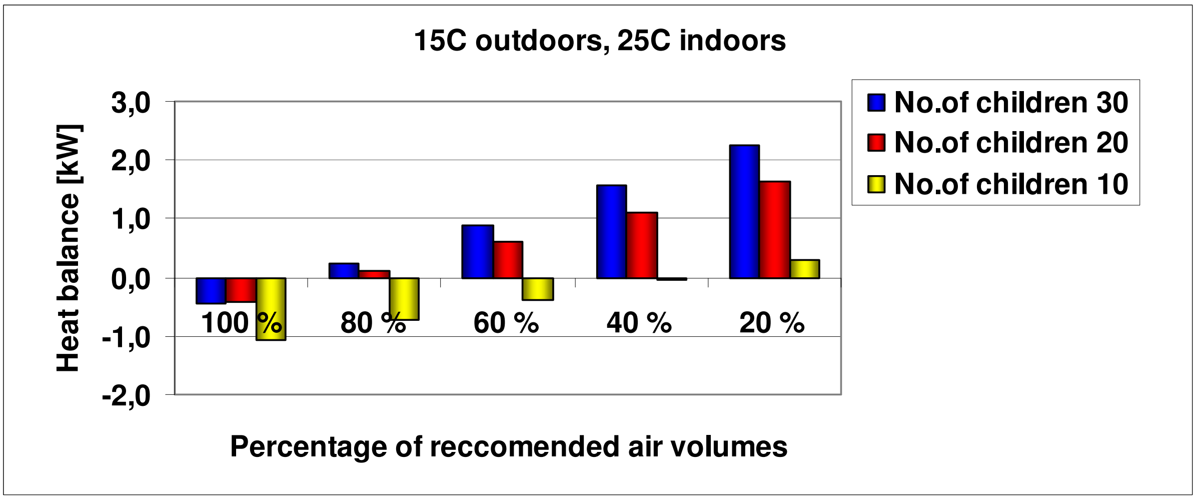

Figure 30 Heat balance at 15ºC outdoors, 25ºC indoors

In a summer situation, the situation is rather reversed where removal of heat surplus is dimensioning for the necessary air volumes. At 15 ºC, which is the average temperature for June and August in Oslo, the air volumes necessary to keep an indoor temperature of 20 ºC have to almost double of recommended air flow (see figure 29). One might then allow temperatures up to 25-26 ºC to limit the use of fan power (see figure 30).

The practical way to make the heat balance regulate the air volumes is to use a thermostat.

3.3.2 Ventilation/air volumes for control of CO_2 concentrations

The idea in the Swedish naturally ventilated schools is that normally there will be a cooling need and therefore air volumes will be high enough to keep CO_2-levels at a low. However, in wintertime, with low inlet air temperature, it is important to find the marginal air volumes necessary to keep acceptable CO_2-levels while at the same time being energy efficient. This means that air volumes might have to be increased from those determined by the temperature, which in turn requires space heating.

With displacement flow, one often operates with the so-called two zone model (Skåret, 2000 p 110). The occupied zone has fresh air layer at 1.1 m where the breathing zone is (Ståbi, 2004 p153). This is the layer height for sitting people as in a classroom

The concentration of pollutants in each zone is considered homogeneous, so that the concentration in the outlet, C_e, is the same as in the upper zone.

$$C_{upper} = C_e = \frac{P_{CO2}}{q_{v,e}} \qquad (3.5)$$

The air flow through the outlet has a different temperature, T_e, (and density) than the supply air, T_s. Therefore $q_{v,e} = q_s \frac{T_e}{T_s}$

The CO_2 concentrations in the occupied zone, $C_{occupied}$, is the result of the mix of fresh air, $q_{v,supply}$, and the induced air flow, $q_{v,induced}$, entering the occupied zone. For calculation purposes it can be assumed that it is a convection wall plume with the fresh air supply placed at floor level.

$$C_{occupied} = \frac{q_{v,induced} C_{upper} + q_{v,\sup ply} C_{\sup ply}}{q_{v,induced} + q_{v,\sup ply}} \qquad (3.6)$$

Where $q_{v,\ induced}$ *is* the total air flow of the plume at the height of the occupied zone subtracted the supplied air. Resolving this equation for the desired concentration in the occupied zone yields the necessary supply air volumes. The desired concentration by the end of a class is debatable as mentioned in the description of the Swedish model.

The CO_2-levels themselves are not harmful but indicate the amount of body odours etc that have been produced. Outdoor levels might be about 400 ppm although this varies with an urban or rural environment(see chapter 2.4). In this case the concentration in the supply air is neglected as it is the concentration above outdoor level which is in fact what is of interest. A level of 900 ppm above outdoor level yields a predicted percentage dissatisfied of 25 %.

Having compared the buoyancy model and vertical model air flow, finding that the latter corresponds best with CFD simulations, it is used for the CO_2 calculations (see Matlab script in appendix 1). As the induced air flow is dependant on the supply air temperature, different CO_2 concentrations will occur at the different supply air temperatures.

CO_2-control is primarily important in wintertime when trying to keep air volumes at a low. The supply air needs to be heated to 0 ºC to avoid frost in the culvert and draft problems. Therefore a relevant case would be to examine 0 ºC outdoor temperature and supply air temperature to find the lowest possible air volumes.

Acceptable CO_2-levels in the Swedish model schools is a debateable issue. Although the recommended values are maximum 600-700 ppm above outdoor level (see chapter 2.4) the Swedish schools often allow higher at times.

- Vertical and convection wall plume model used for calculations of induced air.
- Radius of inlet/convection source is 0,2 m
- 7 inlets
- Preheating of ventilation air to 0ºC
- Assuming steady state will occur before end of class
- Full classroom =30 people
- Designed room temperature 20 ºC
- CO_2 concentrations above outdoor level
- 19 l CO_2/h per child

Variables:
- Inlet temperature
- Inlet velocity /air flow

Table 3 CO_2-levels above outdoor levels at 0ºC supply air temperature using vertical plume model

Ventilation rate Percentage of REN	CO2 [ppm]	Uo [m/s[	Minimum plume temperature [ºC]	Archimedes no.
100 %	372	0.773	10.3	0.39
80 %	542	0.618	11.6	0.61
60 %	649	0.464	13	1.08
40 %	999	0.309	14.6	2.42
20 %	2062	0.155	16.6	9.70
***40,29 %**	**1013**	**0.312**	**14.7**	**2.49**
	900			
* Air flow where no heating is required			Grashof no. 2.0185e8	

Ventilation rate Percentage of REN	CO2 [ppm]	Uo [m/s]	Minimum plume temperature	Archimedes no.	Grashof no
100 %	356	0.773	10.3	0.39	3.2639e9
80 %	459	0.618	12.0	0.61	2.8128e9
60 %	631	0.464	13.1	1.08	2.3219e9
40 %	979	0.309	14.6	2.42	1.7719e9
20 %	2037	0.155	16.6	9.69	1.1162e9
40,3 %	972	**0.312**	**14.6**	**2.39**	**1.7806e9**
43,30 %	900	**0.335**	**14.3**	**2.07**	**1.8673e9**

Table 4 CO_2-levels above outdoor levels at 0ºC supply air temperature using convection plume model

Table 3 and 4 show the CO_2 levels above outdoors with the varying air flow described as *Percentage of REN* (explained in chapter 3.3.1). The two different plume models described in the chapter 3.1 is used to calculate the induced air flow entering the occupied zone. The two bottom lines of table 3 and 4 show respectively the ventilation rate for which no space heating is required and the ventilation rate necessary to keep carbon dioxide at 900 ppm above outdoor level. The calculation of the CO_2 concentration varies little between the two models, although the convection plume models yield slightly lower numbers. The CO2 levels rise exponentially and there is a dramatic difference in reducing the ventilation rate from 40% to 20 % of REN recommendation. To strictly meet the recommendation of maximum 1000 ppm absolute, or 600-700 ppm above outdoors, one would have use about 60 % of REN recommendation.

At temperatures as low as 5ºC it is possible to achieve acceptable CO_2-levels without space heating. With a resulting air flow rate of 56 % of REN recommendations (5,2 l/s/person) a CO_2 level of 1265 mg/m^3 (700 ppm) above outdoor level was calculated in the occupied zone. This is little more than a 1000 ppm absolute value. Even for the case of lower temperatures it is common practice in the naturally ventilated schools to accept higher CO_2 levels to avoid the need for space heating as well as avoiding draft. In colder weather, the transmission and infiltration heat losses are not so moderate, making the choice/dilemma between acceptable

CO_2 levels and energy use more real. At higher temperatures, ventilation is purely for removal of heat surplus.

A CO_2-sensor controlled ventilation rate does not take into account emissions from materials, meaning a lightly occupied classroom might still have an olfactory load. It is therefore important that well documented materials are used.

3.3.3 Ventilation for Control of Relative Humidity

CO_2-levels set aside, decreasing the air flow to a minimum poses an additional threat in the form of high relative humidity (RH). At thermal bridges, such as window frames, the temperature can get so low that condensation forms. As moisture leads to microbiological growth, this must be avoided. The surface temperature of a window frame, T_s, can be denoted as(Gjerstad et al, 2007 p 145):

$$T_s = T_i - \frac{R}{R_{tot}}(T_i - T_o) \qquad (3.7)$$

T_i – indoor temperature, 20 ºC
T_o - outdoor temperature
R - thermal resistance Km^2/W =1/U-value of window
R_{tot} - total thermal resistance, sum of internal and external convection resistance and R.

In appendix 2 calculations of the worst case scenarios are made for an imagined window frame with a U-value of 1,5 W/m^2K.The relative humidity outdoors is assumed 100 % and the humidity ratio is found through a Molliere[7] diagram. The stationary humidity produced by people (see chapter 2.4) is added and the dew point temperature found for the various outdoor temperatures. The temperature in the window frame is similar to the dew point temperature at 20 % of recommended air volumes which gives risk of condensation. At 40 % of recommended air volumes the risk is greatly reduced as the dew point temperature is 2-4 ºC lower than the window frame surface temperature. For outdoor temperatures above 5 C air volumes should be kept above 40 % of REN recommendations.

Another aspect of humidity control is that you wish to keep it between 40-60 %RH for reasons of thermal comfort and health (see chapter 2.4). At outdoor temperatures above 5 ºC the RH levels are alarmingly high for ventilation rates below 40 % of REN recommendations. This will certainly lead to a poorly perceived air quality by the occupants.
At times of extreme cold, less than -10 ºC the RH levels for ventilation rates above 40 % of REN reccomendations will be very low. One might then tolerate higher CO_2 levels, to keep the RH-levels within a comfortable level.

It is imperative to set an absolute minimum ventilation rate independently of thermostat- or CO_2-sensor control. As found in the chapter 3.3.2, ventilation rates below 40 % of REN recommendations do not comply with keeping the carbon dioxide at adequate levels.

[7] Psycometric chart

3.4 Model in ANSYS CFX

n this chapter the development of a CFD model of a classroom is described as well as the adaptation of the calculations used for dimensioning.

3.4.1 Modelling of Children

CFD modelling of people is a difficult task because of the complexity and variation of the human being. Heat generation and CO_2-production can be estimated as described in chapter 2.4 and model geometry made with a great level of detail. Yet, real people are not rigid, and a realistic shape would be too time-consuming to make and require a complex grid. Figure 31 is the cross section of an early model of a person. It is 1,1 m tall, the height of a seated person. The characteristic shape of a buoyancy plume (See Figure 1) can be seen above. The heat of the body causes the air to rise pulling in air from below.

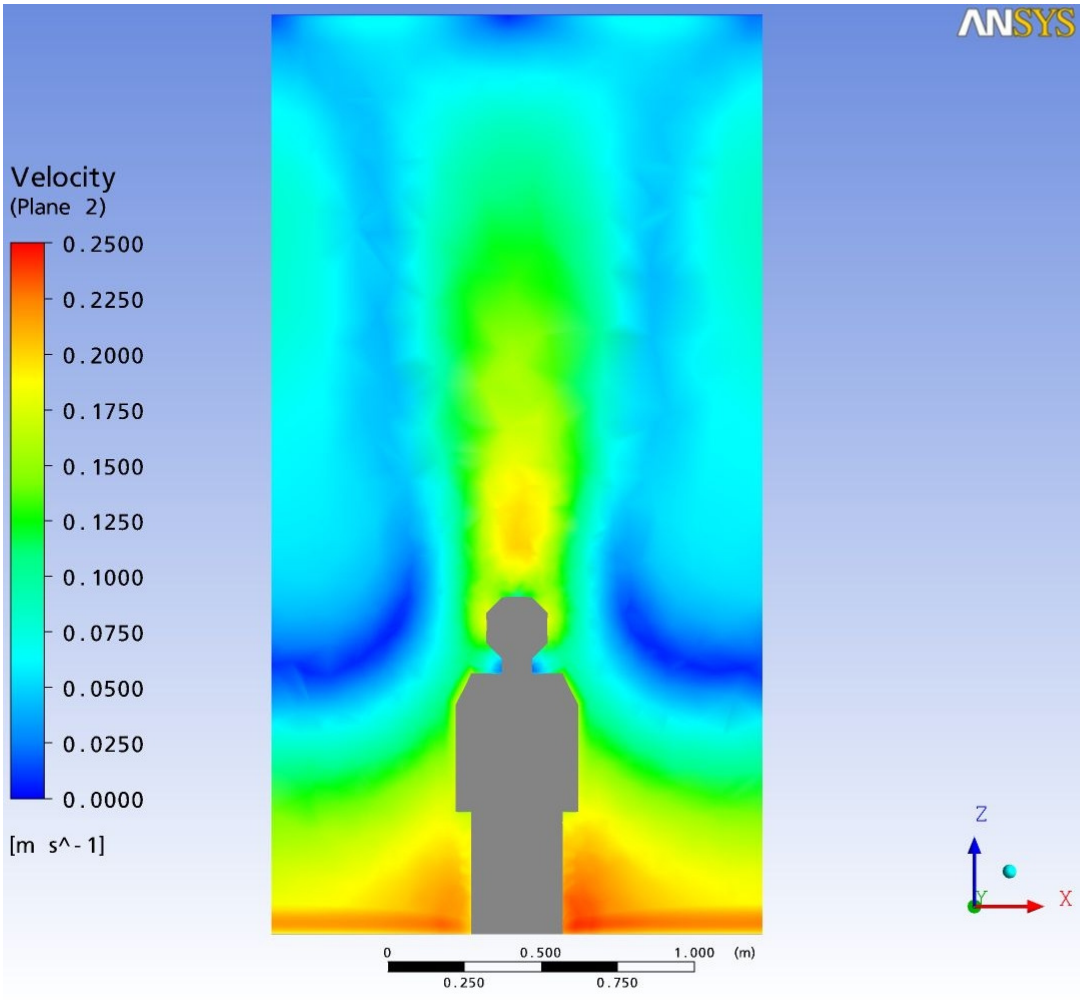

Figure 31 Cross section of plume above person

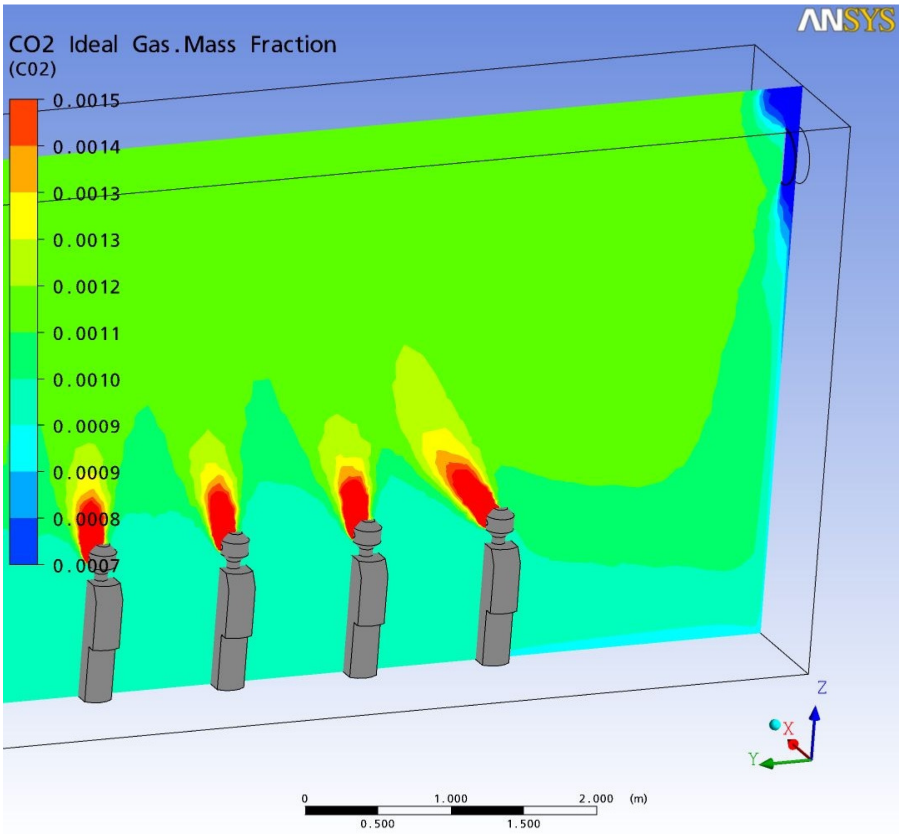

Figure 32 Sample of room with people producing CO_2

Early simulations of CO_2-production (Figure 32) show the expected layering occurs. The CO_2 rises with the warm breath and the body heat and is replaced by the cooler fresh air. Air from the upper zone mixes in with the supply air resulting in a higher CO_2 concentration. Details of this simulation will not be discussed further here, but simply note that the CO_2 modelling behaves as expected.

3.4.2 Room Height

The height is important so that the cold supply air plume has time to induce surrounding air and preheat. It is also crucial for all types of displacement flow ventilation to profit from buoyancy forces to have a higher ceiling than for displacement flow ventilation. Increasing the room volume has an effect on the heat loss. Especially when calculating the infiltration loss this has an impact. The ceiling is flat height was set at 4m above floor level.

3.4.3 Inlet: Number, Size, Shape and Placement

The ideal spacing of the inlets means that the plumes should not influence each other before reaching the occupied zone approximately 1m above the floor. They would then start inducing air from the other plume, instead of the room, not reaching the desired heating effect. As the plumes have an inlet diameter 0.2 m and expands with approximately 12,5°, the plume width at the occupied zone would be approximately 1,5 m with a 4 m ceiling height. For a classroom wall of 10 m, this means 7 inlets could fit. Each would supply air to 4-5 children.

For calculation purposes, the inlet diameter must be such that 6-10 times its length occurs before the area of interest, i.e. the occupied zone. This distance, from the inlet to the breathing zone is, 2-3 m in a classroom giving a diameter of approximately 0,3 m for which calculations will be applicable.

Of lesser importance is the velocity through the inlet opening. As long as it is lower than 1m/s it represents only an insignificant pressure drop. This is not necessarily the case in summertime when great air volumes are required to handle the cooling load. In warm periods, additional ventilation through the windows is necessary. The inlet diameter was set at 0,2 m with the shape of a semicircle placed in the ceiling like the vertical wall plume model.

3.4.4 Outlet

The outlet is placed in the ceiling, away from the inlet with the same area. The dimension of the outlet has an effect on the pressure drop, and should be considered carefully. The necessary height of the outlet channel to profit from the stack effect is not treated in this report. The only reason it is chosen so small in the CFD simulation model is because sometimes boundary condition errors occur if velocities in the room creates negative pressure that tries to suck the air at the outlet back into the room. It then tries to block this as the mass flow can only go out of an outlet boundary condition. It could then end up blocking most of the outlet and the solution will be instable.

3.4.5 Classroom Cross section partition

To save simulation time, only a slice of the classroom was modelled. This can be done assuming that the classroom consists of 7 equal slices that are symmetrical. The slice of the classroom will have an inlet with the air volumes dimensioned for the number of people and area which represents approximately $1/7^{th}$ of the classroom. In reality the side walls will influence the air flow pattern. However, this will give an idea of the air flow in the middle of the room. Even though the placement of the people can be in several different ways, the slice has 4 persons in a row as shown in figure 33 below. These were made with a relatively high degree of details and a mouth for the simulation of breathing. Each person will have an equivalent heat and CO_2 production as the average of $1/7^{th}$ of 30 children the age 14-16(see chapter 2.4). The metabolic rate per square meter is adjusted accordingly. The same applies for the façade and window which are slightly bigger than $1/7^{th}$ of the classroom; the U-value is adjusted so that the transmission losses stay the same as in the heat balance calculations. The façade wall to the right faces the outdoor temperature. All the surfaces have a no-slip boundary condition. The model has a window on the façade wall, with a heater underneath to prevent potential cold drafts. In the ceiling is lighting which emits $1/7^{th}$ of the heat in the heat balance calculations in chapter 3.3.1. Ansys CFX offers a *Best Practices Guide for HVAC* which has been used to decide reasonable settings concerning buoyancy, thermal radiation (not used) and conjugate heat transfer (CHT) between fluids and solids.

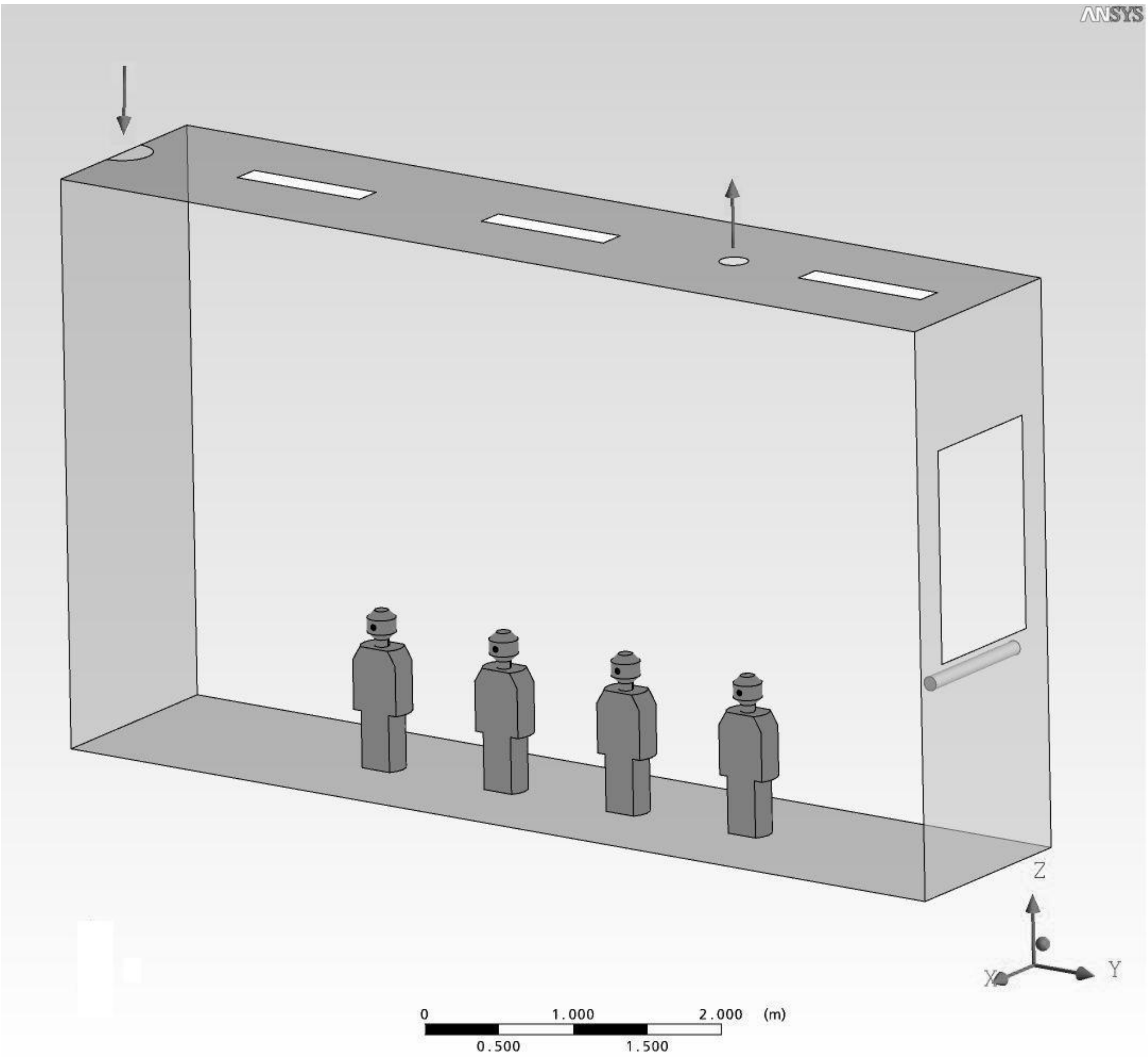

Figure 33 Cross section partition of classroom as modelled in Ansys CFX

Table 5 Area of the respective surfaces in CFD model

Surface	Area
Children	4,26 [m^2]
Floor	10,28 [m^2]
Interior wall	6,00 [m^2]
Facade	4,50 [m^2]
Window	1,50 [m^2]
Heater	0,33 [m^2]
Inlet	0,06 [m^2]
Outlet	0,03 [m^2]
Lighting	0,60 [m^2]
Mouth	0,005 [m^2]

The classroom is in fact only one domain/volume. The persons, inlet and outlet etc. are merely surfaces in the classroom volume, which are later assigned different properties and boundary conditions. The area of the various surfaces the classroom model consists of is listed in table 5. Boundary conditions vary for each case and will be listed together with the simulation results in chapter 4.

As the model is only a slice, the two side surfaces act as symmetry planes. This means that the boundary condition on this surface assumes similar behaviour to the fluid from the neighbouring domain. However it implies that no radiation model can be applied, as there are no surfaces from which it can reflect. This has impact on the heat transfer.

The simulation type is steady state. This is a simplification as CO_2 levels and temperature vary over time. With a constant ventilation rate, they will begin at a relatively low level before increasing until stationary levels at some point in time during a class. In the Swedish naturally ventilated schools, the ventilation is demand controlled meaning the ventilation rate, temperature and CO_2 levels will in fact vary throughout the class.

The component gases in the domain are air and CO_2 ideal gases. Air is governed by a transport equation, and CO_2 is a constraint of this meaning the mass fraction can never surpass 1. The children produce pure CO_2 at 25 ºC which is the temperature of breath. Breath consists of air and water vapour as well, but it is not possible to simulate air entering and exiting the body. Density of carbon dioxide at set at 1.8 kg/m^3 (Incropera et.al, 2002) for conversion from 19 l/h per person to kg/s per person.

The heat transfer model is thermal energy, the turbulence model k-epsilon, the turbulent wall functions are scalable, buoyancy is applied with a reference density of 1,2 kg/m^3. The inlet option (inlet and mouth) has medium turbulence intensity (5%) and the flow direction is normal to boundary condition.

3.4.6 Simulation Input

The simulation input can be found in Appendix 2. An extract and explanation of this appendix is follows here:
The general input for the CFD simulations are found in Table 6 where a 5 ºC supply air temperature is used. The CO_2 and heat production is 1/7th of a full classroom. In table 7 the optimal air volumes for heat balance at 20 ºC and the resulting CO_2 level in the occupied zone are calculated (from chapter 3.3). In the scenario of 5 ºC outdoor temperature, and assumptions as in chapter 3.3.1 approximately 56% of REN recommendations will achieve both energy balance and acceptable CO_2 levels. No space heating is required, and the CO_2 level in the occupied zone is 1265 mg/m^3 (approximately 700 ppm). The input data necessary to use in the CFD simulations is then put in table 8, where the inlet mass flow is divided by 7. The initial value of the CO_2 concentration is the weighted average of the occupied and upper zone as the best guess for the whole domain makes the simulation converges more quickly.

Table 6 General input for CFD simulations

CFD Input			
CO2_production	0,0000407134	kg/s	
Generation of heat from people	342,9	W	80.42 W/m^2
Temperature Breath	25	C	
Lighting, 0,6 m2	100	W	166,7 W/m^2
Heat transfer coefficient, facade	0,187	W/m^2K	(area facade 1,5m*4m)
Heat transfer coefficient, window	1	W/m^2K	

Table 7 Calculations for dimensioning at typical spring/autumn temperature

Typical Spring/Autumn Temperature		
Temperature outdoors:	**5**	
Inlet Temperature	5	**Heat Balance [kW]**
	Number of people	30
	Internal generation of heat	3,1
	Transmission losses	-0,246
	Total	**2,9**
	% of REN recommendation	**Ventilation Heat Loss [kW]**
Air flow at energy balance	56 %	-2,9
		Total Balance [kW]
		0,000
	Space Heating required	0,000
Archimedes number	0.9242	
Grashof number	1.5139e+008	
Inlet velocity, Uo	0,4338	[m/s]
Minimum plume temperature	14,5	[C]
Inlet Mass Flow	0,242	[kg/s]
CO2 mass concentration occupied zone	1265	[mg/m^3]
CO2 mass concentr. upper zone	1418	[mg/m^3]

Table 8 Input information for CFD simulations

Input		
Outdoor Temperature	5	ºC
Inlet Mass Flow	0,0346	[kg/s]
Inlet temperature	5	
Heater	0	W
Initial Conditions		
CO2 Ideal gas mass concentration	1380	[mg/m^3]
CO2 Ideal gas mass fraction	0,0011498	kg/kg
Air Ideal gas mass fraction	0,9988502	kg/kg
Initial Room Temperature	20	ºC

Simulations for typical Oslo winter, spring and summer temperatures (Eklima, 2008):

- At 0 ºC, one simulation is dimensioned after CO_2 levels and requires heating, and one is dimensioned with a lower ventilation rate and no heating.
- At 5 ºC air volumes are dimensioned as to give respectively 20 ºC and 22 ºC designed average room temperature.
- At 15 ºC both 20 ºC and 24 ºC are the designed room temperatures.

No simulations were made for outdoor temperatures below zero. The reason being that -3,8 ºC is the average temperature for December-February. The inlet temperature should be kept above 0 ºC by a heating coil excluded from the CFD simulation. The transmission loss difference from 0 ºC to -3,8 ºC is not very great with such low U-values. The dimensioning outdoor temperature of -20 ºC could have been interesting because of the effects of cold draft from the window and the performance of the heater to prevent this. This was unfortunately not prioritized.

Although 15,6 ºC is the average temperature for summer months in Oslo, day temperatures can often be above 20 ºC in summer. No simulations were made for really hot temperatures as the Swedish model is founded on the possibility of ventilating by opening windows on hot days.

3.4.6 Initial Conditions

Making a qualified guess of the steady state solution and using it as initial conditions will resolve the simulation more quickly. Besides setting the room temperature and CO_2 concentration several other parameters are set based on Ansys HVAC tutorial (Ansys CFX, 2007e):

- Initial velocity is 0 in X, Y, Z direction. (Intended for simulations of wind tunnels etc.)
- Static pressure is 0 Pa
- Turbulence Kinetic Energy has a fractional intensity of 0.05
- Turbulence eddy Dissipation is automatically selected by CFX
- Eddy length scale is 0.25 m

4 Results

4.1 Winter Simulations 20 ºC designed indoor temperature

The following results are from two simulations with 0 ºC supply air temperature and outdoor temperature; Simulation 1 with air volumes designed so that no heating is required, Simulation 2 with increased air volumes to achieve a lower CO_2 concentration, but where heating is required.

4.1.2 Simulation 1: Winter simulation, no heating

Table 9 Boundary Physics for 0 ºC, no heating

Name/Location	Type	Settings
Inlet	Inlet	Static Temperature = 0 [ºC] Mass Flow Rate = 0.0235 [kg s^-1]
Facade	Wall	Outside Temperature = 0 [ºC]
Heater	Wall	Heat Transfer = Adiabatic
Window	Wall	Outside Temperature = 0 [ºC]
Initial conditions		
Average Room temperature: 20 [ºC] Air Ideal gas mass fraction: 0.9992836 [kg/kg] (CO_2 ideal gas mass concentration 1939 mg/m^3)		

Table 10 User Data 0 ºC, no heating

Measuring points	[x y z]	Velocity [m/s]	Temperature [K]	CO2 mass concentration [mg/m^3]
Ankle Height	[0 1.5 0.1]	0.284	291.3	1928 (1071ppm)
Centre of gravity	[0 1.5 0.6]	0.101	291.8	1998 (1110ppm)
Head	[0 1.5 1.1]	0.013	292.2	2078 (1154ppm)

Zone	Cross section planes	Average velocity [m/s]	Temperature average [K]	CO2 mass concentration [mg/m^3]
Occupied zone	1m from walls, z=1	0.100	292.3	1953 (1085 ppm)
Upper zone	1-4 m above floor	0.085	292.6	2045 (1136 ppm)
Classroom cross section	7x4m	0.092	292.5	2026 (1126 pm)

	Velocity average [m/s]	Mass flow [kg/s]	Area [m^2]	Air flow [m^3/s]
Inlet	0.288	0.0235	0.0627	0.018

Table 9 and 10 show the settings and results of a simulation with 0 ºC outdoor and inlet temperature which is dimensioned to keep 20 ºC indoor temperature. Air volumes are dimensioned so that no heating is required. Initial conditions correspond to the room temperature from which the heat balance was calculated and the resulting average CO_2 mass concentrations. High CO_2 levels were accepted. The measuring points are at 1m from the inlet wall where the velocity, temperature and CO_2 concentration are measured for purposes of evaluating thermal comfort. The CO_2 levels are above outdoor level. The zone average values are the area average values of a cross section plane in the occupied and upper zone as well as average for the whole classroom slice. The occupied zone temperature is 19.15 ºC and the classroom average is 19.35 ºC. The velocity at ankle height is 0.284 m/s, somewhat higher than recommended for thermal comfort. The temperature difference between ankle and head is 0.9 ºC.

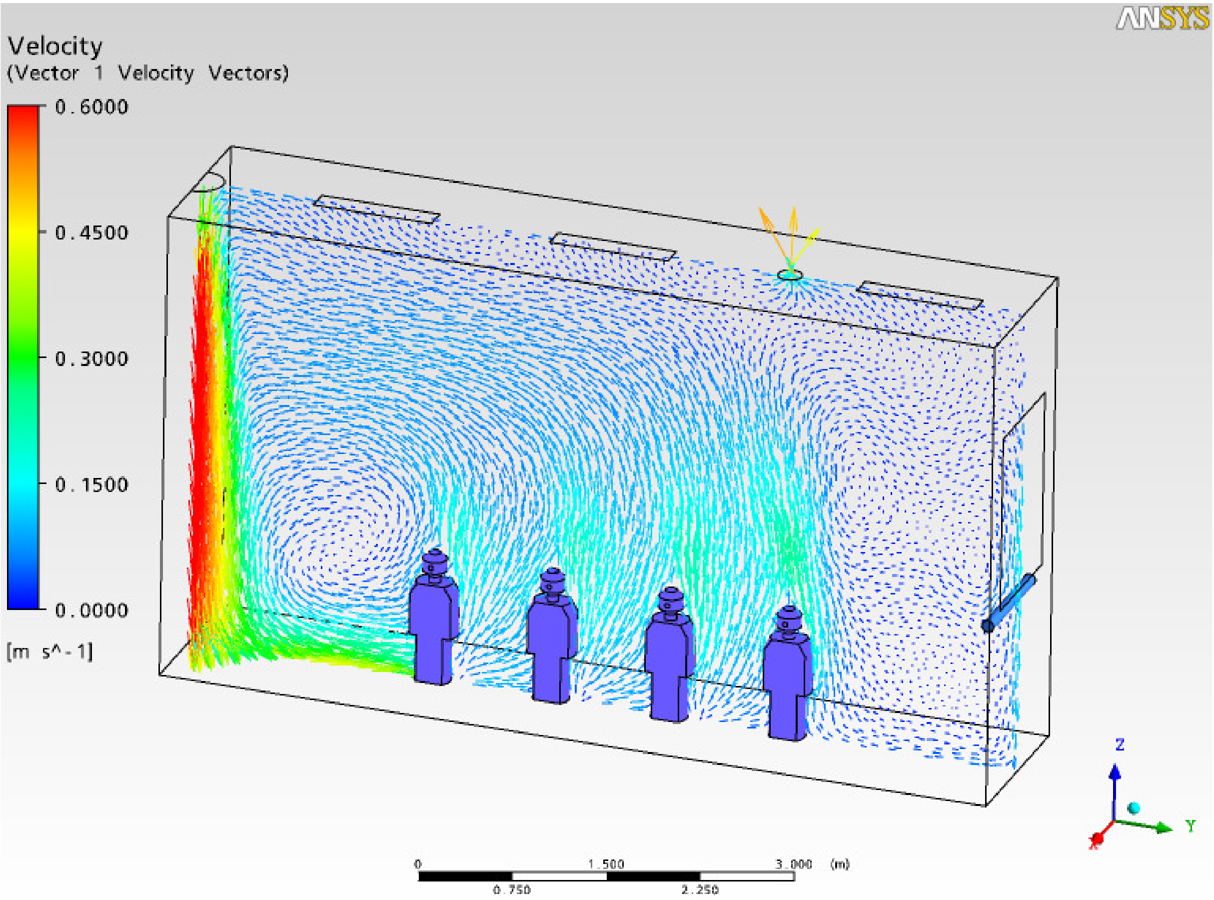

Figure 34 Velocity vectors at 0 for 0 ºC, no heating

Figure 34 and 35 show respectively the velocity vectors and the temperature contours of the classroom. The velocity vectors are graded by size and colour, where a long, red vector indicates the highest velocity. They show that the plume has a maximum velocity of 0,6 m/s in order of magnitude. It induces the surrounding air, causing the air above the children to bend towards the plume where it mixes in and is sucked back down. The air flow between the inlet plume and the first child is a large eddy where the air in the middle is stagnant. The air close to the ceiling as well as between the child closest to the façade and the façade wall is stagnant. There is a clear convection plume above the children. The temperature contours indicate the thermal stratification that takes place. The temperature at floor level is 18 ºC while next to the ceiling it is above 20 ºC. The temperatures next to the façade wall and window are lower than the surrounding air, indicating that a cold draft is taking place.

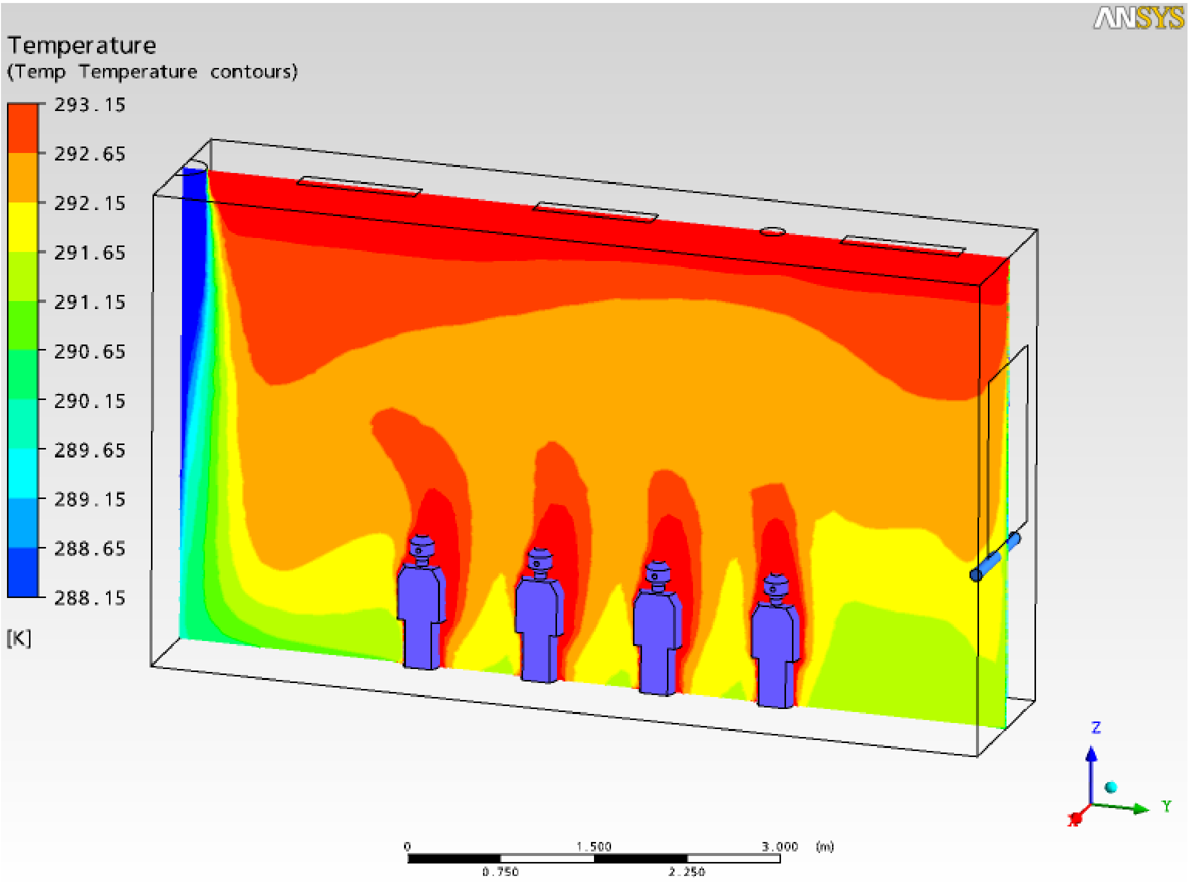

Figure 35 Temperature contours for 0 ºC, no heating.

4.1.2 Simulation 2: Winter, with heating

Table 11 Boundary Physics for 0 ºC with heating

Name/Location	Type	Settings
Inlet	Inlet	Static Temperature = 0 [ºC] Mass Flow Rate = 0.0277 [kg s^-1]
Facade	Wall	Outside Temperature = 0 [ºC]
Heater	Wall	Heat Flux in = 143 [W m^-2]
Window	Wall	Outside Temperature = 0 [ºC]
Initial conditions		
Average Room temperature: 20 [ºC] Air Ideal gas mass fraction: 0.9992884 [kg/kg] (CO_2 ideal gas mass concentration 1731 mg/m^3)		

Table 12 User data for 0 ºC with heating

Measuring points	[x y z]	Velocity [m/s]	Temperature [K]	CO2 mass concentration [mg/m^3]
Ankle Height	[0 1.5 0.1]	0.297	290.5	1860 (1033ppm)
Centre of gravity	[0 1.5 0.6]	0.104	291.1	1935 (1075ppm)
Head	[0 1.5 1.1]	0.011	291.6	2019 (1122ppm)

Zone averages	Cross section planes	Average velocity [m/s]	Temperature average [K]	CO2 mass concentration [mg/m^3]
Occupied zone	1m from wall, z=1	0.105	291.6	1880 (1044ppm)
Upper zone	1-4 m above floor	0.090	292.0	1971 (1095ppm)
Classroom cross section	7x4m	0.097	291.8	1951 (1083ppm)

	Velocity average [m/s]	Mass flow [kg/s]	Area [m^2]	Air flow [m^3/s]
Inlet	0.338	0.0277	0.0627	0.0212

Tables 11 and 12 show the settings and results of a simulation with 0 ºC outdoor and inlet temperature where the air flow is increased to keep CO_2-levels below 900 ppm. The occupied zone temperature is 18,5 ºC and the classroom average is 18.7 ºC. The velocity at ankle height is 0.297 m/s which is somewhat higher than recommended for thermal comfort. The temperature difference between ankle and head is 1.1 ºC.

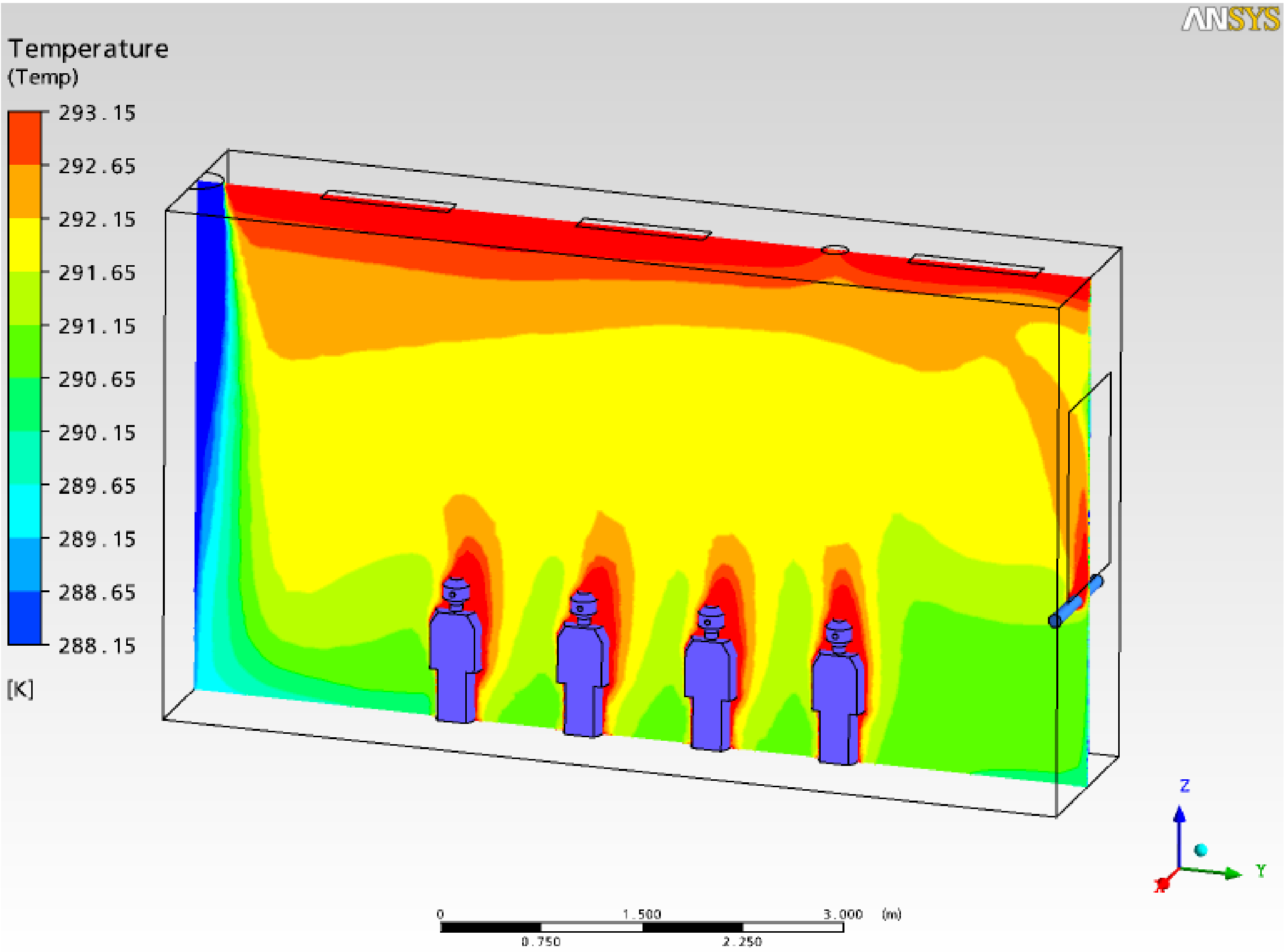

Figure 36 Temperature contours 0 ºC, with heating

The temperature contours in figure 36 indicates the thermal stratification that takes place. The temperature at floor level is 17-18 ºC while next to the ceiling it is 20 ºC. The heater makes the air rise. The temperature next to the wall below the heater is lower than the surrounding air. The inlet plume temperature assimilates later with the surrounding air than that in figure 35.

4.1.3 Cold draft and performance of heater

In figure 37 and 38 the velocity vectors next to the walls are shown for the two winter simulation results shown above. To the left the cold draft caused by an outdoor temperature of 0 ºC is shown. The order of magnitude is no larger than 0.15 m/s. To the right is the same outdoor temperature, but with the heater in use. This reverses the cold draft from the window, but there is still some created by the wall.

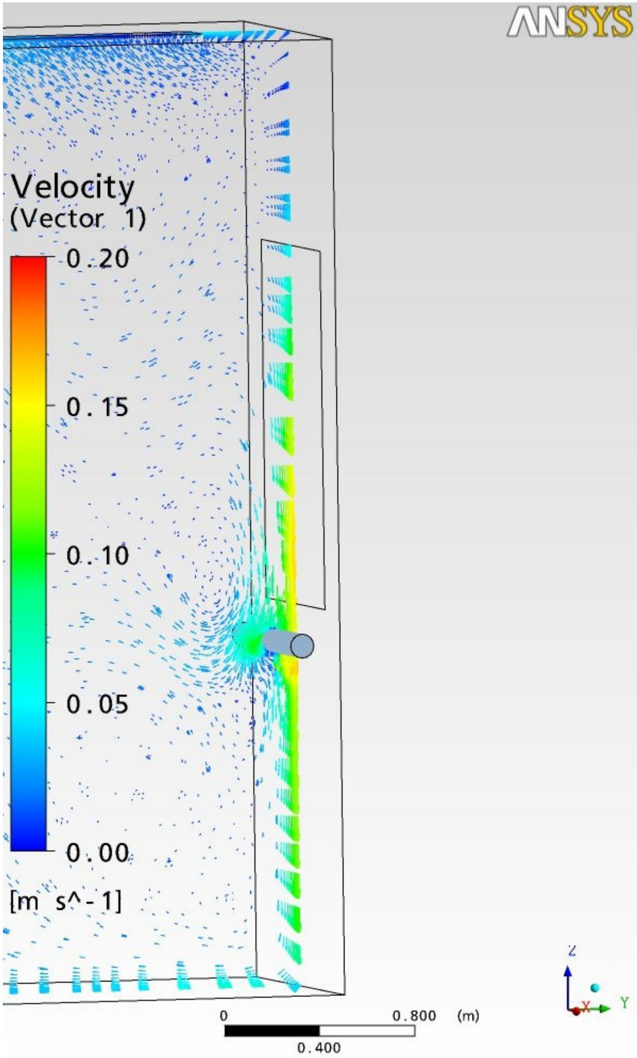

Figure 37 Cold draft 0 ºC outdoor temperature

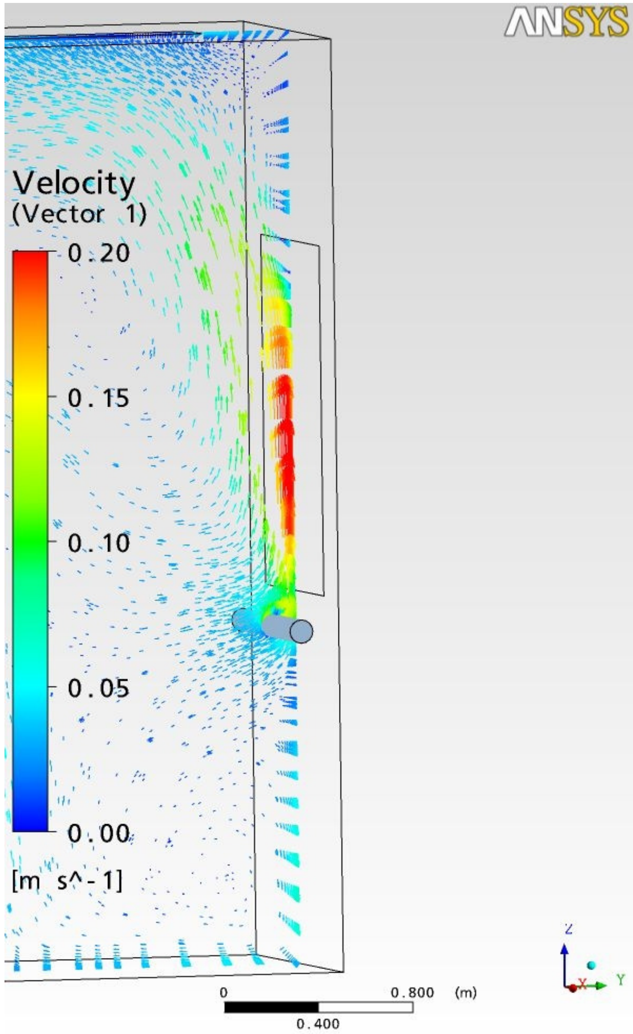

Figure 38 Heater in use at 0 ºC outdoor temperature

4.1.4 Carbon dioxide concentrations for winter simulations

Winter is the most critical time for high CO_2 concentrations. With air volumes dimensioned so low that no heating is required, the CO_2 concentration is 1953 mg/m^3 (1085 ppm) above outdoor level. With the increased air volumes, the concentration is 1880 mg/m^3 (1044ppm). In figure 39 the CO_2 distribution in the cross section sample plane is shown. Also here the concentration should be read as above outdoor level. The effect the heater has on the air flow can also here be observed: Fresh air from the occupied zone (light green) is transported to the upper zone. The inlet plume pulls the polluted air towards it making the CO_2 concentration the highest in the middle of the room near the inlet wall, rather than in the upper zone. The fresh air can be seen to follow the bodies of the children, so that the concentration that reaches the breathing zone is 1800 mg/m^3 (900ppm) which is less than the average for the occupied zone.

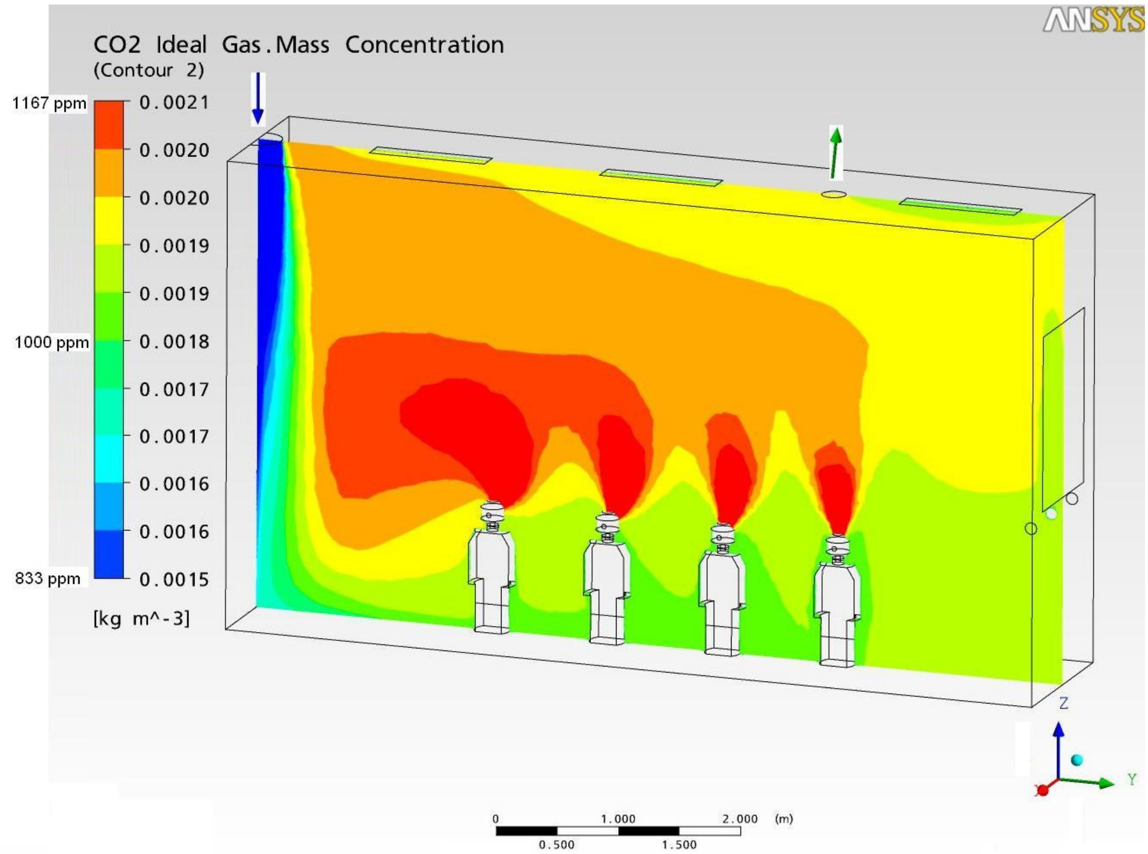

Figure 39 CO_2 concentration above outdoor level, winter simulation with heating

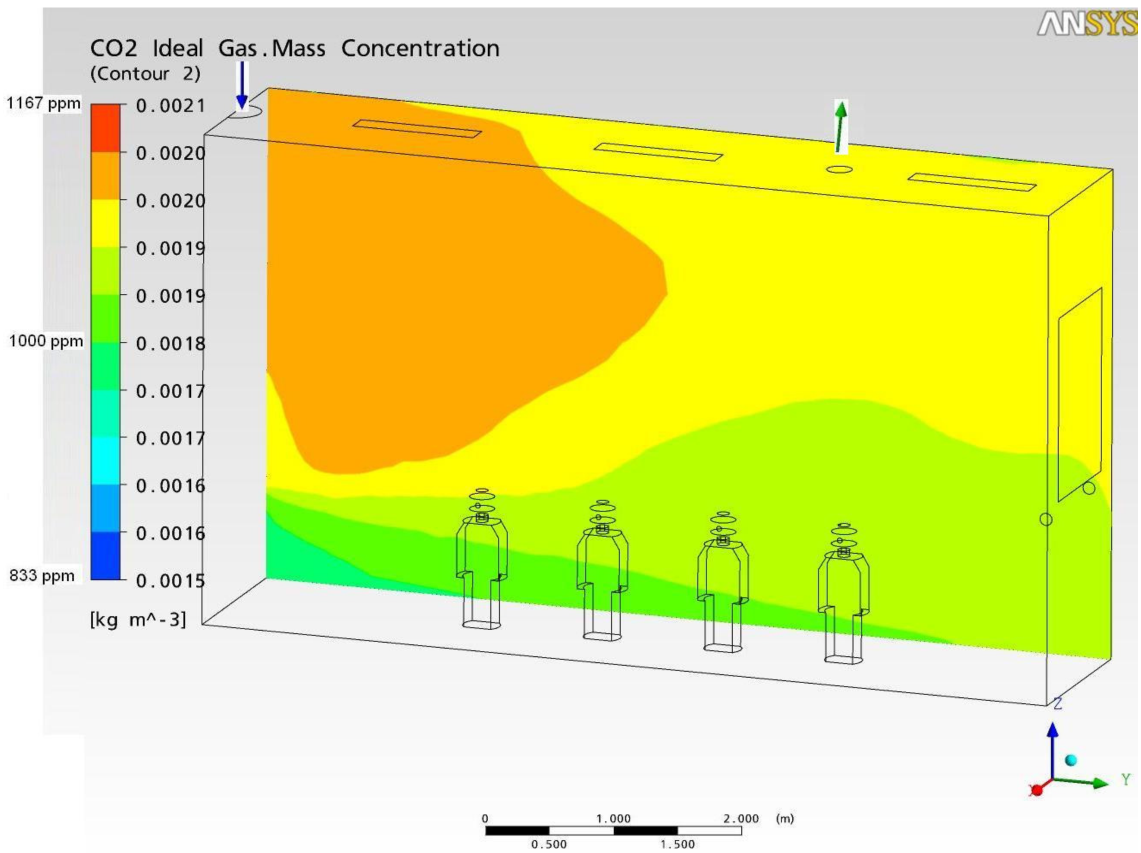

Figure 40 CO_2 concentration at symmetry plane, winter simulation with heating.

By moving the cross section sample plane behind the children, a clearer picture emerges of the distribution in the middle of the classroom (figure 40). This is next to the symmetry plane towards a similar slice, meaning this is a cross section plane almost half way between two rows of children. Ideally horizontal layers should occur, but, as shown in figure 40, this is not the case. Although it is more evenly distributed here, the highest concentration is still displaced towards the inlet wall.

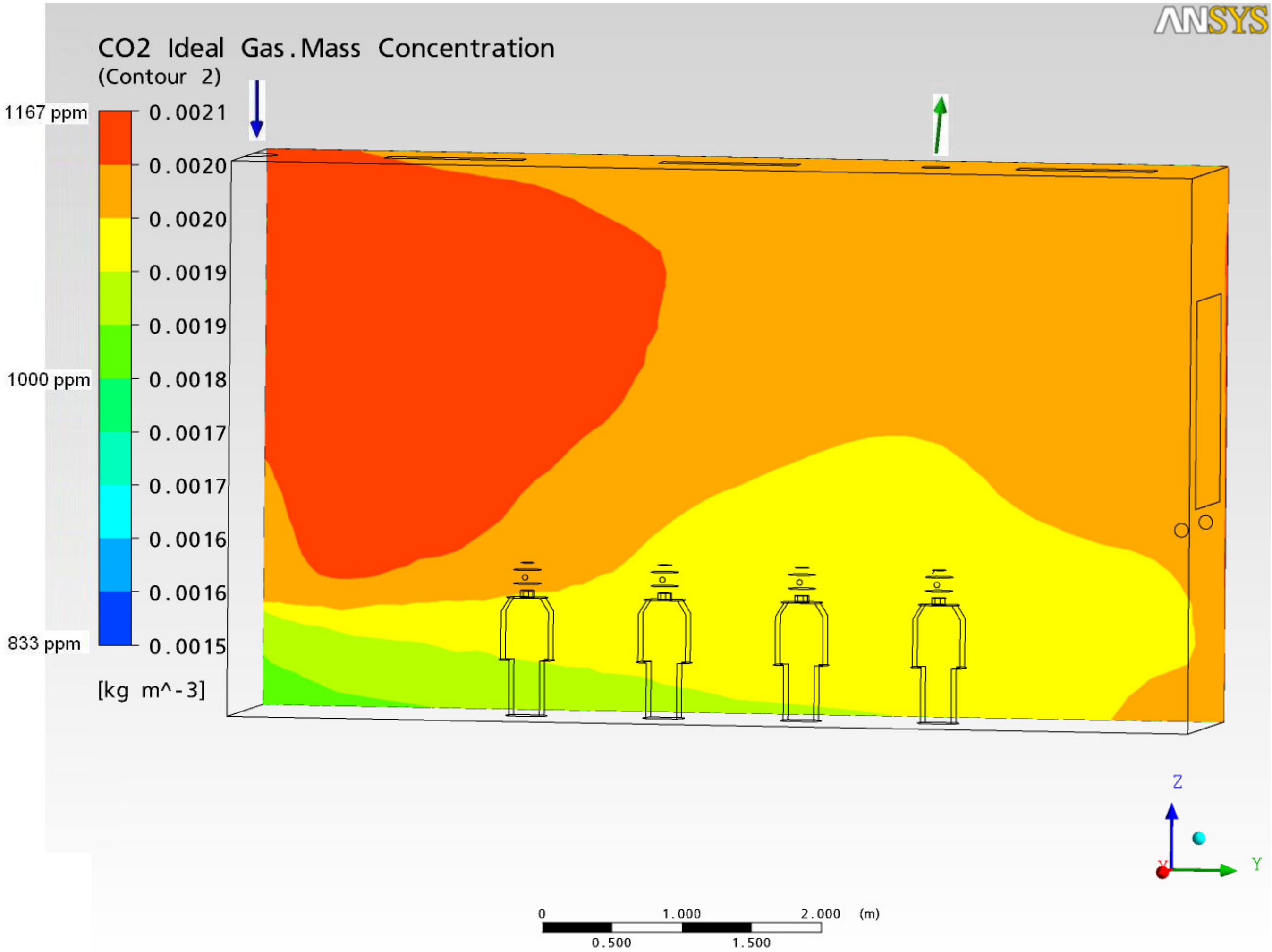

Figure 41 CO_2 concentration above outdoor level, winter simulation no heating

Figure 41 shows the CO_2 contours for the symmetry plane of the winter simulation with no heating and lowest ventilation rate. It has a similar distribution as the winter simulation with heating but higher levels. In addition, the effect of the cold draft can be observed to the right in the figure. Air from the upper zone (orange) is flowing next to the façade wall down into the occupied zone. The person closest to the inlet has a poorer air quality in the breathing zone.

4.2 Spring/autumn simulations

The following results are from two simulations with 5 ºC supply air and outdoor temperature: Simulation 3 with air volumes dimensioned to give 20 ºC indoor temperature and Simulation 4 with reduced air volumes to give 22 ºC.

4.2.1 Simulation 3: Spring/autumn, 20 ºC design indoor temperature

Table 13 Boundary Physics for 5 ºC, 20 ºC indoor temperature

Name/ Location	Type	Settings
Inlet	Inlet	Static Temperature = 5 [ºC] Mass Flow Rate = 0.0334 [kg s^-1]
Facade	Wall	Outside Temperature = 5 [ºC]
Heater	Wall	Heat Transfer = Adiabatic
Window	Wall	Outside Temperature = 5 [ºC]
Initial conditions		
Average Room temperature: 20 [ºC] Air Ideal gas mass fraction: 0.9988502 [kg/kg] (CO_2 ideal gas mass concentration 1380 mg/m^3)		

Table 14 User Data for 5 ºC, 20 ºC indoor temperature

Measuring points	[x y z]	Velocity [m/s]	Temperature[K]	CO2 mass Concentration [mg/m^3]
Ankle Height	[0 1.5 0.1]	0.295	290.4	1374 (763ppm)
Centre of gravity	[0 1.5 0.6]	0.109	290.9	1454 (807ppm)
Head	[0 1.5 1.1]	0.019	291.3	1546 (859ppm)

Zone averages	Cross section planes	Average velocity [m/s]	Temperature average [K]	CO2 mass concentration [mg/m^3]
Occupied zone	1m from wall, z=1	0.104	291.4	1399 (777ppm)
Upper zone	1-4 m above floor	0.088	291.7	1492 (828ppm)
Classroom cross section	7x4m	0.095	291.6	1472 (817ppm)

	Velocity average [m/s]	Mass flow [kg/s]	Area [m^2]	Air flow [m^3/s]
Inlet	0.413	0.0334	0.0627	0.026

In Table 13 and 14 are the settings and results of a simulation with a typical spring/autumn outdoor and inlet temperature of 5 ºC. Air volumes were dimensioned so that no heating is required for a design temperature of 20 ºC the air flow is higher than for the simulations at 0ºC. The occupied zone temperature is 18,3 ºC and the classroom average is 18.5 ºC. The velocity at ankle height is 0.295 m/s which is somewhat higher than recommended for thermal comfort. The temperature difference between ankle and head is 0.9 ºC. The CO_2 concentration is lower than required.

The temperature contour of this simulation found in Figure 42 shows a similar flow pattern to the other simulations. Because the air flow is larger, the plume temperature is only 15-16 ºC when reaching the floor. The temperature in the occupied zone is generally lower.

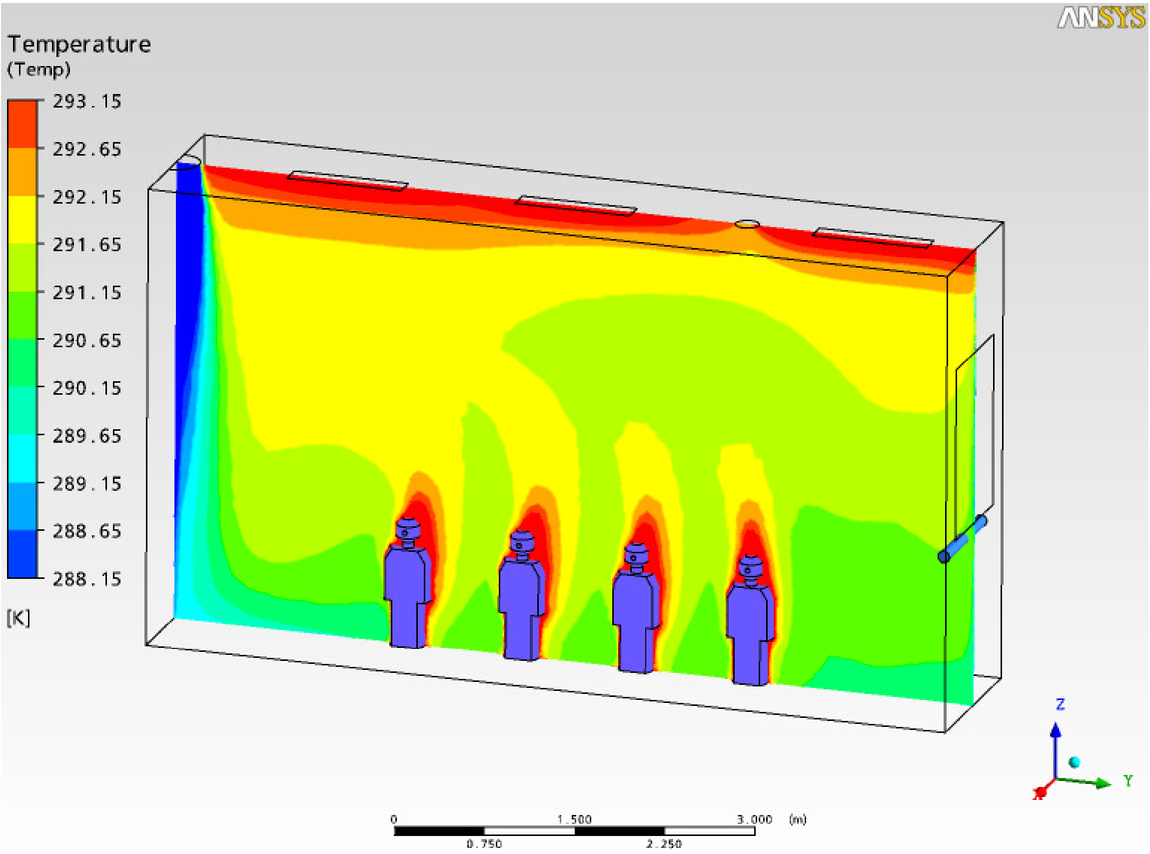

Figure 42 Temperature contours 5 ºC

4.2.2 Simulation 4: Spring/autumn, 22 ºC design indoor temperature

Table 15 Boundary Physics for 5 ºC, 22 ºC design indoor temperature

Name/Location	Type	Settings
Inlet	Inlet	Static Temperature = 5 [C] Mass Flow Rate = 0.0185 [kg s^-1]
Facade	Wall	Outside Temperature = 5 [C]
Heater	Wall	Heat Transfer = Adiabatic
Window	Wall	Outside Temperature = 5 [C]
Initial conditions		
Average Room temperature: 20 [ºC] Air Ideal gas mass fraction: 0.9992836 [kg/kg] (CO_2 ideal gas mass concentration 1576 mg/m^3)		

Table 16 User Data 5 for 5 ºC, 22 ºC design indoor temperature

Measuring points	[x y z]	Velocity [m/s]	Temperature [K]	CO2 mass concentration [mg/m^3]
Ankle Height	[0 1.5 0.1]	0.266	294.9	1666 (926ppm)
Centre of gravity	[0 1.5 0.6]	0.104	295.4	1731 (961ppm)
Head	[0 1.5 1.1]	0.022	295.7	1808 (1004ppm)

Zone averages	Cross section planes	Average velocity [m/s]	Temperature [K]	CO2 mass concentration [mg/m^3]
Occupied zone	1m from wall, z=1	0.097	295.8	1680 (933ppm)
Upper zone	1-4 m above floor	0.089	296.0	1761 (978ppm)
Classroom cross section	7x4m	0.094	295.9	1743 (968ppm)

	Velocity average [m/s]	Mass flow [kg/s]	Area [m^2]	Air flow [m^3/s]
Inlet	0.232	0.018	0.0627	0.0145

In Table 15 and 16 are the settings and results of a simulation that is similar to the typical spring/autumn temperature one. The difference is that air volumes were dimensioned to keep a design temperature of 22 ºC without heating required. The air flow is therefore lower than the simulation at 5 ºC. The occupied zone temperature is 22.7 ºC and the classroom average is 22.8 ºC. The velocity at ankle height is 0.266 m/s which is somewhat higher than recommended for thermal comfort. The temperature difference between ankle and head is 0.8ºC. The CO_2 concentration is slightly higher than recommended.

The temperature contour of this simulation found in figure 43 shows that the temperature in of the plume reaching the floor is 20 ºC. (Note that the Temperature legend has an upper bound of 25 ºC) The temperature next to the window and façade wall is slightly lower, but as it is 21ºC, it will not be perceived as a cold draft.

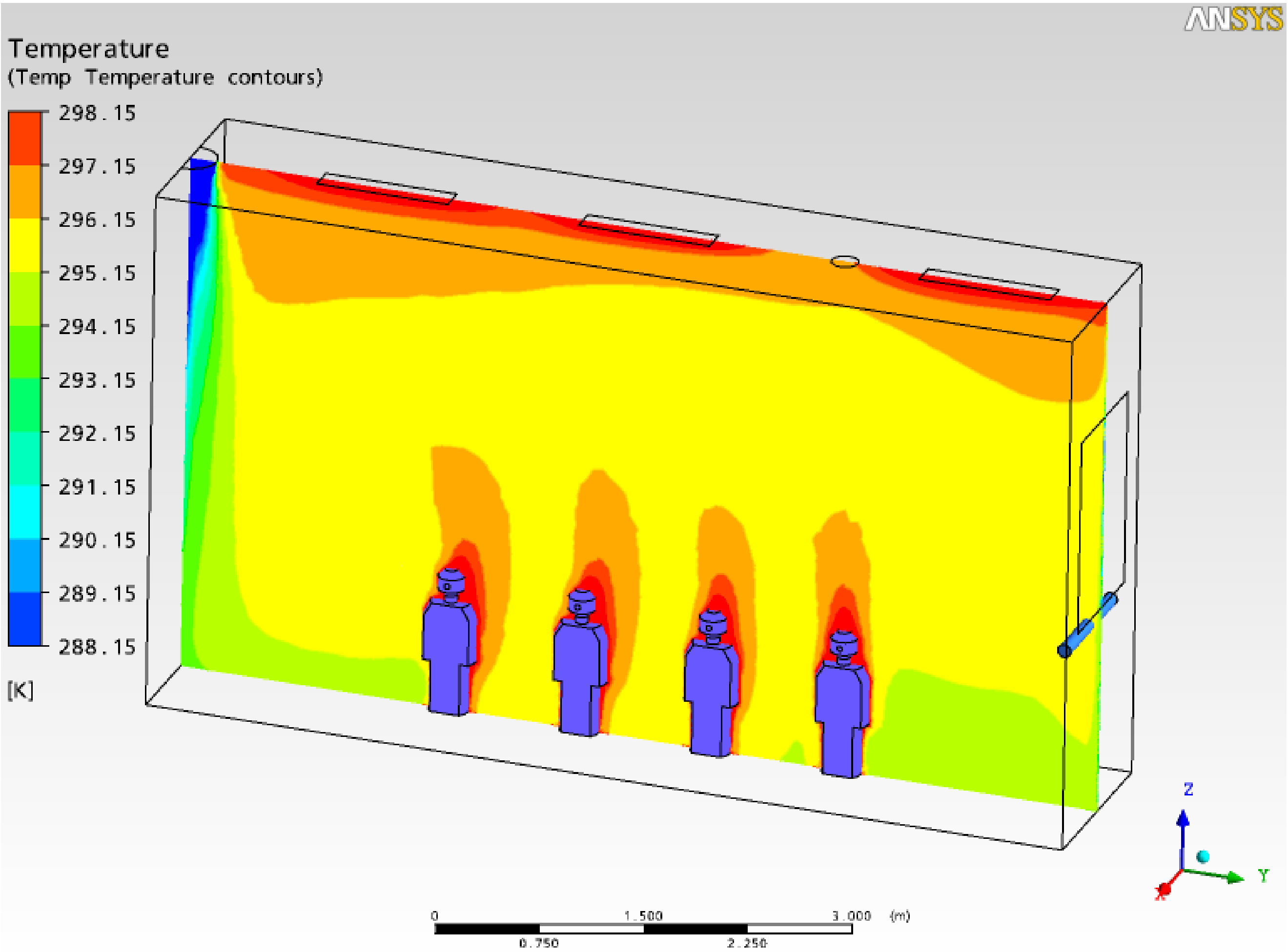

Figure 43 Temperature contours 5 ºC 22ºC design indoor temperature

4.3 Summer Simulations

The following results are from two simulations with 25 ºC supply air and outdoor temperature: Simulation 3 with air volumes dimensioned to give 20 ºC indoor temperature and Simulation 4 with reduced air volumes to give 24 ºC.

4.3.1 Simulation 5: Summer, 20 ºC design indoor temperature

Table 17 Boundary Physics for 15 ºC, 20 ºC design room temperature

Name/Location	Type	Settings
Inlet	Inlet	Static Temperature = 15 ºC Mass Flow Rate = 0.0969 [kg s^-1]
Facade	Wall	Outside Temperature = 15 [ºC]
Heater	Wall	Heat Transfer = Adiabatic
Window	Wall	Outside Temperature = 15 [ºC]
Initial conditions		
Average Room temperature: 20 [ºC] Air Ideal gas mass fraction: 0.9996144 [kg/kg] (CO_2 ideal gas mass concentration 463 mg/m^3)		

Table 18 User data for 15 ºC , 20 ºC design room temperature

Measuring points	[x y z]	Velocity [m/s]	Temperature [K]	CO2 mass concentration [mg/m^3]
Ankle Height	[0 1.5 0.1]	0.403	291.7	460 (255ppm)
Centre of gravity	[0 1.5 0.6]	0.182	292.2	547 (303ppm)
Head	[0 1.5 1.1]	0.052	292.6	633 (351ppm)

Zone averages	Cross section planes	Average velocity [m/s]	Temperature [K]	CO2 mass concentration [mg/m^3]
Occupied zone	1m from wall, z=1	0.128	292.6	491 (272ppm)
Upper zone	1-4 m above floor	0.136	292.6	568 (315ppm)
Classroom cross section	7x4m	0.140	292.5	549 (305ppm)

	Velocity average [m/s]	Mass flow [kg/s]	Area [m^2]	Air flow [m^3/s]
Inlet	1.235	0.0969	0.0627	0.077

A simulation was done with 15 ºC outdoor and inlet temperature and a 20 ºC designed indoor temperature. Air volumes were very large so as to remove the heat surplus. The temperature difference between head and ankle height in front of the closest person to the inlet wall is 0.9ºC and the velocity 0.4 m/s. The temperature in the occupied zone is 19.5 ºC. The CO_2 concentration in the occupied zone is very low, only 491 mg/m^3. The temperature contours (figure 44) show that the plume has a very strong pull on the surrounding air. The warm air above the children is induced in the plume. As this is the simulation with the highest ventilation rate, the velocity through the inlet is also the highest.

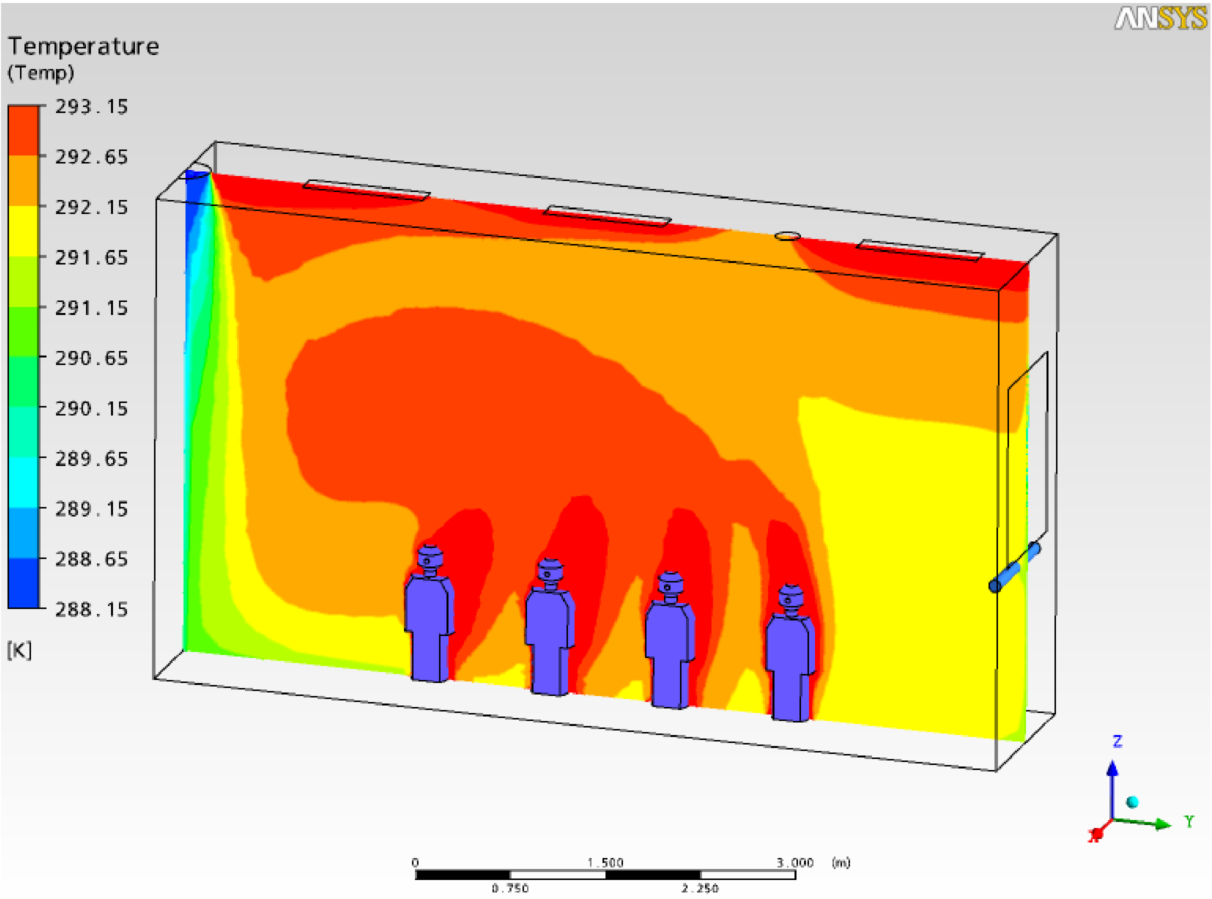

Figure 44 Temperature contours at 15 ºC and 20 ºC designed indoor temperature

This was the simulation with the highest velocities at ankle height. Figure 45 shows the velocity at 0.1 m above floor, starting at the inlet wall and ending at the façade wall. The first child is seated 2 m from the inlet wall. The velocity is then reduced to 0.35 m/s. After about 3.5 m the velocity is reduced to 0.25 m/s.

The other simulations yielded similar graphs. The lowest velocities were for the simulation with 5 ºC and 22 ºC designed indoor temperature. Where the maximum velocity at ankle height, 0.28 m/s, occurred at 0.9 m/s. the velocity is reduced to 0.25 m/s before reaching the first child.

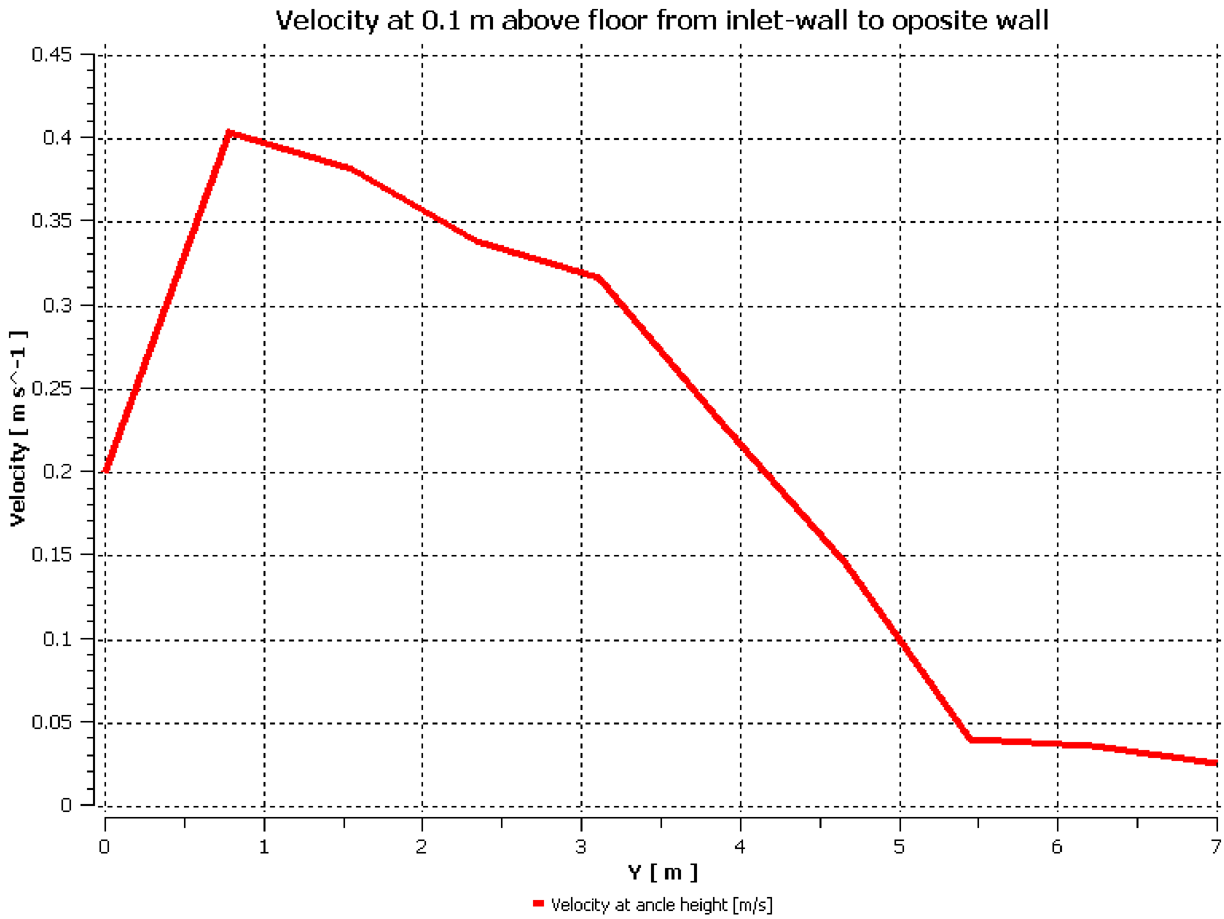

Figure 45 Velocity at ankle height from inlet wall to opposite wall

4.3.2 Simulation 6, Summer: 24 ºC design indoor temperature

Table 19 Boundary Physics for 15 ºC, 24 ºC design room temperature

Name/Location	Type	Settings
Inlet	Inlet	Static Temperature = 15 [ºC] Mass Flow Rate = 0.03704 [kg s^-1]
Facade	Wall	Outside Temperature = 15 [ºC]
Heater	Wall	Heat Transfer = Adiabatic
Window	Wall	Outside Temperature = 15 [ºC]
Initial conditions		
Average Room temperature: 24 [ºC] Air Ideal gas mass fraction: 0.9993071 [kg/kg] (CO_2 ideal gas mass concentration 832 mg/m^3)		

Table 20 User data 15 ºC, 24 ºC design room temperature

Measuring points	[x y z]	Velocity [m/s]	Temperature [K]	CO2 mass concentration [mg/m^3]
Ankle Height	[0 1.5 0.1]	0.290	296.8	902 (501ppm)
Centre of gravity	[0 1.5 0.6]	0.116	297.3	975 (541ppm)
Head	[0 1.5 1.1]	0.021	297.7	1056 (587ppm)

Zone averages	Cross section planes	Average velocity [m/s]	Temperature [K]	CO2 mass concentration [mg/m^3]
Occupied zone	1m from wall, z=1	0.101	297.7	917 (509ppm)
Upper zone	1-4 m above floor	0.093	297.9	1003 (557ppm)
Classroom cross section	7x4m	0.098	297.8	983 (546ppm)

	Velocity average [m/s]	Mass flow [kg/s]	Area [m^2]	Air flow [m^3/s]
Inlet	0.473	0.0370	0.0627	0.0297

To avoid excessive air volumes and unnecessary use of fan power, a simulation allowing 24 ºC was done. The air volumes were reduced to less than half of that with 15 ºC and 20 ºC designed indoor temperature (table 20).

The temperature difference between head and ankle height in front of the closest person to the inlet wall is 0.9 ºC and the velocity 0.29 m/s. The temperature in the occupied zone is 24.6 ºC. The CO_2 concentration in the occupied zone is 917 mg/m^3, well below the recommended level. The temperature contours (figure 46).

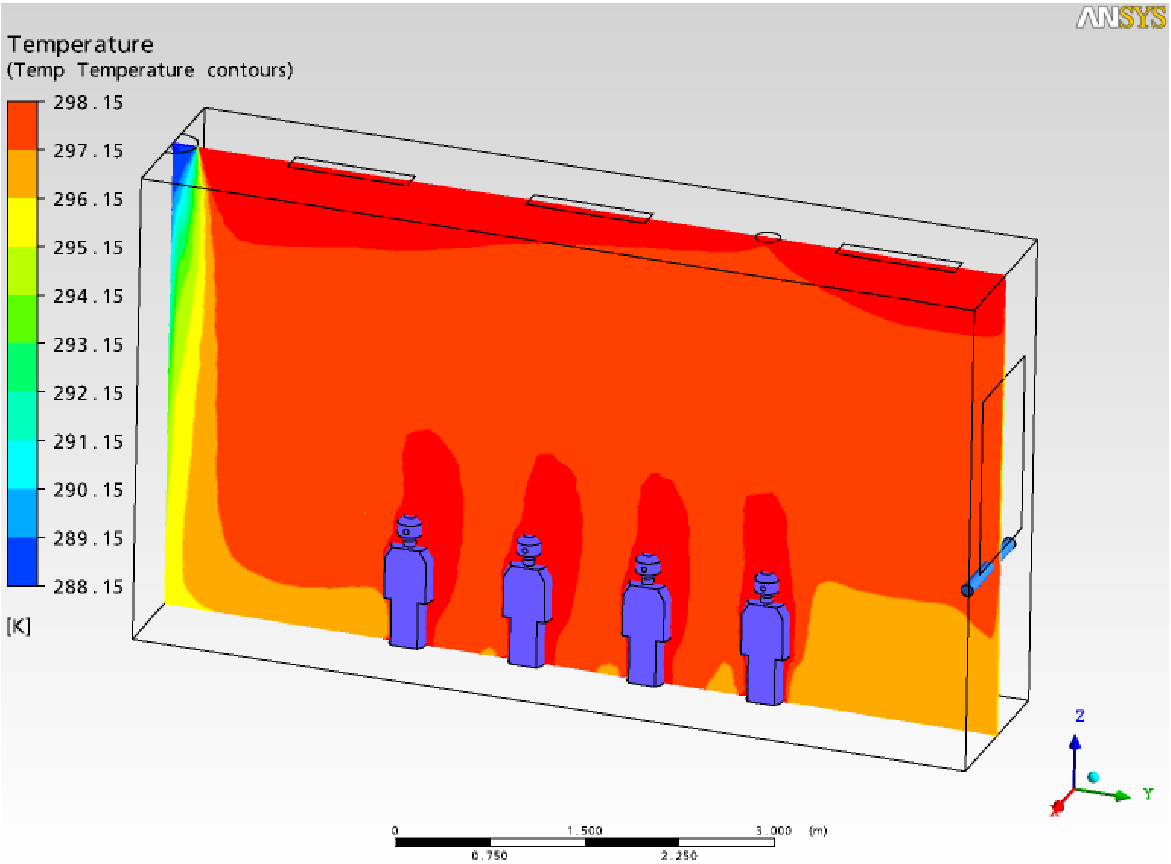

Figure 46 Temperature contours at 15 ºC outdoor temperature and 24 ºC designed indoor temperature

5 Discussion

5.1 Discussion of Method

Both the vertical plume model and the convection plume model corresponded quite well to the CFD simulations. They can probably both be used for preliminary dimensioning. The calculation of the carbon dioxide levels are highly idealized when using a two zone model assuming a clear partition in the occupied zone and upper zone concerning the degree of contamination. As can be seen from the CO_2 contours this is not the case. The velocity is not uniform and one directional, but with eddies created by the inlet plume and the cold draft. There is no reason why the same method could not be applied for a conference room.

Heat balance calculations etc. were done simultaneously with some of the CFD simulations because of the time consuming nature of these. This resulted in several errors being found and corrected as the simulations were in process. Although the errors were rather insignificant, it means that the simulation input mass flow does not altogether match the dimensioned air volumes.

The common practise in CFD modelling is to compare and confirm that the air flow profile of the inlet device corresponds to product data. In this case none was available. At any rate, the inlet was simplified to a hole in the ceiling rather than placed on the wall.

5.2 Evaluation of Simulation Quality

The simulations were steady state. One can not see the effect of heat storage in the wall, nor varying air volumes. Normally in Swedish naturally ventilated schools, the ventilation rate is demand controlled either by temperature or carbon dioxide. It will take time for the carbon dioxide to accumulate before a sensor at the height of the breathing zone activates and increases the ventilation rate.
The simulations serve as examples of the order of magnitude air volumes might be at different temperatures, and the resulting indoor environment.

The CO_2 mass fractions are very small, in the size of parts per million or mg per cubic metre. For the results to be interesting, the simulation requires a great degree of accuracy. The CFX simulator therefore has a convergence criterion at a 10^{-6} residual target[8]. This means the simulation can take over 24 hours. In addition to this, a conversion in the CO_2 production from l/h to kg/s which is the input in Ansys CFX means an extra degree of inaccuracy in the number of decimals and the density. The air flow input is entered as either velocity or mass flow, and the heat balance calculations volume flow is used. Any decimal error or inaccurate air density means the simulation air flow differs from the calculated one. Temperature of breath does not influence heat balance as CO_2 air flows are so low.

[8] Where the error from one iteration to the next is less than 10^{-6}. Normally 10^{-4} is sufficient.

The resolution of the grid is important for the simulation quality. Especially with high velocities when the boundary layer is very thin, a high resolution of the mesh is required next to the wall. One way of measuring the quality of this is by the y+ value. This should be below 5 (Nørsett, 2008). The highest y+ values listed in Table 21 were from simulation 6 with the highest ventilation rate. These are all well below the value of 5. This means that the mesh could have been coarsened in some areas, which would have saved simulation time.

Table 21 y+ values for the surfaces in the fluid domain

Surface	Maximum value of y+
Window	0.475401
Facade	0.555870
Floor, walls	3.555480
Children	1.519970
Heater	0.141365

Only modelling a slice of the classroom meant using symmetry boundary conditions. This means that the effect of the walls in the front and back of the classroom would not influence the flow pattern. Another rather important factor is that it disables the use of radiation models. Normally a Monte Carlo radiation model would be used, but as the side surfaces of the slice are just symmetry planes, Ansys CFX is unable to calculate where the radiation hits a surface and reflects. This has an impact on the heat transfer from the children, heater, lighting from ceiling to floor etc. thereby affecting the temperature distribution and the air flow. In a real room with displacement flow the floor temperature would be higher than the supply air, and the ceiling cooler than the exhaust air due to the radiative heat exchange between the surfaces in the room (Nilsson et al., 2003 p231). As the walls are adiabatic, the effects of thermal mass as well as radiation absorption are lacking. Not only does this give less of a preheating effect of the supply air plume than desired, but a warmer floor would also contribute to the upward pull of the supply air. A warmer inlet wall would heat the plume, reducing the draft problem as the velocity decreases with the temperature difference of the plume and surrounding air. The heater had no positive effect on the temperature in the occupied zone. It was placed below the window to prevent cold draft, which worked well, but as for heating the occupied zone proved itself highly inefficient. A real heater should have a much larger area, and would thereby contribute to raising the operational temperature as well as transporting heat to floor and surrounding walls through radiation. The lighting heated convectively the air close to the ceiling. This heat would have been transported to the walls, floor and children through radiation. The biggest error however is that all the dry heat from the children is transferred convectively as the boundary condition is a heat flux. In fact the radiative and convective heat exchanges are almost equal in size (chapter 2.4). This means that the convection plume each child creates is much larger than it should be. This results in a large eddy created by the inlet plume pulling downward and the convection plume of the children pulling upward. This is somewhat unfavourable; it creates a stagnant area between the inlet wall and the first child which has a negative effect on the removal of contaminants.

The dimensioning (chapter 3.3) was also used for finding steady state conditions to use as initial conditions in the simulation. Making a qualified guess of these helps the simulation to reach convergence more quickly. Ideally it should not matter much what the initial conditions are if allowing enough simulation time, but with the low velocities and small mass fractions involved in HVAC simulations convergence is often difficult to obtain.

In the simulations, although residual errors being smaller than 10^{-6}, either the temperature or the CO_2 level user points had a tendency to not converge towards a constant value. Although sloping only slightly, an “infinite” number of iterations would have landed it elsewhere. The number of iteration has therefore in some cases had an influence on the result. Stopping after 500 iterations might yield a slightly different result than after 1500. The error could be as great as 1 ºC or 500 mg/m^3 gas mass concentration.

5.3 Discussion of results

5.3.1 Ventilation efficiency

When discussing ventilation efficiency (removal of contaminants), it is natural to compare it with ideal mixing ventilation (ε=1). As no data for the concentration of carbon dioxide at the exhaust are available, the upper zone average is used, improving the result somewhat. The efficiencies from the different simulations are listed in table 21. The ventilation efficiency is about 1.06 in average.

Table 22 ventilation efficiency

Simulation	Ventilation Efficiency
S1: Winter, no heating	1,05
S2: Winter, heating	1,05
S3: Spring, 20C indoor	1,07
S4: Spring, 22C indoor	1,05
S5: Summer, 20C indoor	1,03
S6:Summer, 24 C indoor	1,09

Traditional displacement flow with air supplied at floor level would perform much better, but there would be a serious draft problem as there would be no time for the air to preheat. Mixing flow ventilation would need preheating of supply air to a couple of degrees below room temperature. With mixing flow ventilation, the concentration would be homogenous throughout the room. With the same air volumes as these simulations, one could achieve no lower concentration than in the upper zone.

CO_2 mass concentrations in the occupied zone as well as for the whole classroom were all higher than calculated. A likely explanation might be that the upper and occupied zones are not strictly separated as mentioned above. It may also be due to miscalculations and the simplifications made in the dimensioning. Especially the conversion from volume flow to mass flow might result in a different ventilation rate than intended.

The CO_2 contour plots in chapter 4.1.4 give an indication for an optimal placement of a CO2 sensor. For practical reasons it needs to be placed at a wall and neither the inlet wall or the façade wall will give a representative value of the concentration in the occupied zone. One is then left with either the front or back of the room where the sensor should be placed symmetrically.

The ventilation principle seemed to function well for inlet temperatures above 5º where no heating was required and CO_2 levels kept acceptable.

5.3.2 Thermal comfort and air quality

The vertical temperature difference from ankle to head is approximately 1ºC next to the person closest to the inlet wall where the difference was assumed to be greatest. This was the case for all the simulation cases and as this is well below the limit of 3 ºC, it can be ruled out as a risk. The room temperature was slightly lower than dimensioned, especially in the occupied zone. Dimensioning air flow for 21ºC in stead of 20ºC at low supply air temperatures would probably solve this problem. However, decreasing the air flow increases the carbon dioxide concentration.

The risk of draft is substantial with velocities up to 0.4m/s.The feet are not so sensible to velocities as for example the back of the neck where the velocities are drastically reduced. The worst case was the simulation at 15 ºC with the largest air volumes. As mentioned with the lack of a radiation model, the walls are not heated, and so the plume is not heated as quickly as in reality. This would slow it down as the temperature difference is the driving force. This may influence the amount o fresh air entering the occupied zone in a negative way.

Table 23 Ventilation to keep 20 C indoor temperature, measurements at ankle height

Simulation	CO2 1.5m from inlet wall [ppm]	Velocity 1.5m from inlet wall [m/s]	Velocity 3m from inlet wall [m/s]
S1: Winter, no heating	1071	0,284	0,22
S3: Spring, 20ºC indoors	763	0,295	0,229
S5: Summer, 20ºC indoors	255	0,403	0,321

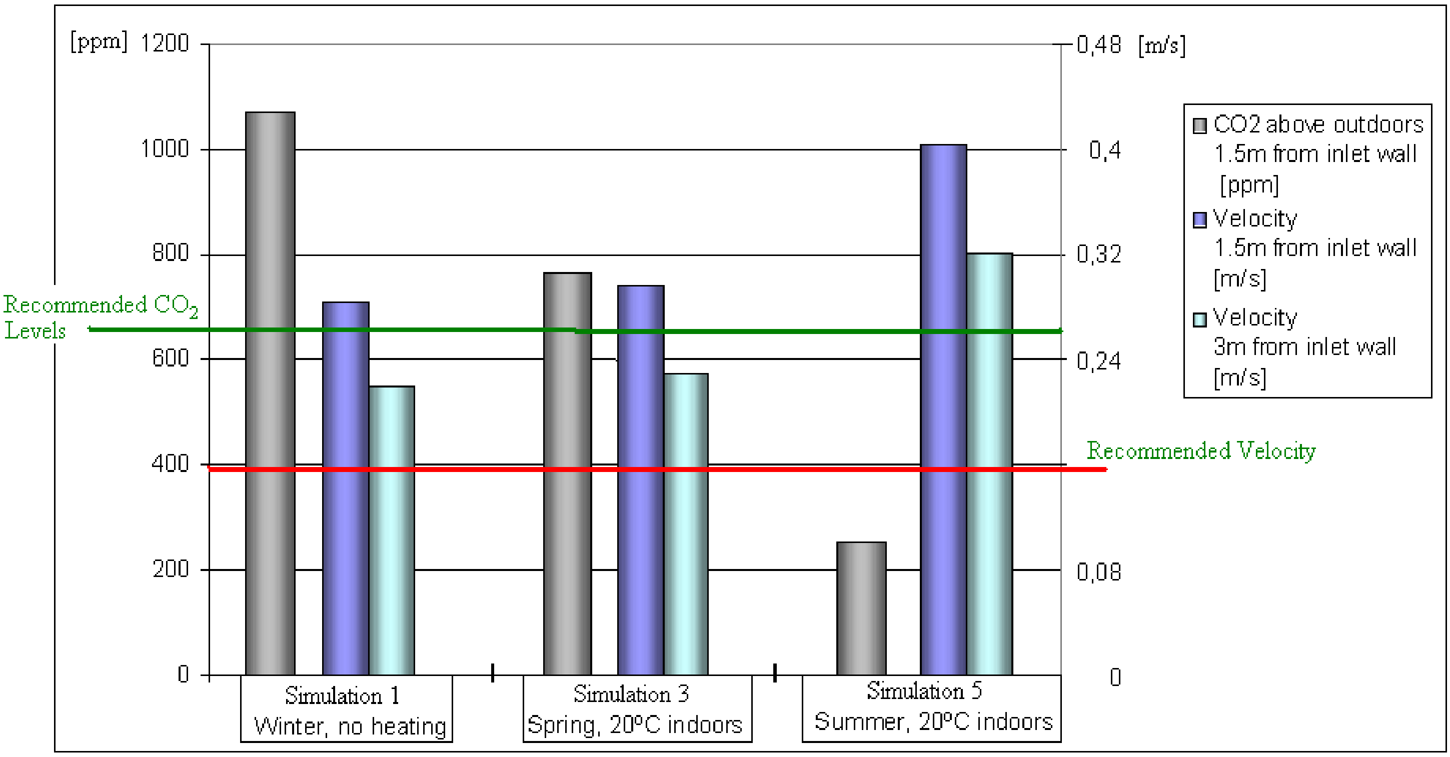

Figure 47 Simulations with ventilation for keeping indoor temperature at 20ºC

In table 23 and figure 47 results from simulations 1, 3 and 5 are summarized. These are the simulations where the air volumes are dimensioned to give 20ºC indoor temperatures. In some respect this is ventilation for cooling purposes. The CO_2 concentration is measured at ankle height 1.5 m from the inlet wall. As can be seen from the CO_2 contours in figure 39 the concentration here is the same as that around the head, and is therefore representative for the concentration in the breathing zone. The velocity is measured 1.5m and 3m from the inlet wall at ankle height, where the first child is placed at 2m. The green and red horizontal lines represent the recommended concentration and velocity. As can be seen, the velocities are generally too high, although this can partly be explained by lack of radiation model (see chapter 5.2). However, as for the summer simulation, one could allow a smaller ventilation rate, higher temperature and would achieve a lower velocity. The critical simulation is the winter simulation with 0ºC supply air temperature. As the simulations did not include the supply air culvert, it is probable that the temperature of the supply air entering the room will in fact be higher than 0ºC being preheated when passing beneath the floor and behind the inlet wall. This means that in fact 5ºC is a more likely minimum supply air temperature provided the air is preheated to 0º C in the culvert for frost protection.

Table 24 Ventilation to keep 20 C indoor temperature, measurements at ankle height

Simulation	CO2 1.5m from inlet wall [ppm]	Velocity 1.5m from inlet wall [m/s]	Velocity 3m from inlet wall [m/s]
S3 Spring, 20ºC indoors	763	0.295	0.229
S4: Spring, 22ºC indoors	926	0.266	0,205

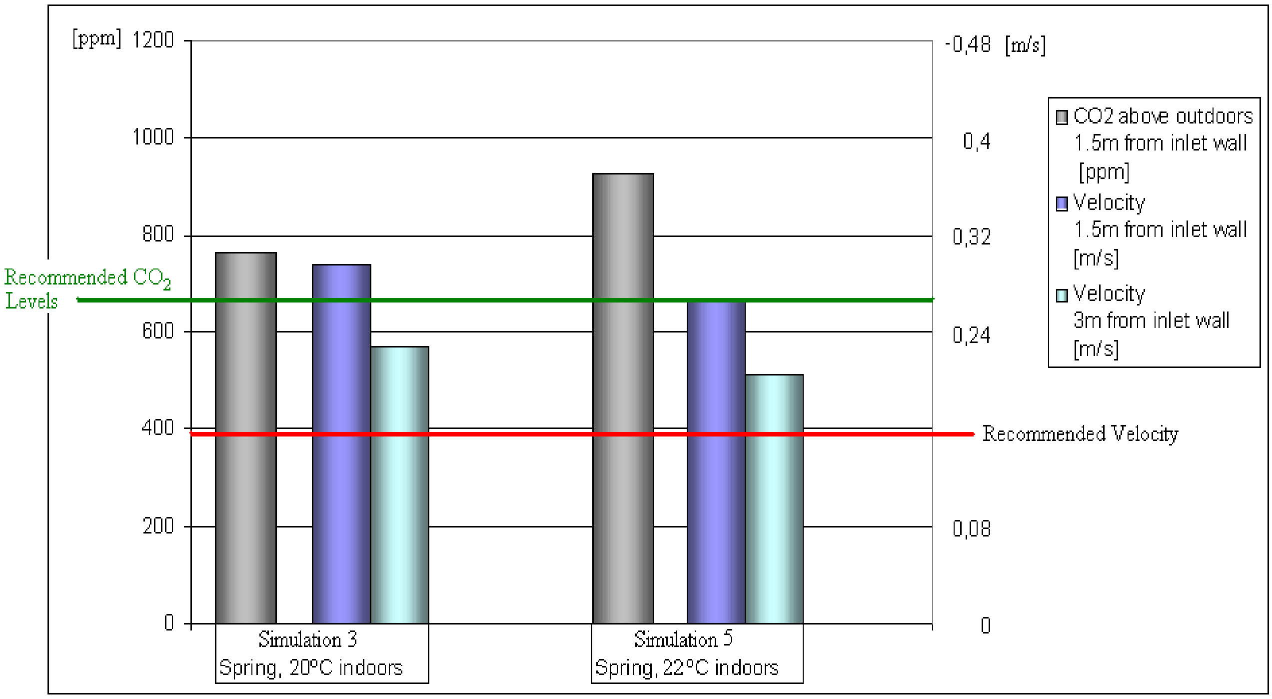

Figure 48 Simulations with 5ºC supply air temperature, different indoor temperature

In table 24 and figure 48 the two simulations with 5ºC supply air temperature are compared. Here also the measurements are at ankle height. The ventilation rate was dimensioned to give respectively 20 ºC and 22 ºC indoor temperature. The ventilation rate is therefore smaller at 22 ºC indoor temperature resulting in a higher CO_2 concentration, but lower velocities around the person seated closest to the inlet wall. For outdoor temperatures above 5ºC the CO_2 concentrations could be kept well below the 650 ppm above outdoors as air volumes will be increased to remove heat surplus. The velocities would stay constant with higher outdoor temperature and ventilation rate, unless a higher indoor temperature is used as a set point. In very warm weather, high velocities are welcomed as an additional cooling effect.

Table 25 Calculated and simulated temperatures of plume and at ankle height

Simulation	Design room temperature	Calculated minimum plume temperature 1m above floor	CFD Simulation: Temperature at ankle height 1.5 m from inlet wall
Simulation 1	20	14.7	18.2
Simulation 2	20	14.3	17.4
Simulation 3	20	14.5	17.3
Simulation 4	22	16.5	21.8
Simulation 5	20	15.5	18.6
Simulation 6	24	18.7	23.7

It is important when considering risk of draft to not look at velocities isolated from the temperatures. Table 25 shows the minimum plume temperatures 1m above floor level and the temperatures at ankle height 1.5 m from the inlet wall. It shows that by the time the plume reaches the floor it has assimilated well, but is still 2 degrees below room temperature when entering the occupied zone. As discussed in chapter 5.2 the radiation might improve the heating of the plume flowing over the floor. The radiation the inlet wall absorbs would contribute to heating the fresh air before it enters the room and probably not contribute any further to heating the plume on the room side. Table 25 also show that one can expect the plume to have increased its temperature by 3-3.5 degrees by the time it reaches the occupied zone. This means that for dimensioning purposes, the minimum plume temperature 1 m above floor should probably be at least 15ºC.

Note that in the results the occupied zone is set 1m from the walls. The minimum requirement (see chapter 2.4) is 0.8 m from windows, heating sources and supply air inlets although the latter varies with the inlet device. For economic reasons, the specific area use (m^2/person) will be kept at a minimum. The velocities and temperatures 1.5 m from the inlet wall do not meet standards for thermal comfort, meaning an even larger near zone is needed. Studying appendix 4 shows that the velocity at ankle height reaches 0.2 m/s at 3m from the inlet walls. This, many will agree, is a too great area to leave unoccupied.

The body heat will cause an upward pull from the clean supply air layer at floor level up to the breathing zone as can be observed from the CO_2 contours. Nilsson (2003 p237) writes about investigations that show that as long as the cleaner air supply layer is thicker than 0.3 m. enough fresh air will reach the person. However; this only valid for a person who is not moving. One can easily imagine that a teacher or a conference chairman etc. being both taller than those seated and more active, could have a different perception of the indoor climate. But it is important to note that the divison in an upper and lower zone is not as prominent as for traditional displacement flow.

5.3.3 Control strategy

Controlling the ventilation rate by means of a thermostat only would cause the ventilation rate to stop completely in the event of low occupation and very cold weather. (The transmission loss is greater than the heat surplus). There is therefore a need to assure minimum ventilation is present. This could either be controlled by a relative humidity sensor or carbon dioxide sensor.

The occurance of outdoor temperatures below freezing point has yet to be mentioned in this report. This would require room heating as well as preheating ventilation air in which case a run around heat recovery unit could be used. Considering that it is possible to supply air at low temperatures, the need for a high temperature efficiency in the heat recovering unit is not essential. It is essential that the heat recovering unit does not add significantly to the pressure drop.

The results emphasize that the system is best suited for mild coastal climates.

5.4 Further work

5.4.1 Determination of draft risk

According to comfort criteria (See chapter 2.4) the vertical temperature difference should be less than 3ºC meaning the absolute minimum should be 17 ºC at ankle height (0.1m) and the velocity should be kept below 0.15-0.25 m/s depending on the season. To determine the risk of draft in the occupied zone it is much more important to know the temperature and velocity after the plume hits the floor. The central temperature and velocity of the plume can be calculated (see chapter 2.1). but this is the minimum temperature of the plume while it is still has a downward direction. It will have gained heat from surrounding air, walls and floor by the time it reaches the nearest person. Little research exists concerning what happens to vertical fluid flow such as cold drafts when the boundary layer flow meets the surface vertical to it. However experiments by Heiselberg (1994) on cold draft show the following correlations of the horizontal flow in the region x =0.4 m - 2 m from the vertical surface from which the cold draft originates.

$$U_{\max}(x) = 0.096\frac{\sqrt{(h\Delta T}}{x+1.32} \text{ [m/s]} \qquad (5.1)$$

Where $\sqrt{h\Delta T}$ is the maximum velocity of the cold draft at the bottom of the cold surface.

$$T_f(x) = T_r - (0.3 - 0.034x)\Delta T \text{ [ºC]} \qquad (5.2)$$

T_r is the reference/room temperature and ΔT is the difference between the cold surface(minimum temperature of draft) and the reference temperature.

This can obviously not be directly applied to the scenario of a supply air plume such as in the Swedish model, but it gives an idea of the behaviour and order of magnitude from which the velocity is reduced and temperature increased after reaching the floor. If, for example, one is interested in the velocity and temperature 1.5 m from the wall. this would be approximately 34 % of the maximum vertical velocity and the temperature difference would have diminished to 25 %. This leads to an estimate that to avoid draft in the occupied zone the maximum plume velocity (downward direction) 1m above floor should be maximum 3 times as large as the draft velocity criteria (0.15-0.25 m/s depending on season). The temperature difference should be no more than 4 times the vertical temperature difference criteria of 3 ºC. Investigating the applicability of equations (5.1) and (5.2) to vertical inlet plumes needs to be done in order to use this in in guidelines for dimensioning.

The velocity and temperature at ankle height is as the plume central velocity and temperature dependant of the inlet temperature. and air flow. One can imagine a graph like Figure 49 that shows the correlation of the draft in the occupied zone versus the inlet temperature for various inlet air flows, q_s, where the horizontal velocity at a chosen distance from the wall is plotted in stead of the plume velocity. The tolerance for draft would increase somewhat with higher temperatures, so a line could be drawn for velocities at 0.15-0.25 m/s (see chapter 2.4.2) for which one should remain below (marked in yellow). One could then find the combination of minimum supply air temperature and maximum supply air flow through an inlet where there is no draft problem. This might or might not be agreeable with the air flow dimensioned from CO_2 criteria and heat balance. The line will correspond to acceptable velocities where the nearest person is seated. This would be the near zone. and the length and width of it may be limited by size and use of room.

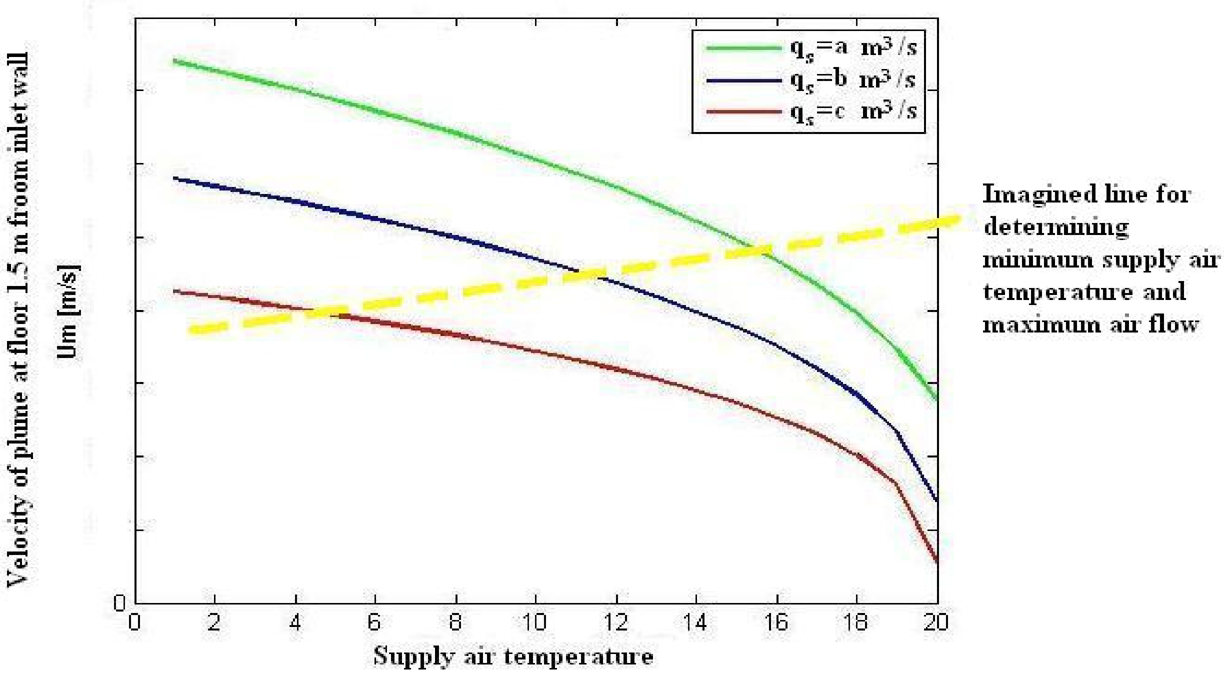

Figure 49 Maximum horizontal velocity of plume entering occupied zone 1.5 m from inlet wall

5.4.2 Simulation of whole classroom

Simulations should be done for an entire classroom in order to study the effects the remaining walls have on the air flow pattern. Displacement flow creates thermal stratification in a room. In order for the temperature in the occupied zone to be 20 ºC. the average temperature in the room should be 21 ºC. As discussed with the lack of a radiation model. the vertical temperature difference is probably larger than in reality. This emphasises the importance of doing simulations for a whole classroom applying a radiation model. It might also be interesting doing transient simulation to see the development of the air quality and thermal comfort during class.

5.4.3 Improvements and alternative design of inlet

To further investigate the validity of the theoretical expressions, a comparison of a typical inlet used in Swedish natural ventilation schools with the approximations of the theory done in chapter 3.1 should be done. Different designs may be relevant; circular, rectangular, with a plate in front, or with a device for spreading the air flow shaped like a "cup" in front to help better mixing with the surrounding air. These should be studied as the width of the plume may influence the number of inlets possible to place on a wall as well as the central temperature, velocity and air flow, and thereby CO_2 concentration. draft etc.

5.4.4 Study of ventilation components

A study of the ventilation components should be done in order to assist in developing a concept that takes into consideration the fan power, heat recovery, cooling capacity of the culvert etc. The pressure drops throughout the system should be estimated as to study the need for fan power.

6 Conclusion

This report has aimed to develop a method for dimensioning classroom ventilation using the Swedish natural ventilation model as well as studying typical scenarios for winter, spring and summer. This was done with the use of empirical expressions of plume behaviour and CFD simulations.

The simulation of a classroom cross section showed the expected air flow pattern, although suffered from the use of symmetry planes and lack of a radiation model. This led to high velocities in the occupied zone.

One of the reasons why displacement flow is not common practise as classroom ventilation is because of the area requirements of a near zone. This often comes in conflict with the furniture, either by limiting the use of area, or by the furniture obstructing the ventilation. The Swedish model is no exception; 1.5 m from the inlet wall conditions concerning draft were not acceptable although somewhat aggravated by the lack of a radiation model.

For supply air temperatures above 5ºC the ventilation rate was such that the internal heat production covers the heating need and the carbon dioxide levels are at adequate levels.

The empirical expressions corresponded well to the CFD simulations although the air flow deviated somewhat. These can therefore be used with success for preliminary design and dimensioning. Some areas are still left to be developed such as guidelines for inlet design and placement and a study of how the resulting inlet air flow corresponds to theory.

7 References

Airson (2008), *Skoleventilasjon* [online] Available at: www.airson.no Accessed 02.05.08

Andersson , T., Håkan Gillbro, H*, Fredkulla- och Risebergsskolan - två av många moderna självdragsskolor.*
Bygg & Teknik, 6/96 DELTAte, Göteborg [online] Available at:
http://www.deltate.se/artiklar/Sjalvdrag/Sjalvdragsskolor/fredk_riseberg.htm Accessed 19.06.08

ANSYS CFX Help(2007a), *Reynolds Averaged Navier-Stokes (RANS) Equations*, Release 11.0

ANSYS CFX Help(2007b), *Best Practices Guide for HVAC*, Release 11.0

ANSYS CFX Help(2007c), *Creating a Surface Plot of y+*, Release 11.

ANSYS CFX Help(2007d), *Modeling Flow Near the Wall,* Release 11.0

ANSYS CFX(2007e) *Tutorial 17: air conditioning simulation* (Release 11.0)

Auranor (2008) *Siv-inn 1 og Siv-inn 2000 tilluftsventiler for fortrengningsventilasjon.* [online] Available at:
http://www.auranor.no/no/service/download_center/downloads/pdf/produktePdf/siv_inn/si_1_kvart_halv.pdf Accesed: 02.05.08 (in Norwegian)

Byggforsk- Norwegian Building Research Institute (1996), *Fordeling av ventilasjonsluft i rom,* 552.351 [online]. Available at: http://bks.byggforsk.no/index.asp?docNumber=552351 Acessessed 14.06.08 (in Norwegian)

Byggforsk- Norwegian Building Research Institute (2005). *Inneklima og ventilasjon i skoler.* [online] Available at:
http://bks.byggforsk.no/index.asp?docNumber=552311 Accessed: 19.05.08 (in Norwegian)

Byggforsk- Norwegian Building Research Institute (1999*). Krav til luftmengder i ventilasjonsanlegg 421.505* SINTEF[online]
http://bks.byggforsk.no/index.asp?docNumber=421503 Accessed 31.07.08 (in Norwegian)

Byggforsk- Norwegian Building Research Institute (1999)*Temperaturforhold og lufthastighet. betingelser for termisk komfort*. 421.501 [online] Available from :
http://bks.byggforsk.no/index.asp?docNumber=421501 (in Norwegian)

CFD Online (2006)*Dimensionless wall distance (y plus)* [online] Available at:
http://www.cfd-online.com/Wiki/Dimensionless_wall_distance_%28y_plus%29).
Accessed 13.05.08

Direcorate for Primary and Secondary Education (Utdanningsdirektoratet) (2004) *Nasjonal Rådgivningstjeneste: Skoleanlegg*. [online] Available at: http://skoleanlegg.utdanningsdirektoratet.no/index.gan?id=1436&subid=0 Accessed: 03.05.08

Dokka, T. H., Mysen, M., Schild, P. & Tjelflaat, P. O. (2003) *Bygningsintegrert ventilasjon: en veileder*. [Oslo], Økobygg.

Dyrhaug, P.E, *Ventilasjon av klasserom-hva ønsker vi å oppnå og hvilke kriterier legger vi til grunn for valg av ventilasjonsprinsipp?* [online] Available at: www.airson.no [Acsessed 20.06.2008]

eklima(2008), *Månedsnormaler(1961-1990) [online]*
Available at http://eklima.met.no Accessed 30.07.08 (Climate data from The Norwegian Meteorological Institute for Oslo)

EN 13779(2004) *Ventilation for non residential buildings* CEN/TC156/WG7

Fanger, P. O. (1972) *Thermal comfort: analysis and applications in environmental engineering,* New York, McGraw-Hill.

Gjerstad, Frode Olav, Hanssen,Sten Olav, Novakovic, Vojislav, Thue, Jan Vincent & Wangensten, Ivar(2007)., *Enøk i bygninger,* Trondheim, Gyldendal undervisning.

Halla, H. (2006) *Romventilasjon i naturlig ventilerte klasserom.* Department of Energy and Process Engineering, Norwegian University of Science and Technology.

Heiselberg, P (1994) *Draught risk from cold vertical surfaces.* Building and Environment, Vol. 29, No. 3, pp. 297-301, Pergamon.

Incropera, F. P. & Dewitt, D. P. (2002) *Fundamentals of heat and mass transfer,* New York, Wiley.

Mysen, M. (2005) Ventilation systems and their impact on indoor climate and energy use in schools: studies of air filters and ventilation control. Trondheim, Norwegian University of Science and Technology, Faculty of Engineering Science and Technology, Department of Energy and Process Engineering.

National Office of Building Technology and Administration (2007), FOR 1997-01-22 nr 33: Forskrift om krav til byggverk og produkter til byggverk (TEK). 4th edition, (Statens bygningstekniske etat), Oslo. [online]
Available from: http://www.lovdata.no/for/sf/kr/xr-19970122-0033.html
Acessed 04.05.08 (in Norwegian)

National Office of Building technology and Administration (2007), *Veiledning til teknisk forskrift til plan-og bygningsloven 1997*, 4th edition, (Statens bygningstekniske etat), Oslo. [online]
Available from: http://www.be.no/beweb/regler/veil/tekveil07/tekveilinnh2007.html
Acessed 04.05.08 (in Norwegian) (often referred to as REN)

Nilsson, P. E.(ed.) (2003) *Achieving the desired indoor climate: Energy Efficiency Aspects of System Design,* Lund, Studentlitteratur.

NILU, (2003) *Problematisk inneklima i mange norske skoler*, press release 12th nov 2003, Availble at:
http://www.nilu.no/index.cfm?ac=topics&text_id=10819&folder_id=4461&view=text&crit=SKOLE
Accessed: 08.05.08

Norwegian board of Health Supervision, The(Statens helsetilsyn),1998, *Veileder til forskrift om miljørettet helsevern i barnehager og skoler m.v.* [online]. Oslo. Available from :
http://www.helsetilsynet.no/upload/Publikasjoner/veiledningsserien/veileder_forskrift_miljoerrettet_helsevern_skoler_barnehager_ik-2619.pdf
Acessed: 16.05.2008 (in Norwegian)

NS 3031 (2007) *Beregning av bygningers energiytelse: metode og data,* Lysaker, Standard Norge.

Nørsett, S. E.(2008)l, personal communication, Summer 2008

PrNS3563(2002), *Ventilasjon i bygninger –Dimensjoneringsmetoder for inneklima (Ventilations for buildings design methods for indoor climate),* Lysaker, Standard Norge.

Rustad, P.O(2008) *Email communication.* Undervisningsbygg Oslo KF

Schild, P.G. (1997) *Accurate prediction of indoor climate in glazed enclosures*, Department of Refrigeration and air conditioning, Faculty of Mechanical Engineering, Norwegian University of Science and Technology. (Phd dissertation)

Skåret, E.(2000), *Ventilasjonsteknisk håndbok*, Byggforsk håndbok 48, Oslo: Østfold trykkeri AS.

Tjelflaat, P.O (2008), Personal communication, Summer 2008.

Vik, T.A. (2008), Personal communication, Summer 2008.

APPENDIX 1: Matlab scripts

This appendix comprise the matlab scripts based on the equations (2.1-2.7) in the theory chapter as well as (3.5) and (3.6) in the method chapter. There are 3 scripts:

1. Main script: Plume central temperature, central velocity and air flow.
2. Input to main script: vertical plume model
3. Input to main script: CFD simulation output (used in chapter 3.2)

1. Main script: Plume central temperature, central velocity and air flow.

```
%%-------------------------------------------------------------------------
clc %clear screen
clear
close all
%Inlet air plume modeled as imaginary cold plate in ceiling next to wall.
%Equations from Skåret, 2000 pp 65-68. (Theory, chapter 2)

%% Input


cp=1005;         % [kJ/kgK]
g=9.81;          % [m/s^2]
beta=1/300;      % [1/K]

ro=0.2;          % [m] inlet opening radius
r=ro             % [m] diameter of imaginary heat source. A'=2Ao,
r'=ro*2^0.5
yp= 0.7*2*r;     % [m] pole distance for free plume

room_temp=20;    % [K] Temperature upper zone
inlet_temp=0;    % [K] Supply air temperature/outdoor temp
delta_T=room_temp-inlet_temp

rho=1.205*(273.15+room_temp)/(273.15+inlet_temp);% [kg/m3]
%qvo=            % [m3/s]Supply air flow max 270 l/s / 7inlets=0.038 m3/s
Uo= 0.334509


% qvo/(pi*r^2)
qvko=Uo*pi*ro^2;
%[kW], convective heat as free plume, double of actual wall plume.
Pk= rho*cp*(qvko)*(delta_T)/1000;
ny=14.4 *10^-6; % kinematic viscosity for air at 283 K


% Grashof number
L=2*ro;
Gr_conv=g*beta*(20.9 * Pk^(2/3) * (yp)^(-5/3))*(L)^3/ny^2
Gr_vert=g*beta*(       delta_T               )*(L)^3/ny^2;
%Cb=tand(12.5)
%As=pi*(Cb*(2.5+yp))^2

%% Central velocity / maximum velocity of buoyancy plume
%y [m], distance from source.
```

```
for k= 1:40      % y=0.1-4 m
  y=(k-1)/10;
% Central velocity
Um(k) =1.28*( (Pk/(y+yp))^(1/3)) ;% [m/s]

% Central Temperature
deltaTm(k) = 20.9 * Pk^(2/3) * (y+yp)^(-5/3) ;
%if deltaTm(y)> (room_temp-inlet_temp) % The temperature can never be lower
 %  Temperature_convection(y)=10 ;                   % than the inlet
temperature.
%else
Temperature_convection(k)=room_temp-deltaTm(k);
%end

% Volume Flow in Plume
qvk(k)=  0.5*(0.055*Pk^(1/3) * (y+yp)^(5/3)) ; % [m3/s] a wall plume is the
half of a free plume

end
temp_conv_occupied=Temperature_convection(31)

%% Run vertical plue calculations and The CFD simulation output

run CFD_plume
run Vertical %Vertical plume

%%
_____________Plot__________________________________________________________


figure
 hold on;
 plot(Um_y, 'r-', 'linewidth',2)   %vertical
 plot(Um, 'y-', 'linewidth',2)      %convection
 plot(Um_vertical_CFD(:,2), 'g-', 'linewidth',2) %decreases as surrounding
air is mixed in
 plot(Um_vertical_wall_CFD(:,2), 'b-', 'linewidth',2)
 plot(Um_convection_CFD(:,2),'k-', 'linewidth',2)
 plot(Um_convection_wall_CFD(:,2),'m-', 'linewidth',2)
   axis([1 40 0 1])
 box
legend( 'Vertical', 'Convection','CFD Vertical', 'CFD Vertical wall', 'CFD
Convection','CFD Convection wall')
%title(' Comparison of plumes')
ylabel('Central velocity, Um [m/s]','fontweight','b');
xlabel('Distance from inlet, y [m]','fontweight','b');
  set(gcf,'color',[1,1,1]) %white background
   set(gca,'XTick',0:10:41)
     set(gca,'XTickLabel',{'0','1','2','3','4'})



    figure

     hold on;
     plot(Temperature_vertical, 'r-', 'linewidth',2)
     plot(Temperature_convection, 'y-', 'linewidth',2)
```

```
      plot(Tm_vertical_CFD(:,2),'g-', 'linewidth',2)
      plot(Tm_vertical_wall_CFD(:,2),'b-', 'linewidth',2)
      plot(Tm_convection_CFD(:,2), 'k-', 'linewidth',2)
      plot(Tm_convection_wall_CFD(:,2),'m-', 'linewidth',2)
      axis([1 41 0 22])
      box
      legend(  'Vertical ','Convection ','CFD Vertical','CFD Vertical
Wall','CFD Convection  ', 'CFD convection wall', 'location','southeast')
      ylabel('Tm [C]','fontweight','b')
      xlabel('Distance from inlet, y [m]','fontweight','b')
      %title('Central Temperature of Plume [C]')
      set(gcf,'color',[1,1,1])
      set(gca,'XTick',0:10:41)
      set(gca,'XTickLabel',{'0','1','2','3','4'})



figure
hold on;
plot(qvk, 'r-', 'linewidth',2) %convection plume
plot(qv_y, 'linewidth',2)      %vertical plume
 box
      legend('Convection Plume', 'Vertical plume')
%title(' Air Flow')
ylabel('Air Flow, qvk [m3/s]','fontweight','b')
xlabel('Distance from inlet, y [m]','fontweight','b')
 axis([1 41 0 0.2])
 set(gcf,'color',[1,1,1])
 set(gca,'XTick',0:10:41)
 set(gca,'XTickLabel',{'0','1','2','3','4'})
%____________________________________________________________________________
close all
%% CO2 concentrations

%CO2 production, P_CO2

no_inlets=7;
qvko_total= 0.5*qvko*no_inlets     % Fresh air supply for classroom
                                   % OBSOBS, half of free plume

co2_child= 19/(3600*1000);        % [m3/s] (19 l/h/ 14-16 yr old)

P_CO2= co2_child*30;              % [m3/s]


qv_e= qvko_total*((room_temp+273.15)/(inlet_temp+273.15));     %air flow
through outlet
C_upper= (1e6)*P_CO2/(qv_e);      %=Ce ppm [m3/s / m3/s ]
C_upper_mg_m3=1.8*C_upper;         % mass concentrarion
C_inlet= 0;                       % 400 ppm in outdoor air/supply air

%CO2 level convection plume model
%Air flow at 1m above floor (entering occupied zone/fresh air layer) with
ceiling height 4 m.
 air_flow_3m=qvk(31)*no_inlets;        % [m3/s] air flow at 1 m above floor,
y=3 m.
qv_induced= air_flow_3m-qvko_total;    % Amount of air in plume from upper
zone
```

```
C_occupied_convection_ppm= (qv_induced*C_upper + 0.5*qvko*C_inlet) /
air_flow_3m  %ppm
C_occupied_convection_mg_m3= 1.8*(qv_induced*C_upper + 0.5*qvko*C_inlet) /
air_flow_3m;% mg/m3

%CO2 level vertical plume model
%Air flow at 1m above floor (entering fresh air layer) with ceiling height
4 m.
 air_flow_3m=qv_y(31)*no_inlets;       % [m3/s] air flow at 1 m above
floor, y=3 m.
qv_induced= air_flow_3m-qvko_total;   % Amount of air in plume from upper
zone
C_occupied_vertical_mg_m3= 1.8*(qv_induced*C_upper + 0.5*qvko*C_inlet) /
air_flow_3m;  %ppm



REN=(0.009*30 + 0.001*70);
Percentage_of_REN=100*qvko_total/REN

%%-----------------------------------------------------------------------------
```

2. Input to main script: vertical plume model

```
%%-----------------------------------------------------------------------------
% Um empirical and CFD
%Vertical plume with buoyancy


%Input
%ro=0.1;         % [m] radius of imaginary opening.
Ao= pi*ro^2;    % [m2]Inlet area. Double of that of a wall plume.
Cb=tand(12.5); % The angle of expansion is approximately 12,5 degrees
yp=ro/Cb;       % [m]pole distance
g=9.81;         % [m/s2] acceleration of gravity
beta=1/300;     % [1/K]
%room_temp=20;  % [deg celcius]
%inlet_temp=10;

b=Cb*(2+yp);
%% Vertical plume varying with distance, y, from inlet

%Uo=0.2;                 % [m/s]initial velocity varies from 0-
qvo=Ao*Uo;               % Air flow through inlet
%delta_T=room_temp-inlet_temp;            % Temperature difference between
plume and room
Aro=g* Ao^0.5 *beta * delta_T * (1/Uo^2) % Archimedes number

for i=1:41              % i is a counter
y=(i-1)/10;                 % [m]Distance from inlet. max 4 m

%Calculation of Um, central velocity p 59 Skåret
Um_y(i)= Uo* ( Aro*(4.3/Cb^2)*((Ao^0.5)/(y+yp)) +
((1.54/Cb)*((Ao^0.5)/(y+yp)) )^3)^(1/3); % [m/s]Central velocity
```

```
%Calculation of central temperature Skåret p59
deltaTm_y(i)=delta_T* ( (4.3/Cb^2)*(Ao/(y+yp)^2) *  (
Aro*(4.3/Cb^2)*((Ao^0.5)/(y+yp)) + ((1.54/Cb)*((Ao)^0.5/(y+yp)) )^3)^-(1/3)
);
Temperature_vertical(i)=room_temp-deltaTm_y(i);


% Calculation of qv
    b= Cb*(y+yp);       % width of plume at distance y, r_max=b
 As=pi*b^2;          % Cross section area of free plume at y m from inlet

    % r is the distance normal to the central axis of the plume till b.

    for j=2:(1+round(100*b))   %j is a counter. r goes from 0 to b

        r(i,1)=0; %j=1 is the center of the axis
        r(i,j)= j/100;

        U(i,1)=Um_y(i);
        U(i,j)= U(i,1)* (1 - (r(i,j)/b)^1.5)^2;

      % Integral of velocity over the cross section area
        qv_partial(i,1)=0.5*pi*U(i,1)*((r(i,2)^2 - r(i,1)^2));        %Air
flow
        qv_partial(i,j)=  0.5*pi*( U(i,j)*((r(i,j)^2- r(i,j-1)^2) )); %Air
flow qv(i,j-1)+

    end %j
    qv_y(i)=sum(qv_partial(i,:)); % integral from r=0 to b

end

temp_vert_occupied=Temperature_vertical(31)

%%-------------------------------------------------------------------------
```

3. Input to main script: CFD simulation output (used in chapter 3.2)

```
% CFD

%% Um along central axis
% OBS! Y=0 is at floor level, Y 4m = ceiling
% Y [ m ],Velocity [ m s^-1 ]

% Vertical isoterm Um

Um_vertical_isoterm= [4, 0.199841
3.89744, 0.203647
3.79487, 0.204773
3.69231, 0.202054
3.58974, 0.197925
3.48718, 0.193251
3.38462, 0.189782
3.28205, 0.185144
3.17949, 0.178008
3.07692, 0.175012
```

```
%% Vertical plume delta T =10
Um_vertical_CFD=[4, 0.201829
3.89744, 0.299858
3.79487, 0.366609
3.69231, 0.406319
3.58974, 0.425754
3.48718, 0.437601
3.38462, 0.453305
3.28205, 0.463517
3.17949, 0.465699
3.07692, 0.465684
```

```
2.97436, 0.16828
2.87179, 0.161778
2.76923, 0.154664
2.66667, 0.14548
2.5641, 0.139722
2.46154, 0.133739
2.35897, 0.128565
2.25641, 0.12268
2.15385, 0.117533
2.05128, 0.11332
1.94872, 0.109105
1.84615, 0.104461
1.74359, 0.100551
1.64103, 0.0969856
1.53846, 0.0934921
1.4359, 0.0903379
1.33333, 0.0871181
1.23077, 0.0847369
1.12821, 0.0822012
1.02564, 0.0795893
0.923077, 0.0771415
0.820513, 0.0742626
0.717949, 0.0717524
0.615385, 0.0688848
0.512821, 0.065122
0.410257, 0.0601933
0.307693, 0.0529665
0.205128, 0.0417751
0.102564, 0.0247598
2.23517e-007, 0.00157532];


%Vertical plume temperature

Tm_vertical_CFD=[0, 283.15
1, 283.437
2, 284.775
3, 285.746
4, 286.742
5, 287.615
6, 288.186
7, 288.671
8, 289.143
9, 289.562
10, 289.959
11, 290.275
12, 290.561
13, 290.833
14, 291.054
15, 291.274
16, 291.469
17, 291.672
18, 291.843
19, 291.978
20, 292.079
21, 292.162
22, 292.238
23, 292.318
24, 292.413
25, 292.505
```

```
2.97436, 0.461695
2.87179, 0.459145
2.76923, 0.455839
2.66667, 0.449878
2.5641, 0.445527
2.46154, 0.437687
2.35897, 0.429072
2.25641, 0.416499
2.15385, 0.404111
2.05128, 0.395141
1.94872, 0.388875
1.84615, 0.383808
1.74359, 0.37946
1.64103, 0.372567
1.53846, 0.362468
1.4359, 0.350558
1.33333, 0.336604
1.23077, 0.323805
1.12821, 0.31095
1.02564, 0.298439
0.923077, 0.286845
0.820513, 0.269589
0.717949, 0.255677
0.615385, 0.242854
0.512821, 0.22554
0.410257, 0.206528
0.307693, 0.189178
0.205128, 0.175257
0.102564, 0.176762
2.23517e-007, 0.199503];


%% Vertical wall plume delta T =10,
vs. chart count (0-4 m
Um_vertical_wall_CFD=[0, 0.201876
1, 0.295071
2, 0.352289
3, 0.385665
4, 0.400823
5, 0.410809
6, 0.425476
7, 0.436083
8, 0.440573
9, 0.444125
10, 0.444068
11, 0.445411
12, 0.44597
13, 0.44361
14, 0.443325
15, 0.440845
16, 0.438802
17, 0.434676
18, 0.430771
19, 0.429794
20, 0.429629
21, 0.429425
22, 0.4291
23, 0.428306
24, 0.426258
25, 0.423648
```

```
26, 292.595
27, 292.672
28, 292.738
29, 292.795
30, 292.84
31, 292.898
32, 292.937
33, 292.964
34, 292.993
35, 293.018
36, 293.022
37, 292.989
38, 292.907
39, 292.948]-273.15;


Tm_vertical_wall_CFD=[0, 283.15
1, 283.529
2, 284.969
3, 285.966
4, 286.919
5, 287.71
6, 288.201
7, 288.601
8, 288.979
9, 289.306
10, 289.625
11, 289.873
12, 290.106
13, 290.338
14, 290.522
15, 290.702
16, 290.854
17, 291.011
18, 291.147
19, 291.252
20, 291.338
21, 291.411
22, 291.482
23, 291.547
24, 291.621
25, 291.691
26, 291.761
27, 291.812
28, 291.859
29, 291.901
30, 291.945
31, 291.998
32, 292.029
33, 292.053
34, 292.086
35, 292.117
36, 292.139
37, 292.158
38, 292.164
39, 292.358]-273.15;
```

```
26, 0.419968
27, 0.417989
28, 0.415977
29, 0.413304
30, 0.40959
31, 0.403003
32, 0.397784
33, 0.390629
34, 0.377744
35, 0.358669
36, 0.33114
37, 0.286611
38, 0.200526
39, 0.0247023 ];


%% Convection plume
Um_convection_CFD=[4, 0.124383
3.89744, 0.45782
3.79487, 0.475052
3.69231, 0.485452
3.58974, 0.479421
3.48718, 0.473436
3.38462, 0.473897
3.28205, 0.472681
3.17949, 0.468621
3.07692, 0.467131
2.97436, 0.461572
2.87179, 0.45748
2.76923, 0.451744
2.66667, 0.442335
2.5641, 0.43549
2.46154, 0.428493
2.35897, 0.423934
2.25641, 0.419441
2.15385, 0.414641
2.05128, 0.410024
1.94872, 0.403473
1.84615, 0.394557
1.74359, 0.385442
1.64103, 0.377611
1.53846, 0.370972
1.4359, 0.364991
1.33333, 0.358554
1.23077, 0.35195
1.12821, 0.344376
1.02564, 0.335305
0.923077, 0.324082
0.820513, 0.314154
0.717949, 0.3035
0.615385, 0.291107
0.512821, 0.278098
0.410257, 0.261363
0.307693, 0.237653
0.205128, 0.20377
0.102564, 0.158822
2.23517e-007, 0.13696];
```

```
% Convection plume Temperature
Tm_convection_CFD=[0, 198.166
1, 255.935
2, 272.145
3, 278.307
4, 282.211
5, 284.694
6, 286.231
7, 287.324
8, 288.18
9, 288.82
10, 289.408
11, 289.87
12, 290.282
13, 290.674
14, 290.977
15, 291.235
16, 291.434
17, 291.612
18, 291.768
19, 291.909
20, 292.046
21, 292.182
22, 292.303
23, 292.404
24, 292.488
25, 292.56
26, 292.627
27, 292.688
28, 292.752
29, 292.813
30, 292.879
31, 292.93
32, 292.977
33, 293.023
34, 293.057
35, 293.084
36, 293.106
37, 293.114
38, 293.086
39, 293.154]-273.15;


%% Convection Plume, velocity along
axis through max 1.5 cm
%% from wall
Um_convection_wall_CFD=[0, 0.152212
1, 0.296673
2, 0.345572
3, 0.378066
4, 0.385655
5, 0.389589
6, 0.402019
7, 0.408149
8, 0.405788
9, 0.40159
```

```
Tm_convection_wall_CFD=[0, 228.359
1, 273.18
2, 279.257
3, 282.374
4, 284.709
5, 286.285
6, 287.203
7, 287.94
8, 288.606
9, 289.166
10, 289.639
11, 289.983
```

```
10, 0.397147
11, 0.396108
12, 0.396182
13, 0.395264
14, 0.395067
15, 0.392118
16, 0.388557
17, 0.382774
18, 0.378594
19, 0.378271
20, 0.380482
21, 0.383669
22, 0.386661
23, 0.387604
24, 0.3863
25, 0.384126
26, 0.380834
27, 0.378989
28, 0.37748
29, 0.375777
30, 0.373914
31, 0.368473
32, 0.364751
33, 0.359627
34, 0.348598
35, 0.331603
36, 0.306904
37, 0.26639
38, 0.189566
39, 0.0668623];
```

```
12, 290.267
13, 290.517
14, 290.727
15, 290.938
16, 291.126
17, 291.318
18, 291.466
19, 291.571
20, 291.638
21, 291.684
22, 291.73
23, 291.786
24, 291.856
25, 291.925
26, 291.99
27, 292.038
28, 292.077
29, 292.108
30, 292.134
31, 292.173
32, 292.191
33, 292.201
34, 292.221
35, 292.241
36, 292.251
37, 292.256
38, 292.244
39, 292.385]-273.15;
```

APPENDIX 2: Dimensioning of ventilation rate spreadsheets

The following pages comprise prints of the Excel spreadsheetsused for dimesioning ventilation rate.

1. Effect Balance classroom
2. CO2-levels calculated from the two plume models
3. Optimal air volumes
4. Simulation input
5. Relative humidity levels

1 Effect Balance classroom

Settings		
Indoor Temperature, Ti	20	degC
Floor area	70	m2
Volume=7*10*4	280	m3
Facade =10m*4m	29,5	m2
Window (15% of floor area)	10,5	m2
U value facade	0,2	W/m2K
U value window	1	W/m2K
sumU*Area	16,4	W/K
Infiltration, n	0	air ch/h
Heat capacity air, cp	1,005	kW/kgK
Density air , rho	1,205	kg/m3
Reccomended air volumes /pers	**0,0070**	**m3/s/pers**
Reccomended air volumes/m2	**0,0010**	**m3/s/m2**
Heat from lighting , Pl	0,7	kW
Heat emission per person , Pp	**0,080**	kW
Transmission losses	Pt =sumU*Area*delta_T	
Infiltration losses	Pi =n*V*rho*cp*delta_T	
Ventilation losses	Pv =q*rho*cp*delta_T	
Internal Heat Generation	Pp+Pl	
Room Heating required, Ph	**Ph+Pt+Pi+Pv+Pp+Pl=0**	

Reccomended air flow: 100%REN = 7 l/s/pers* and 1 l/s/m2

Air flow [m3/s]			
Percentage of REN	**Number of people**		
	30	20	10
100 %	**0,280**	**0,210**	**0,140**
80 %	**0,224**	**0,168**	**0,112**
60 %	**0,168**	**0,126**	**0,084**
40 %	**0,112**	**0,084**	**0,056**
20 %	**0,056**	**0,042**	**0,028**

Reccomended air flow 30 people: 100%REN = 7l/s/pers* + 1/s/m2*70m2/30pers= 9,33l/s/pers

Air flow: l/s/person			
Percentage of REN	**Number of people**		
	30	20	10
100 %	**9,3**	**10,5**	**14**
80 %	**7,5**	**8,4**	**11,2**
60 %	**5,6**	**6,3**	**8,4**
40 %	**3,7**	**4,2**	**5,6**
20 %	**1,9**	**2,1**	**2,8**

Temperature outdoors: -10 degC

delta T	**-30**		
	No.of children		
	30	20	10
Internal generation of heat	3,1	2,3	1,5
Transmission losses	-0,492	-0,492	-0,492
Infiltration losses	0,00	0,00	0,00
Sum	**2,6**	**1,8**	**1,0**

Percentage of REN	**Ventiltation Heat Losses [kW]**		
100 %	-10,17	-7,63	-5,09
80 %	-8,14	-6,10	-4,07
60 %	-6,10	-4,58	-3,05
40 %	-4,07	-3,05	-2,03
20 %	-2,03	-1,53	-1,02

Percentage of REN	**Total Balance [kW]**		
100 %	**-7,6**	**-5,8**	**-4,1**
80 %	**-5,5**	**-4,3**	**-3,1**
60 %	**-3,5**	**-2,8**	**-2,0**
40 %	**-1,5**	**-1,2**	**-1,0**
20 %	**0,6**	**0,3**	**0,0**

Temperature outdoors: -5 degC

delta T	**-25**		
	No.of children		
	30	20	10
Internal generation of heat	3,1	2,3	1,5
Transmission losses	-0,41	-0,41	-0,41
Infiltration losses	0,00	0,00	0,00
Sum	**2,7**	**1,9**	**1,1**

Percentage of REN	**Ventiltation Heat Losses [kW]**		
100 %	-8,48	-6,36	-4,24
80 %	-6,78	-5,09	-3,39
60 %	-5,09	-3,81	-2,54
40 %	-3,39	-2,54	-1,70
20 %	-1,70	-1,27	-0,85

Percentage of REN	**Total Balance [kW]**		
100 %	**-5,8**	**-4,5**	**-3,1**
80 %	**-4,1**	**-3,2**	**-2,3**
60 %	**-2,4**	**-1,9**	**-1,5**
40 %	**-0,7**	**-0,7**	**-0,6**
20 %	**1,0**	**0,6**	**0,2**

Temperature outdoors: 0 degC			
delta T -20			
	No.of children		
	30	20	10
Internal generation of heat	3,1	2,3	1,5
Transmission losses	-0,328	-0,328	-0,328
Infiltration losses	0,00	0,00	0,00
Sum	**2,8**	**2,0**	**1,2**
Percentage of REN	**Ventiltation Heat Losses [kW]**		
100 %	-6,78	-5,09	-3,39
80 %	-5,43	-4,07	-2,71
60 %	-4,07	-3,05	-2,03
40 %	-2,71	-2,03	-1,36
20 %	-1,36	-1,02	-0,68
	Total Balance [kW]		
100 %	**-4,0**	**-3,1**	**-2,2**
80 %	**-2,7**	**-2,1**	**-1,5**
60 %	**-1,3**	**-1,1**	**-0,9**
40 %	**0,1**	**-0,1**	**-0,2**
20 %	**1,4**	**1,0**	**0,5**

Temperature outdoors: 5 degC			
delta T -15			
	No.of children		
	30	20	10
Internal generation of heat	3,1	2,3	0,8
Transmission losses	-0,246	-0,246	-0,246
Infiltration losses	0,00	0,00	0,00
Sum	**2,9**	**2,1**	**0,6**
Percentage of REN	**Ventiltation Heat Losses [kW]**		
100 %	-5,09	-3,81	-2,54
80 %	-4,07	-3,05	-2,03
60 %	-3,05	-2,29	-1,53
40 %	-2,03	-1,53	-1,02
20 %	-1,02	-0,76	-0,51
Percentage of REN	**Total Balance [kW]**		
100 %	**-2,2**	**-1,8**	**-2,0**
80 %	**-1,2**	**-1,0**	**-1,5**
60 %	**-0,2**	**-0,2**	**-1,0**
40 %	**0,8**	**0,5**	**-0,5**
20 %	**1,8**	**1,3**	**0,0**

Temperature outdoors: 10 degC				
delta T	**-10**			
		No.of children		
		30	20	10
	Internal generation of heat	3,1	2,3	1,5
	Transmission losses	-0,164	-0,164	-0,164
	Infiltration losses	0,00	0,00	0,00
	Sum	**2,9**	**2,1**	**1,3**
	Percentage of REN	**Ventiltation Heat Losses [kW]**		
	100 %	-3,39	-2,54	-1,70
	80 %	-2,71	-2,03	-1,36
	60 %	-2,03	-1,53	-1,02
	40 %	-1,36	-1,02	-0,68
	20 %	-0,68	-0,51	-0,34
	Percentage of REN	**Total Balance [kW]**		
	100 %	**-0,5**	**-0,4**	**-0,4**
	80 %	**0,2**	**0,1**	**0,0**
	60 %	**0,9**	**0,6**	**0,3**
	40 %	**1,6**	**1,1**	**0,7**
	20 %	**2,3**	**1,6**	**1,0**

Temperature outdoors: 12 degC				
delta T	**-8**			
		No.of children		
		30	20	10
	Internal generation of heat	3,1	2,3	1,5
	Transmission losses	-0,1312	-0,1312	-0,1312
	Infiltration losses	0,00	0,00	0,00
	Total	**3,0**	**2,2**	**1,4**
	Percentage of REN	**Ventiltation Heat Losses [kW]**		
	100 %	-2,71	-2,03	-1,36
	80 %	-2,17	-1,63	-1,09
	60 %	-1,63	-1,22	-0,81
	40 %	-1,09	-0,81	-0,54
	20 %	-0,54	-0,41	-0,27
	Percentage of REN	**Total Balance [kW]**		
	100 %	**0,3**	**0,1**	**0,0**
	80 %	**0,8**	**0,5**	**0,3**
	60 %	**1,3**	**0,9**	**0,6**
	40 %	**1,9**	**1,4**	**0,8**
	20 %	**2,4**	**1,8**	**1,1**

Temperature outdoors: 15 degC

delta T **-5**

	No.of children		
	30	20	10
Internal generation of heat	3,1	2,3	1,5
Transmission losses	-0,082	-0,082	-0,082
Infiltration losses	0,00	0,00	0,00
Sum	**3,0**	**2,2**	**1,4**

Percentage of REN	**Ventiltation Heat Losses [kW]**		
200 %	-3,39	-2,54	-1,70
180 %	-3,05	-2,29	-1,53
160 %	-2,71	-2,03	-1,36
140 %	-2,37	-1,78	-1,19
120 %	-2,03	-1,53	-1,02
100 %	-1,70	-1,27	-0,85
80 %	-1,36	-1,02	-0,68
60 %	-1,02	-0,76	-0,51

	Total Balance [kW]		
200 %	**-0,37**	**-0,33**	**-0,28**
180 %	**-0,03**	**-0,07**	**-0,11**
160 %	**0,31**	**0,18**	**0,06**
140 %	**0,64**	**0,44**	**0,23**
120 %	**0,98**	**0,69**	**0,40**
100 %	**1,32**	**0,95**	**0,57**
80 %	**1,66**	**1,20**	**0,74**
60 %	**2,00**	**1,46**	**0,91**

Temperature indoors 25

Temperature outdoors: 15 degC

delta T **-10**

	No.of children		
	30	20	10
Internal generation of heat	3,1	2,3	0,8
Transmission losses	-0,164	-0,164	-0,164
Infiltration losses	0,00	0,00	0,00
Sum	**2,9**	**2,1**	**0,6**

Percentage of REN	**Ventiltation Heat Losses [kW]**		
100 %	-3,39	-2,54	-1,70
80 %	-2,71	-2,03	-1,36
60 %	-2,03	-1,53	-1,02
40 %	-1,36	-1,02	-0,68
20 %	-0,68	-0,51	-0,34

Percentage of REN	**Total Balance [kW]**		
100 %	**-0,5**	**-0,4**	**-1,1**
80 %	**0,2**	**0,1**	**-0,7**
60 %	**0,9**	**0,6**	**-0,4**
40 %	**1,6**	**1,1**	**0,0**
20 %	**2,3**	**1,6**	**0,3**

2 CO2-levels calculated from the two plume models

CO2 levels calculated in Matlab script (Inlet.m) in appendix XX
Compare CO2-levels to corresponding air flow.
Minimum plume temperature is the lowest temperature of the plume entering the occupied zone.

Vertical plu	**CO2-levels[ppm] above outdoor level**				
Grashof=2.0	**Percentage of REN**	**CO2 [ppm]**	**Uo [m/s]**	**n plume tem**	**Archimedes no.**
r=0,2	100 %	372	0,773	10,3	0,39
delta_T= 20	80 %	542	0,618	11,6	0,61
30 persons	60 %	649	0,464	13	1,08
7 inlets	40 %	999	0,309	14,6	2,42
	20 %	2062	0,155	16,6	9,70
Energy bala	**40,29 %**	**1013**	**0,312**	**14,7**	**2,49**
Recomended	44,20 %	900	0,341	14,3	1,99

Ti=20, To=0 mg/m3 (600 ppm above outdoor, 900-1000ppm absolute)

Convection	**CO2-levels[ppm] above outdoor level**				
of REN	**CO2 [ppm]**	**Uo [m/s] ****	**plume**	**Archimedes**	**Grashof no**
100 %	356	0,773	10,3	0,39	3.2639e+009
80 %	459	0,618	12,0	0,61	2.8128e+009
60 %	631	0,464	13,1	1,08	2.3219e+009
40 %	979	0,309	14,6	2,42	1.7719e+009
20 %	2037	0,155	16,6	9,69	1.1162e+009
***40,29%**	972	**0,312**	**14,6**	**2,39**	**1.7806e+009**
43,30 %	900	**0,335**	**14,3**	**2,07**	**1.8673e+009**

Ti=20, To=0 *Energy balance=0
r=0,2, 30 per * *No actual initial velocity for a convection plume model, but a corresponding one

Percentage of REN	**CO2 vertical**	**CO2 convection**
100%	372	356
80%	542	459
60%	649	631
40%	999	979
20%	2062	2037
40,29 %	**1013**	972

3 Optimal Air Volumes

Assuming full classroom, 30 people
Using Vertical Plume model for CO2 calculations

r=0,2m
Indoor Temperature(or else noted) 20 [C]

Acceptable CO2level 900ppm above outdoors

¤ Find minimum air flow for given inlet temperature.
¤ Find heating required
¤ If minimum air flow gives heat surplus, increase air volumes untill energy balance is zero.

Dimensioning Winter Temperature		
Temperature outdoors:	**-20**	
Inlet Temperature	0	**Heat Balance [kW]**
	Number of people	30
	Internal generation of heat	3,1
	Transmission losses	-0,66
	Sum	**2,44**
	% of REN reccomendation	**Ventiltation Heat Loss [kW]**
Minimum air flow [1620 mg/m3]	44,2 %	-3,00
	Heating coil*	-3,00
		Total Balance [kW]
	Total heating required	**3,55**
	Space Heating required	0,55
*Outdoor air needs to be heated to 0 degrees in culvert		
Archimedes number	1,9887	
Grashof number	2.0185e+008	
Inlet velocity, Uo	0,34143	[m/s]
Minimum plume temperature	14,3	[C]
Inlet Mass Flow	0,194	[kg/s]
CO2 mass concentration	**1620**	[mg/m3]
CO2 mass concentr. upper zone	1768	[mg/m3]

Typical Winter Temperature		
Energy balance=0, no heating, CO2 levels high		
Temperature outdoors:	**0**	
Inlet Temperature	0	**Heat Balance [kW]**
	Number of people	30
	Internal generation of heat	3,1
	Transmission losses	-0,328
	Total	**2,8**
	% of REN reccomendation	**Ventiltation Heat Loss [kW]**
Air flow at energy balance	40,9 %	-2,77
		Total Balance [kW]
		0,000
	Space Heating required	0,000
Archimedes number	2,4865	
Grashof number	2.0185e+008	
Inlet, velocity, Uo	0,2951	[m/s]
Minimum plume temperature	14,7	[C]
Inlet Mass Flow	0,171	[kg/s]
CO2 mass concentration	**1823**	[mg/m3]
CO2 mass concentr. upper zone	1977	[mg/m3]
Heating required, CO2-levels OK		
Temperature outdoors:	**0**	
Inlet Temperature	0	**Heat Balance [kW]**
	Number of people	30
	Internal generation of heat	3,1
	Transmission heat loss	-0,328
	Sum	**2,8**
	% of REN reccomendation	**Ventiltation Heat Loss [kW]**
Minimum air flow [1620mg/m3]	44,2	-2,25
		Total Balance [kW]
		0,525
	Space Heating required	-0,525
*Outdoor air needs to be heated to 0 degrees in culvert		
Archimedes number	1,9887	
Grashof number	2.0185e+008	
Inlet, velocity, Uo	0,3414	[m/s]
Minimum plume temperature	14,3	[C]
Inlet Mass Flow	0,194	[kg/s]
CO2 mass concentration	**1620**	[mg/m3]
CO2 mass concentr. upper zone	1768	[mg/m3]

Typical Winter Temperature		
Temperature outdoors:	**-5**	
Inlet Temperature	0	**Heat Balance [kW]**
	Number of people	30
	Internal generation of heat	3,1
	Transmission losses	-0,41
	Sum	**2,7**
	% of REN reccomendatio	**Ventiltation Heat Loss [kW]**
Minimum air flow [900ppm]	44 %	-3,00
	Heating coil*	-0,75
		Total Balance [kW]
	Total heating required	**1,05**
	Space Heating required	0,31
*Outdoor air needs to be heated to 0 degrees in culvert		
Archimedes number	1,9887	
Grashof number	2.0185e+008	
Inlet, velocity, Uo	0,34143	[m/s]
Minimum plume temperature	14,3	[C]
Inlet Mass Flow	0,194	[kg/s]
CO2 mass concentration	**1620**	[mg/m3]
CO2 mass concentr. upper zone	1768	[mg/m3]

Typical Spring/Autumn Temperature		
Energy balance=0, CO2-levels OK		
Temperature outdoors:	**5**	
Inlet Temperature	5	**Heat Balance [kW]**
	Number of people	30
	Internal generation of heat	3,1
	Transmission losses	-0,246
	Total	**2,9**
	% of REN reccomendatio	**Ventiltation Heat Loss [kW]**
Air flow at energy balance	56 %	-2,85
		Total Balance [kW]
		0,000
	Space Heating required	0,000
Archimedes number	0.9242	
Grashof number	1.5139e+008	
Inlet, velocity, Uo	0,4338	[m/s]
Minimum plume temperature	14,5	[C]
Inlet Mass Flow	0,242	[kg/s]
CO2 mass concentration o.z	**1265**	[mg/m3]
CO2 mass concentr. upper zone	1418	[mg/m3]

lower temperature in occuoied zone than average, dimension for 22 degrees indoors

Typical Spring/Autumn Temperature		
Energy balance=0, CO2-levels OK		Indoor temperature 22
Temperature outdoors:	**5**	
Inlet Temperature	5	**Heat Balance [kW]**
	Number of people	30
	Internal generation of heat	3,1
	Transmission losses	-0,279
	Total	**2,8**
	Percentage of REN	**Ventiltation Heat Loss [kW]**
Air flow at energy balance	49 %	-2,82
		Total Balance [kW]
		0,000
	Space Heating required	0,000
Archimedes number	1.3768	
Grashof number	1.7157e+008	
Inlet, velocity, Uo	0.3783	[m/s]
Minimum plume temperature	16,5	[C]
Inlet Mass Flow	0,129	[kg/s]
CO2 mass concentration o.z	**1461**	[mg/m3]
CO2 mass concentr. upper zone	1614	[mg/m3]

Energy Balance =0, CO2-levels OK		
Temperature outdoors:	**10**	
Inlet Temperature	10	**Heat Balance [kW]**
	Number of people	30
	Internal generation of heat	3,1
	Transmission losses	-0,164
	Total	**2,94**
	% of REN reccomendatio	**Ventiltation Heat Loss [kW]**
Air flow at energy balance	87 %	-2,94
		Total Balance [kW]
		0,000
	Space Heating required	0,000
Archimedes number	0,2964	
Grashof number	1.0093e+008	
Inlet, velocity, Uo	0,6253	[m/s]
Minimum plume temperature	14,8	[C]
Inlet Mass Flow	0,340	[kg/s]
CO2 mass concentration oc.z	847	[mg/m3]
CO2 mass concentr. upper zone	1001	[mg/m3]

Typical Summer Temperature		
Temperature outdoors:	**15**	**Inddor temp= 20 C**
Inlet Temperature	15	**Heat Balance [kW]**
	Number of people	30
	Internal generation of heat	3,1
	Transmission losses	-0,082
	Total	**3,0**
	% of REN reccomendation	**Ventiltation Heat Loss [kW]**
Air flow at energy balance	178 %	-3,02
		Total Balance [kW]
		0,000
	Space Heating required	0,000
Archimedes number	0,0351	
Grashof number	5.0463e+007	
Inlet, velocity, Uo	1,2856	[m/s]
Minimum plume temperature	15,5	[C]
Inlet Mass Flow	0,678	[kg/s]
CO2 mass concentr. occ zone	366	[mg/m3]
CO2 mass concentr. upper zone	495	[mg/m3]

Allowing higher room temperature to avoid high inlet air velocities (high SFP).

Temperature outdoors:	**15**	**Indoor temp= 24 C**
Inlet Temperature	15	**Heat Balance [kW]**
	Number of people	30
	Internal generation of heat	3,1
	Transmission losses	-0,1476
	Total	**3,0**
	% of REN reccomendation	**Ventiltation Heat Loss [kW]**
Air flow at energy balance	93,50 %	-2,85
		Total Balance [kW]
		0,099
	Space Heating required	-0,099
Archimedes number	0.1997	
Grashof number	9.0833e+007	
Inlet, velocity, Uo	0.7228	[m/s]
Minimum plume temperature	18,7	[C]
Inlet Mass Flow	0,259	[kg/s]
CO2 mass concentr. occ zone	719	[mg/m3]
CO2 mass concentr. upper zone	869	[mg/m3]

4 Simulation input

Facade area in CFX is 1,5*4m (window area the same) to make U*A equal to calculations. U value in siulation= 0,187W/m2K
CO2 production in full classroom: 1,87 kg/m3*19l/h/pers*30pers/(3600s/h*1000l/m3)
0,000284994 kg/s

Air mass flow, heat production, heat loss etc.: 1/7 of full classroom

¤ Co2-levels at initial conditions is between upper zone and lower zone. The better the guess, the quicker the solution converges.

¤ Heat loss through facade is equal to calculations.

¤ (The better the guess of the steady state soution, the quicker it resolves)

CFD Input

CO2_production	0,0000407134	kg/s	
Generation of heat from people	342,9	W	80.42 W/m2
Temperature Breath	25	C	
Lighting, 0,6 m2	100	W	166,7 W/m2
Heat transfer coefficient, facade	0,187	W/m2K	(area facade 1,5*4)
Heat transfer coefficient, window	1	W/m2K	
Other surfaces	adiabatic		

turbulence intensity osv. se report.html

Simulation1:Dimensioned by energy efficiency		no heating	**Simulation 2: Dimensioned by CO2 requirements**		
Energy balance=0, CO2 levels high			**Heating required, CO2-levels OK**		
Outdoor Temperature	**0**		Outdoor Temperature	**0**	C
Inlet Mass Flow	0,024	[kg/s]	Inlet Mass Flow	0,0277	[kg/s]
Inlet temperature	0		Inlet temperature	0	C
Heater	0	kW	Heater (area=0.328 m2)	**-0,075**	kW
			Flux	-0,229	kW/m2
Initial Conditions			**Initial Conditions**		
CO2 Ideal gas mass concentration	1939	[mg/m3]	CO2 Ideal gas mass concentration	1731	[mg/m3]
CO2 Ideal gas mass fraction	0,0016154	kg/kg	CO2 Ideal gas mass fraction	0,0007116	kg/kg
Air Ideal gas mass fraction	0,9983846	kg/kg	Air Ideal gas mass fraction	0,9992884	kg/kg
Initial Room Temperature	20	C	Initial Room Temperature	20	C

Simulation 3: Autumn/Spring	**Indoor temp 20C**		**Simulation 4: Autumn/Spring**	**Indoor temp 22C**	
Outdoor Temperature	5		Outdoor Temperature	5	
Inlet Mass Flow	0,0346	[kg/s]	Inlet Mass Flow	0,0185	[kg/s]
Inlet temperature	5		Inlet temperature	5	
Heater	0	W	Heater	0	W
Initial Conditions			**Initial Conditions**		
CO2 Ideal gas mass concentration	1380	[mg/m3]	CO2 Ideal gas mass concentration(ave)	1576	[mg/m3]
CO2 Ideal gas mass fraction	0,0011498	kg/kg	CO2 Ideal gas mass fraction	0,001313	kg/kg
Air Ideal gas mass fraction	0,9988502	kg/kg	Air Ideal gas mass fraction	0,998687	kg/kg
Initial Room Temperature	20	C	Initial Room Temperature	22	C

Summer 20 C room temp			Summer 24 C room temp		
Outdoor Temperature	15		Outdoor Temperature	15	
Inlet Mass Flow	0,096925714	[kg/s]	Inlet Mass Flow	0,03704265	[kg/s]
Inlet temperature	15		Inlet temperature	15	
Heater	0	W	Heater	0	W
Initial Conditions			**Initial Conditions**		
CO2 Ideal gas mass concentration	463	[mg/m3]	CO2 Ideal gas mass concentration	832	[mg/m3]
CO2 Ideal gas mass fraction	0,000386	kg/kg	CO2 Ideal gas mass fraction	0,000693	kg/kg
Air Ideal gas mass fraction	0,9996144	kg/kg	Air Ideal gas mass fraction	0,9993071	kg/kg
Initial Room Temperature	**20**	C	Initial Room Temperature	**24**	C

5 Relative humidity levels

Assumptions:

Outdoor RH: **(95-)100%**

Indoor temperature 20

Outdoor temp [C]	T_window frame [C]	Humidity content	RH after heating to
-20	13,8	0,0006	5 %
-15	14,6	0,0010	7 %
-10	15,3	0,0015	11 %
-5	16,1	0,0025	19 %
0	16,9	0,0038	25 %
5	17,7	0,0055	37 %
10	18,4	0,0076	54 %

REN=40%	REN=20%	REN=40%	REN=20%	REN=40%	REN=20%
total x[kg/kg]	total x[kg/kg]	Tdp [C]	Tdp [C]	RH [%]	RH [%]
0,0037	0,0006	10	13	25	47
0,0041	0,0010	10,5	14,5	29	50
0,0046	0,0015	11	14,5	33	54
0,0056	0,0025	12	15,5	39	60
0,0069	0,0038	13	16	47	68
0,0086	0,0055	15	17,5	58	80
0,0107	0,0076	16,5	19	90	95

REN=100%		
x	Tdp	RH
0,0018	7,5	9
0,0022	8	15
0,0027	9	20
0,0037	10	25

APPENDIX 3: General settings CFD simulation

This appendix comprises of the general settings for the CFD simulations in ANSYS CFX.

Table 1. Mesh Information		
Domain	Nodes	Elements
Default Domain	240080	776437

Table 2. Domain Physics				
Name	Location	Type	Materials	Models
Default Domain	B253	Fluid	Ideal air and CO2	Component:Air Ideal Gas = Transport Equation Component:CO2 Ideal Gas = Constraint Heat Transfer Model = Thermal Energy Turbulence Model = k epsilon Turbulent Wall Functions = Scalable Buoyancy Model = Buoyant Domain Motion = Stationary

Table 3. Boundary Physics				
Domain	Name	Location	Type	Settings
Default Domain	Inlet	Inlet	Inlet	Air Ideal Gas Mass Fraction = 1 Component:Air Ideal Gas = Mass Fraction Flow Direction = Normal to Boundary Condition Flow Regime = Subsonic Heat Transfer = Static Temperature Static Temperature = xx [C] Mass Flow Rate = xxx [kg s^-1] Mass And Momentum = Mass Flow Rate Turbulence = Medium Intensity and Eddy Viscosity Ratio
Default Domain	Mouth	Mouth	Inlet	Air Ideal Gas Mass Fraction = 0.0 Component:Air Ideal Gas = Mass Fraction Flow Direction = Normal to Boundary Condition Flow Regime = Subsonic Heat Transfer = Static Temperature

				Static Temperature = 298 [K] Mass Flow Rate = 4.07134e-05 [kg s^-1] Mass And Momentum = Mass Flow Rate Turbulence = Medium Intensity and Eddy Viscosity Ratio
Default Domain	Outlet	Outlet	Outlet	Flow Regime = Subsonic Mass And Momentum = Average Static Pressure Relative Pressure = -1.6 [Pa] Pressure Averaging = Average Over Whole Outlet
Default Domain	Symmetry1	Symmetry1	Symmetry	
Default Domain	Symmetry2	Symmetry2	Symmetry	
Default Domain	Children	Default 2D Region	Wall	Heat Flux in = 80.4 [W m^-2] Heat Transfer = Heat Flux Wall Influence On Flow = No Slip Wall Roughness = Smooth Wall
Default Domain	Default Domain Default	F254.253, F255.253, F258.253	Wall	Heat Transfer = Adiabatic Wall Influence On Flow = No Slip Wall Roughness = Smooth Wall
Default Domain	Facade	Facade	Wall	Heat Transfer Coefficient = 0.187 [W m^-2 K^-1] Heat Transfer = Heat Transfer Coefficient Outside Temperature = x [C] Wall Influence On Flow = No Slip Wall Roughness = Smooth Wall
Default Domain	Heater	Heater	Wall	Heat Transfer = Adiabatic Wall Influence On Flow = No Slip Wall Roughness = Smooth Wall
Default Domain	Lighting	Lighting	Wall	Heat Flux in = 167 [W m^-2] Heat Transfer = Heat Flux Wall Influence On Flow = No Slip Wall Roughness = Smooth Wall
Default Domain	Window	Window	Wall	Heat Transfer Coefficient = 1 [W m^-2 K^-1] Heat Transfer = Heat Transfer Coefficient Outside Temperature = x [C] Wall Influence On Flow = No Slip Wall Roughness = Smooth Wall

APPENDIX 4: Velocities at ankle height

Figure 50 Simulation 1: Winter simulation, no heating

Figure 51 Simulation2: Winter simulation with heating

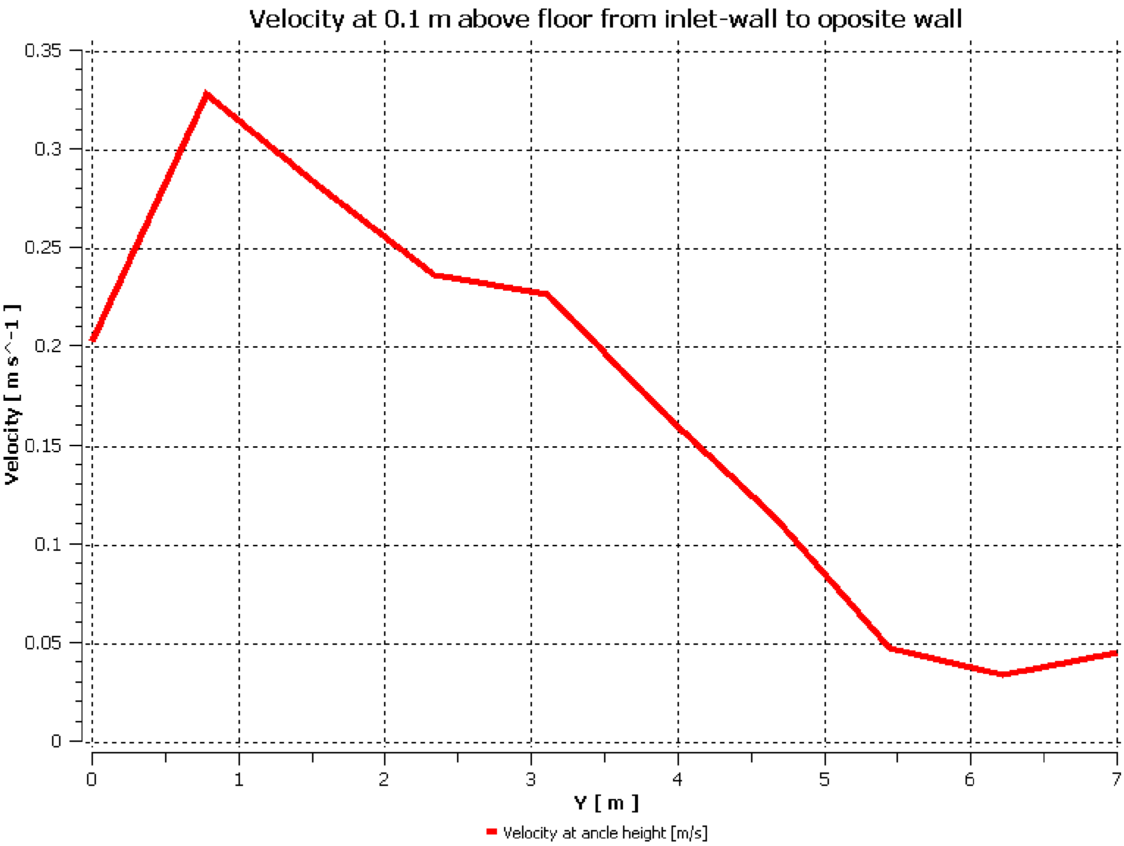

Figure 52 Simulation3: Spring simulation, 20ºC designed indoor temperature

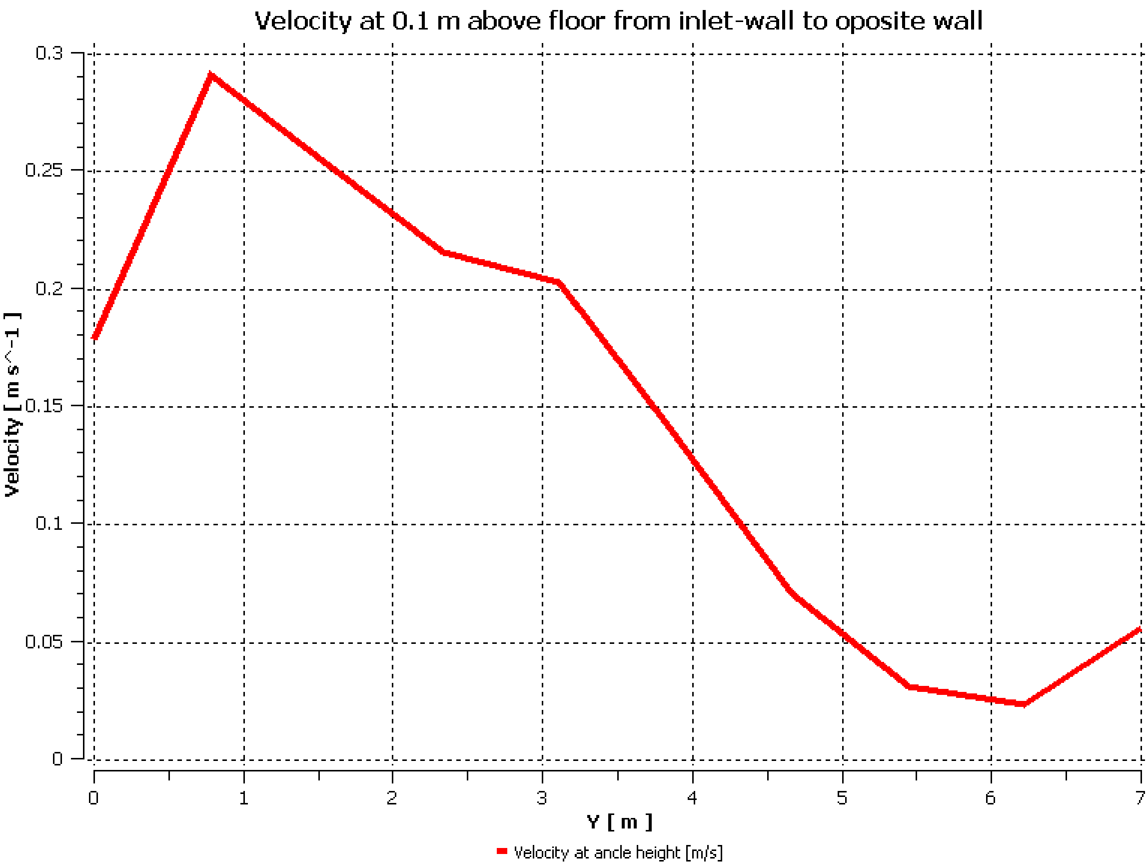

Figure 53 Simulation4: Spring simulation, 22ºC designed indoor temperature

Figure 54 Simulation5: Summer simulation, 20ºC designed indoor temperature

Figure 55 Simulation6: Summer simulation, 24ºC designed indoor temperature

APPENDIX 5: Guidelines for Preliminary Design and Sizing for Climatization of Rooms with Cooling Needs using the Swedish Model.

This guide will attempt to give advice on how to size ventilation for a classroom or conference room ventilated by the Swedish model. The guide is based on the investigations done in this report and is meant as help for sizing of supply openings based on empirical equations. This is an extension of the use of such empirical equations developed by Skåret(2000).

The guide will take into consideration the number of inlets, size and height, different ventilation rates, supply air- outdoor- and designed room temperature, CO_2, humidity and heat emissions from people. Sizing should be done for the maximum amount of people that will occupy the room.

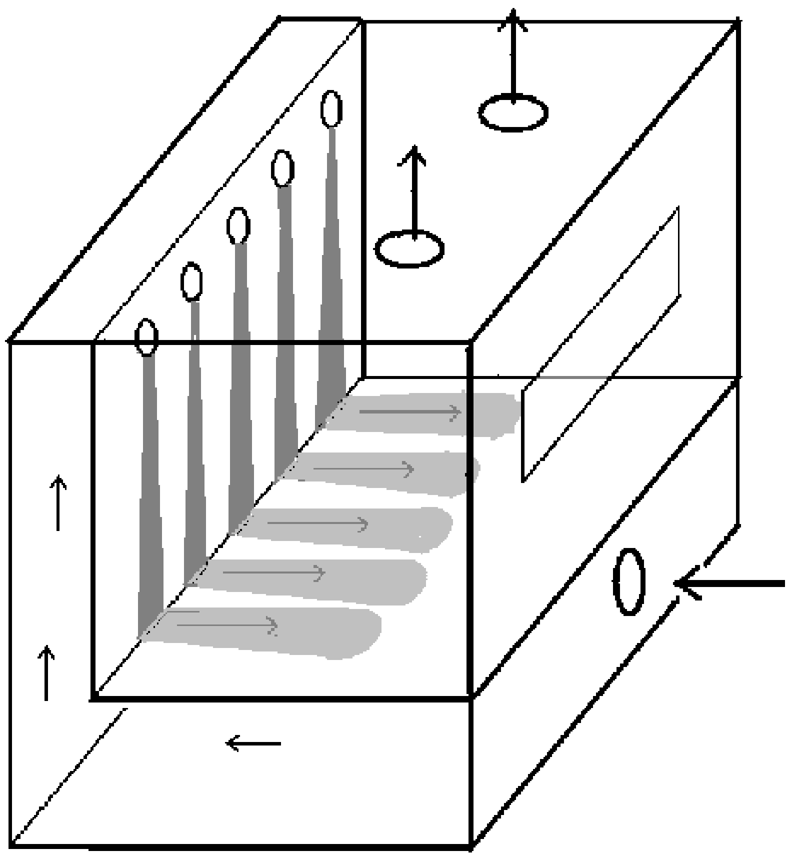

Figure 1 Sketch of room with Swedish ventilation model

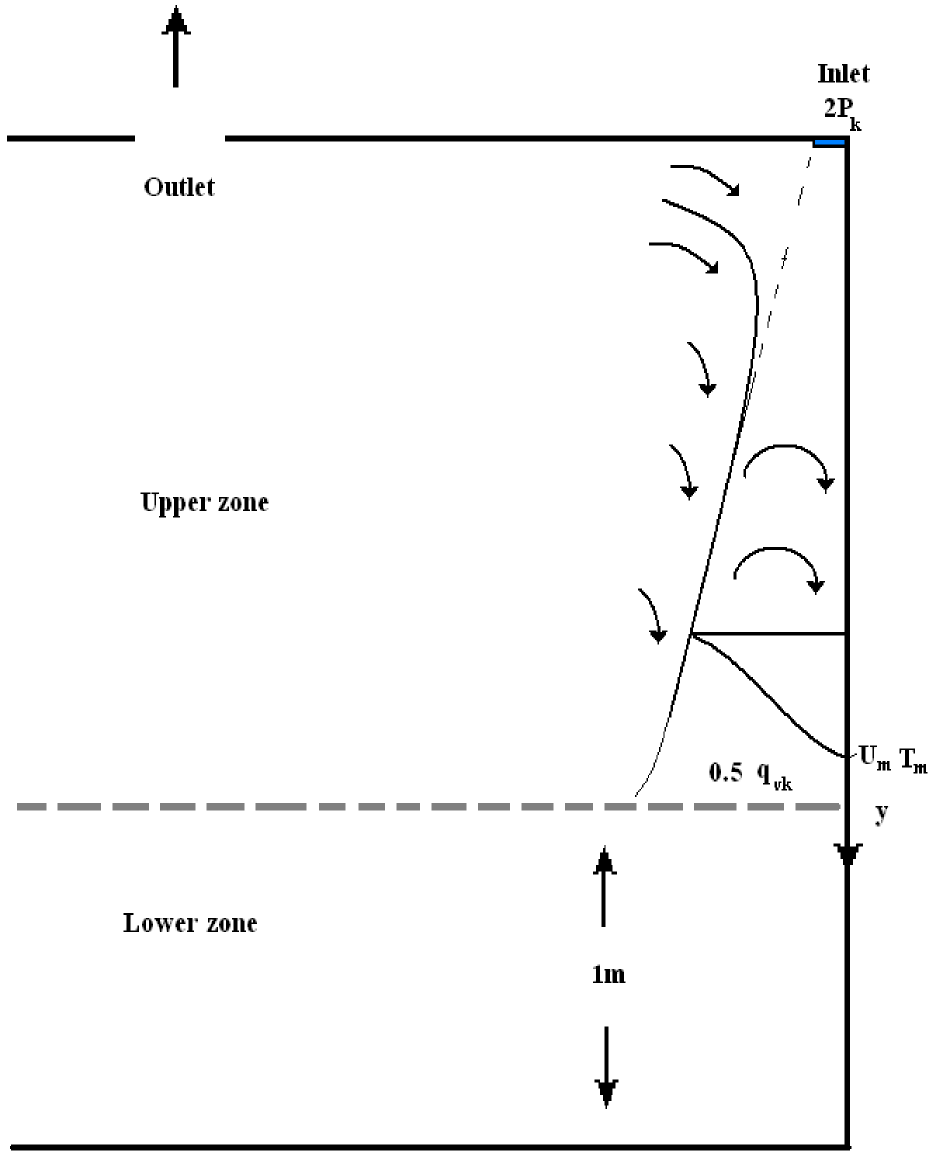

Figure 2 Cross Section of Classroom

Explanation of Basis for Calculation

Figure 1 shows a sketch of a room with the Swedish ventilation model. The air is supplied through the culvert, behind the inner wall through supply openings high on the inner wall. Figure 2 shows the cross section of one such plume. The grey dotted line marks a theoretical division between upper and lower zone 1m above the floor used for the calculation of CO_2 levels. An adaption to a real inlet is made by placing the inlet in the ceiling as a cold "source" causing an upside down convection plume.

In the table below the basis for calculation is listed. In the column to the right example calculations are shown.

The number of inlets could be set so that the plumes do not influence each other before they reach the occupied zone. In the example calculations, the inlet is semicircular and placed in the ceiling. It then expands with approximately 12.5º. A real inlet should be placed on the wall, and might have a different shape, possibly with a plate in front, or a device to assist spreading the air flow. This means the plume will behave differently, and perhaps be much wider.

The transmission and infiltration losses depend entirely of the type of building, façade area, air tightness, volume, U values etc. Finding the correct supply air flow, q_s, requires an iteration process. The ventilation heat loss is given by the supply air flow, q_s, which is not necessarily equal to the air volumes that give heat balance, q_o. q_o, might not give satisfactory CO_2 concentrations, in which case the air volumes should be increased. That in turn well lead to a heat deficit, meaning heating is required. The other case might be that air volumes are too massive, in which case the inlet velocity is too high. One might then try recalculating the heat balance with a higher indoor temperature. If velocities are still too high, one should consider increasing the inlet area, or number of inlets. The example calculations with 10 ºC outdoor and supply air temperature give acceptable CO_2 levels as well as heat balance. However the inlet velocity and temperature at 1 m above floor give a draft risk. To avoid draft in the occupied zone the maximum plume velocity at 1m above floor level should be at a maximum 3 times as large as the draft velocity criteria (0.15-0.25 m/s depending on season). The minimum plume temperature at 1m above floor height should be at least 15ºC. The air flow could be reduced somewhat without compromising the carbon dioxide levels or the temperature.

Basis for Calculations		
Input	Symbol	Example values
Length of inlet wall	L [m]	10
Convection source area [m^2], circular/square	$A=\pi d^2/4$ or d^2 [m^2]	r=0.2 A=0,1256
Actual inlet area	$0.5A$	0.0628
Room temperature, T_e - supply air temperature, T_s	$\Delta T = T_e - T_s$ [K]	(273.15+20)-(273.15+10)= 10
Inlet height above occupied zone	y [m]	3m

Pole distance	*yp=0,7 times diameter of inlet*	0.14
Width of plume at distance y from inlet	$2b = 2tg(12{,}5^o)(y + y_p)$ [m]	1.4
Number of inlets	*N=L/b*	7
Density of air	ρ [kg/m^3]	1.205
Specific heat capacity of air	c_p [kJ/kgK]	1.005
Maximum number of people	n_p	30
CO_2 production	$P_{CO2} = n_p 19l/h/person = n_p 19/3.6 \cdot 10^6$ m^3/s	1.58 10^4
Equations		Example calculations
Internal heat generation	Heat from people, lighting, equipment. Calculated using NS 3031(2007) [kW]	3.1
Transmission/infiltration heat loss	Calculated using NS 3031(2007)	-0.2
Ventilation heat loss	$P_v = q_s \rho c_p \Delta T$ [kW]	-2.9
Heat balance	Sum	0
Air flow that yields heat balance	q_o [m^3/s]	0.29
Supply air flow	q_s [m^3/s]	0.29
Inlet velocity	$U_0 = \frac{q_s/n}{0.5A}$ [m/s]	0.68
Exhaust air flow	$q_{v,e} = q_s \frac{T_e}{T_s}$	0.28
Convection plume cold source	$P_k = q_s c_p \rho \Delta T$	-2.9
Air flow at distance y from inlet (plume entering occupied zone)	$q_{Vk} = 0.5n\left(0{,}055(2P_k)^{1/3}(y + y_p)^{5/3}\right)$	1.42
Induced air	$q_{v,induced} = q_{vk} - q_s$	1.13
CO_2 concentration in upper zone(exhaust) (above outdoor level)	$C_{upper} = C_e = \frac{P_{CO2}}{q_{v,e}}$ [ppm]	520

CO_2 concentration in occupied zone (above outdoor level)	$C_{occupied} = \frac{q_{v,induced} C_{upper} + q_{v,\sup ply} C_{\sup ply}}{q_{v,induced} + q_{v,\sup ply}}$	410

Estimation of draft risk		
Maximum velocity of plume at y (1m above floor)	$U_m = 1{,}28 \left(\frac{P_k}{y + y_p} \right)^{1/3}$ [m/s]	1.2
Maximum temperature difference of plume and surrounding air at y (1m above floor)	$\Delta T_m = 20{,}9 P_k^{2/3} (y + y_p)^{-5/3}$	6.3 (20-6.3=13.7)

Design stages:

For outdoor temperatures below 0ºC:
- Calculate effect necessary for preheating of supply air to 0ºC
- Find air volumes that give heat balance at 20 ºC indoor temperature
- Check if the resulting air volumes yield acceptable CO2 levels.
- Increase air volumes
- Check draft risk
- Calculate heating required to achieve desired room temperature

For outdoor temperatures from 0ºC-5 ºC:
- Find air volumes that give heat balance at 20 ºC indoor temperature
- Check if the resulting air volumes yield acceptable CO2 levels
- Increase air volumes if necessary
- Check draft risk
- Calculate heating required to achieve desired room temperature

For outdoor temperatures 5-15 ºC:
- Find air volumes that give heat balance at 20 ºC indoor temperature
- Check if velocity through inlet is too high
- Check draft risk
- Reduce air volumes and allow higher indoor temperature

For outdoor temperatures above 15 ºC:
- Additional ventilation through windows.

If too high velocities occur already at low temperatures, consider increasing the inlet area. If draft occurs, the air volumes through each supply opening are too high and one might consider adding one more inlet or increasing the height of it.