Diffuse Wall Ventilation
A CFD Study of Thermal Comfort and Contaminant Removal Effectiveness

Truls Ramstad, Gaute Tveit
Indoor Environmental and Energy Engineering, IEEE10, 2020-06

Master’s Thesis
**Title:**
Diffuse Wall Ventilation

**Theme:**
Scientific Theme

**Project Period:**
Spring Semester 2020

**Participants:**
Truls Ramstad
Gaute Tveit

**Supervisor:**
Chen Zhang

**Copies:**
1

**Number of pages:**
100

**Date of Completion:**
June 15, 2020

---

**Abstract:**
This thesis investigates the performance of a diffuse wall ventilation concept in terms of contaminant removal effectiveness, and compares it to other concepts based on thermal comfort. Evaluations are made using CFD simulations for three temperature differences and inlet geometries, using an alternative validation approach as lab experiments could not be conducted. It performed well for the requirements set in this thesis, and outperformed most of the other reviewed ventilation concepts regarding operating range based on thermal comfort. It is conclusively found that the diffuse wall is a well functioning and flexible concept for ventilating an office space, provided that the inlet area is placed beneath the contaminant stratification layers, to avoid spreading of contaminants in the occupied zone.

*The content of this report is freely available, but publication (with reference) may only be pursued due to agreement with the author.*
Contents

Preface vii

1 Introduction 1
  1.1 Project Aim ............................................. 2
  1.2 Limitations .............................................. 3
  1.3 Workflow ................................................ 4

2 Theory 5
  2.1 Numerical Approach ...................................... 5
  2.2 Building Requirements .................................. 11
  2.3 Draught .................................................. 12
  2.4 Contaminant Removal Effectiveness ..................... 13
  2.5 Design Chart ............................................ 14
  2.6 Annex 20 ................................................ 15

I Preliminary Study 17

3 Literature Review 19
  3.1 State of the Art ........................................... 19
  3.2 Relevant Projects ....................................... 20
  3.3 CFD Models used in relevant projects ................... 21

4 Comparison Project 23
  4.1 Background ............................................... 23
  4.2 Flow Element Analysis ................................... 23
  4.3 Experiment from Comparison Project .................... 27
  4.4 CFD Model Development ................................ 29
  4.5 Validation results ....................................... 34
## Main Study

5 **Diffuse Wall**

5.1 Problem Description ........................................ 39  
5.2 Hypothesis .................................................... 40  
5.3 CFD Model Development .................................... 42

6 **Simulation Results** ......................................... 51  
6.1 Case $\Delta T3$ .................................................. 52  
6.2 Case $\Delta T6$ .................................................. 57  
6.3 Case $\Delta T10$ ............................................... 57  
6.4 Baseline case comparison .................................... 58

7 **Design Chart** .................................................. 59

8 **Parameter Study** ............................................. 61  
8.1 Half Wall Inlet .................................................. 62  
8.2 Lamella Inlet ................................................... 65

## Review

9 **Discussion** .................................................... 73

10 **Conclusion** .................................................. 75

11 **Further Work** ................................................ 77

Bibilography ...................................................... 79

## Appendix

IV **Appendix** .................................................... 83  
A Mesh Independence Study .................................... 85  
B Porous Media Parameters .................................... 87  
C Boundary Conditions ........................................... 91  
D Design Chart Development ................................... 93  
E Additional Simulation Results ............................... 95
Preface

This thesis is made as the final project in the Master of Science in Indoor Environmental and Energy Engineering at the Department of the Built Environment, Aalborg University, Denmark. It corresponds to a total of 30 ECTS, and was conducted from January to June of 2020.

Aalborg University, June 15, 2020

Truls Ramstad
tramst18@student.aau.dk

Gaute Larsen Tveit
gtveit18@student.aau.dk
Chapter 1

Introduction

The importance of maintaining a good indoor environment in buildings is becoming more apparent these days, increasing the need for well designed ventilation systems. This has traditionally been achieved by using a mixing or displacement ventilation concept, often resulting in extensive installations with long stretches of ventilation ducts.

Although the function of such systems might be satisfactory, problems occur due to the need of installation space, and in many cases are design changes made at the expense of system performance. This is a recurring issue especially in renovation projects, where the possibility of facilitating for large technical installations is limited.

The ventilation industry has embraced the challenge of finding a good compromise between space and function, and smart solutions such as implementing decentralized air handling units, AHU, in the room as partitions or pieces of furniture have been created.

When choosing a ventilation concept, the installation cost is often essential, and the development of a cost and space efficient ventilation concept would be beneficial. A Norwegian company has therefore proposed a "diffuse wall" concept, designed especially with focus on cost and space optimizing.
Chapter 1. Introduction

1.1 Project Aim

The diffuse wall concept was initially designed as part of an office renovation project by a Norwegian consultant firm, where it was not accepted to lower the ceiling or install visible ducts, making traditional ventilation concepts unsuitable. It was suggested to install several air handling units in small technical rooms at strategic locations, supplying fresh air to nearby plenum chambers. A concept of having these chambers behind simple wall structures with permeable fabric supplying air into the room was suggested as shown in figure 1.1.

Figure 1.1: Illustration of concept

Inlet air is supplied into the chamber, for example by using fabric ducts as in the illustration. The textile front of the wall is in the figure covered with wooden lamellas mounted vertically to make the installation visually appealing, which was part of the initial concept suggestion. Due to the near proximity of technical rooms, a minimal amount of ducting is needed.

Another benefit of creating a diffuse wall, especially when a suspended ceiling is not an option, is that other technical installations might utilize parts of the plenum space by for example mounting cable trays inside, depending on the necessary inlet area for air supply into the room. According to the developers of the concept, several building owners have expressed their interest to implement it in projects.
Upon writing this thesis, the concept has only been analyzed analytically, leading
to this project’s aim. Through obtained experimental results and computational
fluid dynamics (CFD), this thesis aims to evaluate the diffuse wall ventilation in
terms of thermal comfort and operating range, before comparing its performance
to other ventilation concepts using the design chart method, not considering
energy. Vertical temperature gradients and draught rates are chosen as key
parameters as they portray the indoor environment well.

The aim is divided into the following three questions which are going to be answered:

1. Does the ventilation concept satisfy the indoor environmental requirements
   according to Norwegian standards?
2. How sensitive is the performance to changes in parameters?
3. How does it compare to other ventilation concepts?

1.2 Limitations

This thesis was written during the COVID-19 pandemic, which resulted in a
national shutdown of all educational facilities. Lab experiments, which initially
were a major part of analyzing the diffuse wall ventilation concept were terminated
and an alternative approach had to be developed, involving the use of experiment
data from another project for comparison. The findings in this thesis can therefore
not be fully validated due to lack of experiment data, although they will be based
on thorough research.

The sudden shutdown made it necessary to redefine both the workflow and aim of
the thesis, making the many hours used on preparation for experiments redundant.
This limited the available time to carry out the redefined tasks, reducing the extent
of the parametric study compared to what was originally intended. An adequate
selection is nevertheless maintained to make qualified evaluations of the ventilation
concept.
1.3 Workflow

A crucial part of CFD modeling is to verify and validate the model to make sure it predicts accurate flows. The common approach is to follow a step-by-step procedure, where experimental data is compared with the CFD simulations. As this thesis has no experimental data available, an alternative approach is made. Figure 1.2 shows a workflow chart, where dashed red boxes indicate previously conducted work made by others.

Instead of comparing CFD results with lab experiments, a CFD model is created and validated for the comparable concept, before settings are used as input parameters in the diffuse wall CFD model. Evaluation of the concept can then be made knowing that the applied input parameters have proven suitable under similar conditions.

Figure 1.2: Workflow
Chapter 2

Theory

When analysing systems involving fluid flow, there are mainly three approaches to consider: experimental, analytical and numerical studies. An experimental approach involves running scale experiments, giving thorough understanding of how a system works in reality. This can often be complicated, costly and time consuming. A simpler approach is to use analytical studies where quick and cost effective results can be obtained for simple problems through the application of physics. [Zhai, 2020]

2.1 Numerical Approach

A numerical approach uses computational fluid dynamics (CFD) to solve governing equations, making it possible to resolve complex flows at a low cost. This requires user knowledge and experience, as its accuracy is sensitive to user inputs.

The software used in this study is provided by ANSYS and consists of Workbench 2020R1 for integrating all the processes, Design Modeler to generate geometries, Fluent with Mesh for meshing domains and solving the flow, and CFD-Post for extracting graphics.

When generating a CFD code, a geometry or computational domain must be defined. This is divided into a grid of smaller sub-domains, cells, through a meshing procedure. Partial differential equations for fluid properties, e.g. velocity, temperature or pressure, are solved at a node within each cell using the finite element method. The properties are combined to calculate derivatives for gradients occurring in the main flow, making the solution dependent on the number of cells and mesh quality.
2.1.1 Convergence

An iterative process of solving governing equations requires a criterion for when to accept a solution to be converged. This is typically evaluated based on simulation residuals, total balance of a variable or by monitoring values at a point of interest.

Residuals give information about the variation between two iterations for a given variable. Convergence is obtained when these are stable at a low level, indicating no significant change in the solution for new iterations. Convergence can also be evaluated based on heat or mass balance. If this is not achieved within an acceptable range, the solution is not stable and changes should be made.

It is also important to monitor parameters at locations of interest to ensure stability, e.g. ankle or neck velocities. Although these values should converge, it must not be confused with obtaining constant values. If a flow is turbulent in a location, the result could be considered as converged when a repeating pattern is observed.

2.1.2 Governing equations

The software solves governing Navier-Stokes equations for fluid flows, i.e. the continuity, momentum and energy equations. For simplicity, these can be represented by the general advection-diffusion transport equation shown in (2.1).

$$\frac{\partial (\rho \Phi)}{\partial t} + \text{div}(\rho \Phi \mathbf{u}) = \text{div}(\Gamma_{\Phi} \nabla \Phi) + S_{\Phi}$$

(1) (2) (3) (4)

The equation consists of $\Phi$, representing any transferable variable of interest, density $\rho$, velocity field $\mathbf{u}$, diffusion coefficient $\Gamma_{\Phi}$ and source term $S_{\Phi}$. For conservation of mass and momentum, the correlations between the general equation and the variables are shown in table 2.1, where $\mu$ represents viscosity. A similar relationship yields for conservation of energy among others.

<table>
<thead>
<tr>
<th>Equation</th>
<th>$\Phi$</th>
<th>$\Gamma_{\Phi}$</th>
<th>$S_{\Phi}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>continuity</td>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>x-momentum</td>
<td>$u$</td>
<td>$\mu$</td>
<td>$-\frac{\partial p}{\partial x} - \rho g_x$</td>
</tr>
<tr>
<td>y-momentum</td>
<td>$v$</td>
<td>$\mu$</td>
<td>$-\frac{\partial p}{\partial y} - \rho g_y$</td>
</tr>
<tr>
<td>z-momentum</td>
<td>$w$</td>
<td>$\mu$</td>
<td>$-\frac{\partial p}{\partial z} - \rho g_z$</td>
</tr>
</tbody>
</table>

Table 2.1: Variables for the general transport equation
There are four parts to the transport equation (2.1). Term (1) is the temporal variation of $\Phi$. (2) is the convective term, defining the amount of $\Phi$ flowing through the control volume boundaries. (3) defines the flux of $\Phi$ through the boundaries from diffusion and (4) is a source term which defines the creation of $\Phi$ within the control volume. [Versteeg and Malalasakera, 1995; Dirk et al., 2013]

2.1.3 Turbulence modeling

Turbulence occurs in most fluid flows, resulting in random fluctuations which are difficult to solve analytically. Turbulence models helps predict the fluctuations, and three approaches are mainly used: Direct Numerical Simulation (DNS), Large Eddy Simulation (LES) and Reynolds-Averaged Navier-Stokes (RANS) equations.

Both DNS and LES resolves turbulent eddies, which are the swirls created by the turbulence. DNS computes the governing Navier-Stokes equations directly, and requires very fine grids to resolve the smallest eddies, making it unfeasible for most applications. To cope with this problem, LES introduces a boundary between small and large eddies, making it possible to resolve the large ones directly while applying approximations to the small eddies, reducing the fine grid requirements [Zhai, 2020].

Most common is the RANS approach, which decomposes instantaneous flow parameters into an average and a turbulent term, e.g. $u(t) = U + u'(t)$ as illustrated in figure 2.1.

![Figure 2.1: Turbulent velocity decomposition](Versteeg and Malalasakera, 1995)

RANS introduces additional time-averaged terms to the momentum equations called Reynolds stresses. These terms are often predicted using a relationship between the Reynolds stresses and an eddy viscosity, $\mu_t$, before a turbulence model is applied to model this viscosity term. [Versteeg and Malalasakera, 1995]
The research on application of turbulence models to the RANS approach is vast, making it difficult to choose an applicable model. For buoyancy driven flows, two-equation eddy viscosity models have traditionally been used with a Boussinesq approximation, treating density as a constant in the RANS equation except in the buoyancy term. The Reynolds stresses can then be related to the mean flow as functions of the eddy viscosity, $\mu_t$ and turbulent kinetic energy, $k$. These models have proven to be the most versatile, as they do not require prior knowledge about turbulence in the system.

Two-equation models solves the eddy viscosity term through an equation including $k$ and another variable, $\varepsilon$ or $\omega$. The $k$-$\varepsilon$ models have traditionally been used for modeling indoor air flows, where the standard, renormalization group (RNG) and realizable $k$-$\varepsilon$ are most common.

Standard $k$-$\varepsilon$ calculates transport equations for turbulent kinetic energy, $k$, and turbulence dissipation rate, $\varepsilon$, by applying a constant $C_\mu$ to the viscosity equation as shown in (2.1.3), as well as a number of constants in the RANS equation.

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$  \hspace{1cm} (2.2)

RNG $k$-$\varepsilon$ uses the equations from the Standard model with different constant values and additional terms and functions in the calculation of $k$ and $\varepsilon$, improving accuracy for rapidly strained and swirling flows. Low-Reynolds number effects are accounted for, presuming that an adequate near wall treatment is applied. [Zhai, 2020]

Realizable $k$-$\varepsilon$ has proven to outperform the standard model for flows involving swirl and separation by calculating values for $C_\mu$ instead of treating it as a constant.

It has in later years been an increased interest in using modified $k$-$\omega$ models for indoor air flows, where $\omega$ is defined as the ratio of $k$ over $\varepsilon$ [Zhai, 2020]. These models have shown better performance near walls than $k$-$\varepsilon$, but have problems predicting the flow in free stream regions. A variation of the model called shear stress transport (SST) $k$-$\omega$ using a blending function to toggle between $k$-$\omega$ and $k$-$\varepsilon$ depending on the distance from the wall has therefore been developed. Zhai [2020] addresses how recent studies indicates that SST $k$-$\omega$ may perform better than both standard and RNG $k$-$\varepsilon$ for indoor air flows.
2.1.4 Near Wall Treatment

Resolving turbulent flows near a surface is a complex task. The flow changes significantly within a short region near the wall called boundary layer, which is generally divided into three regions: the viscous sub layer, buffer layer and log-law region.

No analytic approach has yet proven to solve boundary layers accurately, and approximation models have been developed. These are divided into two main categories, i.e. wall functions and near wall models. Wall functions resolves the flow through the log-law layer, while the buffer and viscous sub layers are solved using general functions developed through semi empirical experiments. Several approaches are available, where the Standard Wall Function is set as default in most CFD solvers. This approach works well in some cases, but presupposes that all the cells in the mesh near wall is in the log law region. If the geometry is complex or the cells are of unknown size near walls, the Scalable Wall Function is advisable as it forces the mesh cells to be in the log-law region.

The near wall models solves the flow all the way, requiring a finer mesh. This approach is accurate if applied correctly, but requires significantly more computational power due to the large amount of cells.

2.1.5 Meshing

Ensuring good quality meshes is essential for creating converged CFD simulations. Cells may be created in different shapes and sizes, where 3D meshes, like used in this thesis, are differentiated as structured or unstructured. The structured meshes offer an easier implementation in simple domains, while an unstructured mesh is more versatile to adapt to different shapes and bodies within the domain. Regardless of the mesh structure, shape distortion and size ratios to adjacent cells must be kept below certain values to ensure converged simulations.

Since changes in the main flow are determined by the mesh, it is necessary to aim for fine meshes in areas where strong gradients occur, often through high air velocities or strong convective heat fluxes. To keep computational costs down, mesh independence tests are made, where simulations using different mesh sizes are compared. When an increase in mesh size no longer affects the solution, mesh independence is found and an adequate mesh size can be chosen.
2.1.6 Solution methods

Fluent provides density- and pressure-based solvers to calculate flow parameters. The density-based method was initially intended for solving high-speed compressible flows, while the pressure-based method was developed to model incompressible flows at low speeds.

Indoor air flows can be approximated to be incompressible, making density changes insignificant [Heinsohn and Simbala, 2003, p.520]. If buoyancy forces are governing, density still needs to be accounted for in equation terms regarding this. Due to the incompressibility approximation, a pressure-based solver is applied in this thesis.

Mass conservation of the velocity field, seen as continuity in the software, is achieved by solving a pressure equation derived from the governing continuity and momentum equations. The equations are solved independently, and iterations are therefore needed, as all governing equations are solved for every iteration until convergence is reached. [ANSYS, 2020a]

The iterative process is controlled by a pressure-velocity coupling scheme, treating the governing equations as segregated or coupled. This means that they are solved one after another or at the same time. Four pressure-velocity coupling schemes are available for steady-state flows in Fluent, i.e. SIMPLE, SIMPLEC, PISO and Coupled, where the first three uses the segregated approach.

Pressure-velocity coupling

By calculating the face flux of a cell using a pressure estimation, the SIMPLE algorithm checks if this satisfies the continuity equation. If it does not, a correction factor is added to the face flux which is then converted into a pressure correction value.

The cell pressure gets updated based on the correction value and an under-relaxation factor between 0-1, controlling the amount of the current solution to be updated. A high under-relaxation factor may ensure a fast drop in residual values at the risk of causing fluctuations. A low value can smooth these fluctuations, but convergence might be slow.

SIMPLEC has as proven to speed up convergence for some basic laminar flows, while PISO requires significantly more computational power. SIMPLE performs adequately in most cases, and comes at a reasonable computational cost, making it the chosen pressure-velocity coupling scheme for the simulations in this thesis. [ANSYS, 2020c]
2.2. Building Requirements

Spatial discretization
The solution method definition also involves setting spatial discretization schemes for the variables being solved. As Fluent stores variable values at the cell centers, these schemes determine how the cell centered values are interpolated to the cell faces. [ANSYS, 2020a]

Recommendations for choosing a pressure discretization scheme in ANSYS [2020b] states that PRESTO! should be used when a porous media is involved and a Body Force Weighted scheme when large body forces are present. Test simulations are therefore made using both schemes in the search for an accurate model. The recommendations explicitly states not to use the Second Order pressure scheme with porous zones. The remaining variables are defined with the Second Order Upwind scheme, as this is provides better accuracy than First Order, especially for complex flows and meshes that are not aligned.

2.2 Building Requirements

As this thesis has its origin from the Norwegian ventilation industry, the diffuse wall is evaluated based on Norwegian standards and building requirements. These are given specifically for indoor environment in office buildings.

The Norwegian Building Authority [2017] has published Regulations on technical requirements for construction works with minimum requirements defined for fresh air supply as three parts: A, B and C. The result of (A+B) or C which gives the highest required air flow rates will be governing, ensuring satisfactory air supply for all parameters.

A) Minimum 26 m$^3$/h · person should be supplied for sedentary workers with Met 1.1 [ISO, 2005]

B) Occupied rooms should have 1.2 m$^3$/h per m$^2$ floor area, while 0.7 m$^3$/h · m$^2$ is sufficient when the room is not in use

C) Rooms with significant pollution, i.e. toilets, elevator shafts and basements should maintain defined minimum exhaust rates for the given room type

The regulations address thermal comfort, stating that ventilation systems should facilitate for good comfort and health. They do not provide exact requirements, but rather guidelines for obtaining a good indoor environment. A maximum temperature difference between feet and neck is recommended to be be less than...
Chapter 2. Theory

3-4 °C, an operating temperature of 19-26 °C for light work is advised, and air temperatures should be below 22 °C in the heating season. It does not however state any limits regarding draught.

Ministry of Labour and Social Affairs [2006] gives recommendations on air velocities in Arbeidstilsynets Veiledning 444 to obtain an indoor environment complying with the Norwegian Working Environment Act. It states that air velocities at work stations should not exceed 0.15 m/s, while other literature uses 0.20 m/s as the maximum velocity to avoid draught [Skistad et al., 2002].

2.3 Draught

An international standard is developed for evaluating thermal comfort criteria for the body as a whole and local discomfort based on human sensation [ISO, 2005].

The whole body evaluations focus on whether people are too warm or cold, while the local discomfort uses draught rate and vertical temperature difference as key parameters. By using percentage dissatisfied, PD, a value for the expected ratio of persons feeling local discomfort under the given thermal conditions is obtained. As the Norwegian building requirements give recommendations on temperature gradients, only draught evaluations are based on this standard.

A draught rate, DR, estimates the percentage of people expected to experience discomfort due to draught. This is calculated according to equation (2.3), designed to determine draught at neck level for people with sedentary activity. If applied at lower heights, it tends to overestimate the draught rate.

\[
DR = (34 - t_{a,l}) \cdot (\bar{v}_{a,l} - 0.05)^{0.62} \cdot (0.37 \cdot \bar{v}_{a,l} \cdot Tu + 3.14)
\]  

(2.3)

<table>
<thead>
<tr>
<th>DR</th>
<th>Draught rate</th>
<th>[%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>(t_{a,l})</td>
<td>Local air temperature</td>
<td>[°C]</td>
</tr>
<tr>
<td>(\bar{v}_{a,l})</td>
<td>Local mean air speed, &lt; 0.5 m/s</td>
<td>[m/s]</td>
</tr>
<tr>
<td>(Tu)</td>
<td>Local turbulence intensity, 10-60 %</td>
<td>[%]</td>
</tr>
</tbody>
</table>

Turbulence intensity can be estimated to 40% if unknown. Using CFD, it is possible to determine this value more accurately from a turbulent kinetic energy, \(k\), as shown in equation (2.4) [ANSYS, 2020d].

\[
Tu = \sqrt{\frac{2k}{\bar{v}_{a,l}}}
\]  

(2.4)
Three thermal environment categories are defined with different maximum values for thermal comfort. Category A, B and C allows maximum DR of 10%, 20% and 30%, respectively. Norwegian regulations do not require any limitations on which category to follow, and a middle category B is therefore used for evaluation in this thesis.

2.4 Contaminant Removal Effectiveness

When facilitating for a good indoor air quality, the distribution of fresh air and removal of contaminants is important. The contaminant removal efficiency evaluates a system’s ability to remove contaminants by comparing the concentrations at different locations. The efficiency is generally expected to be above 1.0 for a displacement ventilation system, as the concentration at the outlet should be higher than in the occupied zone. As this study focuses on indoor environment for occupants and cross contamination between them, the contaminant removal efficiency in the occupied zone, $\varepsilon_{oz}$, and personal exposure, $\varepsilon_{exp}$, are calculated using the equations below.

$$\varepsilon_{oz} = \frac{c_e - c_s}{c_{oz} - c_s}$$  \hspace{1cm} (2.5)

$$\varepsilon_{exp} = \frac{c_e - c_s}{c_{exp} - c_s}$$  \hspace{1cm} (2.6)

The equations require measurements of contaminant concentration in the exhaust, $c_e$, supply, $c_s$, occupied zone, $c_{oz}$ and mouth of the occupants $c_{exp}$. The occupied zone is defined by Skistad et al. [2002] to be 0.5 m from walls, and up to a height of 1.3 m for seated or 1.8 m for standing persons.

When multiple heat sources are present in a room with displacement ventilation and the contaminant source is not the warmest, contaminant and velocity distributions are expected as shown in figure 2.2.

![Figure 2.2: Contaminant distribution for displacement ventilation Skistad et al. [2002]](image)
The figure indicates that the contaminant concentration increases until it reaches a stratification height, creating a layer above the contaminant source. In this layer, the concentration is near constant until a stratification layer from a warmer source is entered. Since this plume contains more fresh air, the concentration decreases up until the ceiling. The temperature is expected to be at the lowest just above floor level and increasing until reaching the ceiling.

2.5 Design Chart

The design chart is a tool developed with the purpose of presenting operating limits for different ventilation concepts. Several parameters can be included in the chart as limits, although air quality and thermal comfort are the most commonly evaluated, as seen in figure 2.3.

The lines drawn in the chart defines boundaries for whether or not the system performs within the limit parameters, for example as a minimum flow rate to ensure acceptable air quality, or as a maximum temperature gradient or flow rate to avoid draft. The grey areas indicate that unacceptable conditions occurs, leaving the white region as the systems operating area.

The chart has been widely used for analysing the performance of different ventilation concepts in a defined room geometry with internal heat sources referred to as Annex 20 room. They have gradually been added to the chart, and a well developed basis for comparison of concepts has been made. This is shown in figure 2.4 where studies up until the year 2007 are included.
The limits used in the design chart is a minimum fresh air supply of 0.02 m\(^3\)/s, maximum ankle velocity of 0.15 m/s and a maximum temperature gradient from ankle to neck of 3 °C. By interpreting the design chart curves, the performance of a diffuse ceiling ventilation concept shows a significantly more flexible performance than the rest, while displacement ventilation has one of the smallest operating ranges.

### 2.6 Annex 20

A standardized room for evaluating air flow patterns in buildings, Annex 20, was introduced in 1991 by the International Energy Agency. The room measures 4.2 x 3.6 x 2.5 m, with walls made of wood and a minimum of 10 cm insulation.
Part I

Preliminary Study
Chapter 3

Literature Review

Previous projects are reviewed with the aim of finding suitable data for the alternative validation approach. Reports and other sources which supports choices made in this thesis are presented, especially in relation to geometry simplifications, meshing, turbulence modeling and near wall treatment.

3.1 State of the Art

The idea of supplying air from inlet areas covering entire walls has been around for years, although the study made as part of this thesis shows that thorough research and documentation lacks on the performance of such concepts.

There are not many standardized products or solutions currently on the market, and the most similar designs work according to the mixing ventilation principle where diffusers or room units with top mounted inlets are used.

An example of a product design satisfying many of the same needs as the diffuse wall is a decentralised ventilation unit, AM-900VD, created by AirMaster A/S. This is delivered as a complete unit with all functions integrated, only to be connected with electricity and ducting.

It is developed for larger rooms like class rooms and open office spaces, and delivered with either a mixing or displacement configuration. From a design perspective, the unit is compact, i.e. 2.3 m tall, 0.8 m wide and 0.7 m deep, and has a maximum capacity of 720 m$^3$/h. The inlet is one meter tall, starting at floor level, while the exhaust is 0.3 m tall placed in the top of the unit, as seen in figure 3.1.

![Figure 3.1: AM-900VD](image)
Chapter 3. Literature Review

Compared to the diffuse wall, this unit is fully decentralized with all functions and necessary components integrated. From a visual perspective, the diffuse wall concept is an integrated design element of the room, natural to include in renovations or new buildings, while units like AirMaster’s solution can be installed in any room at any time.

3.2 Relevant Projects

A study by Nielsen [1995] is one of few mentioning a diffuse wall concept, referred to as a cooling wall in this case. Scale experiments and 2D CFD simulations were conducted in a room with 450 m$^2$ floor area and 18 m height, of which the first 12 m was utilized as inlet. The concept acted as a displacement ventilation system, and was dimensioned for a heat load of 40 kW from occupants. Results were later validated with on-site full scale measurements, where the CFD gave similar flow patterns and velocities using the $k$-$\varepsilon$ turbulence model, showing a large cooling capacity with reasonable air velocities in the occupied zone.

A diffuse ceiling ventilation concept, where a suspended ceiling acted as a plenum chamber for supplying fresh air into a room through ceiling mounted permeable plates was investigated by Zhang et al. [2014]. The study focused on thermal comfort and cooling capacity, with special attention to draught in the occupied zone, much like in this thesis.

Limitations of the system was later plotted into a design chart by Kristensen and Jensen [2015]. This was done using CFD simulations validated through experimental studies, and showed high flexibility for the diffuse ceiling concept. The study contained a real world application at a school and a parametric study made with laboratory measurements and CFD simulations. It concluded that the plenum works as a pressure chamber with equal air distribution through the porous surface.
3.3  CFD Models used in relevant projects

User experience often plays a big part in developing CFD models, and previous projects are investigated with the aim of obtaining relevant knowledge for use in this thesis.

3.3.1  Mesh

Meshing is by many described more as an art than exact science. As good mesh development is crucial for getting accurate results, a thorough evaluation of how meshes are created in similar projects is made.

A cartesian mesh was used for the 2D cooling wall by Nielsen [1995]. This has similar properties to a cut cell hexagonal or hexcore mesh more recent CFD software. The diffuse wall ventilation study by Zhang et al. [2014] used a non-uniform structured hexahedral mesh with refinement near surfaces. Kristensen and Jensen [2015] applied a hexcore mesh for a similar geometry using tetrahedral elements and refined inflation near surfaces. The final mesh had 490 K cells, as the software limited the total amount to 512 K.

Balafas [2014] concluded that hexa- and polyhedral elements outperform tetrahedral with faster convergence and less iterations. The hexahedral mesh was best for simple geometries, while polyhedral was the most flexible and stable performer. The same conclusions were drawn in a quantitative analysis conducted by Iqbal and Chan [2015].

3.3.2  Manikins

Thermal and breathing thermal manikins are the two main models for physically representing people when investigating indoor environment. The research on this is vast, and many types of manikins have been tested experimentally.

The representation of a person in CFD is a complex task due to the geometry of the body. A computer simulated person (CSP) developed by Brohus [1997] has been widely used, modeling a standing woman with a height of 1.7 m and surface area of 1.62 m².

This model was designed specifically to represent a breathing thermal manikin made at the Technical University of Denmark, and was intended for both contaminant and thermal comfort evaluations. An opening of 1.3 cm² was placed
in the face region, representing the breathing nostrils of the manikin. Other research projects have treated this opening differently, many using a 3.0 cm tall and 4.0 cm wide surface, and by this reducing the cell sizes needed for meshing the regions near the exposure point.

In this thesis, only seated persons are going to be modeled, and the CSP is therefore slightly altered. Research by Topp et al. [2002] shows that the level of detail in the geometry of the manikin makes an impact to the local air flow pattern. Thus, smaller changes to the geometry are insignificant on the flow at larger distances from the manikin.

Minor geometrical changes were further investigated by Topp et al. [2003] who looked into the impact of modeling the space between legs of a sitting person. It was proven to be of importance, as air is not forced to flow around, but rather between the legs. This generally applied to flows normal to the direction of the person.

3.3.3 Near wall treatment

Literature on the differences between wall modeling and functions are studied to make a decision on what to use. The wall function approach is preferable to wall modeling in general CFD cases as resolving the laminar and buffer zone is uneconomical. The approach does not work well in low Re cases, but is economical, robust and reasonable accurate for high Re flows. The scalable wall function has been used with good results in similar cases and is effective when the thickness of layers near walls is unknown. [Versteeg and Malalasakera, 1995]
Chapter 4
Comparison Project

An alternative procedure for CFD model validation is conducted as described by the workflow in section 1.3. The diffuse ceiling study by Zhang [2016] is chosen for comparison, as it looks into thermal comfort parameters using the design chart method and has available measurement data.

4.1 Background

The performance of the diffuse ceiling was investigated for several scenarios through experiments and CFD simulations. The performance was then evaluated based on both thermal comfort and energy through 20 cases, including the use of thermally active building constructions (TABS) and solar heat gain through a window. As this thesis does not focus on energy, Case 12 is chosen for validation as it does not consider these extra parameters.

4.2 Flow Element Analysis

A diffuse ventilation concept has an air flow behaviour mainly governed by buoyancy forces. Fresh air is supplied into a plenum chamber separated from the room by a porous material acting as a diffuser. A pressure drop is created across the material, with a magnitude depending on its porosity. The over-pressure in the plenum forces air into the room at low velocities due to the large inlet area, and the concept is mainly focused on ventilation combined with the possibility of cooling.

Based on this knowledge, assumptions on flow behavior are made for the diffuse wall prior to running simulations. Expected flow patterns for the different elements are portrayed, before they are combined into hypothetical models for the ventilation concept.
4.2.1 Heat loads

In the diffuse ceiling experiment made by Zhang et al. [2014], the room contained heat sources which influenced the supply distribution. This was caused by thermal plumes developing around heat sources, as illustrated for a person in figure 4.1.

![Figure 4.1: Thermal plume from heat source](image)

Buoyancy forces move air in an upwards direction until it reaches a neutral height, where the driving forces stop and a horizontal stratification layer is developed. Due to the vertical kinetic energy still present in the flow, parts of it will continue until it reaches a maximum height where the momentum is equal to zero. At this point, the air flows back down into the normalization height and stabilizes.

4.2.2 Diffuse Wall Ventilation

The diffuse wall concept supplies air through a wall made from a thin ventilation fabric. This is built in front of a plenum chamber with even pressure distribution, as shown together with an expected flow pattern in figure 4.2.
4.2. Flow Element Analysis

The concept is expected to act similar to a displacement ventilation principle. By supplying air into a room with a given temperature difference, air moves down the face of the wall and spreads along the floor. Past knowledge about displacement ventilation has shown that this flow can be simplified as a layer of approximately 20 cm height, continuing at constant velocity until it is obstructed [Skistad et al., 2002].

4.2.3 Diffuse Ceiling Ventilation

Diffuse ceiling ventilation has been investigated in several reports where it has shown a flexible performance with a wide range of acceptable air flow rates and temperature differences. It can be a visually pleasing system that reduces the need of ductwork, although it restricts the available room height.
Chapter 4. Comparison Project

According to previous studies, inlet air will distribute evenly across the porous face [Zhang, 2016; Kristensen and Jensen, 2015]. This is illustrated in figure 4.3, with air supplied into the chamber at the right hand wall. To ensure an even pressure distribution, it is necessary to have a correct resistance in the material. If this is low, the uniform pressure will not develop, and by having a too high resistance, the required pressure will be unnecessarily large.

**Room flow**

When combining flow elements, the expected behaviour gets more complex. Figure 4.4 illustrates the flow pattern for a diffuse ceiling case based on results obtained by Zhang [2016].

![Figure 4.4: Section view of diffuse ceiling flow pattern](image)

The figure shows that the upward momentum from the plume counteracts the downward momentum from the diffuse ceiling. This prevents the air from being distributed evenly across the ceiling panels, and a non-uniform velocity profile is generated.

Past studies show that a bias towards the inlet wall occurs when plumes hit the ceiling. Several factors play in on why this happens. Firstly, the wall on the right hand side has a slightly lower u-value, hence a colder surface temperature making for more down draught and more circulation in that area. Secondly, the studies showed that the air from the inlet at high temperature differences is prone to exit the porous zone at a higher rate close to the inlet, also contributing to a higher circulation.
4.3 Experiment from Comparison Project

To make use of the experimental results obtained by Zhang [2016] for validating the CFD models developed in this thesis, it is necessary to understand how the experiments were conducted. The lab setup with geometries and boundary conditions, as well as the measurement procedure is therefore explained in the following section.

4.3.1 Lab setup

The experiments were conducted in a room with length, width and height of $4.8 \times 3.3 \times 2.335$ m, simulating a two person office with manikins, computers and lamps as heat sources. The computers had heat loads of 55 and 45 W, the monitors 16 and 21.5 W, lamps 54 and 59 W, and the manikins provided 100 W each, giving a total of 450.5 W. Manikins are placed centered in the room at their respective desks facing each other as shown in figure 4.5.

![Figure 4.5: Experiment manikin setup [Zhang et al., 2014]](image)

A section view of the experimental room geometry is shown in figure 4.6. Porous ceiling panels from Troldtekt, initially intended for sound damping, with a thickness of 0.035 m worked as the diffuse ceiling, separating it from a 0.35 m high plenum. The plenum ceiling was made of concrete slabs containing tubes for the TABS.
The inlet was placed within the plenum on the east wall facing a cold chamber with the exhaust placed beneath it inside the room. A window with dimensions 2.3 \times 1.4 \text{ m} was placed between the inlet and outlet for simulating the solar heat gain cases. A plan view of the room is shown in figure 4.7, alongside measurement locations for temperature and velocity measurements.
4.4. CFD Model Development

4.3.2 Measurements

Measurements were made using 3 columns with equipment mounted at specific heights. By moving them between 4 locations and repeating the same experiments, a total of 12 measurement locations were obtained. Velocities were measured using anemometers at the heights 0.1, 0.7, 1.1, 1.7 and 2.285 m. Temperature measurements were done with thermocouples mounted at each anemometer location, as well as at 2.135 and 2.235 m on every pole. Velocity, temperature and pressure measurements were also made in the plenum. [Zhang et al., 2014]

Anemometers used for velocity measurements had an accuracy of ±0.01 m/s plus 5% of readings and the thermocouples were defined with an accuracy of ±0.15 K. [Zhang, 2016]

4.4 CFD Model Development

Although CFD results from Zhang [2016] are available, new models are created to validate the experimental results in a controlled manner. Zhang [2016] created both a porous media model and a radiation model, and it must be emphasised that only the porous media model is created in this thesis. Due to time restrictions and the main project aim being to investigate the diffuse wall concept, the surface temperatures from the existing radiation model are utilized as boundary conditions without further research.

The porous media model was used to define the diffuse ceiling, where the ceiling panels were treated as a fluid zone instead of solid material. Air that passed through this medium was subject to a pressure loss, simulating the panels used in the experiments. As a result of this, there were no solid surfaces facing the plenum and room, and surface radiation could therefore not be modeled directly.

The lack of radiation was accounted for by the radiation model, using a solid ceiling with rectangular slot inlets to represent the same properties as the diffuse ceiling. By running simulations with this model, temperatures for all surfaces were obtained along with a source term value for the porous zone, simulating the radiative heat flux from the surfaces. Lab measurements resulted in a heat unbalance, and a source term was also added in the room to ensure total heat balance.
Chapter 4. Comparison Project

4.4.1 Geometry

The room geometry is similar to the experiment setup with length, width and height equal to $4.8 \times 3.3 \times 2.72$ m, where the height includes the diffuse ceiling thickness of 0.035 m and plenum height of 0.35 m. The manikins, computers and lamps working as heat sources are located as shown in figure 4.8. The work stations are defined "side 1" and "side 2", with "side 1" being the northernmost station.

![Figure 4.8: Plane view of diffuse ceiling geometry](image)

The manikins made by Zhang [2016] were simplified into cuboids. A more refined geometry is made in this thesis, as the manikins are facing perpendicular to the bulk air flow, and the assumption is made that modeling the thighs, knees and legs of a seated person is of importance. A gap between the legs is not considered to be worth the computational cost, since the table is assumed to block the upwards flow. Surface areas of the manikins, tables, computers and lamps are modeled the same way as done by Zhang [2016], with the dimensions shown in figure 4.9.
4.4 CFD Model Development

Air is supplied into the plenum through a rectangular slot of $2.3 \times 0.03 \text{ m}$ placed above the window, while the outlet is a $0.14 \times 0.14 \text{ m}$ square placed at the end of a $0.7 \text{ m}$ long duct near the floor. The duct is included for the flow to stabilize before reaching the outlet surface, as convergence and backflow problems may occur otherwise. The window is included in the CFD model to match the boundary conditions obtained from the radiation model, although solar heat gain is not investigated.

4.4.2 Mesh

Since the geometry is simplified into shapes with straight edges, it is beneficial to make use of hexahedrons, and a hexcore mesh is therefore applied. As the name suggests, the core of a zone is meshed using hex cells while regions near walls and heat sources are meshed with either polyhedrons or tetrahedrons.

Research presented in section 3.3 shows that polyhedral meshes in many cases outperform the others, although tetrahedral meshes provide fairly accurate results. Convergence difficulties were experienced when using polyhedral meshes in this thesis, and it was therefore decided to use tetrahedrons which allowed convergence of the solution.

A mesh independence study was made for cell counts of $420 \text{ K}$, $750 \text{ K}$ and $1.1 \text{ M}$. Velocities and temperatures were monitored, obtaining the results shown in appendix A. The coarsest mesh differs significantly from the others, and it was concluded that mesh independence was reached with the $750 \text{ K}$ mesh shown in figure 4.10 for a clipping plane not including lamps.
4.4.3 Porous media model

In the porous media model, ceiling panels are represented as a uniform porous zone defined with viscous and inertial forces to obtain a given pressure drop. The viscous and inertial resistances were found to be $1.14 \times 10^8 \text{ m}^{-2}$ and $33055 \text{ m}^{-1}$ respectively by using pressure measurements from the experiments. The derivation of these values can be seen in appendix B.

A fluid porosity of zero and a wood-cement material for the ceiling was defined, resulting in heat transfer being modeled through the zone based on the material properties. The air flow is defined as laminar, meaning that no turbulence is calculated in this zone.

4.4.4 Boundary conditions

The boundary conditions are based on data from experiments and necessary assumptions. All values are shown in appendix C. Wall surfaces are defined with constant temperatures according to the radiation model values. Heat sources are assigned surface heat fluxes, and the inlet temperature is set to $8.95 \degree \text{C}$ for an air change rate of 2 ACH.
As the radiation model accounted for radiation heat by adding surface temperatures, only a convective portion of the energy is assigned to the heat sources as fluxes. The ratios of the total heat belonging to the convective portions are found through the radiation model, as listed in table 4.1.

<table>
<thead>
<tr>
<th>Heat source</th>
<th>Area [m²]</th>
<th>Total [W/m²]</th>
<th>Convective [W/m²]</th>
<th>Convective ratio [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Person 1</td>
<td>1.92</td>
<td>52.1</td>
<td>26.6</td>
<td>0.51</td>
</tr>
<tr>
<td>Person 2</td>
<td>1.92</td>
<td>52.1</td>
<td>25.8</td>
<td>0.50</td>
</tr>
<tr>
<td>PC 1</td>
<td>0.56</td>
<td>127</td>
<td>106</td>
<td>0.84</td>
</tr>
<tr>
<td>PC 2</td>
<td>0.56</td>
<td>119</td>
<td>99.3</td>
<td>0.84</td>
</tr>
<tr>
<td>Lamp 1</td>
<td>0.06</td>
<td>900</td>
<td>658</td>
<td>0.73</td>
</tr>
<tr>
<td>Lamp 2</td>
<td>0.06</td>
<td>983</td>
<td>719</td>
<td>0.73</td>
</tr>
</tbody>
</table>

Table 4.1: Convective heat fluxes [Zhang et al., 2014]

Air flow in the porous zone is defined as laminar, and energy source terms of 206.8 W and 40.59 W are added for the porous zone and the room volume, respectively. The room source term is applied as a correction for the heat unbalance rate in the measurement. It was below an accepted 10% error range due to measurement and calibration accuracy, and the experiment was considered to be in balance [Zhang et al., 2014].

4.4.5 Solution settings

The SST $k-\omega$ and RNG $k-\epsilon$ turbulence models were investigated for both the PRESTO! and Body Force Weighted pressure discretization schemes under the solution conditions shown in table 4.2. A scalable wall function was applied for the $k-\epsilon$ models.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure-velocity coupling, Scheme</td>
<td>SIMPLE</td>
</tr>
<tr>
<td>Spatial Discretization, Gradient</td>
<td>Least Squares Cell Based</td>
</tr>
<tr>
<td>Spatial Discretization, rest</td>
<td>Second Order Upwind</td>
</tr>
<tr>
<td>Under Relaxation Factor</td>
<td>0.2/1.0/1.0/0.5/0.8/0.6/0.8/0.8</td>
</tr>
</tbody>
</table>

Table 4.2: Diffuse ceiling solution settings
4.5 Validation results

Three measurement poles at two different locations according to figure 4.7, i.e. pole 1, 2 and 3 at location A and B are used as examples to show the difference between the tested turbulence models. These are presented as velocity and temperature plots in figures 4.11 and 4.12, along with lab results adjusted for measurement accuracy as described in section 4.3. The accuracy of the profiles between points shown from the lab measurements is not possible to determine, as no measurements are available at these heights.

Figure 4.11: Validation velocity results, pole A1, A2 and A3

Figure 4.12: Validation temperature results, pole A1, A2 and A3
The velocity plots show that several of the turbulence models predict accurate velocities above 0.1 m, although the SST $k-\omega$ with a Body Force Weighted scheme performs slightly better. At 0.1 m it overestimates the velocity, indicating that the mesh should be refined near the floor, which will be taken into consideration when developing the diffuse wall mesh.

The plots in figure 4.12 show that all models perform well in predicting temperatures at the selected poles. A comparison between the models is shown in table 4.3, with average errors and total heat imbalances calculated from all measurement locations.

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>Pressure scheme</th>
<th>Avg. velocity error</th>
<th>Heat imbalance</th>
</tr>
</thead>
<tbody>
<tr>
<td>SST $k-\omega$</td>
<td>PRESTO!</td>
<td>26 %</td>
<td>$-$0.20</td>
</tr>
<tr>
<td>RNG $k-\epsilon$, scalable</td>
<td>PRESTO!</td>
<td>25 %</td>
<td>$-$0.05</td>
</tr>
<tr>
<td>SST $k-\omega$, BFW</td>
<td>BFW</td>
<td>20 %</td>
<td>$-$0.08</td>
</tr>
<tr>
<td>RNG $k-\epsilon$, scaleable</td>
<td>BFW</td>
<td>27 %</td>
<td>$+$0.40</td>
</tr>
</tbody>
</table>

Table 4.3: Turbulence model error

The CFD simulations have some uncertainty, as they are based on a radiation model. Since radiation heat is simplified into surface temperatures and source terms, a certain amount of error is expected, which in that case will be passed on to new CFD simulations. This influence is seen as temperatures differ noticeably from the measurements near surfaces. By considering the magnitude of the deviations and the height of their occurrence, these are evaluated to be within an accepted range, although it must be accounted for when drawing conclusions on model validation.

Based on the evaluation of profiles and percentage errors, the CFD simulations are considered to be accurate for velocity and temperature prediction within the occupied zone using the SST $k-\omega$ Body Force Weighted model. This predicts velocities better than the other ones tested. The heat imbalance is also well within the accepted range, as ANSYS [2020a] recommends the heat imbalance to be less than 0.5 % of the total heat load.
Part II

Main Study
Chapter 5

Diffuse Wall

After validating CFD simulations from the diffuse ceiling comparison project, a diffuse wall model can be developed. The settings, turbulence model and mesh is based on the validated model, while choices regarding geometry and which cases to run still have to be made. A hypothesis for expected flow patterns is developed to support assessments made from the simulation results.

5.1 Problem Description

In the wake of the COVID-19 pandemic, an increased awareness of a ventilation system’s ability to remove contaminants and inhibit spreading of diseases is expected. Contaminant removal effectiveness is therefore looked into, in addition to thermal comfort and system performance.

The diffuse wall concept is investigated for three baseline cases. These are defined with temperature differences between inlet and room air of 3, 6, and 10\( ^\circ \)C, and with constant convective heat loads of 322 W, corresponding to what has been done in Annex 20. Air flow rates are calculated accordingly, making three fully defined test cases.

As the concept is intended for use in rooms with different properties, sizes and loads, its flexibility and operating range will be determined by using the design chart method described in section 2.5. A selection of temperature differences and air flow rates beyond the baseline cases is needed to find the concept limits by inter- or extrapolating between these.
5.2 Hypothesis

A hypothesis on how the ventilation concept works in this specific case is made based on the individual flow elements present in the room.

The flow is divided into three parts, i.e. down draught along the cold diffuse wall, buoyancy driven plumes from heat sources and supply air through the fabric, shown in figure 5.1 as q1, q2 and q3, respectively. By calculating the air flow rates generated by each element, a simplified estimation is made for the air velocity at ankle level of a seated person, providing an initial idea of the system performance.

Velocities at 0.1 and 1.1 m above floor level is of particular interest as occupants are seated. Although it is common in modern offices to have the possibility of doing work standing at height adjustable desks, this is not considered in this report.

\[ q_1 = 2.8 \times 10^{-3} \cdot |\Delta T|^{0.4} \cdot y^{0.7} \cdot w \]  
\( \Delta T \): Temp. difference between surface and room [K]  
\( y \): Distance measured from ceiling [m]  
\( w \): Width of the surface [m]

Calculations are made at 0.1 m above the floor, with a wall width of 3.6 m. It is assumed that the horizontal flow along the floor has a standard thickness of 0.20 m, which gives an air flow of \( q_1 = 181.71/\text{s} \). [Ingebrigtsen, 2015, eq. 8.91]
Air flow in plumes from heat sources is calculated by defining virtual point sources. The height from these points to the top of the actual heat source is defined as \( z_0 = 2.3 \cdot D \), where \( D \) is the width of the source [Skistad et al., 2002, p. 17]. Based on this, distances between ceiling and point sources are found, which is useful for determining the air flow in a plume given by equation (5.2).

\[
q_2 = 2.38 \cdot \Phi_{cf}^{3/4} \cdot s^{-5/8} \cdot Z_1
\]  
\[
s = \Delta T \cdot H^{-1} \cdot k
\]  

The temperature at the outlet is assumed equal to the room, with a symmetric temperature gradient in the vertical direction, corresponding to \( k = 0.5 \).

The variable \( Z_1 \) is calculated based on a dimensionless height, \( z^* \), according to equations (5.4) and (5.5), where \( z \) is a given height above the point source.

\[
Z_1 = 0.004 + 0.039 \cdot z^* + 0.380 \cdot (z^*)^2 - 0.062 \cdot (z^*)^3
\]  
\[
z^* = 2.86 \cdot z \cdot \Phi_{cf}^{3/8} \cdot \Phi_{cf}^{-1/4}
\]  

By defining the heights above point sources as \( z \), the air flow rate from each heat source is calculated to the values in table 5.1, where similar sources have equal values.

<table>
<thead>
<tr>
<th>Heat source</th>
<th>Location above floor [m]</th>
<th>Height [m]</th>
<th>( \Phi_{cf} ) [W]</th>
<th>( z ) [m]</th>
<th>( q_2 ) [l/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Person</td>
<td>0.0</td>
<td>1.3</td>
<td>50</td>
<td>1.79</td>
<td>47.7</td>
</tr>
<tr>
<td>PC</td>
<td>0.7</td>
<td>0.4</td>
<td>67.2</td>
<td>1.99</td>
<td>50.3</td>
</tr>
<tr>
<td>Lamp</td>
<td>1.2</td>
<td>0.1</td>
<td>43.8</td>
<td>1.35</td>
<td>31.8</td>
</tr>
<tr>
<td>Total (x2)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>259.4</td>
</tr>
</tbody>
</table>

\textbf{Table 5.1: Plume air flow rates}

As seen from the diffuse ceiling flow elements in chapter 4, air spreads along the ceiling with a bias towards one of the walls. An equal behaviour is assumed for the diffuse wall, defining that 60% of the flow moves towards the inlet wall. By including the flow through the diffuse wall, \( q_3 \), a total air flow rate is calculated to 271.71 l/s using equation (5.6).

\[
q_{tot} = q_1 + 0.6 \cdot \sum q_2 + q_3
\]  

(5.6)
An adjacent zone is defined in front of the diffuse wall as $l_w = q^{0.7}$ [Nielsen et al., 2007]. From this, a maximum velocity is calculated at a length $x$ from the diffuser in equation (5.7).

$$v_x = 0.2 \cdot l_w \cdot x^{-1}$$  \hspace{1cm} (5.7)

A person is located at a distance of $x = 0.955 \text{ m}$ from the diffuse wall, where the ankle velocity is calculated to $0.084 \text{ m/s}$ for $\Delta T3$. Using the same assumptions and formulas for the other cases, the velocities are $0.075 \text{ m/s}$ at $T6$ and $0.072 \text{ m/s}$ at $T10$.

The calculations show a decrease in ankle velocities with increasing temperature differences, which would be expected as the air flow rate also is decreased. These velocities are well within the thermal comfort limits, and the ventilation system shows potential for supplying air without draught at ankle level. It has to be emphasized that these calculations are simplified and do not take into account complex flow patterns such as recirculating flows, which will be investigated through CFD.

### 5.3 CFD Model Development

Since parameters from the validated comparison project are used as inputs for the diffuse wall, the model is developed using a similar approach. A porous media model was initially intended to represent the diffuse wall by assigning parameters based on pressure measurements. This was later simplified due to the limitations explained in section 1.2.

#### 5.3.1 Initial porous media model

As pressure drop measurements could not be obtained experimentally, values from manufacturers data sheets were used as a theoretical approach. A detailed explanation of how porous parameters are calculated can be seen in appendix B.

This caused uncertainties for the model before any evaluations were made. Simulations exposed a flow highly sensitive to changes in porosity values, resulting in simulation and convergence difficulties, proving that pressure measurements are important to model a correct flow through a porous media.

Cell sizing problems also occurred in the porous zone, due to its 0.0004 m thickness. *Porous jump*, a 1D simplification of the zone, was tested to avoid the
problem of meshing a thin body by defining the pressure drop across a face [ANSYS, 2009]. Although this approach resolved the volume mesh challenges, similar inputs for porosity values were required, resulting in similar simulation problems. Considering the uncertainty introduced by having porous media without measurement data, it was decided to do additional model simplifications in order to minimize the potential sources of error.

5.3.2 Simplified simulation model

The plenum chamber and porous fabric were excluded in the model, defining the inlet as a surface with uniform velocity distribution instead. This is based on findings by Kristensen and Jensen [2015], showing that the velocity from the porous face is uniformly distributed when a sufficient pressure drop across the wall is achieved. It is also assumed that the inlet distribution profile is unaffected by thermal plumes, as the inlet is along a wall instead of the ceiling.

The diffuse wall concept is intended to operate with significant pressure drops, and momentum from the inlet is assumed to be governing for the flow direction. The simplification is therefore considered to be within reason, given that a correct plenum pressure is obtained.

Although the concept suggestion has lamellas in front of the fabric, the baseline cases are investigated for a full wall inlet as shown in figure 5.2. This is done to test the initial performance before two additional inlet geometries are investigated through a parameter study.

The CFD model is made by defining the room as one fluid zone. As no experimental measurements are made, information regarding surface radiation is not available and radiation is excluded from the simulations.
5.3.3 Geometry

Annex 20, described in section 2.6, is chosen for this study due to the vast amount of work on other ventilation principles in the same room, making direct comparisons possible. Dimensions are shown in figure 5.3, with the diffuse wall measuring $3.6 \times 2.5$ m on the west side, and "side 1" being the work station closest to it. [Lemaire et al., 1993]

![Figure 5.3: Diffuse wall CFD geometry](image)

Manikins are designed as in the majority of experiments done in Annex 20, with a surface area of $1.9908$ m$^2$, based on the standard CSP size proposed by Brohus [1997]. The seated geometry is developed based on research by Topp et al. [2003, 2002], with dimension as shown in figure 5.4. Mouths used for contaminant testing measure $0.04$ m wide and $0.03$ m tall, and are marked as blue.
To evaluate the simulations, 4 monitoring points are defined in figure 5.5 as A1, A2, B1 and B2, along with 10 points to monitor flow elements along the floor, marked as C.
Mesh

Hexcore meshes with tetrahedrons near surfaces are used with refined meshing near surfaces, with special attention around mouths. Using experiences from the comparison project, the height of the refined zone above the floor was increased in an attempt to capture flow patterns more accurately.

Mesh independence study

Three meshes for a full wall inlet with 500 K, 700 K and 1 M cells were created for the mesh independence study, running simulations with a 3 K temperature difference. Velocities and temperatures are presented using the two points near manikins, A2 and B2, shown in figure 5.6.

![Figure 5.6: Diffuse wall mesh independence](image)

Based on the velocity and temperature plots, it is clear that meshes with 700 K and 1 M cells give similar results, while 500 K differs significantly. To evaluate whether the visual interpretation of the plots corresponds to monitored values, the meshes are evaluated against the finest 1 M mesh by calculating average velocity and temperature deviations. These are presented in table 5.2 along with total heat transfers and contaminant mass fraction at the outlet for stability monitoring.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1 M</td>
<td>-</td>
<td>-</td>
<td>-0.689</td>
<td>1.00 × 10⁻⁴</td>
</tr>
<tr>
<td>700 K</td>
<td>0.00672</td>
<td>0.242</td>
<td>-0.325</td>
<td>1.00 × 10⁻⁴</td>
</tr>
<tr>
<td>500 K</td>
<td>0.0137</td>
<td>1.96</td>
<td>-1.77</td>
<td>2.10 × 10⁻⁴</td>
</tr>
</tbody>
</table>

Table 5.2: Diffuse wall mesh study
Comparison of the three meshes showed a satisfactory independence with 700 K cells. This mesh is shown in figure 5.7, and is used for further investigations in this thesis.

![Mesh section view of person and surfaces](image)

**Figure 5.7: Mesh section view of person and surfaces**

### 5.3.5 Boundary conditions

The boundary conditions are defined using information from the comparison project regarding convective heat.

**Heat load**

Convective heat transfer ratios are obtained from the radiation model in the comparison project, as listed in table 5.3. Heat fluxes for each heat source is calculated to the values in table 5.4, corresponding to a sum of 480 W total or 322 W convective heat load.

<table>
<thead>
<tr>
<th>Heat source</th>
<th>Area [m²]</th>
<th>Load [W]</th>
<th>Convective ratio [-]</th>
<th>Convective flux [W/m²]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Person 1</td>
<td>1.99</td>
<td>100</td>
<td>0.50</td>
<td>25.1</td>
</tr>
<tr>
<td>Person 2</td>
<td>1.99</td>
<td>100</td>
<td>0.50</td>
<td>25.1</td>
</tr>
<tr>
<td>PC 1</td>
<td>0.56</td>
<td>80</td>
<td>0.84</td>
<td>120</td>
</tr>
<tr>
<td>PC 2</td>
<td>0.56</td>
<td>80</td>
<td>0.84</td>
<td>120</td>
</tr>
<tr>
<td>Lamp 1</td>
<td>0.06</td>
<td>60</td>
<td>0.73</td>
<td>730</td>
</tr>
<tr>
<td>Lamp 2</td>
<td>0.06</td>
<td>60</td>
<td>0.73</td>
<td>730</td>
</tr>
</tbody>
</table>

*Table 5.3: Convective heat fluxes*
Surfaces
The floor, ceiling, tables and all walls are defined as adiabatic. This is done as no radiation is considered, and due to lack of information about surface temperatures. Surfaces on heat sources are defined with constant heat fluxes per surface area. The inlet surface is defined as a velocity inlet, while the outlet is defined as an outflow surface.

Air supply
The ventilation energy loss in the room must correspond to the applied heat load to ensure heat balance as indicated by equation 5.8. The necessary flow rate for a given temperature difference between inlet and room air is then derived from equation (5.9).

\[ Q_{\text{trans}} + Q_{\text{vent}} = Q_{\text{heat loads}} \]  
\[ Q_{\text{vent}} = \Delta T \cdot q \cdot C_p \cdot \rho \]  

| \( Q_{\text{vent}} \) | Ventilation loss [W] |
| \( \Delta T \) | Temp. difference between inlet and outlet [K] |
| \( q \) | Air flow rate \([m^3/s]\) |
| \( C_p \) | Heat capacity \([J/(kg K)]\) |
| \( \rho \) | Density \([kg/m^3]\) |

Air flow rates are calculated for the three baseline cases, i.e. 3, 6 and 10 K, as shown in table 5.4 along with ACH, inlet velocity and temperature for an inlet area of 9.0 m². A ventilation loss corresponding to the convective heat load of 322 W is used.

<p>| ( \Delta T ) | Airflow rate | ACH | Inlet temperature | Inlet velocity |</p>
<table>
<thead>
<tr>
<th>( [K] )</th>
<th>([m^3/s])</th>
<th>([1/h])</th>
<th>([^\circ C])</th>
<th>([m/s])</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0.0872</td>
<td>8.30</td>
<td>19.9</td>
<td>0.0097</td>
</tr>
<tr>
<td>6</td>
<td>0.0436</td>
<td>4.15</td>
<td>16.9</td>
<td>0.0048</td>
</tr>
<tr>
<td>10</td>
<td>0.0262</td>
<td>2.49</td>
<td>12.9</td>
<td>0.0029</td>
</tr>
</tbody>
</table>

Table 5.4: Airflow rates

A turbulence intensity of 4 % is defined for the velocity inlet based on assumptions. As previous experiments made in Annex 20 have been conducted for a room temperature of 22.9 \(^\circ C\), this is used as basis for determining inlet air temperatures.

Contaminants
\( \text{N}_2\text{O} \) is used as a tracer gas, as it is not present in the room air. It is also frequently used as tracer gas in lab experiments, making no need for back calculations of the chemical composition of the air at an observed surface.
A seated person of medium activity is expected to exhale 15 to 20 l/h of CO$_2$ [Mysen, 2016]. Based on this, a mass flow of $1.1088 \times 10^{-5}$ kg/s N$_2$O at 37$^\circ$C is used, simulating a person’s breath. This is supplied from the mouth of the person closest to the diffuse wall by defining it as a mass flow inlet. By observing the concentration on the other person’s mouth, it can be determined if contamination will occur between persons.

### 5.3.6 Solution settings

Following the alternative procedure for validation, the solution settings used in the comparison project are applied, as repeated in table 5.5.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulence model</td>
<td>SST $k\omega$</td>
</tr>
<tr>
<td>Pressure-velocity coupling, Scheme</td>
<td>SIMPLE</td>
</tr>
<tr>
<td>Spatial Discretization, Gradient</td>
<td>Least Squares Cell Based</td>
</tr>
<tr>
<td>Spatial Discretization, Pressure</td>
<td>Body Force Weighted</td>
</tr>
<tr>
<td>Spatial Discretization, rest</td>
<td>Second Order Upwind</td>
</tr>
<tr>
<td>Under Relaxation Factor</td>
<td>0.2/1.0/0.8/0.5/0.8/0.6/0.8/0.8/0.8</td>
</tr>
</tbody>
</table>

Table 5.5: Diffuse wall solution settings
Chapter 6

Simulation Results

Results obtained from the three baseline cases described in section 5.1 are presented in this chapter using the boundary conditions in section 5.3.5.

Velocities are displayed using streamlines, and contour plots for temperatures. Surface streamlines are made in a horizontal plane at 0.1 m height and 3D lines are shown from the inlet. Side view contours are plotted in a plane crossing the manikin’s torsos, with the inlet being on the left to visually interpret the flow development and temperature distribution.

The measurement points defined in figure 5.5 are used to monitor temperatures and contaminant distributions as functions of room height. Additional horizontal monitoring planes extending 0.01 m from the manikins in all directions are placed at 0.1 and 1.1 m height of each person to evaluate the thermal comfort, as shown in figure 6.1. These are used to capture the flow characteristics around the person, not only at the side facing the inlet.

Figure 6.1: Monitoring planes for thermal comfort
6.1 Case ∆T3

For a temperature difference of 3 °C, the corresponding air flow rate is 0.0872 m$^3$/s, with an inlet velocity of 0.0097 m/s for the 9 m$^2$ inlet area. Plots of velocity, temperature and contaminants are shown in the following section, along with explanations of what each figure shows.

6.1.1 Velocity

Figure 6.2 shows a high vertical velocity in the supplied inlet air due to gravitational forces. Parts of the air diverges from the bulk flow and stabilizes at certain heights. The remaining flow continues to fall, before most of it is drawn into either a heat plume or a stratification layer. This happens as the momentum in the downward flow is not strong enough to reach the floor level, preventing air supplied at the top of the wall from spreading horizontally along the floor. Inlet air from lower parts of the diffuse wall flows below the stratification layers, supplying the room with fresh air.

![Image of flow pattern, ∆T3](image-url)

**Figure 6.2:** Flow pattern, ∆T3
6.1. Case $\Delta T_3$

Heat sources have a significant influence on the flow pattern as they draw the surrounding air towards them. Inlet air is directed towards the sources around person 1, especially at heights above ankles. Person 2 has a noticeable effect on the flow at 0.1 m, where most of the air is drawn to its ankles, creating higher velocities and a vortex in the back corner. These observations are further supported by the section view in figure 6.3.

Figure 6.3: Velocities, $\Delta T_3$

Figure 6.3 shows how plumes are generated by the manikins, similar to what was seen from lamps and computers during post processing of results. The plumes hit the ceiling before spreading horizontally, where air moved by the plume of person 1 gets entrained back into the vertical supply flow of the inlet, while vortexes are created where the spreading of the two plumes meet. From equation (2.3) and (2.4), the draught rate at neck level of both persons is calculated, showing that person 2 experiences the highest draught rate of 7.6 %, compared to 6.2 % for person 1. The corresponding ankle velocities were measured, where the maximum value of 0.033 m/s occurred at person 2.

From the measurement points C, defined in figure 5.5, inlet velocity profiles are created in a centre plane of the room. These are presented using dimensionless velocities in figure 6.4.
Chapter 6. Simulation Results

Figure 6.4: Velocity profile, $\Delta T_3$

The velocity profiles show a horizontal layer developing along the floor after about 0.2 m. Its depth is challenging to determine, as the distribution is obstructed by the table placed 0.585 m from the inlet. A visual interpretation estimates the depth to be about 0.6 m deep.

6.1.2 Temperature

The $3 \, ^\circ C$ temperature difference restricts any large temperature gradients in the occupied zone, as shown in figure 6.5.

Figure 6.5: Temperature, $\Delta T_3$

The temperature contours indicate how stratification layers occur, a tendency similar to what happens for displacement ventilation systems. The vertical gradient is low at the first meter from the floor, changing less than $1 \, ^\circ C$, before increasing up to a total of $2.5 \, ^\circ C$ difference in the remaining 1.5 meters.
6.1.3 Contaminants

The contaminant distribution from the mouth of person 1 is displayed relative to the exhaust concentration in figure 6.6, showing how the heat plume moves contaminants upwards before spreading out like seen in the velocity figures.

Contaminants are entrained into the inlet air, circulating back to person 1. Fresh air flows below the contaminated air, supplying person 2 with a clean flow along the floor. Air is then entrained by the plume, resulting in marginal contamination at the mouth of person 2.

Since contaminants are drawn into the supply, the result of having the other person as the contaminant source is investigated. A simulation is made using the same boundary conditions, showing the difference between the two situations in figure 6.7 and 6.8, with the outlet marked as red.
The figures clearly show how contaminant diffusion happens as the flow hits the ceiling, although contaminants are removed in both situations without affecting the other occupant.

The contaminant removal effectiveness is calculated in the case with person 1 as source. This is done both for the occupied zone and the personal exposure using equation (2.5) and (2.6), obtaining values of 1.15 and 2.16, respectively.
6.2 Case \( \Delta T_6 \)

For a 6 \(^\circ\)C temperature difference, the air flow rate is 0.044 m\(^3\)/s with an inlet velocity of 0.0048 m/s. A similar research as in \( \Delta T_3 \) is made, with figures presented in appendix E, section E.1. Like seen in \( \Delta T_3 \), the flow from the inlet is affected by gravity resulting in a high downward momentum along the wall. The vertical velocity is greater than in the \( \Delta T_3 \) case as colder air has a higher density.

The flow develops more evenly throughout the room than previously seen. Velocity profiles show a flow depth of about 0.4 m. Due to the increased downward velocity, this starts to develop closer to the inlet than in \( \Delta T_3 \), as momentum forces no longer dissipate before reaching the floor. As a result of this, the flow keeps its momentum and continues horizontally along the floor, instead of diverging on its way down into stratified layers. Deviation is still seen against person 2, who experiences the highest ankle velocity of 0.028 m/s. The even flow along the floor does reduce this significantly as momentum drives it forward, not allowing it to deviate as much as in \( \Delta T_3 \).

Plumes are generated above both persons and hit the ceiling, where parts of the plume from person 1 is entrained into the inlet flow and distributed along the floor. This results in an increase of contaminants around person 2, reducing the contaminant removal effectiveness for both the occupied zone and personal exposure to 1.00 and 0.94, respectively.

More stratification layers are seen as the inlet temperature is decreased. This results in higher temperature gradients in the occupied zone, with a maximum of 1.56 \(^\circ\)C difference from ankles to neck. Draught rates are decreased compared to \( \Delta T_3 \), where person 2 again experiences the highest rate at 7.2 %.

6.3 Case \( \Delta T_{10} \)

At 10 \(^\circ\)C temperature difference, the air flow rate is 0.026 m\(^3\)/s and the inlet velocity 0.0029 m/s. Figures are presented in appendix E, section E.2, where similarities to \( \Delta T_6 \) are seen despite the lower air flow rate. Increased velocities are observed, both vertically at the inlet and horizontally along the floor. A maximum ankle velocity of 0.031 m/s is observed around person 2, and the maximum temperature gradient between ankles and neck is 1.7 \(^\circ\)C. This gives the highest draught rates among the three cases, with a value of 8.6 % for person 1.

The contaminant removal effectiveness is the lowest among the three cases, with a value of 0.98 for the occupied zone and 0.93 for personal exposure. Contour plots show how air from the plume of person 1 is entrained into the inlet flow, contaminating the entire room.
6.4 Baseline case comparison

The three baseline cases are compared in 6.9, using dimensionless contamination and temperature ratios at point A2 as an example.

Both ratios show similarities to the expected profiles in figure 2.2. The contaminant ratio increases steadily in all cases until the stratified layer of contaminated air is entered, where the rate of change increases. This happens sooner with a greater temperature difference, indicating that stratification is closer to the occupied zone. As the maximum concentration is in this layer, the curve turns and the ratio decreases at locations above this point.

The temperature ratio of ΔT6 and ΔT10 are close to equal. The ΔT3 case has a lower gradient in the occupied zone, resulting in a higher value beneath the ceiling. The slopes indicate that heat sources are placed above 0.6 m, as temperatures are fairly steady below this. A summary of the observed thermal comfort and contaminant removal parameters for each case is seen in table 6.1.

<table>
<thead>
<tr>
<th>Case</th>
<th>Max ankle velocity [m/s]</th>
<th>Location</th>
<th>Max DR [%]</th>
<th>Max T_{grad} [°C]</th>
<th>ε_{oz} [-]</th>
<th>ε_{exp} [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>ΔT3</td>
<td>0.033</td>
<td>Person 2</td>
<td>7.6</td>
<td>1.15</td>
<td>1.15</td>
<td>2.16</td>
</tr>
<tr>
<td>ΔT6</td>
<td>0.028</td>
<td>Person 2</td>
<td>7.2</td>
<td>1.55</td>
<td>1.00</td>
<td>0.94</td>
</tr>
<tr>
<td>ΔT10</td>
<td>0.031</td>
<td>Person 1</td>
<td>8.6</td>
<td>1.74</td>
<td>0.98</td>
<td>0.93</td>
</tr>
</tbody>
</table>

Table 6.1: Thermal comfort and contaminant removal effectiveness for full wall inlet
Chapter 7

Design Chart

The diffuse wall ventilation concept is in this chapter added to the design chart presented in figure 2.4. By running multiple cases of different temperature and airflow combinations, limit values are found through inter- and extrapolation using trendlines.

To create a sufficient number of cases, the previously used heat load of 322 W convective heat, or 480 W total, must be altered by adding heat to the manikins, computers and lamps. This approach is used in previous design chart studies, even though it represents unnatural heat loads from persons. A room temperature of 22.9 °C is endeavoured in all simulations.

Several of the ventilation concepts in the design chart are investigated through experiments, meaning that radiation heat is present in the results. These studies have shown that velocities created by radiation are significantly lower than those generated by buoyancy forces, and it is therefore acceptable to compare convective CFD simulations with the experimental evaluations.

The defined limit values for temperature gradient and ankle velocity of the design chart presented in section 2.5 correspond well to the Norwegian building requirements in section 2.2. By applying the minimum requirements in part A and B of the regulations for Annex 20, a minimum air supply of 0.195 m³/s is required. The previously used design chart limits are therefore suitable for evaluating the concept against Norwegian standards.

To calculate the limiting cases, temperature and flow rate combinations shown in figure 7.1 are simulated. A minimum of three simulations are made for the same temperature difference or flow rate, to create a trendline through the points as indicated by the dotted lines in the figure. By defining limit values for temperature gradients, ankle velocity and draught rate, the cases calculated to be at the limits are marked with red crosses. A detailed description of the design chart development is shown in appendix D.
By drawing a line through the limiting cases, the diffuse wall ventilation concept is added to the design chart using the power trendline $\Delta T = 2.6781 \cdot q^{-0.541}$. This is shown in figure 7.2, where the entire chart by Nielsen [2014] is remade for better overview.
Chapter 8

Parameter Study

This chapter aims to investigate how the diffuse wall performs with different inlet configurations. Two inlets, half wall and lamellas, both with an supply area corresponding to 50% of the full wall are created. The half wall inlet supplies air through the lower half, while the lamellas cover the entire wall height with openings being 0.10 m wide, making a total of 18 slots as seen in figure 8.1.

Simulations are made for a temperature difference of 3 °C. Convective heat loads and ventilation losses of 322 W with corresponding air flow rates to maintain heat balance are applied. Similar meshes and converged results are also ensured.

The obtained data is represented similarly as for the baseline cases, using planes for thermal comfort evaluations and the predefined monitoring points for contaminant and temperature.
8.1 Half Wall Inlet

By halving of the supply area, the inlet velocity is doubled to a value of 0.019 m/s. As a result of lowering the inlet height, the vertical velocities along the wall are negligible, and an aligned flow pattern is observed in heights up to about 0.9 m, as seen in figure 8.2.

![Figure 8.2: Flow pattern, half wall](image)

The inlet air maintains its horizontal momentum, resulting in a more even pattern along the floor than previously seen. Air is still diverted towards person 2, and a corner vortex is created. The buoyancy induced velocities are significantly lower, although the momentum forces generate high velocities in the vortex. This results in an observed maximum ankle velocity of 0.032 m/s at person 2, an observation supported by the streamlines in figure 8.3. A low draught rate is also seen, with a maximum 6.9 % at person 2.
The circulation of air between person 1 and the inlet air experienced in the full wall cases is not seen for the half wall. Air moved by heat plumes stratifies into horizontal layers after colliding with the ceiling, instead of being entrained in the inlet flow as seen before, ensuring a clean air supply into the room.

Velocity profiles in figure 8.4 from the half wall show a near uniform velocity distribution in the entire height of the flow. The flow depth decreases after being supplied from the inlet, reaching a height of 0.6 m after a distance of 0.6 m.

The stratification layers are easily seen from the temperatures in figure 8.5. In the occupied zone, the temperature distribution is near constant, and a maximum temperature gradient of 0.90 °C is observed.
Multiple layers are created above the occupied zone, where both warm air and contaminants spread horizontally. Contaminants are forced up and out of the occupied zone, as seen in figure 8.6, resulting in a contaminant removal effectiveness of 2.54 and 11.84 for personal exposure. This shows a remarkably better ability to avoid contamination between occupants compared to the full wall cases.
8.2 Lamella Inlet

Compared to the full wall $\Delta T3$ case in section 6.1, vertical velocities at the inlet are similar, while the flow distributes more evenly along the floor with higher velocities, as seen in figure 8.7.

Lower velocities are observed as the flow approaches person 2, and the corner vortex previously generated is not present. This results in the lowest ankle velocity observed among all cases, with a magnitude of 0.024 m/s, as well as the lowest draught rate of 5.5%. The same distribution along the floor is also seen in figure 8.8.
When interpreting the section view in figure 8.8, higher velocities than in the full wall case are observed in the air separating from the inlet flow. A sudden change of velocity magnitude is seen right beneath table 1, which is easy to misinterpret for a rapid change in flow direction, even though this is not the case.

To understand the flow behaviour in this region, an additional plan view around person 1 is presented for a 0.5 m height in figure 8.9. The plane viewed in figure 8.8 is indicated with a black line across the torso. This shows how air is angled towards the heat sources to a greater extent than seen in the other cases, as spacing between lamellas intensifies the effect. This explains why a sudden change in velocity is observed in figure 8.7, as the flow with the high velocity diverts sideways, thus disappearing from the section view.
Velocity profiles in figure 8.10 show a flow depth of 0.6 m before getting obstructed by the table.

A temperature distribution similar to the full wall ΔT3 case is seen in figure 8.11, with a maximum temperature gradient of 1.27 °C.

The distribution of contaminants is similar to the full wall case. The occupants are supplied with fresh air rising from the floor with the plumes as figure 8.12 shows. The contaminant removal effectiveness for the occupied zone is calculated to 1.30, while the personal exposure contaminant removal effectiveness is 5.04.
To make the differences between the inlet configurations more visible, all three ∆T3 cases are presented using contaminant and temperature ratios, as well as with thermal comfort and contaminant removal parameters in table 8.1.

<table>
<thead>
<tr>
<th>Inlet config.</th>
<th>Max ankle velocity [m/s]</th>
<th>Location</th>
<th>Max DR [%]</th>
<th>Max $T_{grad}$ [°C]</th>
<th>$\epsilon_{oz}$ [-]</th>
<th>$\epsilon_{exp}$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full, ∆T3</td>
<td>0.033</td>
<td>Person 2</td>
<td>7.6</td>
<td>1.15</td>
<td>1.15</td>
<td>2.16</td>
</tr>
<tr>
<td>Half, ∆T3</td>
<td>0.032</td>
<td>Person 2</td>
<td>6.9</td>
<td>0.90</td>
<td>2.54</td>
<td>11.8</td>
</tr>
<tr>
<td>Lamella, ∆T13</td>
<td>0.024</td>
<td>Person 2</td>
<td>5.5</td>
<td>1.27</td>
<td>1.30</td>
<td>5.04</td>
</tr>
</tbody>
</table>

Table 8.1: Thermal comfort and contaminant removal effectiveness for different inlet configurations
8.2. Lamella Inlet

The ratios clearly show how the half wall supplies fresh air with a depth of about 0.8 m before stratification occurs. The lamella configuration performs slightly better than the full wall in keeping contamination ratios low, thus generating stronger stratification layers at 2.0 m height.

Temperature ratios are similar for the full wall and lamella cases. The horizontal inlet flow from the half wall has no temperature ratio at lower heights, before increasing simultaneously as the contamination ratio. All cases show the same tendency as they approach the ceiling.

Figure 8.13: Contaminant and temperature ratios, half wall and lamella
Part III

Review
Chapter 9
Discussion

The hypothesis presented for a full wall inlet in the main study indicated that maximum ankle velocities were well within the thermal comfort limits, and should decay when the temperature difference increases. Results obtained by using the alternative validation approach complied with expectations based on theory and educated evaluations, and provided convincing flow patterns for evaluating the diffuse wall.

An increase in temperature difference gave higher temperature gradients and contaminant levels in the occupied zone. Draught rates and ankle velocities were stable at low levels regardless of the case, showing negligible adjacent zones. Including lamellas as a design element for the full wall improved the performance by supplying air at a higher velocity, implying that a satisfying thermal comfort and indoor air quality could be obtained by limiting the temperature difference and maintaining a reasonable air flow rate.

By defining the other person as the contaminant source, it was seen that the concept works well regardless of where the source is placed in the full wall $\Delta T_3$ case. It is reasonable to presume that the same effect would occur for other inlet configurations, based on the acquired data and observed flow patterns.

The horizontal momentum forces from the half wall inlet configuration improved the contaminant removal effectiveness by forcing contaminated air up and out of the occupied zone. The vertical temperature gradient was noticeably lower than in all other cases, while still maintaining low draught rates and ankle velocities. This configuration has similarities to conventional wall mounted displacement diffusers, only with a larger inlet area. This reduces inlet velocities, avoiding discomfort from draught under the tested boundary conditions.

For temperature differences below $10\,^\circ\text{C}$ or flow rates above $0.09\,\text{m}^3/\text{s}$, the diffuse wall shows the widest range of the concepts presented in the design chart. For higher temperature differences, diffuse ceiling is the only concept with a wider range of acceptable air flow rates. This indicates that the diffuse wall ventilation concept performs well with a high flexibility.
Chapter 10

Conclusion

The diffuse wall ventilation concept performs well within the Norwegian indoor environmental requirements. When testing its limits of thermal comfort, vertical temperature difference was the restricting factor in all cases, as unacceptable draught rates or velocities never occurred.

Changes in temperature difference between the full wall inlet and room air influenced the contaminant removal effectiveness, where higher differences resulted in more contaminants in the occupied zone. Adding vertical lamellas to the inlet improved the performance, while the best contaminant removal effectiveness and vertical temperature gradient were achieved with a half wall inlet configuration.

By evaluating other ventilation concepts tested under similar conditions using the design chart, diffuse wall performs among the best in terms of thermal comfort. Contaminant removal effectiveness is not compared, as the chart does not include this.

Air supply from a diffuse wall has throughout this thesis proven to be a well functioning concept for ventilating an office space. The concept is capable of handling considerable cooling needs without causing thermal discomfort, and is flexible for adding visual elements such as wooden lamellas, making it versatile for many applications. To avoid contamination between occupants, a diffuse wall ventilation system should be designed with an inlet placed below the stratification height of contaminants, thus ensuring good indoor air quality.
Chapter 11

Further Work

As the findings in this thesis showed that a half wall inlet configuration had the best performance, it is recommended to do further studies on inlet heights below stratification layers of contamination. When initializing further work, the findings should be confirmed through a conventional CFD validation process using lab experiments.

A baseline study similar as made in this thesis is recommended for the half wall configuration, thus adding this to the design chart. Testing of different lamella designs could be included, as this adds to the versatility of the concept.

When experiments are made, a parameter study of the pressure chamber geometry and ventilation fabric properties is of interest. To reduce the need of ductwork, the exhaust location should be addressed. A placement in the top of the diffuse wall, similar to what is seen from the concept illustration in figure 1.1 is of particular interest.
Bibliography


ANSYS, 2020b. ANSYS. 37.2 Choosing the Spatial Discretization Scheme, 2020.

ANSYS, 2020c. ANSYS. 37.3 Choosing the Pressure-Velocity Coupling Method, 2020.


Iqbal and Chan, 2015. M. Iqbal and A. Chan. a Study of the Effect of Element Types on Flow and Turbulence Characteristics Around an Isolated High-Rise Building. 11th


Topp et al., 2002. Claus Topp, Peter Vilhelm Nielsen and Dan N. Sørensen. 
Application of Computer Simulated Persons in Indoor Environmental Modeling. 


Part IV

Appendix
Appendix A
Mesh Independence Study

A.1 Diffuse Ceiling

A.1.1 Velocity plots

Figure A.1: Vel., diffuse ceiling B2

Figure A.2: Vel., diffuse ceiling C1

Figure A.3: Vel., diffuse ceiling C3

Figure A.4: Vel., diffuse ceiling D1
A.1.2 Temperature plots

Figure A.5: Temp., diffuse ceiling B2

Figure A.6: Temp., diffuse ceiling C1

Figure A.7: Temp., diffuse ceiling C3

Figure A.8: Temp., diffuse ceiling D1
Appendix B

Porous Media Parameters

A porous media model is able to model a pressure drop through a material by treating the cells in a cell zone as fluid domains where viscous and inertial loss terms are added as a source term to the governing momentum equations. All definitions in this appendix are sourced from the Ansys Fluent User Guide [ANSYS, 2009].

The source term, $S_i$, defines a pressure loss per meter thickness of the porous zone, where the subscript $i$ indicates the direction, i.e. $x$, $y$ or $z$ as given by equation (B.1). This pressure loss is related to the source term values given in table 2.1, defining the fourth term in the general transport equation (2.1).

$$S_i = -\left(\frac{\mu}{\alpha} v_i + C_2 \frac{1}{2} \rho |v| v_i\right)$$ (B.1)

The velocity component in the porous media is given as superficial velocity, defined as the volume flow rate divided by the surface area of the zone.

The default setting in Fluent is to model turbulence across the porous media, which in many cases will be a misrepresentation of the reality, except from when the materials porosity is large. It is therefore possible to define the flow as laminar through the medium in the porous media settings tab, suppressing turbulence effects in the porous zone.

The software also gives an option of defining a fluid porosity between 0 and 1 for the media. The upper value represents an fully fluid zone with no influence by the solid material properties, while a value of zero means only solid and no fluid. In relation to defining this value, material properties for the solid, e.g. diffuse ceiling panels or ventilation fabric does also require definition.

To make use of a porous zone, values for both a viscous resistance coefficient, $1/\alpha$, and inertial resistance factor $C_2$ must be provided to the software. These can be obtained from experimental data through interpretation of a plot for pressure drop as function of velocity through the material. By creating a trendline in the form
where \( y = ax^2 + bx \) for this function, the two factors are found as \( a \) and \( b \) corresponding to equation (B.2).

\[
\Delta p = \left( \frac{\mu}{\alpha} v_i + C_2 \frac{1}{2} \rho |v_i| \Delta n \right) \Delta n \quad \text{(B.2)}
\]

| \( \Delta p \) | Pressure drop \([\text{Pa}]\) |
| \( \Delta n \) | Material thickness \([\text{m}]\) |

The derivation of the porous media parameters for the porous media formulations used in this thesis are described in the following.

### B.1 Diffuse Ceiling

In the diffuse ceiling model, the ceiling panels are defined as an uniform porous zone with a thickness of 0.035 m.

Pressure measurements across the entire diffuse ceiling were conducted by Zhang et al. [2014] for different flow rates and presented as a function of superficial velocity in figure B.1 along with a quadratic trendline.

![Graph](image.png)

**Figure B.1:** Pressure drop, diffuse ceiling

Using the trendline values, the viscous and inertial resistance factors are calculated using equation (B.2) to values of \( 1/\alpha = 1.14 \times 10^8 \text{m}^{-2} \) and \( C_2 = 33055 \text{m}^{-1} \) respectively.
B.2 Diffuse Wall

Prior to the lab facility shutdown, Fabric Air A/S provided manufacturer data for a suitable fabric intended for use in the experiments. Two fabrics were supposed to be tested, where one was intended for a full wall inlet with lamellas, and the other for half the wall. Both were made using a Combi 20 fabric type on the supply area, while areas with no airflow were defined with a non-permeable fabric [FabricAir A/S].

The Combi 20 fabric was rated for a superficial velocity of 128.6 m$^3$/m$^2$ at a 40 Pa pressure drop and with a thickness of 0.0004 m. The same fabric was defined for half wall inlet, only with additional laser cut perforations to create a higher porosity, rating it for 257.1 m$^3$/m$^2$ at 40 Pa.

The manufacturer stated that linear relations between pressure and velocity could be expected, as shown by the trendlines in figure B.2.

![Figure B.2: Pressure drop, diffuse wall](image)

The linearity reduces the functions to only provide values for the viscous resistance factor, calculated as $1/\alpha = 4.13 \times 10^7$ m$^{-2}$ for full wall and $2.57 \times 10^7$ m$^{-2}$ for half according to equation (B.2).
Appendix C
Boundary Conditions

This appendix shows the applied boundary conditions for the different models.

C.1 Diffuse Ceiling

Surface and air temperatures were obtained from the experiments and modified through a radiation model. A source term for the diffuse ceiling was found from the same model, simulating the radiation from ceiling panels, and a room source term was added to account for heat balance in the experiments.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet ($T$)</td>
<td>°C</td>
<td>8.950</td>
</tr>
<tr>
<td>Exhaust ($T$)</td>
<td>°C</td>
<td>26.65</td>
</tr>
<tr>
<td>Plenum ceiling ($T$)</td>
<td>°C</td>
<td>21.61</td>
</tr>
<tr>
<td>Plenum wall, N ($T$)</td>
<td>°C</td>
<td>20.94</td>
</tr>
<tr>
<td>Plenum wall, S ($T$)</td>
<td>°C</td>
<td>20.97</td>
</tr>
<tr>
<td>Plenum wall, W ($T$)</td>
<td>°C</td>
<td>21.55</td>
</tr>
<tr>
<td>Plenum wall, E ($T$)</td>
<td>°C</td>
<td>18.95</td>
</tr>
<tr>
<td>Room wall, N ($T$)</td>
<td>°C</td>
<td>26.46</td>
</tr>
<tr>
<td>Room wall, S ($T$)</td>
<td>°C</td>
<td>26.42</td>
</tr>
<tr>
<td>Room wall, W ($T$)</td>
<td>°C</td>
<td>26.42</td>
</tr>
<tr>
<td>Room wall, E ($T$)</td>
<td>°C</td>
<td>26.10</td>
</tr>
<tr>
<td>Floor ($T$)</td>
<td>°C</td>
<td>26.18</td>
</tr>
<tr>
<td>Window ($T$)</td>
<td>°C</td>
<td>25.34</td>
</tr>
<tr>
<td>Ventilation heat loss ($Q$)</td>
<td>W</td>
<td>-499.0</td>
</tr>
<tr>
<td>Person 1 ($Q$)</td>
<td>W</td>
<td>100.0</td>
</tr>
<tr>
<td>Person 2 ($Q$)</td>
<td>W</td>
<td>100.0</td>
</tr>
<tr>
<td>PC 1 ($Q$)</td>
<td>W</td>
<td>71.00</td>
</tr>
<tr>
<td>PC 2 ($Q$)</td>
<td>W</td>
<td>66.50</td>
</tr>
<tr>
<td>Lamp 1 ($Q$)</td>
<td>W</td>
<td>54.00</td>
</tr>
<tr>
<td>Lamp 2 ($Q$)</td>
<td>W</td>
<td>59.00</td>
</tr>
<tr>
<td>Inlet velocity</td>
<td>m/s</td>
<td>0.334</td>
</tr>
<tr>
<td>Inlet turbulence intensity</td>
<td>%</td>
<td>4.000</td>
</tr>
<tr>
<td>Room source term</td>
<td>W</td>
<td>40.59</td>
</tr>
<tr>
<td>DC source term</td>
<td>W</td>
<td>206.8</td>
</tr>
</tbody>
</table>

Table C.1: Diffuse ceiling boundary conditions
Appendix C. Boundary Conditions

C.2 Diffuse Wall

Adiabatic surfaces are defined for walls, the floor and the ceiling and only convective fractions of heat fluxes are defined. Boundary conditions are given as constant or case variable.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Walls ($Q_{\text{conv}}$)</td>
<td>[W]</td>
<td>0</td>
</tr>
<tr>
<td>Floor ($Q$)</td>
<td>[W]</td>
<td>0</td>
</tr>
<tr>
<td>Ceiling ($Q$)</td>
<td>[W]</td>
<td>0</td>
</tr>
<tr>
<td>Ventilation heat loss ($Q$)</td>
<td>[W]</td>
<td>-322.0</td>
</tr>
<tr>
<td>Person 1 ($Q_{\text{conv}}$)</td>
<td>[W]</td>
<td>50.00</td>
</tr>
<tr>
<td>Person 2 ($Q_{\text{conv}}$)</td>
<td>[W]</td>
<td>50.00</td>
</tr>
<tr>
<td>PC 1 ($Q_{\text{conv}}$)</td>
<td>[W]</td>
<td>67.20</td>
</tr>
<tr>
<td>PC 2 ($Q_{\text{conv}}$)</td>
<td>[W]</td>
<td>67.20</td>
</tr>
<tr>
<td>Lamp 1 ($Q_{\text{conv}}$)</td>
<td>[W]</td>
<td>43.80</td>
</tr>
<tr>
<td>Lamp 2 ($Q_{\text{conv}}$)</td>
<td>[W]</td>
<td>43.80</td>
</tr>
<tr>
<td>Inlet turbulence intensity</td>
<td>[%]</td>
<td>4.000</td>
</tr>
</tbody>
</table>

Table C.2: Diffuse wall constant boundary conditions

<table>
<thead>
<tr>
<th>ΔT</th>
<th>Airflow rate $[\text{m}^3/\text{s}]$</th>
<th>ACH $[1/\text{h}]$</th>
<th>Inlet temperature $[\text{°C}]$</th>
<th>Inlet velocity $[\text{m/s}]$</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0.0872</td>
<td>8.30</td>
<td>19.9</td>
<td>0.0872</td>
</tr>
<tr>
<td>6</td>
<td>0.0436</td>
<td>4.15</td>
<td>16.9</td>
<td>0.0436</td>
</tr>
<tr>
<td>10</td>
<td>0.0262</td>
<td>2.49</td>
<td>12.9</td>
<td>0.0262</td>
</tr>
</tbody>
</table>

Table C.3: Diffuse wall variable boundary conditions
Appendix D
Design Chart Development

This appendix shows a detailed description of how the design chart is developed using a selection of simulation cases. The manikin planes at 0.1 and 1.1 m shown in figure 6.1 are used to monitor velocity magnitudes, temperatures and turbulent kinetic energy for thermal comfort evaluation.

Velocity magnitudes are obtained directly, while temperature gradients are found as the difference between the planes. Draught rates are calculated using equation (2.3) and (2.4). The maximum values from the simulations are shown in table D.1, where values exceeding limits are marked as red.

<table>
<thead>
<tr>
<th>$\Delta T$ [°C]</th>
<th>q [m$^3$/s]</th>
<th>Heat load [W]</th>
<th>Max $v_{0.1}$ [m/s]</th>
<th>Location</th>
<th>Max $T_{\text{grad}}$ [°C]</th>
<th>Location</th>
<th>Max DR [%]</th>
<th>Location</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0.087</td>
<td>322</td>
<td>0.033 Person 2</td>
<td>1.15</td>
<td>Person 1</td>
<td>7.6</td>
<td>Person 2</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.140</td>
<td>517</td>
<td>0.029 Person 1</td>
<td>1.37</td>
<td>Person 1</td>
<td>9.6</td>
<td>Person 1</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.180</td>
<td>665</td>
<td>0.030 Person 1</td>
<td>1.48</td>
<td>Person 1</td>
<td>10.9</td>
<td>Person 1</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.044</td>
<td>322</td>
<td>0.028 Person 2</td>
<td>1.55</td>
<td>Person 1</td>
<td>7.2</td>
<td>Person 1</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.090</td>
<td>665</td>
<td>0.048 Person 2</td>
<td>2.06</td>
<td>Person 1</td>
<td>12.4</td>
<td>Person 2</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.140</td>
<td>1034</td>
<td>0.027 Person 2</td>
<td>2.32</td>
<td>Person 2</td>
<td>11.9</td>
<td>Person 2</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.180</td>
<td>1330</td>
<td>0.033 Person 2</td>
<td>2.59</td>
<td>Person 2</td>
<td>15.3</td>
<td>Person 1</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0.200</td>
<td>1724</td>
<td>0.036 Person 2</td>
<td>2.96</td>
<td>Person 1</td>
<td>17.9</td>
<td>Person 1</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0.300</td>
<td>2585</td>
<td>0.039 Person 1</td>
<td>3.58</td>
<td>Person 1</td>
<td>20.7</td>
<td>Person 1</td>
<td></td>
</tr>
<tr>
<td>7.5</td>
<td>8</td>
<td>0.080</td>
<td>788</td>
<td>0.040</td>
<td>Person 2</td>
<td>2.47</td>
<td>Person 1</td>
<td>11.8</td>
</tr>
<tr>
<td>8</td>
<td>0.100</td>
<td>985</td>
<td>0.049 Person 2</td>
<td>2.71</td>
<td>Person 1</td>
<td>15.4</td>
<td>Person 2</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>0.140</td>
<td>1379</td>
<td>0.035 Person 2</td>
<td>3.10</td>
<td>Person 2</td>
<td>15.8</td>
<td>Person 2</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>10</td>
<td>0.026</td>
<td>322</td>
<td>0.031</td>
<td>Person 1</td>
<td>1.74</td>
<td>Person 1</td>
<td>8.6</td>
</tr>
<tr>
<td>10</td>
<td>0.035</td>
<td>431</td>
<td>0.031 Person 1</td>
<td>2.03</td>
<td>Person 1</td>
<td>9.9</td>
<td>Person 1</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>0.050</td>
<td>616</td>
<td>0.035 Person 2</td>
<td>2.41</td>
<td>Person 1</td>
<td>11.7</td>
<td>Person 1</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>0.070</td>
<td>862</td>
<td>0.037 Person 2</td>
<td>2.86</td>
<td>Person 1</td>
<td>13.8</td>
<td>Person 1</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>0.100</td>
<td>1231</td>
<td>0.053 Person 2</td>
<td>3.24</td>
<td>Person 1</td>
<td>15.5</td>
<td>Person 1</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>0.050</td>
<td>739</td>
<td>0.035 Person 1</td>
<td>2.75</td>
<td>Person 1</td>
<td>13.7</td>
<td>Person 1</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>0.080</td>
<td>1182</td>
<td>0.042 Person 2</td>
<td>3.48</td>
<td>Person 1</td>
<td>16.8</td>
<td>Person 1</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>14</td>
<td>0.040</td>
<td>689</td>
<td>0.038</td>
<td>Person 1</td>
<td>2.77</td>
<td>Person 1</td>
<td>14.4</td>
</tr>
<tr>
<td>14</td>
<td>0.050</td>
<td>862</td>
<td>0.038 Person 1</td>
<td>3.08</td>
<td>Person 1</td>
<td>15.6</td>
<td>Person 1</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>0.080</td>
<td>1380</td>
<td>0.046 Person 2</td>
<td>3.87</td>
<td>Person 1</td>
<td>19.4</td>
<td>Person 1</td>
<td></td>
</tr>
</tbody>
</table>

Table D.1: Diffuse wall design chart cases
Table D.1 shows maximum ankle velocities well within the limit of 0.15 m/s, and that these are observed at person 2 in 64% of the cases. This proves that draught at ankle levels is not a concern within the evaluated operating range, and that other forces that inlet momentum might be governing for the air flow pattern in the room.

The draught rate at neck level was higher for person 1 in most cases, mainly cased by the turbulence intensity being larger at this point. Temperature gradients are similar at both occupants as a result of stratification, although a higher gradient is seen at person 1 due to the floor temperatures being slightly lower close to the inlet.

Linear trendlines are developed between cases with the same temperature difference or air flow rate. Due to the magnitudes of ankle velocities, only temperature gradients and draught rates are considered to give limiting cases. Plots are created for temperature gradient or draught rate as function of either flow rate or temperature difference. Linear equations, \( y = ax + b \), are then solved for the limit values, as shown in equation (D.1) and (D.2).

For a given \( q \):
\[
\Delta T = \frac{\text{[limit value]} - b}{a} \quad [\text{°C}] \tag{D.1}
\]

For a given \( \Delta T \):
\[
q = \frac{\text{[limit value]} - b}{a} \quad [\text{m}^3/\text{s}] \tag{D.2}
\]

When limiting combinations of flow rates and temperature differences are found, they are plotted into a design chart and a limit curve is drawn, as figure D.1 shows.
Appendix E
Additional Simulation Results

E.1 Case \( \Delta T_6 \)

E.1.1 Velocity

Figure E.1: Flow profile, \( \Delta T_6 \)

Figure E.2: Velocities, \( \Delta T_6 \)
Appendix E. Additional Simulation Results

Figure E.3: Velocity profile, ΔT6

E.1.2 Temperature

Figure E.4: Temperature, ΔT6
E.1.3 Contaminants

Figure E.5: Contaminants, ΔT6
Appendix E. Additional Simulation Results

E.2 Case $\Delta T_{10}$

E.2.1 Velocity

Figure E.6: Flow profile, $\Delta T_{10}$

Figure E.7: Velocities, $\Delta T_{10}$
E.2. Case $\Delta T_{10}$

![Figure E.8: Velocity profile, $\Delta T_{10}$](image)

### E.2.2 Temperature

![Figure E.9: Temperature, $\Delta T_{10}$](image)
E.2.3 Contaminants

Figure E.10: Contaminants, $\Delta T_{10}$