MASTER

CFD analysis of the mixing in an enclosure under time-periodic inlet conditions

Thysen, J.

Award date:
2015

Link to publication
CFD ANALYSIS OF THE MIXING IN AN ENCLOSURE UNDER TIME-PERIODIC INLET CONDITIONS

Jo-Hendrik Thysen
R-1862-A

June, 2015
CFD analysis of the mixing in an enclosure under time-periodic inlet conditions

Jo-Hendrik Thysen (839783)

Supervisors
prof.dr.ir. G.J.F. van Heijst
prof.dr.ir. B.J.E. Blocken

Daily supervisor
dr.ir. T.A.J. van Hooff

June 19, 2015
Acknowledgements

I would like to thank my supervisors prof.dr.ir. G.J.F. van Heijst and prof.dr.ir. B.J.E. Blocken to give me the opportunity to accomplish my graduation project at the Building Physics Section of the Department of Civil Engineering at Leuven University. I am also grateful to my daily supervisor dr.ir. T.A.J. van Hooff for assisting me during this project.
Abstract

Efficient removal of contaminants inside a room is important to obtain and maintain a healthy living environment. Therefore, properly mixing the indoor air with fresh supply air from outside is of major concern. This report presents the results of Computational Fluid Dynamics (CFD) simulations of the mixing ventilation in an enclosure that is supplied with a time-varying rate of fresh air and under isothermal conditions. The used numerical approach consists of solving the unsteady Reynolds-Averaged Navier-Stokes (RANS) equations in the vertical midplane of the enclosure. The purpose of the report is to examine whether a time-dependent supply rate leads to a better mixing compared to steady supply conditions.

Before investigating the effects of time-dependent forcing, the flow field inside the enclosure, associated with a constant (steady) rate of the supply air is examined. Several linear eddy viscosity turbulence models are used that provide closure of the RANS equations and a grid-sensitivity study is performed to ensure the numerical predictions obtained with the models to be grid-independent. The performance of the turbulence models in predicting the flow field and the turbulence level is assessed by comparing their predictions with experimental results found in literature. The renormalization group (RNG) $k - \epsilon$ turbulence model shows the best performance and is thus used to investigate the mixing behaviour under time-varying supply conditions. The supply rate varies periodically according to a sine function and the influence of two inlet parameters, i.e. the period and the amplitude, on the velocity field, the turbulent kinetic energy and the vorticity inside the enclosure is examined. This allows for an understanding of the contaminant distribution. Two approaches are considered: the contaminant distribution in the case that a source is present in the enclosure and the decay from an initial concentration level of contaminants. The results are reported and discussed in detail.
# Table of contents

- **Acknowledgements** iv
- **Abstract** v
- **Table of contents** viii

1 Introduction 1

2 Theoretical background 4

2.1 Discretisation of the flow domain ............................... 4
2.2 Governing equations .............................................. 6
2.3 Modelling of turbulence .......................................... 8
  2.3.1 Turbulence models ......................................... 10
  2.3.2 Limitations of the linear eddy viscosity models ............. 14

3 Two-dimensional CFD simulations under steady-state conditions 15

3.1 Experimental setup and measurement results ....................... 15
3.2 CFD simulations: computational grids, boundary conditions and solver settings .................................. 18
  3.2.1 Computational geometry and grid ........................... 18
  3.2.2 Boundary conditions ........................................ 18
  3.2.3 Solver settings ................................................ 19
3.3 Grid-sensitivity analysis ........................................... 21
  3.3.1 RNG $k - \varepsilon$ model ..................................... 21
  3.3.2 LRN $k - \varepsilon$ model ..................................... 24
  3.3.3 SST $k - \omega$ model ........................................ 27
  3.3.4 Comparison of the GCI for the different turbulence models . 30
3.4 Performance of different turbulence models in 2D simulations 30
3.4.1 Comparison of pathline plots 31
3.4.2 Validation and comparison of non-dimensional velocity profiles 32
3.4.3 Validation and comparison of non-dimensional turbulent kinetic energy profiles 35
3.4.4 Discussion 38
3.4.5 Comparison with other work 41
3.4.6 Conclusion 41

4 Two-dimensional simulations with time-dependent inlet conditions 43
4.1 Steady-state reference case 43
4.1.1 Boundary conditions and solver settings 43
4.1.2 Flow field characteristics 44
4.1.3 Concentration field of contaminants with the presence of a uniform constant source 46
4.1.4 Decay of the contamination from an initial concentration 47
4.2 Time-varying inlet velocity 53
4.2.1 Boundary conditions and solver settings 53
4.2.2 Flow field characteristics 55
4.2.3 Concentration field of contaminants with the presence of a uniform constant source 62
4.2.4 Decay of the contamination from an initial concentration 64
4.3 Discussion 80
4.3.1 Movement of the large recirculation cell 80
4.3.2 Influence of the period and amplitude on the time-averaged mean concentration field 82
4.3.3 Influence of the period and amplitude on the decay from an initial concentration 83
Nomenclature

Symbols

\( L \)  Length room (m)
\( H \)  Height room (m)
\( W \)  Width room (m)
\( h \)  Inlet height (m)
\( t \)  Outlet height (m)
\( V \)  Room volume (m\(^3\))
\( x, y, z \)  Cartesian coordinates (m)
\( u_x \)  X-component instantaneous velocity (m/s)
\( u_y \)  Y-component instantaneous velocity (m/s)
\( u_z \)  Z-component instantaneous velocity (m/s)
\( u \)  X-component mean velocity (m/s)
\( v \)  Y-component mean velocity (m/s)
\( w \)  Z-component mean velocity (m/s)
\( U \)  Velocity magnitude (m/s)
\( u' \)  Velocity fluctuations \( x \)-direction (m/s)
\( v' \)  Velocity fluctuations \( y \)-direction (m/s)
\( w' \)  Velocity fluctuations \( z \)-direction (m/s)
\( p \)  Pressure (kg/ms\(^2\))
\( Re \)  Reynolds number
\( y^* \)  Non-dimensional distance between wall and centre of wall-adjacent cells
\( D_h \)  Hydraulic diameter (m)
\( k \)  Turbulent kinetic energy (m\(^2\)/s\(^2\))
\( F_s \)  Safety factor
\( m \)  Formal order of accuracy
\( r \)  Refinement factor
\( E \)  Mean GCI

\( c \)  (Mass fraction of) mean concentration
\( c_i \)  (Mass fraction of) initial mean concentration
\( S_c \)  Source of contaminants (kg/m\(^3\)/s)
\( C \)  Courant number
\( Q \)  Inlet flow (m\(^3\)/s)
\( t \)  Time (s)
\( \Delta t \)  Time step (s)
\( \Delta u_0 \)  Velocity amplitude (s)
\( T \)  Period (s)
\( \rho \)  Density of fluid (kg/m\(^3\))
\( \Gamma \)  Diffusion coefficient (m\(^2\)/s)
\( \epsilon \)  Dissipation of turbulent kinetic energy (m\(^2\)/s\(^3\))
\( \omega \)  Specific dissipation rate (1/s)
\( \mu \)  Dynamic viscosity (kg/ms)
\( \nu \)  Kinematic viscosity (m\(^2\)/s)
\( \mu_t \)  Eddy (or turbulent) viscosity (kg/ms)
\( \omega_z \)  Z-component vorticity (s\(^{-1}\))
\( \tau_n \)  Nominal time constant (s)

Subscripts

\( 0 \)  Inlet
\( i \)  Grid number
\( RC \)  Reference case
1 Introduction

The mixing of fluids is a subject of extensive discussion and research in the fluid dynamics and engineering branch with a wide range of applications. Examples can be found on a small scale such as in microfluidic systems (e.g. devices for DNA analysis) where chaotic mixing is used in microchannels (Stroock et al., 2002), as well as on a larger scale in many industrial processes (Harnby et al., 1997). In this report, the focus is on the mixing of air inside a room with fresh air from outside in order to reduce the concentration of indoor contaminants that cause the air to be of poor quality. Many indoor pollutants exist, among others emission of building materials, microbial growth, tobacco smoking and people (Cao et al., 2014). Therefore properly mixed and diluted indoor air is a necessity in order to obtain and maintain a comfortable and healthy living environment, certainly in air-tight buildings.

Computational Fluid Dynamics is the method used to predict the flow field inside the room in which the indoor air is mixed by mixing ventilation. The supply air is injected near the ceiling in the form of a wall jet whose momentum ensures mixing of the fresh supply air with the room air (van Hooff et al., 2012a) after which the diluted air is extracted from the room by an outlet slot. Mixing ventilation is the most widely used method in ventilation systems (Awbi, 2003) and many publications are devoted to this type of ventilation. Cao et al. (2014) shows a summary of earlier studies in which experiments as well as numerical analyses are used to determine the airflow distribution and the indoor air quality in a room with mixing ventilation according to a variety of inlet and outlet configurations. The large majority of earlier ventilation studies are performed with a statistically constant supply airflow rate resulting in a stationary flow pattern inside the room, which depends on different factors like the location of the ventilation openings, the temperature and speed of the inlet jet, etc. Typically, such a stationary airflow distribution contains stagnant zones characterised by high concentrations of contaminants (Awbi, 2003), leading to a weak ventilation performance and hence locally poor air quality. Figure 1.1 shows the presence of such stagnation zones in the middle of a large recirculation cell observed in the experimental work of van Hooff et al. (2012b), among others indicated by the weak vorticity value along a line in the centre of the enclosure (Figure 1.1).

The current study differs from the aforementioned works since a time-dependent supply flow rate will be used instead of a stationary supply. It also distinguishes from situations in which variable-air-volume ventilation, demand-controlled ventilation or specific ventilation schedules are applied since the time scales involved in these cases are in general relatively large, e.g. one hour, and hence a statistically stationary flow pattern is still present in the room. Temporal forcing at relatively small time scales and the influence on the indoor airflow distribution is of current interest. The aim is to create better mixing of room air with fresh supply air by breaking down stagnation regions in order to improve the ventilation efficiency compared to the case without temporal forcing. Improving ventilation efficiency is also one of the main routes towards a reduction in energy consumption. This study can be put in a broader context than just the ventilation of a room. Also cabins of
airplanes, cars, ships, etc., form alternative enclosures for the application of mixing under time-varying conditions. Furthermore the flow in swimming pools and refrigerators and the mixing of fluids in stirring tanks (like paint and chemical species) are a few additional examples for which this study can be useful.

Former publications about the ventilation of a room under time-dependent conditions seem to be rather scarce, but the results found in literature look promising. The works of Kandzia et al. (2011) and Schmidt et al. (2013) both describe the effect on the airflow pattern in a room when a periodic variation (on the scale of minutes) of the inlet velocity is used. They observed that the circulation cells, which were present under steady conditions (i.e. with a constant inlet velocity), disappear in the mean velocity field of the unsteady case. Another study by Sattari et al. (2011) observed that pulsating inlet conditions result in a growth of the distribution and strength of vortices as well as a reduction of stagnation zones. Both phenomena improved the mixing throughout the room. Also in publications with a main focus on the mixing process itself, instead of being explicitly related to ventilation of buildings, a time-dependent perturbation of the flow field has proven its worth. Aref (1984) presented a simple two-dimensional (2D) model which provides an idealization of a stirred tank where the agitator is modelled by a point vortex. With the agitator held at a fixed position, the model device does not stir very efficiently. If, on the other hand, the agitator is moved in such a way that the (potential) flow is unsteady, chaotic motion is observed leading to efficient stirring. Similar work on chaotic mixing was performed by Solomon and Mezic (2003). They reported experimental and numerical studies of mixing in a laminar vortex flow (i.e. a horizontal vortex chain) that is weakly three-dimensional and weakly time-periodic. It was shown that for certain periods a completely uniform chaotic mixing was obtained with a full exchange of particles between neighbouring vortices.
The report is organised as follows. The equations of flow from which the predictions of the flow field and the contaminant distribution inside the enclosure can be obtained, are shown in Section 2. Also the modelling of turbulence using turbulence models will be described here. In Section 3 the results of 2D simulations of the flow field inside the enclosure when the supply rate of fresh air is constant are presented and compared to measurement data found in literature. A grid-sensitivity study is carried out and the performance of different turbulence models is assessed. The effect of a time-varying supply rate on the flow field and the distribution of contaminants is then examined in Section 4. The results of several 2D simulations are described and discussed. Finally, an overview of future work is presented in Section 5 and the report ends with Section 6 in which the most important conclusions are listed.
2 Theoretical background

Computational Fluid Dynamics (CFD) is a technique to model and analyse a system involving fluid flow by means of computational simulations. It allows for the prediction of the flow field by solving the equations of flow numerically. In order to achieve this, the flow domain should be first discretised in space using a well-defined computational grid as will be explained in Section 2.1. Then, the equations of flow are described in Section 2.2. Since the focus is on turbulent flows additional equations are needed in order to model turbulence and its effects on the mean flow. Section 2.3 is devoted to the modelling of turbulence.

2.1 Discretisation of the flow domain

The computational grid that discretises the flow domain should be constructed with care. It is required that the size of all grid cells changes slowly and smoothly away from the domain boundary as well as within the domain interior to avoid discontinuities in grid size which could have a destabilising effect on the numerical calculations (Tu et al., 2012). Special attention should be devoted to regions that are characterised by high gradients of the flow variables (e.g. the velocity) since high gradients imply that the flow variable changes value over very small distances. In order to capture these changes correctly a higher grid resolution is required in such areas as illustrated in Figure 2.1. This figure shows an example of a computational grid in a cross-section of an enclosure that is supplied with fresh air through the inlet channel, with the flow direction of the inlet air indicated by the arrow. Only part of the inlet channel and the enclosure is represented to illustrate clearly the higher grid resolution used in the shear layer between the inlet air and the room air (marked with I) to solve the high velocity gradients in this area accurately.

From Figure 2.1 it is observed that the near-wall regions are characterised by a higher grid resolution as well, to cover the high velocity gradients in the shear layer which are a result of the no-slip condition at the walls. The grid cells are sufficiently fine near the wall to cover the boundary layer including the viscous sublayer. This near-wall treatment is termed “Low-Reynolds-Number Modelling” (LRNM) since the flow close to the wall, which is dominated by viscous forces and hence low Reynolds numbers, is completely solved. Besides LRNM also “High-Reynolds-Number Modelling” (HRNM) exists in which the flow field in the viscous sublayer is not resolved but instead determined from semi-empirical functions, the so-called wall functions. In the case of HRNM the computational grid is extended towards the wall with a coarser grid resolution and the flow variables in the grid cells nearest to the walls are determined by the wall functions. Both near-wall treatments are represented schematically in Figure 2.2 with the right side of the figure illustrating the low-Reynolds-number modelling and the left side the high-Reynolds-number modelling technique.
Figure 2.1: Cross-section of an enclosure that is supplied with fresh air through an inlet channel. The left image shows half the enclosure whereas the right image represents a zoom of the area surrounded by the red box in the left image. It illustrates the use of a locally higher grid resolution in the area of the shear layer (I). The arrow indicates the flow direction of the supply air.

Figure 2.2: Sketch to illustrate the two near-wall modelling techniques. Left: high-Reynolds-number modelling. Right: low-Reynolds-number modelling. The wall is indicated with the number I, the viscous sublayer and the buffer layer lie in the region marked with the number II, and the turbulent area starts from the point indicated by the number III.

The correct use of both near-wall treatments depends on the distance between the wall and the (centre of) the row of cells nearest to the wall, which is expressed by the dimensionless wall distance \( y^* \) (Casey and Wintergerste 2000):

\[
y^* = y \left( \frac{k}{C} \right)^{1/2} \frac{\rho}{\mu}
\]

with \( \rho \) the density of the fluid, \( \mu \) the dynamic viscosity, \( y \) the normal distance from the wall to (the centre of) the near-wall cells, \( C \) a constant and \( k \) the turbulent kinetic energy (see later). To ensure the correct use of LRNM, the \( y^* \) value for every near-wall grid cell is required to be below 5, preferably around 1 or lower (ANSYS 2013). For HRNM, the near-wall cells should be positioned in the logarithmic layer of the boundary layer, around \( y^* \approx 30 - 300 \). The use of wall-functions reduces the near-wall grid resolution,
and thus the computational demand compared to LRNM. The drawback of using wall-functions, however, is that the calculated flow must be (more or less) consistent with the assumptions made in arriving at these functions. Important to note is that whatever modelling approach is adopted, a sufficient number of mesh points must be packed into a very narrow region adjacent to the walls in order to capture the variation in the flow variables (Casey and Wintergerste, 2000).

In the current study, the LRNM technique will be used for reasons explained later in the report. The flow equations are then solved (numerically) throughout the whole fluid domain, including the viscous sublayer, to determine the flow field in each cell centre of the computational grid.

2.2 Governing equations

The determination of the flow field inside the room and the associated distribution of contaminants are of current interest. A solution of the velocity field and the contaminant concentration as a function of position and time is obtained by performing computational simulations in which the governing equations are solved numerically. For the fluid under consideration, three assumptions are made in the derivation of all the equations:

1. The fluid is incompressible
2. The density of the fluid is constant \( \rho = \rho_0 \) and no temperature gradients are considered (isothermal)
3. The viscosity of the fluid is constant

The equations will be represented in the suffix notation in which the indices \( i \) and \( j \) are used. They take on the values 1, 2 and 3 corresponding to the \( x \)-direction, \( y \)-direction and \( z \)-direction respectively. In the case that an index is repeated (for example \( x_i x_i \)), a summation should be made over all the coordinate-directions (Einstein notation): \( x_i x_i = x_1 x_1 + x_2 x_2 + x_3 x_3 \).

The continuity of mass is expressed as

\[
\frac{\partial u_i}{\partial x_i} = 0 \tag{2.2}
\]

in which \( u_i \) represents the components of the (instantaneous) velocity field \( (u_1, u_2, u_3) \), hereafter denoted as \( (u, v, w) \). It states that the mass of fluid inside an infinitesimal control volume is conserved. The conservation of momentum is governed by the Navier-Stokes equation

\[
\rho \left[ \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} \right] = -\frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial x_j^2} \tag{2.3}
\]

with \( \rho \) the density of the fluid, \( p \) the pressure and \( \mu \) the dynamic viscosity. Equality 2.3 holds for the \( i \)th component of the velocity and it shows that velocity variations in time
and position are a result of the presence of pressure gradients and viscous stresses (diffusion of momentum). Due to the constant density, body forces corresponding to gravity can be neglected (Versteeg and Malalasekera 1995), (Nieuwstadt 2008), (Tu et al., 2012). The transport of species (mass) is given by a convection-diffusion equation according to

\[ \rho \left[ \frac{\partial c}{\partial t} + \frac{\partial c u_i}{\partial x_i} \right] = \rho \frac{\partial}{\partial x_i} \left( \Gamma \frac{\partial c}{\partial x_i} \right) + S_c \]  

(2.4)

in which \( c \) represents the concentration of the species (ANSYS, 2013). The equation expresses that species can be transferred through the flow domain due to convection on the one hand (second term on the left-hand side) and diffusion on the other (first term on the right-hand side) with \( \Gamma \) equal to the diffusion coefficient. In addition, the concentration level can change by the presence of a source which is incorporated in the transport equation through the source term \( S_c \).

Equations 2.2, 2.3 and 2.4 can be solved numerically in order to find the solution of the flow field and the concentration of species inside the room. This technique is called Direct Numerical Simulation (DNS) and it requires a sufficiently fine mesh to capture all the scales present in the flow. However, for fully-turbulent flows that are characterised by high Reynolds numbers (Re), as is the case in the current study, resolving the flow field down to the microscale is practically not realisable. The number of grid cells needed to resolve the full turbulence structure is of the order \( Re^{9/4} \) from which it is clear that high Reynolds number flows would require too much computer memory (Nieuwstadt, 2008), (Tu et al., 2012). In many engineering applications such a full description of the turbulence is not even necessary and a statistical solution of the flow field in terms of mean quantities is preferred (Casey and Wintergerste, 2000). The turbulence is in that case not solved, but modelled. The aforementioned equations, which describe the instantaneous flow field and concentration field, can be transformed to equations of the mean variables by applying the Reynolds decomposition in which an instantaneous flow variable is decomposed into a mean part and a deviation from the mean (fluctuation). The instantaneous velocity, pressure and concentration can then written as

\[ u_i = \bar{u}_i + u'_i \]
\[ p = \bar{p} + p' \]
\[ c = \bar{c} + c' \]  

(2.5)

with the bar and the prime-symbol indicating the mean and the fluctuating part respectively. \( \bar{u}_i \) is one of the components of the mean velocity field \((\bar{u}, \bar{v}, \bar{w})\) and \( u'_i \) equals a component of the velocity fluctuations \((u', v', w')\). By substituting these decomposed quantities into the Equations 2.2, 2.3 and 2.4 and then taking the time average (Versteeg and Malalasekera, 1995), the so-called unsteady Reynolds-Averaged Navier-Stokes (RANS)
equations are derived:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (2.6)$$

$$\rho \left[ \frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} \right] = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\mu}{\partial x_i^2} - \rho \frac{\partial u_i' u_j'}{\partial x_j} \quad (2.7)$$

$$\rho \left[ \frac{\partial \bar{c}}{\partial t} + \frac{\partial \bar{c} \bar{u}_i}{\partial x_i} \right] = \rho \frac{\partial}{\partial x_i} \left( \Gamma \frac{\partial \bar{c}}{\partial x_i} \right) - \rho \frac{\partial c_i'}{\partial x_i} + S_c \quad (2.8)$$

In the case that the turbulent flow is considered to be statistically stationary, i.e. when the mean velocity field is not a function of time, the partial time derivative at the left side of Equation 2.7 can be neglected and the set of equations is referred to as the steady RANS equations (Salim et al., 2011). Equation 2.7 is similar to the Navier-Stokes equation of the instantaneous velocity field (Equation 2.3), except for the third term on the right-hand side (\(-\rho \frac{\partial u_i' u_j'}{\partial x_j}\)) which includes the mean effect of the turbulent velocity fluctuations on the flow: apparently, the averaging procedure that was carried out to obtain the RANS equations is performed in such a manner that the rapid velocity fluctuations are eliminated and become expressed by their mean effects on the flow through this term, the so-called Reynolds or turbulent stresses (Casey and Wintergerste, 2000). The Reynolds stresses form in fact six new additional unknowns,

$$\bar{u}' v', \bar{v}' w', \bar{u}' v', \bar{v}' w' \text{ and } \bar{u}' w', \quad (2.9)$$

which can be interpreted as average momentum fluxes with $$\bar{u}_i' v_j'$$ representing the average flux of momentum in the $$j$$-direction along the $$i$$-direction, which exerts a stress on the mean flow (Kundu et al., 2012). The same holds for Equation 2.8 of the mean concentration field in which the mean effect of the turbulent fluctuations are represented by the turbulent diffusion fluxes (mass fluxes) $$-\rho \frac{\partial \bar{c} u_j'}{\partial x_j}$$ which comprises three additional unknowns:

$$\bar{c} u', \bar{c} v' \text{ and } \bar{c} w' \quad (2.10)$$

For all these new unknowns a mathematical expression should be derived in order to close the system of mean flow equations. This requires the construction of a mathematical model, known as a turbulence model, in which the turbulence related quantities (Equations 2.9 and 2.10) are expressed as a function of other flow variables. It is worth to mention that there is no generally valid universal model of turbulence and many different variants exist.

### 2.3 Modelling of turbulence

The simplest form of turbulence modelling that provides closure relations is based on the presumption that there exists an analogy between the action of viscous stresses and Reynolds stresses on the mean flow (Versteeg and Malalasekera, 1995). Newton’s law of viscosity states that the viscous stresses are proportional to the rate of strain of a fluid element with as proportionality constant the dynamic viscosity $$\mu$$. It was experimentally observed that turbulence decays unless there is shear in isothermal incompressible flows and
also turbulence was found to increase as the mean rate of deformation increases (Awbi, 2003). Hence, Boussinesq proposed in the so-called Boussinesq approximation that the Reynolds stresses are related to the mean rate of strain (i.e. mean velocity gradients) using the so-called turbulent or eddy viscosity $\mu_t$, leading to the following relations for the Reynolds stresses:

$$
- \rho u'u' = 2\mu_t \frac{\partial \bar{u}}{\partial x} - \frac{2}{3} \rho k
$$

$$
- \rho v'v' = 2\mu_t \frac{\partial \bar{v}}{\partial y} - \frac{2}{3} \rho k
$$

$$
- \rho w'w' = 2\mu_t \frac{\partial \bar{w}}{\partial z} - \frac{2}{3} \rho k
$$

$$
- \rho u'v' = \mu_t \left( \frac{\partial \bar{u}}{\partial y} + \frac{\partial \bar{v}}{\partial x} \right)
$$

$$
- \rho u'w' = \mu_t \left( \frac{\partial \bar{u}}{\partial z} + \frac{\partial \bar{w}}{\partial x} \right)
$$

$$
- \rho v'w' = \mu_t \left( \frac{\partial \bar{v}}{\partial z} + \frac{\partial \bar{w}}{\partial y} \right)
$$

with $k$ the turbulent kinetic energy which is based on the velocity fluctuations according to

$$
k = \frac{1}{2} \left[ (u')^2 + (v')^2 + (w')^2 \right] \quad (2.12)
$$

The turbulent transport of mass is modelled in a similar way by the use of the so-called Reynolds extended analogy (Casey and Wintergerste, 2000) in which the turbulent mass fluxes are linked to the mean concentration gradients as

$$
- \rho u'c' = \Gamma_t \frac{\partial \bar{c}}{\partial x}
$$

$$
- \rho v'c' = \Gamma_t \frac{\partial \bar{c}}{\partial y}
$$

$$
- \rho w'c' = \Gamma_t \frac{\partial \bar{c}}{\partial z}
$$

with $\Gamma_t$ equal to the turbulent diffusion coefficient (turbulent diffusivity). This coefficient is calculated from the eddy viscosity $\mu_t$, using a model constant called the turbulent Schmidt number $\sigma_t$ as $\Gamma_t = \mu_t / \sigma_t$. The value for $\sigma_t$ is taken to be 0.7 in many cases and is the default value in several codes. A more thorough analysis of the relations presented in the Equations 2.11 and 2.13 can be found in literature (Versteeg and Malalasekera, 1995) and (Nieuwstadt, 2008).

Turbulence models that express the Reynolds stresses using the eddy viscosity ($\mu_t$) as presented here are called linear eddy viscosity models. The turbulent viscosity is not a property of the fluid but a function of the state of turbulence and it should be determined. Many different eddy viscosity models exist, each of them specifying a number of specific equations from which the eddy viscosity can be found. In the next of the section the focus is on finding an expression for the eddy viscosity.
2.3.1 Turbulence models

In the current study, three different two-equation eddy viscosity models are used, two \( k - \epsilon \) models and one \( k - \omega \) model, in which two separate transport equations are solved to determine the eddy viscosity: the \( k - \epsilon \) models solve the equations of the turbulent kinetic energy \( k \) and the turbulence dissipation rate \( \epsilon \) whereas the \( k - \omega \) model determines \( k \) and the specific dissipation rate \( \omega \). The eddy viscosity \( \mu_t \) can then be defined as a function of these variables.

**Standard \( k - \epsilon \) model**

The standard \( k - \epsilon \) model by Launder and Spalding (1974) is considered the most popular version of two-equation models for practical engineering flow calculations. However, it is known to predict incorrect results under certain circumstances (Casey and Wintergerste, 2000). A known deficiency is the poor prediction in regions characterised by large strain rates as for example in stagnation regions occurring in impinging flows (Lakehal and Rodi, 1997) in which an excessive amount of turbulent kinetic energy is produced. Therefore, this model is not tested in this study and the interest goes out to other turbulence models. However, since the models that will be discussed below are in a sense related to standard \( k - \epsilon \) model, it is worth to comment briefly on its equations for \( \mu_t \), \( k \) and \( \epsilon \).

The eddy viscosity is computed by combining \( k \) and \( \epsilon \) as follows:

\[
\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \tag{2.14}
\]

in which \( C_\mu \) is a dimensionless model constant (see Table 2.1). The turbulent kinetic energy is obtained from the following transport equation:

\[
\rho \left[ \frac{\partial k}{\partial t} + \frac{\partial k \bar{u}_i}{\partial x_i} \right] = \frac{\partial}{\partial x_i} \left[ \left( \mu + \mu_t \sigma_k \right) \frac{\partial k}{\partial x_i} \right] + G_k - \rho \epsilon \tag{2.15}
\]

It is repeated that the assumptions presented in the beginning of Section 2.2 have been taken into account while deriving this equation. From the equation it can be observed that \( k \) can vary throughout the flow field as a result of transport, production and dissipation of \( k \), represented by the first term, second term and third term on the right-hand side of the equation respectively. The production term \( G_k \) is represented as

\[
G_k = \mu_t S^2 \quad \text{with} \quad S \equiv \sqrt{2S_{ij}S_{ij}} \tag{2.16}
\]

with \( S_{ij} \) the mean rate of strain tensor \( S_{ij} = \frac{1}{2} [\partial \bar{u}_i/\partial x_j + \partial \bar{u}_j/\partial x_i] \). Hence, shear effects in the mean flow form the source of turbulent kinetic energy. For the dissipation of turbulence the following equation holds:

\[
\rho \left[ \frac{\partial \epsilon}{\partial t} + \frac{\partial \epsilon \bar{u}_i}{\partial x_i} \right] = \frac{\partial}{\partial x_i} \left[ \left( \mu + \mu_t \sigma_\epsilon \right) \frac{\partial \epsilon}{\partial x_i} \right] + C_1 \frac{\epsilon}{k} G_k - C_2 \rho \frac{\epsilon^2}{k} \tag{2.17}
\]

The equation is similar to the \( k \)-equation with also a transport, source and dissipation term on its right-hand side. \( \sigma_k \) and \( \sigma_\epsilon \) are the turbulent Prandtl numbers for \( k \) and \( \epsilon \).
It are modelling constants, just as $C_{1\epsilon}$ and $C_{2\epsilon}$, with values presented in Table 2.1. For more information, the reader is referred to (Launder and Spalding, 1974), (ANSYS, 2013), (Versteeg and Malalasekera, 1995).

**Table 2.1:** Model constants in the standard $k - \epsilon$ model by Launder and Spalding (1974).

<table>
<thead>
<tr>
<th>$C_\mu$</th>
<th>$\sigma_k$</th>
<th>$\sigma_\epsilon$</th>
<th>$C_{1\epsilon}$</th>
<th>$C_{2\epsilon}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1</td>
<td>1.3</td>
<td>1.44</td>
<td>1.92</td>
</tr>
</tbody>
</table>

**Renormalization Group $k - \epsilon$ model**

The Renormalization Group (RNG) $k - \epsilon$ model by Yakhot et al. (1992) includes small modifications compared to the standard $k - \epsilon$ model, which are a result of the renormalization group theory on which the RNG model is based. These modifications should form an improvement in accuracy and reliability for a wider class of flows (ANSYS, 2013).

The eddy viscosity is determined as in Equation 2.14 for the standard $k - \epsilon$ model, except for the value of the dimensionless model constant $\alpha_k$ which has changed (see Table 2.2). The equations of the turbulent kinetic energy $k$ and its rate of dissipation $\epsilon$ also show similarities with those used in the standard model:

\[
\rho \left[ \frac{\partial k}{\partial t} + \frac{\partial k \bar{u}_i}{\partial x_i} \right] = \frac{\partial}{\partial x_i} \left( \alpha_k \mu_t \frac{\partial k}{\partial x_i} \right) + G_k - \rho \epsilon \tag{2.18}
\]

\[
\rho \left[ \frac{\partial \epsilon}{\partial t} + \frac{\partial \epsilon \bar{u}_i}{\partial x_i} \right] = \frac{\partial}{\partial x_i} \left( \alpha_\epsilon \mu_t \frac{\partial \epsilon}{\partial x_i} \right) + C_{1\epsilon} \frac{\epsilon}{k} G_k - C_{2\epsilon} \frac{\rho \epsilon^2}{k} - R_\epsilon \tag{2.19}
\]

with the production term of $k$, $G_k$, calculated as in Equation 2.16. Two important changes are made compared to the standard $k - \epsilon$ model: firstly, the model constants $\alpha_k$, $\alpha_\epsilon$, $C_{1\epsilon}$ and $C_{2\epsilon}$ (see Table 2.2) are analytically determined according to the RNG theory instead of derived empirically from benchmark experimental data, and secondly an additional term in the equation for $\epsilon$, the source term $R_\epsilon$, is included. It is this extra source term that makes the RNG $k - \epsilon$ model perform better in flows characterised by high strain rates compared to the standard $k - \epsilon$ model which predicts too high turbulence levels in high shear flows as discussed earlier (ANSYS, 2013).

**Table 2.2:** Model constants in the RNG $k - \epsilon$ model of Yakhot et al. (1992).

<table>
<thead>
<tr>
<th>$C_\mu$</th>
<th>$\alpha_k$</th>
<th>$\alpha_\epsilon$</th>
<th>$C_{1\epsilon}$</th>
<th>$C_{2\epsilon}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0845</td>
<td>1.393</td>
<td>1.393</td>
<td>1.42</td>
<td>1.68</td>
</tr>
</tbody>
</table>

The RNG $k - \epsilon$ model is a high- Reynolds-number model. Since in the current study the grid is extended with a high grid resolution through the viscous sublayer ($y^+ \approx 1$), applying the RNG $k - \epsilon$ model with wall functions would lead to erroneous predictions of the flow...
field. To circumvent this problem, the RNG model is extended with an enhanced near-wall treatment (EWT), based on a two-layer model in which the whole flow domain is subdivided into a viscosity-affected region and a fully-turbulent region determined by a wall-distance-based turbulent Reynolds number $Re_y$:

$$Re_y = \frac{\rho y \sqrt{k}}{\mu}$$  \hspace{1cm} (2.20)

with $y$ the wall-normal distance measured from (the centres of) the grid cells to the nearest wall. In the fully-turbulent region ($Re_y > Re^*_y$), the RNG $k - \epsilon$ model is employed whereas in the viscosity-affected near-wall region ($Re_y < Re^*_y$), the low Reynolds number one-equation model of Wolfshtein (1969) is applied. This one-equation model retains the equation of the turbulent kinetic energy as described in the RNG model (Equation 2.18), but uses different expressions for the eddy viscosity $\mu_t$ and the turbulence dissipation rate $\epsilon$:

$$\mu_{t,2layer} = \rho C_{\mu} l_{\mu} \sqrt{k}$$  \hspace{1cm} (2.21)

$$\epsilon = \frac{k^{3/2}}{l_{\epsilon}}$$  \hspace{1cm} (2.22)

with $l_{\mu}$ and $l_{\epsilon}$ length scales determined by certain expressions which will not be given here and $C_{\mu}$ the model constant as in Table 2.2. The two-layer model then ensures that these algebraic expressions of both the eddy viscosity and the dissipation in the viscosity-affected region are smoothly blended with the solutions obtained from solving the transport equations in the fully-turbulent region in order to have a continues definition of all the flow variables over the whole flow domain. For further details about the two-layer model the reader is referred to ANSYS (2013).

**Low-Reynolds-number $k - \epsilon$ model**

The aforementioned $k - \epsilon$ models, the standard and the RNG model, are both high-Reynolds-number models and their near-wall behaviour should be specified by either wall functions or the enhanced near-wall treatment. This is not necessary if a low-Reynolds-number (LRN) model is used since these are developed to solve the flow field down to the viscous sublayer. The LRN model of Chang et al. (1995) is considered in this study and is, among other LRN models (Patel et al., 1985), a modified version of the standard $k - \epsilon$ model. The equations for $\mu_t$, $k$ and $\epsilon$ take on the following form:

$$\mu_t = \rho C_{\mu} f_{\mu} \frac{k^2}{\epsilon}$$  \hspace{1cm} (2.23)

$$\rho \left[ \frac{\partial k}{\partial t} + \frac{\partial k u_i}{\partial x_i} \right] = \frac{\partial}{\partial x_i} \left[ \left( \mu + \mu_t \sigma_k \right) \frac{\partial k}{\partial x_i} \right] + G_k - \rho \epsilon$$  \hspace{1cm} (2.24)

$$\rho \left[ \frac{\partial \epsilon}{\partial t} + \frac{\partial \epsilon u_i}{\partial x_i} \right] = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_{\epsilon}} \right) \frac{\partial \epsilon}{\partial x_i} \right] + C_{1\epsilon} \frac{\epsilon}{k} G_k - C_{2\epsilon} f_{\epsilon} \rho \frac{\epsilon^2}{k}$$  \hspace{1cm} (2.25)

with the production term of $k$, $G_k$, calculated as in Equation 2.16. The similarity with the standard $k - \epsilon$ model is clearly visible and all the model constants ($C_{\mu}$, $C_{1\epsilon}$, $C_{2\epsilon}$, $\sigma_k$ and
σ, ϵ) maintained their values (Table 2.1). The distinction between the LRN model and the standard model is the presence of the parameters \( f_\mu \) and \( f_2 \), which are damping functions that resemble the effects of the wall on the turbulent fluctuations. They are determined from the following algebraic expressions:

\[
f_\mu = [1 - \exp(-A_\mu R_k)]^2 \left( 1 + \frac{B_\mu}{R_i^{5/4}} \right)
\]

\[
f_2 = [1 - \exp(-A_2 R_k)] \left[ 1 - B_2 \exp\left(-R_i^2\right) \right]
\]

with the variable \( R_k \) equal to the turbulent Reynolds number (Equation 2.20) and \( R_t \) described as \( Re_t = \rho k^2/\mu \epsilon \). The coefficients in both expressions are all constants: \( A_\mu = 0.0215, B_\mu = 31.66, A_2 = 0.0631 \) and \( B_2 = 0.01 \).

**Shear Stress Transport \( k - \omega \) model**

The Shear Stress Transport (SST) \( k - \omega \) model by [Menter 1994](#) is the third model that will be used in this study and it solves an equation for \( k \) and one for the specific turbulence dissipation \( \omega \) which can be thought of as the ratio of \( \epsilon \) to \( k \). The SST model forms an improvement on the standard \( k - \omega \) model of [Wilcox 1998](#). The disadvantage of this standard model is its sensitivity to the solutions of \( k \) and \( \epsilon \) in the freestream (i.e. the region a distance from the wall). To circumvent this problem, the SST \( k - \omega \) model in fact combines the standard \( k - \omega \) model with the standard \( k - \epsilon \) model. The latter is insensitive to the freestream values and hence is employed in the far field whereas it gradually blends into the standard \( k - \omega \) model which is applied in the vicinity of the wall. To achieve this, the \( k - \epsilon \) model must be converted into a \( k - \omega \) formulation. It is mentioned that the SST \( k - \omega \) model is a LRN model and hence no extra near-wall treatment should be specified. The equations of the SST \( k - \omega \) model are rather extensive in the sense that most parameters are captured by relatively long expressions and that the dependence on the blending functions is not straightforward. Therefore the equation of \( \mu_t, k \) and \( \omega \) will be represented without giving many details of the terms that appear in these equations.

\[
\mu_t = \frac{\rho k}{\omega} \frac{1}{\max \left[ \frac{1}{\alpha^*}, \frac{S_F}{\alpha \omega} \right]}
\]

\[
\rho \left[ \frac{\partial k}{\partial t} + \frac{\partial k \bar{u}_i}{\partial x_i} \right] = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k - Y_k
\]

\[
\rho \left[ \frac{\partial \omega}{\partial t} + \frac{\partial \omega \bar{u}_i}{\partial x_i} \right] = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_i} \right] + G_\omega - Y_\omega + D_\omega
\]

This definition of the eddy viscosity takes into account the transport of turbulent shear stress. \( S \) represents the strain rate magnitude as expressed in Equation 2.16. \( F_2 \) is one of the two blending functions used in the SST model and \( \alpha^* \) is computed from an algebraic expression. The two Prandtl numbers for \( k \) and \( \omega \), \( \sigma_k \) and \( \sigma_\omega \), are no longer constants (as it was the case in the standard \( k - \omega \) model) and they are determined from certain
expressions that include the blending functions. Also the production and dissipation term of \( \omega \), \( G_\omega \), and \( Y_\omega \), are evaluated with blending functions in their definitions. No blending functions are present in the expression of the dissipation of \( k \), \( Y_k \). The production term of \( k \), \( G_k \), is the same as in the \( k \)-equation of the standard \( k - \epsilon \) model (Equation 2.16). Finally, it is mentioned that the cross diffusion term \( D_\omega \) is a result of the transformation of the standard \( k - \epsilon \) model into equations based on \( k \) and \( \omega \). It also includes a blending function. The exact expressions for all the parameters involved can be found in ANSYS (2013).

In the next of the report, the bar symbol which indicates the mean values will be dropped to obtain a more compact notation. Hence, hereafter, the mean velocity components are represented as \((u,v,w)\) and for the mean concentration the notation \( c \) is used. The fluctuating quantities will still be marked with the prime-symbol.

### 2.3.2 Limitations of the linear eddy viscosity models

A large approximation in the linear eddy viscosity models is the assumption of turbulence isotropy which means that the Reynolds stresses are linearly related to the mean rate of strain by a (spatially dependent) isotropic scalar quantity, i.e. the eddy viscosity (Casey and Wintergerste 2000). This implies that the components \( u', v' \) and \( w' \) are treated equally as can be seen in the Equations 2.11. The isotropic condition is mainly problematic in flow areas characterised by high shear levels, for example in impinging jets or high swirling flows, in which the turbulence tends to be strongly anisotropic. Such flows can only be predicted in detail if for every turbulent stress component another level of the eddy viscosity is considered in order to allow for individual influences of the fluctuating quantities on the flow (Lauder and Spalding 1974), (Schälín and Nielsen 2004). However, the eddy viscosity approach has proved perfectly adequate in shear flows dominated by only one of the turbulent shear stresses such as in, among others, wall boundary layers and mixing layers (ANSYS 2013). It should be mentioned that several publications erroneously report that the eddy viscosity models assume turbulence isotropy in a sense that the normal Reynolds stresses are equal, i.e. \( -\rho u'u' = -\rho v'v' = -\rho w'w' \). This type of isotropy does only hold in the case that the shear stresses in the flow are zero \( (\partial u/\partial x = \partial v/\partial y = \partial w/\partial z = 0) \) and should not be confused with the aforementioned definition of isotropy. Besides the assumption of turbulence isotropy it should be kept in mind that the Boussinesq hypothesis is still a phenomenological description of reality, based on the molecular theory (Nieuwstadt 2008).

For flows in which anisotropic turbulence is important, other types of models such as second-order RANS models can be used. The Reynolds Stress Model (RSM) for example is fundamentally different from the eddy viscosity models since it does not assume the linear relations in Equation 2.11 but instead solves transport equations for the individual Reynolds stresses. It rejects the isotropic assumption and hence is better suited for flows affected by anisotropic effects. Disadvantages of the RSM are, however, the larger computational time and less straightforward numerical convergence by which this model is characterised.
3 Two-dimensional CFD simulations under steady-state conditions

Two-dimensional CFD simulations under steady-state inlet conditions are performed in a ventilated rectangular room which is based on the International Energy Agency (IEA) Annex 20 2D benchmark test described in the work of Nielsen (1990). The benchmark test is chosen since it contains experimental results of the flow field that can be used to validate the simulations performed in this report. The section starts with a description of the Annex 20 room and the different settings needed to perform the CFD simulations. It is ensured that the simulations do not depend on the computational grid by performing a thorough grid-sensitivity analysis. Since three different turbulence models are used for the numerical examination of the flow field, the grid-sensitivity test is performed for all three models. Then, the grid-independent results are compared to the experimental results of Nielsen (1990) to judge the performance of each turbulence model. The section ends with a discussion about the simulation results and a comparison to the work in other publications.

3.1 Experimental setup and measurement results

The IEA Annex 20 test case represents a box-like room subjected to forced mixing ventilation (Nielsen, 1990). The geometry is depicted in Figure 3.1 and its dimensions are: length $L = 9.0$ m, height $H = 3.0$ m and width $W = 3.0$ m. Air is supplied parallel to the ceiling via the inlet placed in the upper left of the room and is exhausted through the outlet located on the lower right. The inlet opening height $h$ measures 0.168 m and is rather large compared with practical opening dimensions. The height of the outlet $t$ is 0.48 m. Both inlet and outlet slots have a width equal to the width of the room. The flow is characterised by the slot Reynolds number which is defined as $Re_0 = u_0 h / \nu$, based on the inlet height, the inlet velocity $u_0$ and the kinematic viscosity of air $\nu$ at room temperature ($20$ °C). The flow problem is considered to be isothermal due to the absence of heat sources and temperature differences between the inflow air and the air in the room. All dimensions will be made dimensionless using the height of the room $H$.

Measurements were performed by Nielsen et al. (1978) and Restivo (1979) in a reduced-scale model with a real size height of $H = 89.3$ mm. The inlet slot was preceded by a smooth plane contraction of area ratio 2 along the $z$-direction, as depicted in Figure 3.1. They applied laser-Doppler anemometry to measure the $x$-component of the mean velocity $u$ and the root mean square (rms) value of the velocity fluctuations in the $x$-direction $\sqrt{u'^2}$. The dependency of these variables on the slot Reynolds number was also examined and the flow appeared to be fully turbulent for a slot Reynolds number of 5,000; the non-dimensional velocity and intensity profiles remained unchanged for higher Reynolds numbers. According to the similarity principle which guarantees that the measurement

\*More information on this ‘similarity principle’ can be found in Nielsen (1974) and Awbi and Nemri (1990).
results in the model completely represent the flow field in the (much larger) room, the slot Reynolds numbers of both the model and the room should be the same. However, the condition of perfectly matching the Reynolds numbers becomes less stringent for a flow in the fully-turbulent regime, since a small deviation in Reynolds number will not cause the flow to change behaviour. Therefore, all measurements presented in Nielsen (1990) were performed at the slot Reynolds number of 5,000, corresponding to an inlet velocity $u_0$ equal to 0.455 m/s.

Both the velocity and the velocity fluctuations were measured along four lines in the vertical midplane $z/W = 0.5$ (see Figure 3.1). The data is represented in Figure 3.2 in a non-dimensional form using the inlet velocity $u_0$. The velocity profiles downstream of the entrance slot (at $x = H$ and $x = 2H$) clearly show a wall jet behaviour near the ceiling ($y/H > 0.8$) with a sharp decrease in velocity at the wall due to the no-slip condition. A shear layer separates the jet from the rest of the fluid and the resulting instabilities cause a small increment in turbulent fluctuations since they convert kinetic energy of the mean flow into turbulent kinetic energy (Nieuwstadt, 2008). This is confirmed by the peak in the rms profile around the inflection points of the velocity curve on the one hand ($y/H \approx 0.9$) and the increase in turbulence level more downstream the inlet slot ($x = 2H$) while the peak in the mean velocity decreases on the other hand (Nielsen et al., 1978). Also the width (spread) of the wall jet increases due to shear as well as diffusion. According to Skovgaard and Nielsen (1991), the turbulence level at the inlet, for which a value of 4% was measured, has very little effect on the amount of turbulence in a distance from the entrance slot, meaning that the shear effects form the major source of turbulence in this flow.

**Figure 3.1:** Sketch of the IEA Annex 20 room and the definition of the coordinates and dimensions. The inlet geometry and the vertical midplane ($z/W = 0.5$) are indicated with a grey and red colour respectively. Also the lines along which the measurements are performed are visualised ($x = H$, $x = 2H$, $y = 0.028H$ and $y = 0.972H$).

\[\text{Specification}\]
\[
\begin{align*}
L/H &= 3.0 \\
W/H &= 0.056 \\
t/H &= 0.18 \\
W/H &= 1.0 \\
H_{room}/H_{model} &= 3.0 \text{ m} \\
H_{model} &= 0.0893 \text{ m}
\end{align*}
\]

\[\parallel\text{Image is a renewed version of the sketch depicted in Nielsen et al. (2010)}\parallel\]
As the jet approaches the wall opposite to the inlet \((x/H = 3)\), it approaches a region of high pressure and the corresponding adverse pressure gradient causes detachment of the wall jet, resulting in a recirculation pattern in the majority of the room, indicated by the negative velocity of the return flow at \(x = H\) and \(x = 2H\) below \(y/H = 0.5\). The recirculation is also clearly visible in the velocity profile measured nearby the floor along the line \(y = 0.028H\) from which it is seen that the flow speeds up and subsequently slows down again while approaching the left wall of the room. The flow speeds up due to the high pressure region in the impinging point of the jet at the floor and slows down again as a result of a higher pressure in the lower left corner. The velocity profiles along the horizontal lines \(y = 0.028H\) and \(y = 0.972H\) reveal two smaller circulation regions, one in the bottom left corner \((x/H = 0.25)\) and the other in the top right corner of the room \((x/H = 2.75)\) respectively. A sketch of the mean flow pattern inside the enclosure is represented in Figure 3.3. The large recirculation cell is shown together with the two corner vortices. The impinging point is marked with a dashed circle.

**Figure 3.2:** Measurement results presented in Nielsen (1990) of the non-dimensional mean \(x\)-velocity \(u/u_0\) (○) and the non-dimensional velocity fluctuations \(\sqrt{u'^2}/u_0\) (▲) along four lines in the vertical midplane \((z/W = 0.5)\) of the Annex 20 room: \(x = H\), \(x = 2H\), \(y = 0.028H\) and \(y = 0.972H\).
Figure 3.3: Sketch of the mean flow pattern inside the enclosure which consists of a large recirculation cell and two corner vortices. The impinging point of the wall jet near the floor is indicated with a dashed circle.

3.2 CFD simulations: computational grids, boundary conditions and solver settings

Two-dimensional simulations are performed and validated using the experimental data of Nielsen (1990) described above. This section comments on the basic settings to carry out the numerical calculations. First, the computational geometry and the corresponding grids to discretise this domain are presented. Then, the used boundary conditions for the inlet, outlet and walls are explained, followed by a description of the solver settings and the convergence criteria.

3.2.1 Computational geometry and grid

Two-dimensional simulations are performed to investigate the flow field in the Annex 20 room. The computational domain represents the midplane (with \( H = 3 \) m) without an inlet and outlet geometry as sketched in Figure 3.4. Gambit 2.4.6 has been used to construct five different non-uniform structured rectangular grids to discretise the domain, with a total number of cells ranging between 5,244 cells (grid1) and 83,536 cells (grid5), based on a grid refinement factor of \( \sqrt{2} \) in both directions. Figure 3.4 shows the five grids and they are constructed according to the requirements discussed in Section 2.1. It is ensured that the non-dimensional wall distance \( y^* \) (Equation 2.1) of the wall-adjacent cells is lower than 5 (and in most cases even lower than 1) in order to completely solve the flow field down to the wall (low-Reynolds-number modelling). In this study, pressure gradients and boundary layer separation play an important role in the determination of the flow inside the room. Therefore the boundary layer should be accurately solved all the way down to the viscous sublayer and hence LRNM is the preferred solution method in this study (van Hooff et al., 2013). Details of the five grids are summarised in Tables A.1 and A.2 in Appendix A.

3.2.2 Boundary conditions

The boundary conditions in the simulations were chosen to represent those of the experiments as close as possible. At the inlet (see Figure 3.4), a uniform constant velocity is
specified with components \((u_0; v_0) = (0.455; 0)\) m/s, according to the intended slot Reynolds number of 5,000 which is based on the inlet height \(h = 0.168\) m and the kinematic viscosity of air at a temperature of 20°C \((15.3 \cdot 10^{-6}\) m\(^2\)/s). From the inlet velocity and the turbulence intensity, which is 4%, the turbulent kinetic energy at the inlet slot \(k_0\) is determined according to \(k_0 = \frac{3}{2} (0.04 \cdot u_0)^2 \approx 4.97 \cdot 10^{-4}\) m\(^2\)/s\(^2\). The dissipation of \(k_0\) is then computed as \(\epsilon_0 = 10k_0^{3/2}/h \approx 6.59 \cdot 10^{-4}\) m\(^2\)/s\(^3\) (Nielsen, 1990). From both \(k_0\) and \(\epsilon_0\), the specific dissipation \(\omega_0\) is obtained, specified by the relation \(\omega_0 = \epsilon_0/(C_\mu k_0) \approx 14.74\) s\(^{-1}\), with \(C_\mu = 0.09\) a model constant (Rong et al., 2008). For the outlet slot, the experimental turbulence parameters are not given and therefore should be estimated. A pressure outlet was considered with a turbulence intensity of 4% and a hydraulic diameter \(D_h = 4Wt/(2(W + t))\) of approximately 0.828 m. The gauge pressure at the outlet is set to zero. The no-slip condition is applied at the walls of the computational domain which are considered to be smooth, stationary and impermeable.

### 3.2.3 Solver settings

The mean flow profile in the room is determined by numerically solving the discretised steady incompressible RANS equations, described in Section 2, using the computational code Fluent 15.0. Three different linear two-equation eddy-viscosity turbulence models are used to estimate the Reynolds stress terms. The RNG \(k - \epsilon\) model by Yakhot et al. (1992) has been extensively used in room air movement simulations (Awbi, 2003) and it
includes small modifications compared to the standard $k-\epsilon$ model which should serve an improvement in accuracy and reliability for a wider class of flows. Since the computational grids are designed to apply LRNM, the RNG $k-\epsilon$ model is used with the enhanced near-wall treatment option. The second turbulence model is the LRN $k-\epsilon$ model by Chang et al. (1995), hereafter denoted LRN ($k-\epsilon$) model. This model was designed to properly predict the flow pattern in a fully-developed turbulent recirculating flow in a sudden pipe expansion. A specification of the near-wall behaviour is not needed since its application is extended down to the viscous sublayer. The third model investigated in this study is the Shear Stress Transport (SST) $k-\omega$ model by Menter (1994). It is capable of accurately predicting adverse pressure-gradient boundary layer flows and flow separation (Bates et al., 2005) as well as accounting for non-equilibrium effects in boundary layers, e.g. close to separation points, due to a modified definition of the eddy viscosity of the standard $k-\omega$ model (Casey and Wintergerste, 2000) and (Tu et al., 2012). The SST $k-\omega$ model does not require extra near-wall specifications since it is designed to be used on LRNM grids. All turbulence models adopt the default model constants (see tables in Section 2.3.1).

The numerical calculations are performed with the pressure-based solver under steady time conditions and with double precision. Every simulation is carried out with the SIMPLEC algorithm for the pressure-velocity coupling together with the second-order scheme for the spatial discretisation of the pressure. The gradients in the governing equations are computed according to the least square cell based method and for the momentum, $k$, $\epsilon$ and $\omega$, the second-order upwind scheme is used. If no convergence could be obtained, the second-order discretisation scheme for the pressure was changed to the standard scheme and/or the pressure-velocity coupling was switched to the SIMPLE scheme or the coupled pressure-based solver. Sometimes, the under-relaxation factors were also lowered in order to reach convergence. The flow-field is always initialised based on the inlet conditions. Table A.3 in Appendix A summarises the solution settings for all simulations.

The convergence of the solution is accurately monitored and checked. First of all, the scaled residuals should show a decreasing behaviour as a function of the number of iterations and it was claimed that their values are well below $10^{-6}$. The second more important check of convergence is the monitoring of the (mean) velocity magnitude at certain points in low velocity regions of the flow domain, as depicted in Figure 3.5. Since a steady-state solution is to be expected, the velocity graphs should eventually become constant with increasing number of iterations. Convergence is assumed to be achieved from the moment that the velocity monitors showed a constant behaviour. At that point, the simulations also fulfilled the condition of the scaled residuals with values well below $10^{-8}$. Additional computations of the net mass flow are also carried out to check the overall conservation of mass since the ANSYS (2013) describes that the net mass flow with respect to the inlet mass flow should be lower than $10^{-1}$ % for a converged solution. The performed simulations showed values from 0 % to $10^{-3}$ % which is certainly low enough.
Figure 3.5: Sketch of the three points in the flow domain where the velocity magnitude is continuously monitored.

3.3 Grid-sensitivity analysis

A grid-sensitivity analysis is conducted using five grids (described in Section 3.2.1) for three turbulence models: RNG $k-\epsilon$ model, the LRN $k-\epsilon$ model and the SST $k-\omega$ model. For all models, the non-dimensional mean x-velocity $u/u_0$ and the non-dimensional mean turbulent kinetic energy $\sqrt{k}/1.1u_0$ are calculated on every grid along the lines $x = H$, $x = 2H$, $y = 0.028H$ and $y = 0.972H$, in analogy to the measurement results presented in Nielsen (1990).

3.3.1 RNG $k-\epsilon$ model

The velocity and turbulent kinetic energy results for the RNG $k-\epsilon$ model are depicted in Figures 3.6 and 3.7 respectively. It is observed that all profiles nearly coincide along every line in the flow field, independent of the grid. The profiles of $\sqrt{k}/1.1u_0$ seem to vary slightly more compared to the velocity profiles, but this is only due to a difference in scaling of the coordinate axes. A closer look at the plotted data reveals that grid3 (20,792 cells) and grid4 (41,600 cells) nearly produce the same outcome, whereas the results obtained on grid1 (5,244 cells) and grid2 (10,400 cells) tend to deviate slightly from these profiles in certain regions of the flow domain. Although the deviations are very small, grid2 yields more accurate results than grid1 in these regions, as illustrated in Figure 3.8 which shows a zoomed area of the turbulent kinetic energy profile at $y = 0.972H$. The very small deviations in the results together with the lesser computational time needed for the calculations on grid2 compared to grid3 and grid4, suggest that it is a good choice to assume the results produced on grid2 to be grid-independent.

To estimate the error of the results obtained on grid2, grid3 and grid4 in order to examine which grid can be considered to produce grid-independent results, the grid-convergence index (GCI), defined by Roache (1997), is calculated according to:

$$GCI = F_s \left| \frac{f_i - f_{i-1}}{1 - r^m} \right|$$

(3.1)

It provides an estimate of the error in a fine-grid solution, $f_i$, by comparing this solution to that of a coarse grid, $f_{i-1}$. Both $f_i$ and $f_{i-1}$ are either velocity or turbulent kinetic energy results in corresponding points along the same line in the flow domain. The grid
Figure 3.6: Comparison of the non-dimensional mean x-velocity ($u/u_0$) along four lines ($x = H$, $y = 2H$, $y = 0.028H$, $y = 0.972H$) obtained on grid1 to grid4 and predicted by the RNG $k - \epsilon$ model. The thin black line corresponds to $u/u_0 = 0$.

Figure 3.7: Comparison of the non-dimensional mean turbulent kinetic energy ($\sqrt{k}/1.1u_0$) along four lines ($x = H$, $y = 2H$, $y = 0.028H$, $y = 0.972H$) on grid1 to grid4 and predicted by the RNG $k - \epsilon$ model.
refinement factor $r$ is taken to be $\sqrt{2}$ since the grids are refined with this factor in each coordinate direction and the power $m$, which represents the formal order of accuracy of the algorithm, is assigned a value of 2 due to the second-order discretisation schemes used for the simulations. The expression encapsulated by the absolute value signs in Equation 3.1 is called the fine-grid Richardson estimator and it forms the best estimate of the error between the grid-solution and the unknown exact solution. To account for the uncertainty in the Richardson-based error estimates, a safety factor $F_s$ is incorporated. In this way, the GCI can be considered as the worst-case estimation of the solution error (Sørensen and Nielsen, 2003). The value of the safety factor in the GCI calculation of grid2 is 3 whereas it is chosen to be 1.25 in the calculations for grid3 and grid4. Roache (1997) recommends $F_s$ equal to 3 for normal use whereas the value of 1.25 is only adequate for scrupulously performed grid-sensitivity studies with three or more grid solutions. Although the calculation for the GCI of grid3 only involves two grids (grid2 and grid3), it is known in advance that the result obtained on the coarser grid (grid1) is less accurate and hence can be neglected. As a consequence, three or more grids are indirectly used in these computations, which supports the use of a safety factor with value 1.25. The same reasoning holds for the GCI calculations of grid4.

The calculated GCI values corresponding to the results obtained with the RNG $k-\epsilon$ model are very small along all lines in the flow domain and they decrease with an increasing number of grid cells, as expected. This is illustrated in Figure 3.9 which shows a histogram of the average GCI ($E$) calculated along the four lines for both the non-dimensional velocity results and the non-dimensional turbulent kinetic energy profiles obtained on grid2 ($E_2$), grid3 ($E_3$) and grid4 ($E_4$). It is observed that the GCIs of the turbulent kinetic energy results are less than these of the velocity profiles. For an unknown reason, the GCIs at $y = 0.028H$ do not meet the condition $E_2 > E_3 > E_4$. The maximum average values appear at $y = 0.972H$ for grid2 and they are only of the order $10^{-3}$. Such small GCIs make it difficult to visualise them, as illustrated in Figure 3.10a in which the local GCI
values are shown as an error band (red area) surrounding the velocity profile along the line $y = 0.972H$ obtained on grid2, for which the average GCI was maximum. Figure 3.10b is a zoom for a better visualisation. From this analysis it is clear that grid2 indeed produces grid-independent results for the RNG $k - \epsilon$ model with an almost negligible discretisation error.

Figure 3.9: Visualisation of the mean GCI calculated from the results obtained with the RNG $k - \epsilon$ model along the four lines in the fluid domain ($x = H$, $y = 2H$, $y = 0.028H$, $y = 0.972H$) on grid2 (E2), grid3 (E3) and grid4 (E4) for (a) the non-dimensional velocity and (b) the non-dimensional turbulent kinetic energy.

Figure 3.10: (a) Non-dimensional mean velocity profile obtained with the RNG $k - \epsilon$ model on grid2 at $y = 0.972H$, surrounded by red error bands corresponding to the local GCIs. (b) Zoom-in of the profile presented in (a) to better visualise the GCI error bands.

3.3.2 LRN $k - \epsilon$ model

The grid-sensitivity analyses for the LRN $k - \epsilon$ model is performed in the same way as for the RNG $k - \epsilon$ model and the simulations predict almost analagical results. The profiles of the non-dimensional velocity and the non-dimensional turbulent kinetic energy obtained
on the four grids are depicted in Figures 3.11 and 3.12 respectively. Again, the results corresponding to grid1 and grid2 are almost similar to these obtained on grid3 and grid4, apart from small deviations in which grid2 performs slightly better than grid1. Figure 3.13 shows a zoom-in on the results to illustrate this. As a consequence of the small variability between the profiles, the computed GCIs of grid2, grid3 and grid4 are small as well. From the histogram presented in Figure 3.14, it is shown that the GCI averaged along every line in the flow domain is only of the order $10^{-3}$ with in general the GCIs corresponding to the velocity profiles higher than these of the turbulent kinetic energy. Figure 3.15 shows the (local) GCIs of the velocity result obtained on grid2 along the line $y = 0.028H$, which is characterised by the largest average GCI, and even in this case a zoom is needed to see the error bands. This forms a visual proof of the small discretisation error on the velocity and turbulent kinetic energy computations obtained on grid2, therefore confirming that this grid yields grid-independent results for the LRN $k – \epsilon$ model.

**Figure 3.11:** Comparison of the non-dimensional mean x-velocity ($u/u_0$) along four lines ($x = H$, $y = 2H$, $y = 0.028H$, $y = 0.972H$) obtained on grid1 to grid4 and predicted by the LRN $k – \epsilon$ model. The thin black line corresponds to $u/u_0 = 0$. 
Figure 3.12: Comparison of the non-dimensional mean turbulent kinetic energy ($\sqrt{k}/u_0$) along four lines ($x = H$, $y = 2H$, $y = 0.028H$, $y = 0.972H$) on grid1 to grid4 and predicted by the LRN $k-\epsilon$ model.

Figure 3.13: Zoom-in on (a) the mean velocity profiles at $y = 0.972H$ and (b) the mean turbulent kinetic energy profiles at $x = 2H$, obtained with the LRN $k-\epsilon$ model. The region surrounded by the rectangular box indicated in the left images is enlarged and reproduced on the right-hand side.
3.3.3 SST $k - \omega$ model

Besides the RNG $k - \epsilon$ model and the LRN $k - \epsilon$ model, also for the SST $k - \omega$ turbulence model a grid-sensitivity analysis is carried out. The profiles are visualised in Figures 3.16 and 3.17. Compared to the former two models, the results obtained with this model show a much larger variability from grid to grid. To properly investigate the grid-independence, a fifth grid (grid5) with even more grid cells (83,536 cells) is used. The profiles obtained on grid1 deviate the most from the other profiles, certainly at $y = 0.028H$ and $y = 0.972H$ for the velocity results. Those obtained on grid2 to grid5 show more or less the same trend, except for the velocity along the line $x = H$ in the region $0 < y/H < 0.3$ where grid2 and grid3 predict for a large part a negative velocity in contrast to the computed positive
velocity obtained on the other (finer) grids. It is believed that grid-independent results are obtained on grid4 or grid5, since they produce nearly the same profiles along every line for both the velocity and turbulent kinetic energy.

A GCI analysis is presented in Figure 3.18 for both grid4 and grid5, again in the form of a histogram which plots the averaged GCI computed along the lines $x = H$, $x = 2H$, $y = 0.028H$ and $y = 0.972H$ ($E4$ for grid4 and $E5$ for grid5). For the velocity profiles, $E4$ is relatively small at $x = 2H$ and $y = 0.972H$ whereas it is quite large for the other lines. Figure 3.19 plots the velocity profiles (obtained on grid4 and grid5) at $x = H$, which are observed to have the largest averaged GCIs, together with error bands and it shows that the error is substantially reduced when grid5 is used instead of grid4. The same reasoning holds for the turbulent kinetic energy results. Therefore, since the profiles along all lines in the flow domain are taken into account, it is best to consider the results obtained on grid5 as grid-independent, despite the large computational time needed for the calculations performed on this grid. It is mainly due to the large computational time that no mesh with more grid cells than grid5 is constructed for a more thorough grid-sensitivity analysis. Yet, it should be mentioned that the errors on the results obtained with grid5 are acceptably low.

**Figure 3.16:** Comparison of the non-dimensional mean x-velocity ($u/u_0$) along four lines ($x = H$, $y = 2H$, $y = 0.028H$, $y = 0.972H$) obtained on grid1 to grid5 and predicted by the SST $k – \omega$ model. The thin black line corresponds to $u/u_0 = 0$. 

28
Figure 3.17: Comparison of the non-dimensional mean turbulent kinetic energy ($\sqrt{k/1.1u_0}$) along four lines ($x = H$, $y = 2H$, $y = 0.028H$, $y = 0.972H$) on grid1 to grid5 and predicted by the SST $k - \omega$ model.

Figure 3.18: Visualisation of the mean GCI calculated from the results obtained with the SST $k - \omega$ model along the four lines in the fluid domain ($x = H$, $y = 2H$, $y = 0.028H$, $y = 0.972H$) on grid4 (E4) and grid5 (E5) for (a) the non-dimensional velocity and (b) the non-dimensional turbulent kinetic energy.
3.3.4 Comparison of the GCI for the different turbulence models

The grid-sensitivity analysis learns that it is worth to do a grid-sensitivity test for each turbulence model separately. Both the visualisation of the profiles and the calculation of the grid-convergence index are useful to determine when results can be considered grid-independent. In this study, the RNG $k - \epsilon$ model and the LRN $k - \epsilon$ model produce grid-independent results for grid2, whereas grid5 is sufficiently fine for the SST $k - \omega$ turbulence model. A comparison of the mean GCIs corresponding to the models is given in Figure 3.20. It is observed that the GCI values of the RNG $k - \epsilon$ model are in general the lowest, followed by the GCIs of the SST $k - \omega$ model and then those of the LRN $k - \epsilon$ model. However, it should be mentioned that the SST model shows relatively large peaks at $x = H$ and $y = 0.028H$, much larger than the ones corresponding to the LRN model. Therefore the LRN model is preferred to the SST model.

3.4 Performance of different turbulence models in 2D simulations

The grid-independent results are examined more in depth via pathlines and contour plots to point out the differences predicted by the three turbulence models. All results are extensively described and the simulations are validated using the measurement data of Nielsen (1990) to check if they do represent the flow field in the Annex 20 room. Based on this validation the performance of each turbulence model is discussed in order to determine which model performs the best in this study. The findings are also compared to numerical results found in other works at the end.
3.4.1 Comparison of pathline plots

The predicted flow patterns according to the three turbulence models are visualised using pathlines as seen in Figure 3.21. The pathlines resemble the trajectories of particles released from nineteen equally spaced vertical lines. The colour corresponds to the magnitude of the non-dimensional mean x-velocity $u/u_0$ which is maximum and equal to one at the inlet slot. The two $k - \epsilon$ models (figures a and b) show an almost similar flow behaviour. Both models predict a large recirculation cell in the room which has a negative vorticity (clockwise rotation). The position of its centre varies a little due to the difference in the detachment points of the wall jet near the ceiling and the return flow near the floor; the jet and the return flow in the LRN turbulence model have a larger throw than in the RNG model. Another consequence of the difference in separation point is the smaller size of the corner vortices in the LRN model with respect to those predicted by the RNG model. For the sake of completeness it is noted that the simulations in fact predict a sequence of circulation cells in the corners, referred to as ‘Moffatt eddies’. More information on this can be found in Appendix B. The pathline pattern obtained with the SST $k - \omega$ model (figure c) is very divergent from the results obtained with the $k - \epsilon$ models due to the presence of a second vortex with an anticlockwise rotation near the left wall. This also causes the large recirculation zone to detach much earlier from the floor compared to the simulations performed with the RNG and LRN model. In addition, the wall jet separates earlier from the ceiling resulting in a larger vortex in the upper right corner of the room. The difference in the point of separation near the ceiling observed in the predictions of the three turbulence models is probably due to a difference in the adverse pressure gradient. Another possible explanation could be a difference in the turbulent kinetic energy close to the wall: boundary layers characterised by higher turbulence levels are more capable of overcoming the adverse pressure gradient.
Figure 3.21: Pathline plots for (a) the RNG $k-\epsilon$ model, (b) the LRN $k-\epsilon$ and (c) the SST $k-\omega$ turbulence model. The four lines $x = H$, $x = 2H$, $y = 0.028H$ and $y = 0.972H$ are indicated. The colour corresponds to the magnitude of the non-dimensional mean x-velocity $u/u_0$.

3.4.2 Validation and comparison of non-dimensional velocity profiles

Figure 3.22 shows the grid-independent profiles of the (non-dimensional) mean x-velocity obtained with the three different turbulence models along the four lines in the flow domain (see Section 3.3) together with the measured results of Nielsen (1990) (Section 3.1), to examine if the simulations accurately represent reality. The blue, green and bordeaux curves correspond to respectively the RNG $k-\epsilon$, the LRN $k-\epsilon$ and the SST $k-\omega$ turbulence model.

Along the vertical line $x = H$, the results predicted by the RNG and LRN $k-\epsilon$ model form a good approximation of the measured velocity data. The wall jet region near the ceiling ($y/H > 0.8$) is accurately represented by the RNG model whereas the LRN model overpredicts the peak velocity and underestimates slightly the spread of the wall jet, probably caused by an underestimation of diffusion and shear. Close to the floor ($y/H < 0.2$), it is the LRN model which performs best. The RNG turbulence model predicts a too negative x-velocity at this position. With small deviations, both models approximate well the shape of the measured data in the middle of the room ($0.2 < y/H < 0.8$). The same flow characteristics of these $k-\epsilon$ models are observed more downstream of the inlet slot, at $x = 2H$, with an acceptable prediction in the middle and again for the RNG model an overestimation of the negative velocity near the bottom. However, in the wall jet region
Figure 3.22: Comparison of the grid-independent non-dimensional mean x-velocities \( \frac{u}{u_0} \) with the measurement results of Nielsen (1990) (○) along four lines \( (x = H, y = 2H, y = 0.028H, y = 0.972H) \). The simulations are performed by the RNG \( k-\epsilon \) model (blue), the LRN \( k-\epsilon \) model (green) and the SST \( k-\omega \) model (bordeaux). The thin black line corresponds to \( \frac{u}{u_0} = 0 \).

The measured peak velocity is now better predicted by the LRN model instead of the RNG model. Furthermore, the wall jet profile in both simulations is too sharp and the peak appears closer to the ceiling in contrast to the more flattened measured profile. From the pathline plots in Figure 3.21, the shape of the velocity profiles along the vertical lines \( x = H \) and \( x = 2H \) is easy to understand: the positive velocity near the ceiling is a result of the wall jet, whereas the negative velocity near the bottom is introduced by the return flow of the large recirculation cell.

Unlike the good similarity between the measurement data and the results predicted by the \( k-\epsilon \) models, the SST \( k-\omega \) turbulence model shows much more deviations. Besides the overestimation of the peak velocity and the underprediction of the spread of the wall jet at \( x = H \), the simulation deteriorates strongly from the measurements in the middle and lower part of the room \( (y/H < 0.8) \). Immediately below the wall jet region the velocity profile takes on negative values and becomes positive again near the floor, in contrast to the measured data for which the x-velocity only changes sign ones from the positive wall.
jet region near the ceiling to a flow with negative x-velocity near the lower half of the room. This discrepancy is caused by the presence of the second circulation cell near the left wall predicted by the SST $k-\omega$ model, as already seen in the pathline plot of Figure 3.21. At $x = 2H$, the turbulence model performs much better with only a small overestimation of the velocities near the ceiling and the floor, an acceptable underestimation of the wall jet spread (comparable to the results of the $k-\epsilon$ models) and a good prediction of the measured data in the middle of the room.

Along the horizontal line $y = 0.028H$, positioned closely to the floor, three distinctive regions of the measured velocity profile should be present in the simulations as well: close to the outlet slot ($x/H = 3$) the non-dimensional mean x-velocity is positive, at the centre of the line ($0.5 < x/H < 2.75$) the large recirculation cell with a clockwise rotation causes the velocity to be negative (i.e. the return flow), and in the lower left corner of the room ($0 < x/H < 0.25$) a small corner vortex with an anticlockwise rotation is present indicated by the positive bump in the velocity profile. The RNG $k-\epsilon$ model predicts a similar shape as the measurement data but with a large underestimation in the velocity. Predicting the correct velocity magnitude near the floor of the room seems to be a deficiency of this model. The LRN $k-\epsilon$ model is more precise in predicting the return flow, but fails to predict the corner vortex. However, it is seen from the pathline plot in Figure 3.21 that the LRN model does predict the circulation cell in the lower left corner, but apparently with an almost negligible velocity along the line $y = 0.028H$. For the SST $k-\omega$ model, the results are again very different from the measurements due to the prediction of the second circulation cell.

The shape of the simulated velocity profiles close to the ceiling, i.e. along the line $y = 0.972H$, is in good agreement with the measurements for all turbulence models. In the right upper corner ($2.75 < x/H < 3$), a corner vortex with an anticlockwise rotation is detected, indicated by the negative value in the velocity profile. The position of this circulation cell is predicted correctly by the RNG $k-\epsilon$ model, however the velocity is too low. Also the simulation results of the SST $k-\omega$ model show the circulation cell, but its dimensions are modelled too large which causes a considerable deviation between simulation and measurement result at this position. For the LRN $k-\epsilon$ model, again no corner vortex is observed in the velocity profile since the predicted velocity is almost around zero. However, the pathline plot in Figure 3.21 shows that the circulation cell is certainly present. Close to the inlet slot, all three turbulence models assume a velocity $u/u_0$ equal to 1, according to the boundary condition $u = u_0 = 0.455 \text{ m/s}$ imposed at the inlet ($x/H = 0$). The difference with the measured value is a consequence of the simulations being performed without an inlet geometry. In the experiments of Nielsen (1990), the speed of 0.455 m/s was probably measured in front of the inlet geometry (at $x/H < 0$). Due to the no-slip condition at the walls, the flow is accelerated in the middle part of the geometry (i.e. along the line $y = 0.972H$) to maintain a constant flow rate with as result a value of $u/u_0$ larger than one at $x/H = 0$. In the first part of Appendix D, the influence of an inlet and outlet geometry on the flow field predicted by the RNG model is investigated. It is observed that indeed the velocity at the inlet region increases towards
the values from the measurements when an inlet channel is added. This also has a small effect in the nearby region \(0.5 < x/H < 2.75\), for which the simulations do approach the measurements more closely in this case.

To conclude which turbulence model approximates best the measured non-dimensional velocity profile, the deviation between the measured and the simulated data is determined via the root mean square error (RMSE),

\[
\text{RMSE} = \left( \frac{1}{n} \sum_{j=1}^{n} (\phi_{j}^{\text{meas}} - \phi_{j}^{\text{cal}})^2 \right)^{1/2},
\]

in which the root-mean-square operation is applied on the difference between the measured data \(\phi^{\text{meas}}\) and the simulation result \(\phi^{\text{cal}}\) at the same position \(j\), with \(\phi = u/u_0\). The squared difference is summed over all measurement points (\(n\) in total). The RMSE value is computed for every line (\(x = H, x = 2H, y = 0.028H\) and \(y = 0.972H\)) and the results are summarised in Table 3.1. As expected, the SST \(k - \omega\) model shows the largest deviations from the measured data. Therefore this model is considered inappropriate to predict the velocities in this study. For the \(k - \epsilon\) turbulence models, the RMSE values are much better and of comparable magnitude. The RNG model performs somewhat better at \(x = H\) and \(y = 0.972H\), whereas the LRN turbulence model has lower RMSE values at \(x = 2H\) and \(y = 0.028H\). Since the LRN model is not able to represent the corner vortices in its velocity profiles, the RNG turbulence model is assumed to give the best estimate of the velocity measurements.

**Table 3.1**: RMSE values of the non-dimensional mean x-velocity simulations. The lowest RMSE values are indicated in bold for every line.

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>(x = H)</th>
<th>(x = 2H)</th>
<th>(y = 0.028H)</th>
<th>(y = 0.972H)</th>
</tr>
</thead>
<tbody>
<tr>
<td>RNG (k - \epsilon)</td>
<td>0.0569</td>
<td>0.0495</td>
<td>0.0760</td>
<td>0.0745</td>
</tr>
<tr>
<td>LRN (k - \epsilon)</td>
<td>0.0738</td>
<td>0.0469</td>
<td>0.1159</td>
<td>0.0539</td>
</tr>
<tr>
<td>SST (k - \omega)</td>
<td>0.1616</td>
<td>0.0677</td>
<td>0.1591</td>
<td>0.1132</td>
</tr>
</tbody>
</table>

3.4.3 Validation and comparison of non-dimensional turbulent kinetic energy profiles

In addition to the profiles of \(u/u_0\), also the grid-independent non-dimensional turbulent kinetic energy profiles calculated with the three turbulence models are compared and validated with experimental data. Because the simulations only give information about the turbulent kinetic energy \(k\) as a result of the Boussinesq hypothesis (see Section 2.3), the measured quantity \(\sqrt{\overline{u'^2}}\) should be estimated in order to compare simulations and measurements. Nielsen (1990) states that the flow can be considered as a flow with two-dimensional wall jet profiles for which holds that \(\overline{v'^2} \sim 0.6\overline{u'^2}\) and \(\overline{w'^2} \sim 0.8\overline{u'^2}\) with \(v'\) and \(w'\) the velocity fluctuations in the \(y\)-direction and \(z\)-direction respectively. From the definition of \(k\),
given in Equation 2.12 it is easily determined that

\[ \sqrt{k} \sim 1.1 \sqrt{u'^2} \]  

(3.3)

and thus \( \sqrt{k}/1.1u_0 \sim \sqrt{u'^2}/u_0 \). The profiles along the four lines are depicted in Figure 3.23. It should be mentioned that these profiles seem to deviate more from the measurements than the velocity profiles in Figure 3.22, but this is only due to the difference in scaling of the coordinate axes. Besides the turbulence profiles along the four lines, also contour plots of \( \sqrt{k}/1.1u_0 \) are presented in Appendix C. These will be used to clarify some features observed in the profiles.

\[ \frac{\sqrt{u'^2}}{u_0} \]

\[ \frac{\sqrt{k}}{1.1u_0} \]

Figure 3.23: Comparison of the non-dimensional turbulent kinetic energy \( (\sqrt{k})/1.1u_0 \) with the measurement results of Nielsen (1990) \( (\triangleright) \) along four lines \( (x = H, y = 2H, y = 0.028H, y = 0.972H) \). The calculations are performed by the RNG \( k-\epsilon \) model (blue), the LRN \( k-\epsilon \) model (green) and the SST \( k-\omega \) model (bordeaux).

The experimental data along the line \( x = H \) shows a peak in the profile \( (y/H = 0.9) \) which indicates a high amount of velocity fluctuations in the wall jet region produced by instabilities due to shearing between the wall jet and the recirculation cell. The RNG and SST model accurately predict the peak value and the shape of the profile in this region whereas the LRN model slightly underpredicts the spread of the wall jet. For \( x/H < 0.8 \)
the SST model deteriorates from the measurements due to the presence of the second recirculation cell. The other two turbulence models perform better in this region, but local deviations with respect to the experimental data are still present. The small increase in turbulence around $y/H = 0.1$ is not observed in the simulation results of both models. Instead, the predictions show a slight increase in turbulence further away from the wall ($0.1 < y/H < 0.5$).

When the wall jet moves along the ceiling from $x = H$ to $x = 2H$, the shear instabilities develop further and the turbulence level is augmented, indicated by the higher peak value and the larger width of the measured data in the wall jet area ($y/H > 0.6$). All three turbulence models predict a similar shape of the turbulent kinetic energy profile comparable to the experimental results but the turbulence level is underestimated. Certainly in the peak region the models show a poor performance. The profile of the RNG model is too diffusive in this region and for the other two models the peak in turbulence level is more pronounced, but still underestimated. Also the position of the peak is slightly shifted with respect to the measurements for the LRN and SST model predictions. The computations of the $k-\epsilon$ models are good in the middle of the room ($0.2 < y/H < 0.6$) but deviate largely from the experiments near the bottom where the predictions are too low. The SST model performs the worst of the three models, with an underprediction of the turbulence level at every point.

The simulation results along the horizontal line $y = 0.028H$ confirm the observations that the turbulence level near the bottom is mainly underpredicted. Only at the outlet slot ($x/H = 3$), the RNG and LRN model overestimate the amount of velocity fluctuations. Both turbulence models predict a similar shape with an increase in turbulence towards the outlet slot in contrast to the experimental profile which is rather constant. The SST $k-\omega$ model performs poorly due to the presence of the second circulation cell which is characterised by a low level of turbulence (see Figure [C.1] in Appendix [C]). This causes the simulation result to drop abruptly ($x/H < 1.5$), leading to a different prediction compared to the other models. However, at the outlet it performs slightly better than the RNG and LRN model.

Along the line $y = 0.972H$, the simulations again underpredict the turbulence level. The RNG $k-\epsilon$ model is observed to approximate well the shape of the measurement profile. The peak near the right wall ($x/H = 2.75$), which is the consequence of the shear between the wall jet and the circulation cell in the right upper corner, is present but less pronounced. This also holds for the LRN model. The prediction of the SST model shows two peaks: the left one is also caused by shearing between the jet and the corner vortex whereas the right peak results from the shear between the corner vortex and another smaller vortex in this corner. The increase in turbulent kinetic energy represented by the second peak is also noticed in the profiles of RNG and LRN model, but much weaker. Compared to the measurements it is clear that the second peak predicted by the SST model is slightly exaggerated in magnitude. Another important difference in the simulated outcomes is noticed for $x/H < 1$. The results of $\sqrt{k}/1.1u_0$ by the LRN and SST models are much lower than the measurements, indicating a better performance of these models in this region.
below the measured value whereas the RNG model calculates a relatively higher turbulence level in this area. This is probably due to the fact that the turbulence values immediately below the inlet slot are much higher for the RNG model compared to the other two models as seen in Figure C.1a (Appendix C). This causes the turbulence level to be higher in the close vicinity of the inlet for the RNG $k-\epsilon$ model, resulting in a faster increase in the turbulence profile along the line $y = 0.972H$ compared to the LRN and SST model. Finally, at $x/H = 0$, it is noted that the three turbulence models predict a turbulence value half that of the measurements due to an apparently incorrect boundary condition. In the second part of Appendix D, simulations with different turbulent kinetic energy values at the inlet are presented and it is shown that the turbulence intensity of 4% at the inlet, in accordance with Nielsen (1990), should in fact be much higher to reach the value of 8% presented in the plot. The turbulence intensity of 4% is hence an incorrect boundary condition presented in the work of Nielsen (1990).

To analyse which model performs the best in predicting the correct turbulence level, the RMSE is computed along the four lines in the flow domain according to Equation 3.2 with $\phi = \sqrt{k}/1.1u_0$ (see Table 3.2). Apparently, the RNG model is the most accurate along the lines $x = H$, $y = 0.028H$ and $y = 0.972H$ whereas the LRN model predicts better the experimental data at $x = 2H$. For the SST model, the difference between the turbulence predictions and the measurements is largest, as expected. Therefore, the RNG $k-\epsilon$ model is preferred to the other turbulence models.

Table 3.2: RMSE values of the non-dimensional mean turbulent kinetic energy simulations. The lowest RMSE values are indicated in bold along every line.

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>$x = H$</th>
<th>$x = 2H$</th>
<th>$y = 0.028H$</th>
<th>$y = 0.972H$</th>
</tr>
</thead>
<tbody>
<tr>
<td>RNG $k-\epsilon$</td>
<td>0.0216</td>
<td>0.0296</td>
<td>0.0696</td>
<td>0.0513</td>
</tr>
<tr>
<td>LRN $k-\epsilon$</td>
<td>0.0259</td>
<td>0.0238</td>
<td>0.0715</td>
<td>0.0528</td>
</tr>
<tr>
<td>SST $k-\omega$</td>
<td>0.0587</td>
<td>0.0428</td>
<td>0.1018</td>
<td>0.0692</td>
</tr>
</tbody>
</table>

3.4.4 Discussion

The small differences between the measurement data of Nielsen (1990) and the two-dimensional simulation results can be attributed to many reasons:

- The boundary conditions in the measurement data of Nielsen (1990) are slightly unambiguous and not entirely complete. At the outlet, no boundary conditions are given and hence these are estimated in the simulations. Furthermore, Nielsen (1990) shows measurement results of the turbulence intensity at the inlet ($x/H = 0$) with a value between 4% and 5% whereas another graph in this work shows a turbulence intensity with a value of more than 8% at the same point, which is remarkable. The simulations were performed with the turbulence intensity of 4% but the investigations in Appendix D show that the measurement results would be better approached for higher turbulence intensities. Although the impact of a difference in boundary conditions between simulations and measurements is
rather limited to a region close to the inlet and outlet, it is a source of deviating predictions.

- The uncertainty of the measurement results in Nielsen (1990) is not reported.
- Adding an inlet and outlet channel creates simulation results which are in better agreement with the experimental data (see Appendix D). According to Voigt (2000) and Heschl et al. (2013) this will also enhance stability in the numerical calculations. Therefore, further investigations in this report will include both an inlet and outlet geometry.
- The turbulent kinetic energy is underestimated at almost every position, certainly near the top wall \((y = 0.972H)\) and the floor \((y = 0.028H)\). A reason of this underprediction could be the estimation of the velocity fluctuations from the turbulent kinetic energy according to Equation 3.3. Choosing another (higher) proportionality constant than \(1.1^{-1}\) will shift the profiles upward but this does not change their shape. An alternative to obtain better turbulence predictions could be the use of a Reynolds Stress Model (RSM), which does not rely on the Boussinesq approximation like the eddy-viscosity turbulence models used here and therefore is assumed to perform better in regions where turbulence tends to be strongly anisotropic. Chen (1996) performed simulations with three different RSMs but it seems that the improvements compared to the simulations presented in the above sections are very poor. The turbulence profiles are still underestimated, certainly near the bottom. Only the circulation cell in the lower left corner is better calculated. In this report, it was tried to perform simulations with the low-Re stress-omega Reynolds Stress Model. Since this model is mainly based on the \(k - \omega\) model, it predicts similar deviating results as the SST \(k - \omega\) model and therefore the results are not presented in this report. Only the wall jet was much better predicted due to the anisotropy by which it is characterised. Unfortunately, the results of other tested RSMs did not converge.
- The flow in the IEA Annex 20 room involves laminar-turbulent transition due to the highly turbulent jet region on the one hand and the presence of low velocity regions on the other (Pedersen and Meyer, 2002). This can hamper the numerical calculations performed with the turbulence models used in this report since these are developed for usage in fully-turbulent flows.
- PIV measurements of Pedersen and Meyer (2002) insinuate the presence of a recirculation zone at the lower left corner of the Annex 20 room which is, apparently, significantly larger than indicated by the simulation results of the RNG \(k - \epsilon\) model, as can be seen in Figure 3.24. This underestimation can lead to deviations in the rest of the flow field. However, a recirculation zone with those dimensions seems not to be present in the experiments of Nielsen (1990). As already mentioned, the experimental data along the line \(y = 0.028H\) shows a vortex in the lower left corner, but with significantly smaller dimensions than the cell in Pedersen and Meyer (2002). Furthermore, Rong et al. (2008) performed simulations with two turbulence models that did predict a circulation.
cell with a comparable size as in Pedersen and Meyer (2002). However, the velocity profiles seem to deviate from the Nielsen (1990) measurements in this area. Therefore, only the information in the work of Nielsen (1990) along the four lines is considered to judge the performance of the turbulence models in this report.

- Another plausible cause why the simulations show deviations from the experiments is the presence of three-dimensional effects. The measurement data is subjected to the influence of the no-slip condition at the side walls whereas the simulation results in two dimensions are not. This is also the reason why 2D calculations are compared with measurements in the midplane since there the effect of the no-slip condition should be minimal. Susin et al. (2009) performed three-dimensional (3D) simulations of the Annex 20 room and they showed via streamline plots in the horizontal plane $y = 0.972H$ near the top wall that the influence of the no-slip condition is restricted to a region close to the side walls. The work of Voigt (2005), however, points out the existence of three-dimensional structures near the bottom plane $y = 0.028H$, as depicted in Figure 3.25. Also Nielsen et al. (2010) showed simulation results in which three-dimensional flow structures appear, although these effects depend on the used turbulence model. From the Nielsen (1990) measurement results it is not clear where, if any, 3D structures appear in the flow field and their possible influence on the results is also not known. Further investigations could focus on measurements outside the midplane in order to estimate the influence of the side walls and to determine how good the 2D simulations represent the flow behaviour in the entire room. In this study it was tried to perform 3D simulations as well, but the calculations were considered unreliable due to problems with numerical convergence.

![Figure 3.24](image_url)  

**Figure 3.24:** Comparison of the streamline plots of the mean velocity field in the left part of the Annex 20 room ($x/H < 1$) obtained in (a) experiments and (b) simulations. The experiments are performed by Pedersen and Meyer (2002) and the simulation results are obtained by the RNG $k - \epsilon$ model.
3.4.5 Comparison with other work

The simulation results obtained with the three turbulence models and described in the above subsections are in agreement with the findings in other works. The computations of the RNG $k-\epsilon$ model for the velocity and turbulent kinetic energy along the four lines in the flow domain are confirmed by the work of Chen (1995). He performed similar 2D simulations for both quantities in the IEA Annex 20 room and recommended the use of the RNG model rather than other $k-\epsilon$ models for the predictions of indoor airflow due to its accuracy and stability. The work of Heschl et al. (2013) shows velocity profiles obtained with this model along the lines $x = H$ and $x = 2H$, similar to those presented in this study with an underprediction of the velocity near the bottom wall. Unfortunately, they did not show simulation results along the other lines and for the turbulent kinetic energy. A validation study for the LRN $k-\epsilon$ model by Chang et al. (1995) was not found in the literature but a comparison with simulations performed with another LRN model, the LRN $k-\epsilon$ model by Launder and Spalding (1974), shows only small differences (Skovgaard and Nielsen, 1991), (Voigt, 2000): the jet region of the velocity profile as well as the small vortex in the upper right corner are predicted slightly better for the Launder and Spalding (1974) model. Nevertheless, these are only small variations and both the velocity and turbulent kinetic energy profiles are in good agreement, indicating the correctness of the simulation results presented here. For the SST $k-\omega$ turbulence model many validation studies of the measurements in the IEA Annex 20 test case can be found (Voigt, 2000), (Rong et al., 2008), (Heschl et al., 2013). They all simulated profiles for the velocity and the turbulent kinetic energy similar to those calculated in this study. Only small variations near the inlet and outlet region are detected, which are due to a difference in boundary conditions at these slots. In addition, the streamline plots in Rong et al. (2008) visualise the same additional recirculation cell near the left wall. All works agree that the SST $k-\omega$ model has very poor performance in the Annex 20 room.

3.4.6 Conclusion

The 2D simulations performed in this report for the velocity and turbulent kinetic energy profiles (Figures 3.22 and 3.23) demonstrate that the RNG and LRN $k-\epsilon$ turbulence model predict similar results as the experimental data, in contrast to the SST $k-\omega$ model.
which is considered to be imprecise. These two models are in close agreement and it is difficult to determine which of the two is preferred to the other since both show good characteristics at distinct regions along the different lines. From the RMSE computations of the velocity profiles (Table 3.1) it is shown that the RNG model predicts slightly better the measurements along the lines \( x = H \) and \( y = 0.972H \) whereas the LRN model performs best at \( x = 2H \) and \( y = 0.028H \). To assess the performance at local regions in the flow domain, the RMSEs of specific parts along the lines are calculated. The RNG model appears to be superior in representing the wall jet but fails to predict the velocity near the floor compared to the LRN model. Both models show similar behaviour in the middle of the room (along \( x = H \) a \( x = 2H \)). Furthermore, the RNG model better represents the corner vortices whereas these are absent in the profiles of the LRN model. Also the predictions of the turbulent kinetic energy for the two models vary not much. Only near the inlet and outlet region a considerable better performance of the RNG model is noticed compared to the LRN model. The RMSEs in Table 3.2 are lowest for the RNG model along the majority of the lines (\( x = H, y = 0.028H \) and \( y = 0.972H \)), but it should be mentioned that the difference with these of the LRN model is almost negligible. From this analysis it can be concluded that two of the three tested turbulence models show acceptable results of the flow field. Despite the almost equal performance of the two \( k - \epsilon \) models, the RNG model is slightly preferred to the LRN model since it better predicts the corner vortices and does have a better overall agreement with the measurements based on the calculated RMSEs. In addition, the error on the predicted results according to the GCI (Section 3.3) is smaller. Therefore, the RNG \( k - \epsilon \) turbulence model will be used for the calculations in the rest of this study.
4 Two-dimensional simulations with time-dependent inlet conditions

The essence of this study is the examination of the mixing of air performance when a time-dependent inlet velocity is used. The section starts with a description of the steady-state flow field, according to a constant inlet velocity, that will form the reference case with which the computations under time-varying inlet conditions are compared. In this way it is possible to determine the effect of such time dependency on different flow characteristics like the velocity, turbulent kinetic energy and the capability of removing indoor contaminants (ventilation performance). All numerical calculations will be performed in the same computational domain as described earlier, i.e. the vertical midplane of the room, but with an inlet and outlet channel added to it like depicted in Figure D.1 (Appendix D). The 2D domain is discretised with grid2 (10,400 cells) and the RNG $k-\epsilon$ turbulence model is used since this model has proved to yield the best predictions. At the end of the section, the results are discussed.

4.1 Steady-state reference case

The steady-state flow field resulting from a constant inlet velocity serves as a reference case with which the results of the simulations under time-varying inlet conditions are compared. The boundary conditions and solver settings are described first after which the predictions of the flow patterns and the contaminant concentration fields in the room are presented.

4.1.1 Boundary conditions and solver settings

The velocity at the inlet $u_{0,RC}$ is 1 m/s, according to a slot Reynolds number of almost 11,000. The subscript $RC$ points out that the quantities are related to the steady-state Reference Case. The other boundary conditions at the inlet and the outlet are the same as described in Section 3.2.2 but the location where the boundary conditions are applied has changed due to the incorporation of the inlet and outlet channel. The steady-state simulations are performed with the SIMPLEC scheme for the pressure-velocity coupling and the least square cell based method for the spatial discretisation of the gradients. All other quantities are spatially discretised with second-order schemes. Convergence of the solution is obtained when the velocity has reached its steady-state value. The scaled residuals are constant and their values are in the range $[10^{-8}; 10^{-6}]$. To ensure the correct use of LRNM, the $y^*$-values of the cells closest to the walls should be lower than 5. This condition is certainly fulfilled: the minimum value is $7.82 \cdot 10^{-4}$, the maximum equals 4.39 and for the mean an outcome of 0.633 is calculated. It is worth to remind that the steady RANS equations are solved (Section 2.2). Hence all the presented results correspond to the mean flow (mean velocity field, mean turbulent kinetic energy, mean concentration field, etc.).
4.1.2 Flow field characteristics

The flow patterns, velocity field, turbulent kinetic energy and vorticity corresponding to a constant inlet velocity are described in this paragraph.

**Flow patterns and velocity field**

The predicted flow pattern is visualised in Figure 4.1a using pathlines which are the trajectories of particles released from nineteen equally spaced vertical lines. The flow pattern looks very similar to the steady-state flow field found previously in Figure 3.21a, despite the difference in the slot Reynolds number of both simulations. The agreement is a consequence of the fully-turbulent behaviour for $Re_0 \geq 5,000$, as already mentioned in Section 3.1. The flow pattern contains two smaller vortices in the corners and a large recirculation cell dominating most of the room. The corner vortices together with the area nearby the left wall are characterised by low velocity values as seen in Figure 4.1b, which plots the distribution of the (non-dimensional) mean velocity magnitude $U/u_0$ where $U$ equals $\sqrt{u^2 + v^2}$ with $u$ and $v$ the mean velocity in the $x$-direction and $y$-direction respectively. The large recirculation cell has a somewhat higher velocity at its boundaries (light blue coloured) with respect to its centre. The highest velocities are detected at the inlet slot and in the wall jet. Also in the neighbourhood of the outlet an increment of $U/u_0$ is noticed. In the upper part of the outlet channel (blue coloured area) the mean velocity in the $x$-direction is slightly negative (not shown in Figure 4.1), pointing out the presence of back flow due to a recirculation cell which is present in this part of the outlet channel.

![Figure 4.1](image)

**Figure 4.1:** (a) Pathline plot to visualise the flow patterns and (b) contours of the non-dimensional mean velocity magnitude $U/u_0,RC$ for the steady-state reference case. The vertical lines in (a) indicate the positions at which vorticity profiles are plotted, presented later in the report.
**Turbulent kinetic energy**

The amount of turbulence in the flow is represented in Figure 4.2 which shows contours of $\sqrt{k}/u_{0,RC}$ with $k$ the turbulent kinetic energy and $u_{0,RC}$ the inlet velocity. The turbulent kinetic energy is highest in the shear layer and in most parts of the large recirculation cell. Also the vicinity of the outlet channel is characterised by a higher turbulence level. The turbulent kinetic energy is produced by shear effects in the mean flow field (see production term of the $k$ in Equation 2.18). In the left half of the room as well as in the two corners, the values of $\sqrt{k}/u_{0,RC}$ are rather low.

![Image of Figure 4.2: Contour plot of the non-dimensional turbulent kinetic energy, represented as $\sqrt{k}/u_{0,RC}$, for the steady-state reference case.](image)

**Vorticity**

Eddies are necessary flow components to achieve a good mixing of the fresh supply air with room air. The mixing capability of the flow is represented by the vorticity, which is directly related to the mean velocity gradients (shear) and rotational effects. High levels of shear and rotation lead to a well mixed flow inside the room. Figure 4.3 shows profiles of the $z$-component of the vorticity $\omega_z = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}$ along several vertical lines that are located at the positions visualised in Figure 4.1a. The vorticity close to the left wall along the line $x = 0.083H$ is almost negligible, except near the bottom ($y/H < 0.2$) where the vorticity is slightly positive due to the presence of the corner vortex. In the shear layer a large peak in vorticity is observed (around $y/H = 0.95$) as a result of the high velocity gradients $\left(\frac{\partial u}{\partial y}\right)$ in this area. The lines at $x = H$ and $x = 2H$ cross the large recirculation cell which is characterised by a small negative vorticity value due to its clockwise rotation. The influence of this recirculation cell is also noticeable in the profile near the right wall at $x = 2.92H$ where the vorticity is slightly negative as well. This line also intersects the vortex in the right upper corner resulting in a positive vorticity towards the ceiling. It can be noticed that the vorticity is rather low throughout the whole room compared to the vorticity level observed in the shear layer and close to the floor and ceiling. Hence the mixing seems to be relatively weak in most of the room.
Figure 4.3: Profiles of the z-component of the vorticity $\omega_z$ along four vertical lines in the flow domain ($x = 0.083H$, $x = H$, $x = 2H$ and $x = 2.92H$) for the steady-state reference case. The range of $\omega_z$ is limited to $[-5; 5]$.

### 4.1.3 Concentration field of contaminants with the presence of a uniform constant source

Stagnant regions should be avoided in the room since the transport of fresh supply air into these regions is limited. The room air in these areas cannot be diluted sufficiently which can lead to local high contamination concentrations and therefore causing a sharp drop in the ventilation performance (Awbi, 2003). Also the amount of turbulent kinetic energy and the vorticity should be high enough to ensure that the supplied fresh air is able to mix as good as possible with room air. The capability of the ventilation system under steady-state inlet conditions to remove indoor contaminants is examined by placing a constant source of pollutants uniformly throughout the room and in the inlet and outlet channel. The source is activated after the simulation has fully converged to the steady-state flow field presented above. Its production rate is $S_c = 10^{-6}$ kg/m²s and the pollutants have exactly the same characteristics as the room air; there is no difference in density, molecular weight and temperature. The fresh supply air is free from any contaminants. The contaminant concentration in the room is represented by the non-dimensional expression $c S_c \rho \tau_n$ with $c$ the (dimensionless) mass fraction of contaminants, $\rho$ the density of air and $\tau_n$ the nominal time constant which is the shortest possible time it takes to replace the indoor air. $\tau_n$ is calculated from the supply rate $Q$ and the volume of the room $V$ (ASHRAE, 2009) according to

$$\tau_n = \frac{V}{Q} \quad (4.1)$$

which results in a value of approximately 161 s for the 2D room ventilated with the constant inlet velocity $u_{0, RC} = 1$ m/s.

Figure 4.4 shows the steady-state distribution of the contaminant concentration in the room. Close to the inlet channel the air is not polluted since the high velocities convect the contaminants away from this area. The whole jet region has a low level of contaminants.
due to the relatively high velocities and the presence of the shear layer which enhances the mixing of fresh air with room air. The jet propagates along the ceiling and the right wall towards the floor where it reattaches, as can be seen in the pathline plot of Figure 4.1. Part of the jet curves immediately towards the outlet which creates a short-circuit of the supply air to the extract point. This can be disadvantageous for the ventilation performance since part of the fresh air is extracted from the room before it could be mixed sufficiently with the polluted air (Awbi, 2003). The other part of the jet curves away from the outlet slot and propagates further along the floor which results in a relatively lower contaminant concentration in this area (green coloured). In the upper part of the outlet channel the concentration is observed to be low as a result of the back flow along which fresh air enters into this part of the channel. The effect of the back flow on the ventilation in the room is assumed to be small since the fresh air is not able to penetrate into the room due to the high velocities in the short-circuiting flow which is directed towards the outlet slot. Performing the simulations with a longer outlet channel would have avoided this. The concentration levels are the highest in the left part of the room, certainly in the lower left corner. The ventilation in this area is poor due to stagnation zones and a very weak amount of vorticity and turbulent kinetic energy. When moving through the room from left to right, the vorticity and the turbulent kinetic energy increase which is translated in a lower level of contaminants. An exception is the region in the right upper corner where a weak increase in the concentration is observed since it is characterised by a high pressure which creates a stagnation zone.

Figure 4.4: Steady-state non-dimensional concentration field $c_{Sc}$ when a constant source $Sc$ is distributed uniformly throughout the room and the room is ventilated with a constant air velocity at the inlet (steady-state reference case).

4.1.4 Decay of the contamination from an initial concentration

Besides the interest in the distribution of the contaminant concentration in the room when a constant source is present, the decay from an initial concentration of contaminants, represented by the mass fraction $c_i$, is studied. No source is present in the room. $c_i$ equals $10^{-6}$ and is uniformly distributed throughout the room, as depicted in Figure 4.5a. The initial concentration is set after the simulation has fully converged to the steady-state flow field. The contaminants have exactly the same characteristics as the room air and
the fresh supply air is free from any pollution. To show the decay of the contaminant field as a function of time, a time-dependent simulation instead of a steady calculation is performed for the reference case. More information about the boundary conditions and solver settings for time-dependent computations will be given later in Section 4.2.1 with the only difference that for the case considered here the inlet velocity remains constant over time.

**Figure 4.5:** Contour plots showing the decay of the concentration field \( c/c_i \) from an initial contaminant mass fraction \( c_i = 10^{-6} \) as a function of time \( t/\tau_n \) for the steady-state reference case. The concentration field is shown for (a) \( t/\tau_n = 0 \) (0 s), (b) \( t/\tau_n = 0.037 \) (6 s), (c) \( t/\tau_n = 0.093 \) (15 s), (d) \( t/\tau_n = 0.17 \) (27 s), (e) \( t/\tau_n = 0.31 \) (50 s), (f) \( t/\tau_n = 1 \) (161 s), (g) \( t/\tau_n = 2 \) (322 s).
The decay of the concentration field is studied over a time equal to two times the nominal time constant \( \tau_n \). Figure 4.5 depicts a series of subsequent contour plots of the contaminant concentration \( c \) over time with the concentration values given in a non-dimensional form using the initial concentration \( c_i \). Note that \( c \) in this case is made non-dimensional only with \( c_i \); the quantities \( \rho \) and \( \tau_n \) are not necessary as it is the case when a source is present which has the unit kg/m\(^2\)s whereas \( c_i \) represents the mass fraction and hence is a non-dimensional quantity. In each of the images fifteen monitoring points are shown for which the monitored data will be described later on in this section. The figure illustrates the spread of the fresh supplied air through the room as a function of time. Close to the inlet, the fresh air penetrates into the room along the ceiling with a high velocity and it pushes away the polluted air (image b). As the supplied air moves further, the surrounding air starts to dilute more and more (image c) due to turbulent mixing caused by the high turbulence level in the shear layer and in the recirculation cell (see Figure 4.2). The fresh air then turns downwards according to the flow pattern of the large recirculation cell and reaches the floor where part of the fresh air is lost directly via the outlet slot and the other part penetrates again into the room (image d). It is observed that the region close to the ceiling and the right wall, i.e. the core of the short-circuiting flow of the supplied air, has the lowest concentrations (green coloured). At this stage of the ventilation process, the air in the middle part of the room has also started to dilute due to the turbulent mixing. As the fresh air moves further along the bottom wall, the concentration levels in the left half of the room start to decrease as well, with the air in the middle region still containing a higher concentration compared to zone near the floor (image e). The area close to the left wall is reached last by the fresh air with in particular the zone under the inlet slot and certainly the region in the lower left corner. Image f shows the concentration pattern after a time equal to the nominal time constant and image g depicts the contaminant distribution for a time span twice as long. These images confirm that the region covered by the short-circuiting flow has the best decay from an initial concentration whereas the area nearby the left wall has the worst decay performance. The concentration levels increase while moving from right to left through the room.

In addition to the contour plots of the concentration field, the decay is also monitored continuously in fifteen uniformly distributed points. These points were already indicated in the images of Figure 4.5 and are sketched here again in Figure 4.6 with their coordinates, a number and a colour. The numbers help to link the monitoring point to the corresponding decay curves which are represented over a time span equal to two times the nominal time constant in Figure 4.7. The curves with a similar decay behaviour are indicated with the same colour:

- The bordeaux curves correspond to the points near the right wall (5, 10 and 15) and they are characterised by a very rapid decay followed by a decay rate which is much lower. The rapid drop in the concentration level is a consequence of the points laying in the core of the short-circuiting fresh air which is characterised by low concentration levels (see image d and e in Figure 4.5) and the drop occurs at the moment the fresh air crosses these points. Once the initial “cloud” of fresh air has passed by, the concentration decreases much slower determined by
Figure 4.6: Positions of the fifteen monitoring points that are uniformly distributed through the room. The $x$-coordinates are $x = 0.17H$, $x = 0.83H$, $x = 1.5H$, $x = 2.17H$ and $x = 2.83H$ whereas the $y$-coordinates are given by $y = 0.17H$, $y = 0.5H$ and $y = 0.83H$.

Figure 4.7: Curves of the concentration decay $c/c_i$ as a function of time $t/\tau_n$ in the fifteen monitoring points (which are visualised in Figure 4.6) for the steady-state reference case.

the transport of contaminants towards nearby regions via mixing and by the extraction of polluted air from the room through the outlet while new fresh air is still pumped in with a constant rate. The three bordeaux curves show the best decay performance from an initial concentration.

- The blue curves represent the points near the floor (2, 3 and 4) and their shape
looks very similar to these of the bordeaux curves. In the beginning the concentration decreases fast since the return flow of fresh air along the floor flushes away a large part of the pollution, but the drop in concentration level is less pronounced than in the bordeaux decay curves. The reason for this is that at the moment the fresh air reaches the floor it contains already higher amounts of contaminants due to more mixing with the indoor air than at the moment it crossed the points 5, 10 and 15 (bordeaux curves). It is also observed that the fast decay rate is somewhat lower, probably due to the slower propagation of the fresh air along these points with respect to the points 5, 10 and 15 as a result of the shearing with the walls and the surrounding fluid. The fast decay is again followed by a much slower decrease in concentration that is controlled by turbulent mixing, air extraction and the supply of fresh air.

- The monitoring points corresponding to the green curves are situated close to the left wall (1, 6 and 11). The difference with the aforementioned curves is that the transition from the rapid to the slower decay is less pronounced, i.e. the initial decay corresponding to the first passage of fresh air across these points is weaker. The cause is that the fresh air contains yet more contaminants and its velocity has again decreased when it arrives near the left wall. The green curves are characterised by the highest concentration values at any moment in time (see also image $f$ and $g$ in Figure 4.5).

- The points with positions in the middle of the room (7, 8 and 9) are represented by the orange decay curves. As already shown in Figure 4.5, these points are not directly exposed to the supplied air as is the case for the points positioned along the wall (bordeaux, blue and green curves). Instead, the fresh air reaches these points through mixing in the large recirculation cell. The decay curves start to decrease at approximately the same moment in time as the bordeaux curves. The fresh air has until then only moved along the ceiling (image $c$ in Figure 4.5), indicating the quick spread of the supplied air towards the middle of the room. The decay curve in point 9 decreases immediately whereas the points 7 and 8 show first a very slow decrease in the concentration before they take on the same (faster) decay rate as in point 9. This faster decrease in concentration starts from the moment the return flow along the floor has passed by. The decay performance of the orange curves shows to be in between the blue curves (points near the floor) and the green curves (points near the left wall), except for point 9 which eventually shows the same decay performance as the blue curves.

- The last three monitoring points (points 12, 13 and 14) are represented by the white coloured curves. These points are positioned close to the ceiling and they are discussed separately from the other points since they show an almost constant part in their decay curves where the concentration barely decreases, certainly in the points 12 and 13. The decay starts again with a drop in the concentration caused by the fresh air that passes along the ceiling. The drop is more pronounced going from point 12 to point 14 since the latter point is situated closer to the core of the short-circuit zone (image $c$ Figure 4.5).
the initial drop the decay curves stagnate until the return flow sets in and fresh air is more and more distributed throughout the room. This is illustrated in the images c, d and e of Figure 4.5 where the green core region above the points 12, 13 and 14 starts to diminish in size during the return flow, indicating the transport of this fresh air towards the nearby regions (i.e. in the direction of these points) via turbulent mixing. Since the indoor air in the middle part of the room is diluted from the right upper corner of the room towards the lower left corner, the stagnation region in the decay curves of points 12 and 13 is longer than in the point 14 which is situated closer to the right wall. A careful observation shows that the neighbouring point (15) of point 14 also has a weak stagnation part in its decay curve.

In summary, three distinct parts in the decay curves are observed: a sharp drop, a stagnation part and a slow decay behaviour. The initial drop is only present in the points close to the ceiling (white curves) and in the boundaries of the large recirculation cell (bottom: blue curves; right wall: bordeaux curves) since the fresh air crosses these points as first and hence did not mix much yet with contaminated indoor air. The drop in concentration increases for the points that are located closer to the core of short-circuit zone where the concentration levels are the lowest. The longer it takes for the fresh air to reach a certain point, the later the decay curve starts to decrease and the more gradual the decay will occur (i.e. a slow decay behaviour without a sharp initial drop in the concentration level). The latter is a consequence of the fresh air being mixed more with polluted air before the point has been reached (left wall: green curves). The orange curves are more or less similar to the green curves, but their decay starts earlier due to their location which is in the middle of the room. The stagnation parts in the decay curves which occur after the initial drop and before the slow decay part, are only clearly visible in the points close to the ceiling (white curves) and depend on the time it takes for the secondary flow (return flow) to reach these points.

The black arrow in Figure 4.7 points out the global evolution of the decay curves in the steady-state reference case while moving from the right side of the room to the left side, in which the smooth transition from the decay curve in point 15 to the decay curve in point 11 is clearly observed. Furthermore, the decay slows down in all the points and all the curves seem to converge over time. A calculation of the average decrease of the concentration in the room based on the concentration values in the fifteen points gives an indication about the ventilation performance under steady-state conditions. After a time equal to the nominal time constant, the contaminant concentration has decreased on average to 47% of the initial concentration. Ventilating the room for a time equal to two times the nominal time constant results in an average decrease in concentration to 21% of the initial value $c_i$. These values will be compared later with the computations performed in the time-dependent simulations.
4.2 Time-varying inlet velocity

The steady-state ventilation method leads to stagnation zones with very weak mixing causing an accumulation of contaminants in these regions which has an unfavourable effect on the ventilation performance. In this section it is investigated if time-dependent forcing can form an improvement to this.

4.2.1 Boundary conditions and solver settings

The velocity of the air supplied at the inlet is varied periodically according to the following sine function:

\[ u_0(t) = u_{0,RC} + \Delta u_0 \sin \left( \frac{2\pi}{T} t \right) \]

(4.2)

It fluctuates around the inlet velocity of the reference case \( u_{0,RC} = 1 \text{ m/s} \) to ensure that the same amount of fresh air is supplied to the room after every period \( T \) as in the reference case at the same moment in time. In this way, both ventilation methods can be compared and changes in the ventilation performance are not a result of a difference in the amount of supplied air. The magnitude of the inlet velocity changes in the range \( u_{0,RC} \pm \Delta u_0 \), with \( \Delta u_0 \) the amplitude of the sine function. Other boundary conditions at the inlet and outlet slot have the same values as described in Section 3.2.2.

The impact of both the period and the amplitude of the inlet velocity on the flow field and on the ventilation performance is investigated. Three different periods and two different amplitudes are tested. Table 4.1 summarises the used values for these parameters in a non-dimensional form. The period is represented with respect to the nominal time constant \( \tau_n \) and to make the amplitude non-dimensional the steady-state velocity is used. The simulations SIM1, SIM2 and SIM3 are performed with a different period (\( T_1, T_2 \) and \( T_3 \) respectively) but with the amplitude kept constant. The period varies on minute scale and is the smallest for SIM1 (\( T_1/\tau_n = 0.2 \) with \( T_1 \approx 32 \text{ s} \)). For SIM2 the period is somewhat longer (\( T_2/\tau_n = 0.5 \) with \( T_2 \approx 80 \text{ s} \)) and SIM3 has the longest period equal to the nominal time constant (\( T_3/\tau_n = 1 \) with \( T_3 \approx 161 \text{ s} \)). These three simulations have a velocity amplitude given by \( \Delta u_0/u_{0,RC} = 0.5 \). In the fourth simulation, SIM4, the amplitude is doubled to a value \( \Delta u_0/u_{0,RC} = 1 \) and the period is equal to the one used for SIM1 (\( T_4 = T_1 \)). The variation in inlet velocity changes the inlet slot Reynolds number \( Re_0 \). For the first three simulations (SIM1 to SIM3) \( Re_0 \) takes on the values [5,490;16,470] whereas for SIM4, which has a larger velocity amplitude, the interval is equal to [0;21,960]. All time-dependent simulations are started from the steady-state reference case presented in Section 4.1. The inlet velocity at the start equals \( u_{0,RC} \) and then increases to a maximum determined by the sine function in Equation 4.2. A full period of the inlet velocity is obtained when the velocity is again increasing and eventually reaches the value \( u_{0,RC} \). This is illustrated in Figure 4.8 which depicts the inlet velocity over a time equal to \( \tau_n \) for all the simulations. Also the constant inlet velocity in the reference case (RC) is plotted (red).

The spatial discretisation of the gradients in the governing equations is computed according
Table 4.1: Simulations performed under time-varying inlet conditions. In SIM1 to SIM3 the period changes while the amplitude is kept constant. SIM4 has a different amplitude than the other simulations and its period is the same as in SIM1.

<table>
<thead>
<tr>
<th>Inlet parameter</th>
<th>SIM1</th>
<th>SIM2</th>
<th>SIM3</th>
<th>SIM4</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T/\tau_n$</td>
<td>0.2</td>
<td>0.5</td>
<td>1</td>
<td>0.2</td>
</tr>
<tr>
<td>$\Delta u_0/u_{0,RC}$</td>
<td>0.5</td>
<td>0.5</td>
<td>0.5</td>
<td>1</td>
</tr>
</tbody>
</table>

Figure 4.8: Velocity at the inlet $u_0/u_{0,RC}$ as a function of time $t/\tau_n$ for the steady-state reference case and the four transient simulations over a time span equal to the nominal time constant $\tau_n$.

to the least square cell based method and the PISO algorithm is the recommended scheme for the pressure-velocity coupling in transient calculations (ANSYS, 2013). For the spatial discretisation of the other variables the second-order upwind scheme is used. Transient calculations require also a specification of the time discretisation, for which the bounded second-order implicit scheme is selected due to its stability and accuracy (ANSYS, 2013). The time-step $\Delta t$ is based on the relation

$$\Delta t = C \frac{\Delta x_{cell}}{U_{char}} \quad (4.3)$$

with $C$ the Courant number, $\Delta x_{cell}$ a typical cell size and $U_{char}$ a characteristic flow velocity. In order to solve the flow correctly even in the smallest cell of the grid, the time step should not be taken too large. Therefore, the typical cell size is taken to be the length of the smallest cell, which is approximately 23 mm, and the chosen characteristic flow velocity is 2 m/s according to the maximum velocity at the inlet. Since for stability reasons it is recommended that $C$ should be smaller than 1 (ANSYS, 2013), a time step of 0.01 s is used for all simulations. Twelve iterations per time step are performed to ensure that the solution is converged within a time step. Convergence is obtained when the monitored velocity and concentration values reach a constant value at the end of every time step. Furthermore, the scaled residuals are also checked to decrease sufficiently during a time.
step and to ensure the correct use of LRNM, the \( y^* \)-values of the near-wall cells are (on average) below 1. For the time-dependent simulations the unsteady RANS equations are solved (Section 2.2), implying that the mean flow field at every time step is calculated. The mean flow field changes in time determined by the period of the inlet velocity and these changes are solved due to the size of the time step which is much smaller than this period.

4.2.2 Flow field characteristics

The flow patterns, velocity field, turbulent kinetic energy and vorticity corresponding to a periodic change of the inlet velocity are described in this paragraph and compared to the results of the steady-state reference case.

Flow patterns

On the left-hand side of Figure 4.9 a sequence of pathline plots is shown to visualise the flow pattern of SIM1 over one period in time, starting from the flow field corresponding to the minimum velocity at the inlet. The colour of the pathlines represents the value of the non-dimensional mean velocity magnitude \( \frac{U}{u_{0,RC}} \) and the times of the plots are represented in a non-dimensional form \( \frac{t}{T1} \). These times are marked on the inlet velocity function with a blue circle (top image) and the time scale is chosen so that the first velocity minimum coincides with \( \frac{t}{T1} = 0 \). It is worth to mention that this simulation has run first for a couple of periods of the inlet velocity in order to avoid the “start-up effects” in the beginning of the simulation. It is observed that the centre of the large recirculation cell appears to move in a clockwise direction according to an elliptic trajectory which is repeated during every periodic cycle of the inlet velocity. The trajectory is sketched by the blue solid line in Figure 4.10 in which the position of the cell centre in the steady-state reference case is indicated by the red dot. The same movement of the cell centre is also observed for SIM2 and SIM3 along a similar elliptic trajectory (see green and black line respectively in Figure 4.10). For longer periods the centre needs more time to do one revolution and hence the recirculation cell moves back and forth less frequently (slower). A more thorough analysis of the movement of the large recirculation cell is given in the discussion (Section 4.3). In all three simulations the position nor the size of the corner vortices changes noticeably over time.

The flow field corresponding to the higher amplitude of the inlet velocity (SIM4) is somewhat different from the flow patterns of the other transient simulations, as noticed in the pathline plots on the right-hand side of Figure 4.9. The associated times are again represented in a non-dimensional form \( \frac{t}{T4} \) and marked on the inlet velocity function (top image) using a blue circle, with the time scale chosen so that the first minimum of the inlet velocity coincides with \( \frac{t}{T4} = 0 \). Start-up effects do not influence the presented results since the simulation has run already over a few periodic cycles. At \( \frac{t}{T4} = 0 \) the velocity at the inlet is at its minimum value of 0 m/s and no fresh air is supplied to the room. With increasing inlet velocity, the penetration depth of the supply jet grows and from the moment the jet reaches the end of the inlet channel \( \frac{t}{T4} < 0.19 \), part of the supplied air curls around the sharp corner leading to the development of a vortex under the supply slot.
Figure 4.9: Left hand side: pathline plots visualising the flow pattern obtained in SIM1 over one period. The times corresponding to these plots are: (a1) $t/T_1 = 0$ (0 s), (b1) $t/T_1 = 0.37$ (12 s), (c1) $t/T_1 = 0.50$ (16 s), (d1) $t/T_1 = 0.62$ (20 s), (e1) $t/T_1 = 0.87$ (28 s). Right hand side: pathline plots visualising the flow pattern obtained in SIM4 over one period. The times for these pathline plots are: (a4) $t/T_4 = 0.19$ (6 s), (b4) $t/T_4 = 0.37$ (12 s), (c4) $t/T_4 = 0.44$ (14 s), (d4) $t/T_4 = 0.59$ (19 s), (e4) $t/T_4 = 0.87$ (28 s). The top images show the inlet velocity as a function of time on which the times associated with the pathline plots are indicated with a blue circle ($\circ$).
with a clockwise rotation. The inlet velocity at this point is still small ($u_0/u_{0,RC} = 0.09$) which explains why this vortex is not observed in the flow field of SIM1, SIM2 and SIM3 where the minimum velocity is $u_0/u_{0,RC} = 0.5$. The vortex increases in size ($t/T4 = 0.19$), eventually detaches and drifts away from the inlet while approaching the centre of the large recirculation cell which has been moving towards the left ($t/T4 = 0.37$). Around $t/T4 = 0.44$ the recirculation cell absorbs the vortex (i.e. merging) and its centre changes direction towards the right ($t/T4 = 0.59$ and $t/T4 = 0.87$). For the corner vortices, no change in position or size is detected over time. The creation of the vortex under the inlet slot and the pattern of the large recirculation cell repeat themselves for every period of the inlet velocity. In Figure 4.10 the trajectory of the centre of the large recirculation cell is sketched (blue dashed line). It is observed that the elliptic trajectory is much more extended than in the simulations with a smaller amplitude due to the merging process. Another change in SIM4 compared to the other simulations is that the vortex present in the upper part of the outlet slot grows when the velocity at the inlet lowers, as is also the case in the other simulations, but then its size eventually extends into the room. This is illustrated in Figure 4.11 which shows a zoom-in on the outlet channel in a couple of pathline plots for times later than presented (on the right side) in Figure 4.9. The inlet velocity at the moment the cell extends its size into the room ($t/T4 = 1.06$) is $u_0/u_{0,RC} = 0.02$, which is below the minimum velocity of the other transient simulations ($u_0/u_{0,RC} = 0.5$) which explains why such a growth of this vortex is not detected in these simulations.

**Figure 4.10:** Sketch of the trajectory (of the centre) of the large recirculation cell in the four transient simulations: SIM1 (blue solid line), SIM2 (green), SIM3 (black) and SIM4 (blue dashed line). The stationary position of the cell centre in the reference case is indicated with a red circle.

**Figure 4.11:** Growth of the vortex present in the upper part of the outlet channel, illustrated with a zoom-in on the outlet slot in the pathline plots at times: $t/T4 = 0.87$ (28 s), $t/T4 = 1.06$ (34 s), $t/T4 = 1.12$ (36 s) and $t/T4 = 1.18$ (38 s). The recirculation cell appears to have a size larger than the channel around $t/T4 = 1.06$. 
**Velocity field**

Figure 4.12 depicts the contours of the (time-averaged) mean velocity magnitude $U/u_{0,RC}$ for all the simulations. Image $a$ represents the velocity field of the steady-state reference case which was already depicted in Figure 4.1, but is shown here again to allow an easy comparison with the results obtained under transient inlet conditions (images $b$-$e$). The contour plots of the time-dependent simulations are the result of averaging the mean velocity field over a sufficiently long time. The averaging started after the transient simulation has already run for a few periodic cycles of the inlet velocity in order to not include the start-up effects, which are present in the beginning of the simulation, and it was stopped when the time-averaged field did not change any more over time. Images $b$, $c$ and $d$ in Figure 4.12 are the contour plots obtained in SIM1, SIM2 and SIM3 respectively, and they do not show large differences compared to the steady-state reference case. The width of the core of the supply jet (yellow colour) is slightly more extended for SIM1 around $x = 2H$ than for the other simulations. Also the velocity in the return flow along the floor increases slightly for smaller periods of the inlet velocity. These findings can be better observed in Figure 4.13 where the profiles along the vertical lines $x = H$ and $x = 2H$ are compared. Furthermore, from the contour plots it is observed that the velocity in the corner regions does not change noticeably.

The time-averaged velocity field of SIM4 is visualised in image $e$ of Figure 4.12 and it is compared to the contour plot of SIM1 (image $b$) in order to investigate the influence of a larger amplitude on the flow field; both simulations do have the same period of the inlet velocity. The wall jet near the ceiling appears to penetrate somewhat deeper into the room (on average) for SIM4 and its width has increased, indicated by the larger spread of the yellow coloured core. Also the areas near the right wall and the floor have a larger width and their velocities are augmented (green/yellow colour). An increase in velocity is noticed as well in the low velocity regions near the left wall and in the centre of the large recirculation cell. The velocity in the corners is not changed. These observations are confirmed by a comparison of the profiles of SIM1 and SIM4 along the lines $x = H$ and $x = 2H$ in Figure 4.13: the overall increase in velocity is clearly noticed, as well as the larger spread of the wall jet around $x = H$.

**Turbulent kinetic energy**

The contour plots of the (time-averaged) turbulent kinetic energy $\sqrt{\kappa}/u_{0,RC}$, presented in Figure 4.14 show more variation between the steady-state reference case (image $a$) and the time-dependent simulations (images $b$-$e$). Again, the results for the transient calculations are obtained after averaging over a long time. SIM1, SIM2 and SIM3 (images $b$-$d$) contain on average clearly more turbulent kinetic energy in the large recirculation cell compared to the steady-state reference case (yellow area). This area of higher turbulence is also more extended towards the left for the transient calculations as can be observed by comparing the field around the line $x = H$. It is observed that the increase of turbulence in the region of the large recirculation cell and near the left wall is somewhat higher when the period of the inlet velocity is smaller. The flow close to the outlet slot is also characterised by higher turbulence values in the transient simulations, with again slightly higher values for shorter...
Figure 4.12: Contour plots of the (time-averaged) non-dimensional velocity field $U/u_{0,RC}$ for the steady-state reference case (a) and for the time-dependent simulations SIM1 (b), SIM2 (c), SIM3 (d) and SIM4 (e).

Figure 4.13: Comparison of the profiles of the non-dimensional (time-averaged) mean velocity magnitude $u/u_{0,RC}$ along two vertical lines, $x = H$ and $x = 2H$, for the reference case (red), SIM1 (blue solid line), SIM2 (green), SIM3 (black) and SIM4 (blue dashed line).
periods. For the corner regions no changes are observed. A comparison of the profiles along the lines \( x = H \) and \( x = 2H \), presented in Figure 4.15, also shows clearly the higher turbulence for shorter periods.

In the case the amplitude is doubled (SIM4) the time-averaged turbulent kinetic energy is increased considerably throughout the whole room as seen in image e of Figure 4.14 (see also Figure 4.15). The amount of turbulent kinetic energy in the large recirculation cell has reached the high turbulence level by which the shear layer is characterised. This high turbulence region is even more extended to the left compared the other transient simulations, resulting in the increment of turbulence close to the left wall. In the corner regions the turbulent kinetic energy is again not raised. All the simulations show a low turbulence in the vicinity of the walls due to the locally dominant viscous forces that dampen the velocity fluctuations.

\[ \frac{\sqrt{k}}{u_{0,RC}} \]

**Figure 4.14:** Contour plots of the (time-averaged) non-dimensional turbulent kinetic energy \( \sqrt{k}/u_{0,RC} \) for the steady-state reference case (a) and for the time-dependent simulations SIM1 (b), SIM2 (c), SIM3 (d) and SIM4 (e).
Figure 4.15: Comparison of the profiles of the non-dimensional (time-averaged) turbulent kinetic energy $\sqrt{k}/u_{0,RC}$ along two vertical lines, $x = H$ and $x = 2H$, for the different simulations: steady-state reference case (red), SIM1 (blue solid line), SIM2 (green), SIM3 (black) and SIM4 (blue dashed line).

Vorticity

In addition to the predictions of the velocity field and turbulent kinetic energy, the profiles of the (time-averaged) z-component of the vorticity ($\omega_z$) for all the simulations are presented in Figure 4.16 along the lines $x = 0.083H$, $x = H$, $x = 2H$ and $x = 2.92H$ (see Figure 4.1a for their position in the flow domain). The three simulations SIM1, SIM2 and SIM3 show almost no changes in vorticity compared to the reference case. Only in the large recirculation cell along the line $x = 2H$ ($0.1 < y/H < 0.8$) a very slight increase in negative vorticity is observed for SIM1.

The vorticity in SIM4 has changed more compared to the other transient simulations. Along the lines $x = H$ and $x = 2H$ an increase in negative vorticity is observed in the area of the large recirculation cell ($0.1 < y/H < 0.8$) and $\omega_z$ appears to be decreased at $x = H$ in the wall jet area (indicated with the number I in image b). The profiles of SIM4 along the other two lines, $x = 0.083H$ and $x = 2.92H$, do not show variations compared to the other simulations apart from two local deviations. The first deviation is observed at $x = 0.083H$ below the large peak of the shear layer, indicated with the number II in image a. This increase in negative vorticity is a result of the recirculation cell that develops under the inlet slot in SIM4. A similar deviation is noticed at $x = 2.92H$ near the floor (see number III in image d) where the vorticity has increased as well compared to the other simulations as a result of the vortex with positive vorticity in the upper part of the outlet slot which, as already shown, extends towards the room for very low inlet velocities in SIM4. Finally it is mentioned that the peak in the shear layer for all the simulations did not changes value (which cannot be seen in image a due to the limited range of $\omega_z$ in the plots).
Figure 4.16: Profiles of the z-component of the vorticity $\omega_z$ associated with the mean flow field obtained in the five simulations along four vertical lines in the flow domain: (a) $x = 0.083H$, (b) $x = H$, (c) $x = 2H$ and (d) $x = 2.92H$. The range of $\omega_z$ is limited to $[-5; 5]$.

4.2.3 Concentration field of contaminants with the presence of a uniform constant source

The influence of the time-dependent inlet conditions on the ventilation performance is studied by distributing a constant source uniformly throughout the room as described in Section 4.1.3. The source is activated after the simulations have already run for a certain time to avoid any influence of start-up effects. The contour plot of the concentration field $c_S \rho / \tau_n$ corresponding to the steady-state reference case is reproduced here in Figure 4.17a whereas the results of the simulations under transient inlet conditions, averaged over a sufficiently long time, are depicted in Figures 4.17b-e. The concentration fields of SIM1, SIM2 and SIM3 (images b-d) point out that the transient inlet conditions do have a positive effect on the ventilation performance compared to the reference case. For these three simulations, the contaminant concentration level throughout the whole room is lower compared to the steady-state reference case. It is shown that a shorter period of the inlet velocity is more beneficial for the ventilation: the large recirculation cell and the area near the left wall clearly contain less contaminants for SIM1 than for SIM2 and SIM3. The
profiles along the lines \( x = H \) and \( x = 2H \), presented in Figure 4.18, show the same observations and it is noticed that the decrease in concentration from SIM3 to SIM2 is smaller than from SIM2 to SIM1, despite the larger difference of the period in the former case \( (\Delta \frac{T}{\tau_n} = 0.5) \) compared to the latter \( (\Delta \frac{T}{\tau_n} = 0.3) \). Finally, in the contour plots it is also seen that in the left corner the ventilation remains the worst with relatively high concentration levels. Also in the right upper corner the small increase in the contaminant concentration persists.

The best results are obtained when the amplitude is doubled, as seen in the contour plot of SIM4 in Figure 4.17. The concentration levels are decreased strongly throughout the room compared to the other transient simulations and the steady-state reference case. The area close to the left wall still contains the highest levels of contaminants, with in particular the lower left corner zone. Also the air in the upper right corner shows again a small increase.

**Figure 4.17:** Contour plots of the non-dimensional (time-averaged) concentration field \( \frac{c_p}{c_{sc} \tau_n} \) when a constant source \( S_c \) is distributed uniformly throughout the room for the steady-state reference case (a) and for the time-dependent simulations SIM1 (b), SIM2 (c), SIM3 (d) and SIM4 (e).
of contaminants. Right behind the inlet slot of the room the fresh supply air appears to be already mixed with contaminated air, indicated by the light blue colour in the core of the jet. This is not the case in the other simulations which gives rise to the idea that the vortex that splits of under the inlet slot (see Figure 4.9a) increases locally the mixing whereby the contaminated air under the entrance slot is transported to this jet region. From a comparison of the profiles along the lines \( x = H \) and \( x = 2H \), presented in Figure 4.18, the lower concentration level in SIM4 compared to the other simulations is clearly noticed. However, near the ceiling the contaminant concentration in the other simulations appears to be smaller than in SIM4.

![Figure 4.18](image)

**Figure 4.18:** Comparison of the profiles of the non-dimensional (time-averaged) concentration field \( \frac{C}{c_i} \rho \tau_n \) along two vertical lines, \( x = H \) and \( x = 2H \), for the different simulations: steady-state reference case (red), SIM1 (blue solid line), SIM2 (green), SIM3 (black) and SIM4 (blue dashed line).

### 4.2.4 Decay of the contamination from an initial concentration

In this subsection the decay from an initial concentration of contaminants, with no source present in the enclosure, similar as was done for the steady-state reference case in Section 4.1.4, is examined. The initial concentration \( c_i \) is set after the simulation has already run for a sufficiently long time span to avoid any influence of the start-up effects and \( c_i \) is uniformly distributed through the room. First, the distribution of fresh air through the room will be visualised using contour plots of the contaminant concentration field. Then, the influence of the period on the decay curves in the fifteen monitoring points is discussed after which the influence of the amplitude will be described. Finally, the average decrease of the concentration in the room is commented for the different simulations.

**Distribution of the fresh air in the room with time**

Figure 4.19 shows a sequence of contour plots of the non-dimensional mean concentration \( c/c_i \) for SIM1 over time, illustrating the distribution of the fresh air under time-dependent inlet conditions. The times corresponding to the contour plots are shown at the top right of the images. At time \( t/T = 0 \) the velocity at the inlet is \( u_0/u_{0,RC} = 1 \) and it starts to increase towards the maximum value \( u_0/u_{0,RC} = 1.5 \) (see the first image in Figure 4.19).
Figure 4.19: Contours of the non-dimensional mean concentration field $c/c_i$ as a function of time for SIM1. The times corresponding to these plots are: (a) $t/T_1 = 0.12$ (4 s), (b) $t/T_1 = 0.25$ (8 s), (c) $t/T_1 = 0.39$ (12.4 s), (d) $t/T_1 = 0.51$ (16.4 s), (e) $t/T_1 = 0.67$ (21.6 s), (f) $t/T_1 = 0.82$ (26.4 s), (g) $t/T_1 = 1.02$ (32.8 s), (h) $t/T_1 = 1.38$ (44.4 s), (i) $t/T_1 = 1.59$ (52.2 s). The first image shows the inlet velocity as a function of time on which the times associated with the pathline plots are indicated with a blue circle (◦).

The fresh air penetrates deeper into the room with time along the ceiling via the wall jet and its core contains the lowest concentration values (images a and b). In addition, the fresh air is transported towards the middle of the room as a result of the high turbulence levels in the shear layer and in the large recirculation cell (see Figure 4.14b). As the fresh air approaches the right wall the velocity at the inlet has reached its maximum value (image b) after which it starts to decrease and the penetration depth of the low concentration core
(blue colour) reduces again (images c, d and e). The reduction in penetration depth of the wall jet induces a split-off of the “head” of the fresh air which continues to move along the right wall and the floor according to the flow pattern of the large recirculation cell (images e and f). This also leads to an increase in the concentration level nearby the walls after the head has passed by since on the one hand the penetration depth is too weak to provide a sufficient amount of fresh air to these areas and on the other hand contaminated air from the left side of the room is able to reach these areas via the large recirculation cell (see arrow in image f). As the head moves further along the bottom wall, the blue core of the fresh air starts to penetrate again deeper into the room due to the velocity at the inlet that started to increase again (image g). From this point the above described process recurs and this for every periodic cycle of the inlet velocity. A few images are added to the sequence of contour plots to show the distribution of the concentration at later stages of the ventilation process. In image h it is observed that the wall jet contains lower concentration values than before as indicated by the large light blue coloured zone (compare images c and h) since the surrounding air is already more diluted and therefore the core of the jet is capable of keeping the low concentration value for a longer time while it mixes gradually with the surrounding air. Image i illustrates more clearly the split-off of the head of the wall jet.

For the simulations with a longer period than SIM1, i.e. SIM2 and SIM3, the head of the wall jet does not split-off any more and the contaminant concentrations in the short-circuit flow between the inlet and outlet slot seem not to increase as much as in SIM1 after the wall jet has passed by. These observations are illustrated with a few contour plots in Figures 4.20 and 4.21 for SIM2 and SIM3 respectively. A possible explanation for this is that for the simulations with a longer period the velocity at the inlet changes slower over time; it takes longer to reach the maximum velocity after which the penetration depth starts to decrease due to the decreasing inlet velocity. This decrease in penetration depth occurs more gradually leading to a smooth reduction of the supplied fresh air to the areas near the ceiling and the right wall. This in contrast to SIM1 where the transport changed more abruptly. In the meanwhile the air with high concentration levels that is transported towards the short-circuit area via the large recirculation cell (in analogy to the situation illustrated for SIM1 in Figure 4.19f) seems to be diluted already to a certain level, more than in SIM1. Hence, the increase in concentration in the short-circuit area after the wall jet has passed by seems to be less for SIM2 and SIM3 than in SIM1.

The sequence of contour plots of the concentration over time for SIM4 is presented in Figure 4.22 and the plots correspond to the same times as those of SIM1 to allow for an easy comparison (which is possible since the periods of the inlet velocity in both simulations are equal). The simulation starts with an inlet velocity equal to $u_0/u_0,RC = 1$ (see first image in this figure). The flow field of SIM4 differs from the other simulations due to the presence of the second recirculation cell under the inlet slot, as discussed in Figure 4.9. This cell moves along with the wall jet, thereby extracting fresh air which leads to a larger spread of the head of the jet compared to SIM1 (images a and b). The location of the recirculation cell is indicated in image a with a box and the circle represents its
Figure 4.20: Contours of the non-dimensional mean concentration field \( c/c_i \) as a function of time for SIM2. The times corresponding to these plots are: (a) \( t/T_2 = 0.24 \) (19.6 s), (b) \( t/T_2 = 0.5 \) (39.8 s), (c) \( t/T_2 = 0.75 \) (60 s) and (d) \( t/T_2 = 1 \) (80.4 s). The first image shows the inlet velocity as a function of time on which the times associated with the pathline plots are indicated with a green circle (◦).

Figure 4.21: Contours of the non-dimensional mean concentration field \( c/c_i \) as a function of time for SIM3. The times corresponding to these plots are: (a) \( t/T_3 = 0.24 \) (39.2 s), (b) \( t/T_3 = 0.5 \) (80 s), (c) \( t/T_3 = 0.75 \) (120 s) and (d) \( t/T_3 = 1 \) (161 s). The first image shows the inlet velocity as a function of time on which the times associated with the pathline plots are indicated with a circle (◦).

centre. Image b corresponds to the situation right after this vortex has merged with the large recirculation cell. The inlet velocity at this stage has reached its maximum and starts to decrease while the jet keeps moving further along the ceiling and the right wall (images c, d and e). It is observed that in the meanwhile a larger part of the air in the left half of the room is diluted compared to SIM1. This is a result of the larger spread of the jet in SIM4 caused by the second vortex, thereby bringing fresh air into the region of the large recirculation cell which distributes the air throughout the room due to high turbulence levels (see Figure 4.14e). In images e and f, the inlet velocity is very close to the minimum value \( (u_0/u_{0,RC} = 0) \) and almost no fresh air is supplied to the room. This again leads to a split-off of the head of the jet and according to the same arguments as described above for SIM1, the concentration increases again after the head of the jet has passed by. As the head moves further along the floor, it appears to contain a larger amount of fresh air compared to SIM1, which is beneficial for the area nearby the left wall.
At this moment, it can be observed from image $g$ that a new period of the inlet velocity has started since air penetrates again deeper into the room. This image also clearly shows the extraction of fresh air by the vortex under the inlet slot. In images $h$ and $i$ the concentration field at later times is presented and, compared to SIM1, the area near the left wall appears to be better decayed.

**Figure 4.22:** Contours of the non-dimensional mean concentration field $c/c_i$ as a function of time for SIM4. The times corresponding to these plots are: (a) $t/T = 0.12$ (4 s), (b) $t/T = 0.25$ (8 s), (c) $t/T = 0.39$ (12.4 s), (d) $t/T = 0.51$ (16.4 s), (e) $t/T = 0.67$ (21.6 s), (f) $t/T = 0.82$ (26.4 s), (g) $t/T = 1.02$ (32.8 s), (h) $t/T = 1.38$ (44.4 s), (i) $t/T = 1.59$ (52.2 s). The first image shows the inlet velocity as a function of time on which the times associated with the pathline plots are indicated with a blue circle ($\circ$).
Influence of the period on the contaminant decay

In addition to the contour plots of the concentration field, the decay of the (non-dimensional) concentration \( c/c_i \) as a function of time is studied in the fifteen monitoring points that were shown in Figure 4.6. In this paragraph, the influence of a change in the period \( T \) is described (SIM1, SIM2 and SIM3) and later the impact of an increment in the amplitude \( \Delta u_0 \) will be considered.

Figure 4.23 shows the decay curves of SIM1, SIM2 and SIM3 together with the curves of the reference case in the fifteen monitoring points. The position of a plot on the page is in agreement with the location of the associated monitoring point in the room in order to have a representation of the decay behaviour according to the location in the room. In the previous paragraph it was shown using contour plots of the concentration field that the contaminant concentration in certain regions increases as the penetration depth of the wall jet decreases due to a decreasing velocity at the inlet. This variation in penetration depth causes periodic fluctuations in the concentration levels throughout the room which is clearly noticed in the decay curves of the fifteen monitoring points. Despite the increment in the contaminant concentration noticed in the decay curves of certain monitoring points, the concentration level does decrease over one cycle of the inlet velocity, leading to an overall decrease in the concentration over time. The increase in the concentration level for the transient simulations is fundamentally different from the decay curves in the reference case which follow a monotonically decreasing function (in all monitoring points), i.e. without any increment in the concentration level. Each transient simulation is characterised by a specific shape of the decay curve which repeats for every period of the inlet velocity as indicated with arrows in the plot of point 15. The shape and the amplitudes of the fluctuations vary slightly throughout the flow domain:

- The decay curves of the points located near the right wall (points 5, 10 and 15) show the largest fluctuations during each periodic cycle. The curves of SIM1 and SIM2 are built up of several dips, consisting of a sharp drop followed by an increase whereas the drop in the decay curve of SIM3 is first followed by a weak increase after which it descends gradually till the next drop. The drops for the three transient simulations are observed to be smaller for the points 5 and 10 compared to point 15.

- The monitoring points close to the floor (points 2, 3 and 4) show similar decay curves as for the points nearby the right wall, with a periodically recurring shape determined by the inlet velocity. For SIM1 the shape of the decay curve looks more like a smooth wave instead of several sharp peaks whereas for SIM2 the shape seems to be the same as described earlier. The dips show a gradual decrease in size for both simulations from point 4 to point 2. In the decay curve of SIM3 the small increase in concentration after each drop disappears which leads to a monotonically decreasing curve for this simulation.

- Near the left wall in the points 1, 6 and 11, the same changes as mentioned above are observed: the peaks in the curves are again much more dampened and the
Figure 4.23: Decay curves of the concentration $c/c_i$ obtained in the fifteen monitoring points (see Figure 4.6) for the transient simulations SIM1 (blue), SIM2 (green) and SIM3 (black). The results of the reference case (red) are also indicated. The position of the decay plots is the same as the location of the associated monitoring point in the room. The asterisks (*) on each transient decay curve point out the concentration level every time a periodic cycle of the inlet velocity has ended.
periodic drops in the concentration occur more gradual, which leads for SIM1 and SIM2 to monotonically decreasing curves.

- Points 7 and 8, which are two of the three points located in the middle of the room, are characterised by a similar decay as in the points nearby the left wall. Point 9, the most right located point of the three points in the middle, shows a different behaviour for SIM1 since the decay curve in this simulation is build up of dips and hence the decay curve is not monotonically decreasing any more.

- The last three monitoring points to be discussed are points 12, 13 and 14 which are located close to the ceiling. The difference with the curves in the other points is again observed for SIM1. For point 12, during every period of the inlet velocity a dip in the concentration level occurs followed by a weak decrease until a new dip appears. It is this decrease between two dips that is different from the decay curves of SIM1 in other monitoring points. The amplitude of the fluctuations in point 14 are more intense compared to point 13 which in turn shows larger amplitudes than in point 12.

By taking a global view at Figure 4.23, the transition between the decay curves corresponding to the monitoring points that are situated along the walls can be observed, starting in point 12 and going to point 11 in the clockwise direction (i.e. the flow direction). The transition is the best observed for SIM1. When the fresh air moves along the ceiling, it crosses the points $12 \rightarrow 13 \rightarrow 14 \rightarrow 15$ (in this order) and the dips of the decay curves increase in size since the monitoring points more downstream are situated closer to the low concentration core of the fresh air supply jet. Point 12 on the contrary is located at the boundary of the wall jet where the concentration level is somewhat larger and hence the dips in the concentration level will be less pronounced compared to these in point 15. During the movement of the fresh air along the right wall, the floor and then the left wall (points $15 \rightarrow 10 \rightarrow 5 \rightarrow 4 \rightarrow 3 \rightarrow 2 \rightarrow 1 \rightarrow 6 \rightarrow 11$) it mixes more and more with the polluted indoor air, thereby causing drops in the decay curve of SIM1 that decrease in amplitude for the points that are reached later by the fresh air. The transition from a peak-shaped decay curve to a wave-shaped decay curve is observed and is a result of the lower velocities whereby the fresh air crosses the points that are reached later: the drop and the increase in the concentration level during each cycle of the inlet velocity occur more and more gradually. For points 7 and 8, situated in the middle, the decay curve of SIM1 is similar to these observed in the points near the left wall since the fresh air reaches these points via mixing throughout the large recirculation cell, thereby causing a weak dip in the concentration level and a wave-shaped decay profile in these points. Point 9 on the other hand, the most right located point of the three points in the middle, is positioned in the boundary of the fresh air jet due to the large width of the jet while moving along the right wall which causes the decay profile of SIM1 to be peak-shaped. A similar transition is noticed for the other two transient simulations SIM2 and SIM3 while going from point 12 via the walls to point 11. Except for point 9 which does not show a deviating decay behaviour any more from the other two points in the middle. This is a result of the spread
of the fresh air jet along the right wall being decreased for SIM2 and SIM3 compared to SIM1.

In the fifteen monitoring points, the concentration level of the transient calculations obtained after each periodic cycle of the inlet velocity can be compared to the concentration in the steady-state case at the same moment in time in order to examine which ventilation method performs the best in the decay from an initial concentration. It should be emphasized that only the concentration values obtained after each periodic cycle are useful to compare with these of the steady-state case since then both ventilation methods have supplied the same amount of air to the room. The concentration at the end of every periodic cycle of the inlet velocity is indicated on the decay curves in Figure 4.23 with an asterisk (*). For SIM1, most monitoring points show lower concentration values than in the steady-state simulation, as for example in point 1 which is depicted here again in Figure 4.24 and can be considered to be representative for the majority of the monitoring points. The concentration values for the steady-state decay curve are also indicated with an asterisk in this figure. Only a few points, located near the right wall and the ceiling (5, 10, 13, 14 and 15), show a decay in which the marked concentration values of SIM1 are above those of the steady-state reference case (see Figure 4.23). This is mainly due to a combination of the large peak amplitudes in these decay curves (resulting in large overshoots) together with a sharp initial decay in the steady-state case. However, in each of these monitoring points the concentration values drop below the steady-state curve after a number of periods. For the other two transient simulations, SIM2 and SIM3, the same is observed with the majority of the monitoring points showing decay values below the steady-state curve. Also in these simulations there are a few monitoring points that do have decay curves for which the marked concentration values lie above the steady-state curve: for SIM2 these monitoring points are located near the ceiling, the right wall and the floor (points 3, 4, 5, 10, 14 and 15) whereas for SIM3 such monitoring points are only positioned near the floor (points 2, 3, 4 and 5). In these points the transient decay curves exceed the curve of the steady-state case during the first periodic cycles of the inlet velocity. From this analysis it can be concluded that transient forcing of the flow has a positive effect on the decay performance compared to the steady-state ventilation method since the concentration values under time-dependent conditions appear to be almost always lower than in the steady-state case.

Not only the decay performance of the transient simulations with respect to the steady-state is of interest. Also the influence of the change in the period on the concentration decay is examined, i.e. a comparison between SIM1, SIM2 and SIM3. Figure 4.25 represents the concentration values in the fifteen monitoring points when the concentration has decayed for a period equal to the nominal time constant (part A of the graph above the grey dashed line) and for a time span twice as long (part B of the graph below the grey dashed line). It is worth to mention once more that only at times that are multiples of \( \tau_n \) the concentration values corresponding to the transient simulations can be compared since then the same amount of air is supplied to the room in all the simulations. The monitoring points are ordered according to their position in the room (floor, middle and ceiling) and
Figure 4.24: Decay curves of the concentration $c/c_i$ in monitoring point 1 corresponding to SIM1 (blue) and the steady-state simulation (red). The concentration values at specific moments in time determined by the period of the inlet velocity, marked with an asterisk (*), are compared.

The concentration level is indicated with a specific symbol for each simulation: reference case (RC) $\times$, SIM1 $\circ$, SIM2 $\square$ and SIM3 $\triangledown$. The decay for a time equal to $t/\tau_n = 1$ will be discussed first (Figure 4.25A):

- The first thing that can be noticed is the decrease in all the concentration values with increasing number of the monitoring point for each location (floor, middle and ceiling), illustrating that the ventilation gets better towards the right side of the room in the area of the short-circuiting flow.

- The steady-state simulation ($\times$) appears to have the largest concentration values in the majority of the points with respect to these of the transient simulations which is in agreement with the previous findings. 60% of the monitoring points (9/15), most of which are located near the ceiling and in the middle of the room, indicate that the steady-state simulation predicts the highest concentration levels. In most of the points near the floor (points 2, 3 and 4) the steady-state concentration level is not the highest of all simulations, but still, the values can be considered relatively high. Only in the points nearby the right wall (points 5, 10 and 15), the steady-state performs relatively good.

- The best decay results after a ventilation time equal to the nominal time constant are obtained with SIM1 ($\circ$) since the concentration values for this simulation are the lowest with respect to the other simulations in 80% of the monitors (12/15). The remaining monitoring points, located downstream of the inlet slot (points 13, 14, 15), are characterised by relatively high concentration values for this simulation.
The concentration values in SIM2 (□) with respect to the other simulations depend on the position in the room. It is shown that the concentration levels in the left half of the room (points 1,2; 6,7,8; 11,12,13) are relatively low whereas the monitors located in the right half are characterised by high concentrations compared to the other simulations. SIM2 ventilates worse than SIM1 in almost all monitoring points (93%; 14/15), except for point 13. It is observed that SIM2 predicts the highest concentration levels of all simulations in the points near the right wall (points 5, 10, 15).

In most points near the floor (points 2, 3 and 4), SIM3 (⊿) appears to have the highest concentration values of all simulations. Also in the other regions of the room (middle and ceiling), SIM3 predicts a relatively high concentration level, except for the points close to the right upper corner (points 9,10; 13,14,15) where the concentrations are relatively low and even the lowest in points 14 and 15. 80% of the monitors (12/15) indicate that SIM3 predicts a higher concentration value than SIM1 (all points except point 13, 14 and 15) and in 67% of the points (10/15) SIM3 is worse than SIM2.

In summary, SIM1 shows the best results in most of the room except for the monitoring points 13, 14 and 15 where SIM3 shows the best performance. However, SIM3 has relatively
high concentration values in the other monitoring points, certainly near the bottom. For SIM2, the concentration values are observed to lie in between these of SIM1 and SIM3. This holds for 60% of the monitoring points, illustrating that in general the decay is better when smaller periods of the inlet velocity are used. These monitoring points are surrounded by a frame in Figure 4.26. All monitoring points in which the decay in the steady-state reference case is worse than under time-varying conditions are marked with the ×-symbol in this figure, from which it is again concluded that forcing under time-dependent conditions leads to an improvement of the decay performance compared to ventilation with a steady inlet velocity (constant over time).

Figure 4.25: The part under the dashed line, shows the concentration level in each monitor after a time $t/τ_n = 2$. Apart from the lower concentration values, a doubling of the decay time does not show large differences in the order of the concentration levels of the different simulations. A few observations are made:

- The same points in which SIM1 showed the lowest concentration values after a time $t/τ_n = 1$ still have the lowest values after the longer time span. In general SIM1 has even improved in most points, as for example in monitors 13, 14 and 15, which causes SIM1 to perform relatively well now downstream of the inlet slot.

- SIM2 is more or less still characterised by relatively low concentration values in the left half of the room and somewhat higher values in the right half. It also performs worse than SIM1 in the same points as for the shorter ventilation time with point 13 the only point in which SIM2 is better than SIM1. However, due to the relatively lower concentration value of SIM1 in this point the difference with SIM2 has shrunk. In addition, SIM2 seems to approach the concentration value of SIM3 in most points were SIM2 has higher concentration values than SIM3 at $t/τ_n = 1$ (see e.g. points 9, 10, 14 and 15). In point 5, the concentration level of SIM2 falls even below this of SIM3 which means that the number of monitoring

Figure 4.26: Visualisation of the monitoring points in which the concentration obtained in SIM2 lies in between the concentrations predicted by SIM1 and SIM3. The solid black line surrounds the monitoring points that show this characteristic after a time $t/τ_n = 1$. The dashed black line together with the solid line indicates all monitoring points that show this characteristic after a time $t/τ_n = 2$. In addition, all the monitors in which the steady-state ventilation has the highest concentration value compared to SIM1, SIM2 and SIM3 are marked with the ×-symbol. The points for which this holds are unchanged for the longer ventilation time ($t/τ_n = 2$).
points in which SIM2 had the highest concentration value reduces from three to two points (points 10 and 15).

- For SIM3, the same observations as for the shorter ventilation time hold: the concentration values are still relatively high, certainly near the bottom, and only in the points 14 and 15 it performs relatively well.

- The steady-state case remains the simulation with the highest concentration values. The same monitoring points that are marked with the ×-symbol in Figure 4.26 still show this characteristic. In a few points the steady-state ventilation has even worsen (or the transient simulations have improved), e.g. in point 4 but also in the points near the right wall where the steady-state case performed relatively well for the smaller ventilation time (points 5 and 10) but now shows relatively higher concentration values.

Hence, for longer ventilation times, the steady-state ventilation method appears to perform worse compared to the transient simulations of which SIM1 remains the best. It also seems that the simulations with a shorter period of the inlet velocity improve with respect to the simulations with a longer period. Because SIM2 has obtained a lower concentration value than SIM3 in point 5, the number of points for which the concentration of SIM2 lays in between the concentration of SIM1 and SIM3, has increased by one. This point is indicated with a dashed line in Figure 4.26.

**Influence of the amplitude on the contaminant decay**

Besides the influence of the period on the decay behaviour, also the impact of an enlargement of the amplitude of the inlet velocity is tested (SIM4). Therefore SIM1 and SIM4 are compared since both simulations have the same period but the amplitude of SIM4 is twice as large. Figure 4.27 plots the results of both simulations in a few monitoring points. The decay curves of SIM4 are represented by the blue dashed line, these of SIM1 by the blue solid line as before and the steady-state results are also added as a second reference. The most important observations are summarised:

- The results in monitoring point 15, which is positioned in the upper right corner, are depicted in Figure 4.27a and almost no differences between SIM1 and SIM4 are observed. SIM4 still overshoots the steady-state decay curve in the first periods of the inlet velocity. However, for an increasing time, it performs somewhat better due to a smaller increase of the concentration level during every periodic cycle compared to SIM1. The results in point 15 are representative for the points 4, 5, 10 and 14.

- The decay curves in point 1, which is located in the lower left corner, are shown in Figure 4.27b. Both simulations show the same shape of the decay curve but SIM4 is characterised by lower concentration values at any moment in time and the decay starts earlier compared to SIM1. Therefore SIM4 performs somewhat better than SIM1. These observations also hold for the points 2, 3, 6 and 7.
Figure 4.27: Comparison of the decay profiles of the (non-dimensional) concentrations $c/c_i$ obtained in SIM4 (blue dashed line) with these obtained in SIM1 (blue solid line) over a time span equal to two times the nominal time constant $\tau_n$ in the following monitoring points: (a) point 15, (b) point 1, (c) point 8, (d) point 9, (e) point 11 and (f) point 12. The steady-state case is also plotted as a reference (red curve).

- For the two points in the middle, points 8 and 9, the decay curves are plotted in Figures 4.27 c and d respectively. The concentrations for SIM4 appear to be lower than SIM1 as well but the shape of the decay curve has somewhat changed. In point 8, SIM1 is monotonically decreasing as a function of time whereas SIM4 shows small increases in the concentration level. However, the decay in SIM4 is better. In the monitoring point 9, the amplitudes of the peaks in SIM4 are
larger than in SIM1. Also in this case the curve in SIM4 falls everywhere below the curve of SIM2, therefore leading to a better decay in SIM4.

- The monitoring points close to the inlet slot, monitors 11 and 12, show the largest changes in the concentration between SIM1 and SIM4 as seen in Figures 4.27e and f respectively. In both points SIM4 is characterised by large dips that are not (or to a lesser degree) noticed in SIM1. The reason for this is the development of the vortex under the inlet slot which detaches and eventually merges with the large recirculation cell. Every time the vortex develops it crosses point 11 which causes a large dip in the decay curve. This also explains the shorter time before the decay starts in SIM4 than in SIM1 for this point. When the vortex detaches it penetrates into the room along the boundary of the wall jet leading to a larger spread of the low concentration core of the fresh air compared to SIM1. This results in larger dips in the decay curves of points 12 and point 13. Also point 9 in the middle of the room shows larger decay amplitudes for this reason.

The concentration values that are obtained after each cycle of the inlet velocity are indicated for point 15 with an asterisk on SIM1 (⋆) and with a circle on SIM4 (●) as seen in Figure 4.27a. At the end of the first period SIM4 has a slightly higher concentration value than SIM1. For longer times the concentration level in SIM1 is always higher. This decay behaviour is only noted for two more monitoring points (points 12 and 13) whereas for all other monitors, SIM4 predicts lower concentration values than SIM1 after every period of the inlet velocity. Hence, an increment of the amplitude is beneficial for the decay performance. These observations are confirmed by comparing the concentration levels of SIM1 and SIM4 in the fifteen monitoring points at the two particular times $t/\tau_n = 1$ and $t/\tau_n = 2$, presented in Figures 4.28a and b respectively. The contaminant concentration is always the lowest for SIM4. In point 11, a large decrease in the concentration is noticed for SIM4 due to the presence of the vortex under the inlet slot.

![Figure 4.28](image-url): Comparison of the decayed concentration values $c/c_i$ predicted by SIM1 (⋆) and SIM4 (●). Image (a) corresponds to a ventilation time of $t/\tau_n = 1$ whereas in (b) the concentration values are obtained after a time $t/\tau_n = 2$. 
Finally, the average decrease of the concentration in the room is calculated at the times \( t/\tau_n = 1 \) and \( t/\tau_n = 2 \) for the different simulations by averaging the concentration values over the fifteen monitoring points. The results are presented in Figure 4.29 in the non-dimensional form \( \langle c/c_i \rangle_{15\text{points}} \), with the operator \( \langle \rangle_{15\text{points}} \) indicating the averaging of the concentration values over all the monitors. For the reference case it was already mentioned in Section 4.1.4 that the contaminant concentration has decreased (on average) to 47\% of the initial concentration in the case the room is ventilated for a time equal to the nominal time constant \( (t/\tau_n = 1) \) and that for a ventilation over a time span that is twice as long \( (t/\tau_n = 2) \) the concentration level has decreased to 21\% of the initial value. The transient simulations perform better than the steady-state case for both ventilation times, which supports the earlier findings. In SIM3 and SIM4, the concentration is decayed (on average) to a level close to that of the steady-state reference case with a somewhat lower concentration level for SIM3 than SIM4. The simulation with the shortest period, SIM2, seems to perform much better and shows the lowest concentration values compared to SIM2, SIM3 and the steady-state reference case. For this simulation the contaminant concentration has decreased to 41\% of the initial value at \( t/\tau_n = 1 \) and to 16\% at \( t/\tau_n = 2 \). An increment of the amplitude of the inlet velocity (SIM4) leads to even lower concentrations and SIM4 shows the best decay behaviour of all five simulations: after a time \( t/\tau_n = 1 \), the concentration averaged over all monitors has decayed to 34\% of the initial concentration value and for the time twice as long \( (t/\tau_n = 2) \) to 12\%.

**Figure 4.29:** Average decrease in the concentration after ventilating the room for two times \( t/\tau_n = 1 \) and \( t/\tau_n = 2 \). The concentration values are obtained by averaging the concentrations of all the monitoring points at the times under consideration.
4.3 Discussion

The discussion comprises a description of the movement of the large recirculation cell and the motivation why using a shorter period and/or a higher amplitude of the inlet velocity results in a better ventilation performance.

4.3.1 Movement of the large recirculation cell

In Figures 4.9 and 4.10 of Section 4.2.2 it was shown that for the transient simulations the centre of the large recirculation cell moves in a clockwise direction according to an elliptic trajectory which is repeated during every periodic cycle of the inlet velocity. It is believed that the position of the cell centre depends on the penetration depth of the wall jet. To illustrate this, Figure 4.30 depicts pathline plots obtained in SIM1 which are overlayed with contour plots of the mean contaminant concentration which visualise the wall jet. The six images describe the flow field over one periodic cycle of the inlet velocity after the flow field has already run over a sufficiently long time to avoid start-up effects. In the pathline plots a few reference lines are indicated in order to better notice the movement of the cell centre. The following observations can be made:

1. As the jet penetrates into the room along the ceiling, the centre of the large recirculation cell moves towards the left in the direction of the “head” of the wall jet (images a and b).

2. The centre of the recirculation cell then starts to move to the right from the moment the wall jet has passed this centre (images b and c).

3. When the jet propagates further along the right wall and the floor, the cell centre continues its movement to the right (images c and d) and reaches its outermost right position (image e).

4. From this point the cell centre moves back (images e and f) towards its position presented in image a.

The same observations hold for SIM2 and SIM3. It appears that the centre of the recirculation cell moves according to the penetration depth of the wall jet and it seems that the centre follows the head of the jet.

Furthermore it is noticed that the cell centre of the large recirculation cell moves relatively fast in the steps 1, 2 and 3 (described above) compared to its movement during step 4. In the first three steps the wall jet propagates along the ceiling, the right wall and the floor, thereby moving the cell centre relatively fast along most part of the elliptic trajectory. Once the head of the wall jet is passed by and the cell centre has reached its outermost right position, the centre moves slowly towards the left, until a new periodic cycle of the inlet velocity starts and the wall jet penetrates again deeper into the room. The time needed for the cell centre to do one revolution is equal to the period of the inlet velocity. Hence, in SIM1 the cell centre moves more frequently back and forth compared to SIM2.
and SIM3. In addition, for increasing period, the time in which the cell centre moves slowly (step 4) increases compared to the time of the fast moving part of the trajectory (steps 1, 2 and 3). For SIM2 and SIM3, the cell centre is thus moving very slowly during most of the time of a periodic cycle. This is not the case in SIM1 where the movement of the centre occurs in a more continuous way.

![Figure 4.30](image)

**Figure 4.30**: Comparison between the position of the cell centre of the large recirculation cell and the movement of the wall jet during one periodic cycle of the inlet velocity of SIM1: (a) $t/T_1 = 0.12$ (4 s), (b) $t/T_1 = 0.25$ (8 s), (c) $t/T_1 = 0.40$ (12.8 s), (d) $t/T_1 = 0.52$ (16.8 s), (e) $t/T_1 = 0.70$ (22.4 s) and (f) $t/T_1 = 0.90$ (28.8 s). The pathline plots show the cell centre, and the wall jet is visualised using contour plots of the dimensionless mean concentration field $c/c_i$. In the images four reference lines are shown to better see the movement of the cell centre.

For SIM4, the movement of the large recirculation cell is similar as described above. The only difference is that the second vortex under the inlet slot develops and moves along with the wall jet (see right side of Figure 4.9). As the jet penetrates deeper into the room, the centre of the large recirculation cell moves towards the left and eventually merges with the second vortex. At this moment, the wall jet has passed the centre of the merged recirculation cell and a similar movement as described in the above steps 2-4 is observed.
Besides the simulations presented in this report, two extra transient simulations with a larger period are performed to further examine the movement of the large recirculation cell. The periods are $T/\tau_n = 1.5$ and $T/\tau_n = 4$, and both simulations have an amplitude of $\Delta u_0/u_{0,RC} = 0.5$. In the first case the recirculation cell appears to move along an elliptic trajectory which is similar but smaller compared to the trajectories observed in SIM1, SIM2 and SIM3. In the case that the period is increased up to $T/\tau_n = 4$, the recirculation cell does not move any more and the position of the cell centre is the same as in the steady-state reference case. Apparently, the time-periodic character should be small enough to induce the movement of the large recirculation cell.

Finally, several steady-state simulations are performed with (constant) velocities of the supply air, in analogy to the steady-state reference case, which cover a range equal to the range used for the transient simulations: from $u_{0,RC} = 1.5$ m/s to $u_{0,RC} = 0.5$ m/s with steps of 0.25 m/s, $u_{0,RC}$ equal to 0.1 m/s and $u_{0,RC} = 0.01$ m/s. A few pathline plots are presented in Figure 4.31 with the colour indicating the velocity magnitude $U$. Images a and b correspond to a constant inlet velocity of $u_0 = 1.5$ m/s and $u_0 = 0.5$ m/s respectively and it is noticed that the centre of the large recirculation cell has barely moved, which also holds for the simulations with the inlet velocity in between these velocities. This is in contrast to the transient simulations (SIM1, SIM2 and SIM3) in which the cell centre did change position for this range of velocities. When the inlet velocity is lowered to a value of 0.1 m/s (image c), the cell centre has moved to the left, and for the lowest tested velocity (0.01 m/s) the flow pattern has strongly changed and the recirculation cell appears to have its centre closer to the left wall (image d). Such a flow field as in image d is not noticed in the pathline plots of SIM4 for which the lowest inlet velocity was 0 m/s. These observations indicate that a particular position on the elliptic trajectory of the cell centre obtained for a certain inlet velocity in the transient simulations does not correspond to the position of this centre in a steady-state simulation performed with the same inlet velocity.

### 4.3.2 Influence of the period and amplitude on the time-averaged mean concentration field

In Section 4.2.3 it was shown that the time-averaged mean concentration field, corresponding to the case in which a source of contaminants is present, decreased throughout the room when the period of the inlet velocity was shortened. For SIM1, the time-averaged velocity and the vorticity field did not show large changes compared to the simulations with a higher period SIM2 and SIM3 (Figures 4.12 and 4.16), but an increase in the turbulent kinetic energy was clearly present (see Figure 4.14). This increase is considered to be the main reason for the decrease in the overall concentration level in the simulations with a shorter period since a higher turbulence level enhances the spread of the fresh air from the near-wall region towards other parts in the room. The turbulence level is mainly augmented in the area of the large recirculation cell, pointing out the higher velocity gradients present in this region in the simulations with the shorter period, which leads to higher amounts of shear in the flow and hence to more production of turbulent kinetic energy (see the production term in the $k$-equation, Equation 2.18 in Section 2.3.1).
Figure 4.31: Steady-state simulations in analogy to the reference case but with other (constant) inlet velocities: (a) $u_{0, RC} = 1.5$ m/s, (b) $u_{0, RC} = 0.5$ m/s, (c) $u_{0, RC} = 0.1$ m/s and (d) $u_{0, RC} = 0.01$ m/s. Two vertical lines are drawn which serve as a reference frame. The colours of the pathlines correspond to the mean velocity magnitude $U$.

In the case the amplitude of the inlet velocity is larger but the period is unchanged (SIM4), the inlet velocity increases much faster over time compared to the simulation with a smaller amplitude (SIM1). This leads to much more shear in the flow domain, especially in the shear layer and in the recirculation zone, and hence to much more turbulent kinetic energy (see Figure 4.14) which is beneficial for the mixing of fresh air with room air. Another difference between SIM4 and SIM1 is the presence of the second recirculation cell located under the inlet slot which also has an influence on the spread of fresh air through the room, since it gradually transports fresh air towards the middle of the room as it detaches from the left wall. All these observations cause SIM4 to have a lower time-averaged concentration field compared to SIM1.

4.3.3 Influence of the period and amplitude on the decay from an initial concentration

In the explanation of Figure 4.25 it was noted that most monitoring points in the fluid domain show a concentration value for SIM2 which is in between the concentration value of SIM1 and SIM3 after ventilating the room for a time $t = \tau_n$ and $t = 2\tau_n$ (see Figure 4.26). In other words, for most of the points the initial concentration has decayed more when the period of the inlet velocity is shorter. It is believed that this is also due to the higher turbulence levels on average for the simulations with a shorter period, which results in a better spread of the fresh supplied air and hence a better decay. Additional investigations are necessary to determine why in certain monitoring points (see Figure 4.26) the observation of a better decay for the simulation with a shorter period does not hold, despite the increase in turbulence. The decay of SIM4 is also assumed to be better
according to the above mentioned reasons: higher shear levels and the second vortex that causes a better spread of the fresh air.
5 Future work

The simulation results presented in Section 4 are obtained by solving the RANS equations in which the turbulence is completely modelled by a turbulence model, in this case the RNG $k - \epsilon$ model. Future work will investigate the flow field using the Large Eddy Simulation (LES) method which has the advantage that it solves part of the turbulence instead of modelling all turbulent scales and hence LES has proved to be more accurate than the RANS method [Nieuwstadt 2008]. In addition, instantaneous values of the velocity field and the turbulence levels can be obtained, in contrast to the RANS equations where only mean quantities are calculated. The detailed turbulent flow information that is available in the LES prediction helps to accurately predict the contaminant distribution inside the ventilated room [Chen 2009]. Another approach to verify the presented simulation results is to perform experiments.

The numerical calculations of the contaminant distribution inside the ventilated room have shown that the ventilation performance is worse when the flow field is subjected to a constant supply rate compared to the ventilation method in which the supply rate periodically varies around this constant value. Hence, in order to obtain the same ventilation performance as in the time-varying method, the constant supply rate should be augmented. Future research should focus on the energy consumption of time-periodic forcing compared to forcing with a constant supply rate and to what extend this increase in supply rate has an impact on the energy consumption for ventilation and air-conditioning.

The predictions also showed that the contaminant concentration has the highest concentration levels near the left wall since this region is reached last by the supplied air. A possibility could be the use of a supply channel in which the direction of the inlet jet is allowed to vary in such a way that the fresh air is directly supplied to the area near the left wall. The change in direction can vary as a function of time, with the inlet velocity decreasing as the angle between the inlet jet and the ceiling becomes larger in order to prevent that the jet penetrates with high velocities into the occupied zone. Another option is to provide the room with an extra inlet slot placed in the right upper corner. In this way the air supplied by the two inlets will collide in the middle along the ceiling, thereby moving downwards and eventually forming two recirculation cells in each half of the room [Kandzia et al. 2011], [Schmidt et al. 2013]. Also in this case varying supply rates can be used and in a controlled fashion between both inlets in order to obtain a well mixed flow field. The time-periodic forcing should not necessarily be sinusoidal. Also pulsatile forcing in which the supply rate changes abruptly over time is worth to try and possibly leads to much more shear and turbulence inside the enclosure.

Future work should focus on the connection between time-dependent forcing and chaotic mixing. 2D stationary flows do not allow for any chaotic behaviour at all whereas in 2D time-periodic flows chaotic advection might be introduced. This can be examined by placing two tracer particles initially very close to each other at several locations in the enclosure and by determining their trajectories as a function of time. An exponential
increase of their distance is an indication of chaotic behaviour. However, chaotic mixing in such time-periodic 2D flows is typically limited by transport barriers that separate regions of ordered and chaotic mixing (Solomon and Mezić 2003). This does not hold for time-dependent 3D flows in which completely uniform mixing is possible. Hence, it is certainly worth to perform 3D simulations in the future in order to resemble reality as close as possible. This will also give an idea about the flow characteristics, e.g. 3D flow structures, and the contaminant distribution in the whole room.

Other possible future work can be the investigation of the room ventilation under non-isothermal conditions by using for example a difference in temperature between the inlet air and room air (Cao et al. 2014). Also placing heat sources in the room to represent occupants is an option. Furthermore, draught in the occupied zone should be avoided in order to ensure thermal comfort of the occupants. Therefore, the effect of the changing velocities throughout the room, which are a result of the varying inlet velocity, on thermal comfort should be examined in the future.
6 Conclusions

The report presents a detailed analysis of the mixing in a rectangular shaped enclosure when subjected to time-dependent forcing and isothermal conditions. The influence of both the period and the amplitude of the inlet velocity on the flow and on the removal of indoor contaminants are examined in the vertical midplane of the enclosure by using 2D CFD simulations conducted with the RNG $k-\epsilon$ turbulence model. Three different periods ($T_1/\tau_n = 0.2$, $T_2/\tau_n = 0.5$ and $T_3/\tau_n = 1$) and two different amplitudes ($\Delta u_0/u_{0,RC} = 0.5$ and $\Delta u_0/u_{0,RC} = 1$) are tested.

In the case of a change in the period of the inlet velocity, with the amplitude held constant, the following conclusions can be made:

- The time-averaged velocity magnitude ($U/u_{0,RC}$) in the enclosure weakly increases for smaller periods. The change is the largest near the floor. The velocities near the left wall and in the corners remain relatively low.
- The time-averaged turbulent kinetic energy ($\sqrt{k}/u_{0,RC}$) increases for smaller periods, mainly in the area of the large recirculation cell. Furthermore this area extends towards the left wall when smaller periods are used.
- The time-averaged vorticity ($\omega_z$) does not show considerable changes.
- The time-averaged contaminant concentrations ($c/S_c$) in the case that a source of contaminants is present in the enclosure decreases for smaller periods. The area near the left wall and in particular in the lower left corner is characterised by the highest concentration levels compared to other parts in the enclosure.
- The decay from an initial contaminant concentration (no source present) shows fluctuations in the concentration levels ($c/c_i$) which are the largest in the wall jet area (ceiling, right wall and floor). The observation that smaller periods of the inlet velocity result in a better decay (lower concentration values) after ventilating for a time $t/\tau_n = 1$ and $t/\tau_n = 2$ holds in most of the room.

In the case that for the simulation with the shortest period, the amplitude of the inlet velocity is increased and the period is kept unchanged, the following conclusion can be made:

- The time-averaged velocity field increases considerably when the amplitude is augmented, certainly in the near-wall areas, and on average the penetration depth of the wall jet is larger. The velocity near the left wall shows a slight increase whereas in the corner regions the velocity does not seem to be changed much.
- A larger amplitude of the inlet velocity results in a strong increase in the time-averaged turbulent kinetic energy, certainly in the zone of the large recirculation cell. Also close to the left wall, the turbulent kinetic energy levels are increased.
- The time-averaged vorticity shows an increase in the area of the large recirculation cell in the simulation with a higher amplitude of the inlet velocity.

- There is a strong decrease in the time-averaged contaminant concentration levels throughout the whole room (with the presence of a source) in the case that a larger amplitude is used. The area close to the left wall is still characterised by higher contaminant concentrations compared to other parts in the enclosure.

- Also the decay over time from an initial concentration is better in the simulation with a higher amplitude. All the fifteen monitoring points which are uniformly distributed throughout the room show lower concentration values, after ventilating the room for a time equal to \( t/\tau_n = 1 \) and \( t/\tau_n = 2 \), in the case that a higher amplitude of the inlet velocity is used.

A comparison of the flow characteristics between the simulation with the shortest period and the simulation performed under steady inlet conditions shows that the (time-averaged) velocity, vorticity and turbulent kinetic energy all increase in the latter case. The time-averaged contaminant concentration values throughout the enclosure and the decayed concentration levels at \( t/\tau_n = 1 \) and \( t/\tau_n = 2 \) are considerably lowered when time-varying inlet conditions are employed. The results even get better if a higher amplitude of the inlet velocity is used. From this it can be concluded that it is more efficient to supply fresh air using time-dependent conditions.
References


A Details of the computational grids and solver settings

The details of the constructed grids and the different solver settings used for the 2D steady-state simulations are summarised in this section. Table A.1 gives an overview of the number of grid cells used for all grids in every region of the flow domain as indicated in Figure A.1. The cells are arranged on the edges according to the bi-exponent formulation in Gambit 2.4.6. In this way, sudden changes between neighbouring cells are avoided and a good quality mesh is obtained. The quality is checked via the size change measure in Gambit, which is calculated as:

\[
\text{size change} = \max \left( \frac{s_c}{s_n} \right).
\]  

(A.1)

\( s_c \) represents the size of a particular cell and \( s_n \) runs over the size of all its neighbours. The maximum ratio is then defined to be the size change of that particular cell. It is aimed to keep this value below 1.4.

Table A.1: Number of cells on the edges of grid1 to grid5 in both the x-direction (horizontal edge) and y-direction (inlet height, middle part, outlet height) (see Figure A.1), together with the maximum cell size change.

<table>
<thead>
<tr>
<th>Number of cells</th>
<th>grid1</th>
<th>grid2</th>
<th>grid3</th>
<th>grid4</th>
<th>grid5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total</td>
<td>5,244</td>
<td>10,400</td>
<td>20,792</td>
<td>41,600</td>
<td>83,536</td>
</tr>
<tr>
<td>Inlet height</td>
<td>25</td>
<td>45</td>
<td>63</td>
<td>83</td>
<td>101</td>
</tr>
<tr>
<td>Middle part</td>
<td>39</td>
<td>45</td>
<td>63</td>
<td>89</td>
<td>139</td>
</tr>
<tr>
<td>Outlet height</td>
<td>28</td>
<td>40</td>
<td>58</td>
<td>88</td>
<td>128</td>
</tr>
<tr>
<td>Horizontal edge</td>
<td>57</td>
<td>80</td>
<td>113</td>
<td>160</td>
<td>227</td>
</tr>
<tr>
<td>Grid quality</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Maximum size change</td>
<td>1.33</td>
<td>1.4</td>
<td>1.22</td>
<td>1.14</td>
<td>1.09</td>
</tr>
</tbody>
</table>

Figure A.1: Visualisation of the different flow regions for which the number of cells are summarised in Table A.1.
The grids are suitable for low-Reynolds-number modelling since their $y^*$-values are all much below 5, and on average even much below 1. Table A.2 gives the minimum, maximum and mean $y^*$-values along the domain boundary for the flow solutions obtained with the three turbulence models used in this study: the RNG $k - \epsilon$ model, the LRN $k - \epsilon$ model of Chang et al. (1995) and the SST $k - \omega$ model. The maximum $y^*$-values are detected at the upper horizontal wall very close to the inlet whereas the minimum values appear near the left wall in the lower left corner of the fluid domain.

**Table A.2:** Minimum, maximum and mean $y^*$-values for the five constructed grids, according to the three turbulence models used in this study.

<table>
<thead>
<tr>
<th>$y^*$</th>
<th>grid1</th>
<th>grid2</th>
<th>grid3</th>
<th>grid4</th>
<th>grid5</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>RNG $k - \epsilon$</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>○ Minimum</td>
<td>$1.95 \times 10^{-3}$</td>
<td>$3.25 \times 10^{-4}$</td>
<td>$2.74 \times 10^{-4}$</td>
<td>$3.26 \times 10^{-4}$</td>
<td>-</td>
</tr>
<tr>
<td>○ Maximum</td>
<td>4.44</td>
<td>2.85</td>
<td>2.42</td>
<td>3.04</td>
<td>-</td>
</tr>
<tr>
<td>○ Mean</td>
<td>0.63</td>
<td>0.27</td>
<td>0.25</td>
<td>0.29</td>
<td>-</td>
</tr>
<tr>
<td><strong>LRN $k - \epsilon$</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>○ Minimum</td>
<td>$2.38 \times 10^{-4}$</td>
<td>$1.80 \times 10^{-5}$</td>
<td>$1.27 \times 10^{-5}$</td>
<td>$1.35 \times 10^{-5}$</td>
<td>-</td>
</tr>
<tr>
<td>○ Maximum</td>
<td>1.53</td>
<td>0.20</td>
<td>0.18</td>
<td>0.23</td>
<td>-</td>
</tr>
<tr>
<td>○ Mean</td>
<td>0.15</td>
<td>0.024</td>
<td>0.024</td>
<td>0.027</td>
<td>-</td>
</tr>
<tr>
<td><strong>SST $k - \omega$</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>○ Minimum</td>
<td>$1.64 \times 10^{-3}$</td>
<td>$3.26 \times 10^{-4}$</td>
<td>$3.96 \times 10^{-4}$</td>
<td>$4.74 \times 10^{-4}$</td>
<td>$4.69 \times 10^{-4}$</td>
</tr>
<tr>
<td>○ Maximum</td>
<td>4.27</td>
<td>2.83</td>
<td>2.44</td>
<td>3.04</td>
<td>2.98</td>
</tr>
<tr>
<td>○ Mean</td>
<td>0.62</td>
<td>0.27</td>
<td>0.24</td>
<td>0.28</td>
<td>0.27</td>
</tr>
</tbody>
</table>

Table A.3 summarises the utilised solver settings and some convergence related parameters for the steady-state simulations with the three turbulence models and for all grids. For most simulations, the SIMPLEC algorithm is chosen for the pressure-velocity coupling, except when convergence problems are encountered and a switch to the SIMPLE scheme or the coupled pressure-based solver has to be made. If still no converged solution could be obtained, the spatial discretisation scheme for the pressure, which is initially always chosen to be second order, has to be changed to the standard scheme. In particular for the SST $k - \omega$ model this adjustment is necessary. The other solver settings are left the same for all simulations: least square cell based method for the calculation of the gradients and the second-order upwind scheme for the other variables (momentum, $k$, $\epsilon$ and $\omega$). To check convergence, the scaled residuals as well as the velocity field in certain points are monitored. The table shows the range of residual values when the solution is considered to be converged (i.e. when the velocity monitors indicate that the steady state is reached). All residual values are well below the suggested value of $10^{-3}$ (ANSYS, 2013). Finally, Table A.3 also lists the ratio of the net mass flow to the inlet mass flow to investigate the overall conservation of mass. The ratio ranges from 0% to $10^{-3}$% which is certainly low enough according to the recommended upper limit of $10^{-1}$% (ANSYS, 2013).
Table A.3: Solver settings used in the steady-state simulations. The pressure-velocity coupling, the spatial discretisation of the pressure, the range of the residuals and the ratio of the net mass flow to the inlet mass flow are represented for every grid and turbulence model.

<table>
<thead>
<tr>
<th>Pressure-velocity coupling</th>
<th>Spatial discretisation pressure</th>
<th>Range residuals</th>
<th>Net mass flow Inlet mass flow (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>RNG $k - \epsilon$</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>o Grid1</td>
<td>Coupled</td>
<td>Second order</td>
<td>$10^{-13} - 10^{-15}$</td>
</tr>
<tr>
<td>o Grid2</td>
<td>SIMPLEx</td>
<td>Standard</td>
<td>$10^{-12} - 10^{-13}$</td>
</tr>
<tr>
<td>o Grid3</td>
<td>SIMPLEx</td>
<td>Second order</td>
<td>$10^{-9} - 10^{-11}$</td>
</tr>
<tr>
<td>o Grid4</td>
<td>SIMPLEx</td>
<td>Second order</td>
<td>$10^{-10} - 10^{-11}$</td>
</tr>
<tr>
<td><strong>LRN $k - \epsilon$</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>o Grid1</td>
<td>SIMPLEx</td>
<td>Second order</td>
<td>$10^{-14} - 10^{-16}$</td>
</tr>
<tr>
<td>o Grid2</td>
<td>SIMPLEx</td>
<td>Standard</td>
<td>$10^{-9} - 10^{-10}$</td>
</tr>
<tr>
<td>o Grid3</td>
<td>SIMPLEx</td>
<td>Second order</td>
<td>$10^{-8} - 10^{-10}$</td>
</tr>
<tr>
<td>o Grid4</td>
<td>SIMPLEx</td>
<td>Second order</td>
<td>$10^{-8} - 10^{-6}$</td>
</tr>
<tr>
<td><strong>SST $k - \omega$</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>o Grid1</td>
<td>Coupled</td>
<td>Standard</td>
<td>$10^{-4} - 10^{-6}$</td>
</tr>
<tr>
<td>o Grid2</td>
<td>SIMPLEx</td>
<td>Standard</td>
<td>$10^{-4} - 10^{-7}$</td>
</tr>
<tr>
<td>o Grid3</td>
<td>SIMPLEx</td>
<td>Standard</td>
<td>$10^{-8} - 10^{-9}$</td>
</tr>
<tr>
<td>o Grid4</td>
<td>SIMPLEx</td>
<td>Second order</td>
<td>$10^{-8} - 10^{-9}$</td>
</tr>
<tr>
<td>o Grid5</td>
<td>SIMPLE</td>
<td>Standard</td>
<td>$10^{-9} - 10^{-11}$</td>
</tr>
</tbody>
</table>
B Details about the corner vortices predicted by the turbulence models

All three turbulence models used in this study predict vortices in the lower left and upper right corner of the room. A more thorough examination shows that the corner vortices are in fact a sequence of cells with alternating vorticity, referred to as ‘Moffatt eddies’ [Moffatt 1964]. Figure B.1 depicts a pathline plot of the lower left corner (obtained with the RNG model) in which three particles are released, tracing different eddies. The red coloured cell corresponds to the corner vortex observed in the pathline plots of Figure 3.21. The zoom-in reveals two other cells indicated with the green and blue colour.

\[\text{Figure B.1: Zoom-in on the lower left corner of the room to visualise the sequence of circulation cells. Three eddies are visualised (red, green and blue coloured).}\]

Moffatt eddies can appear in a Stokes flow for which the Reynolds number is much smaller than one. Such flow conditions are likely to occur in the corner of the room where the viscosity is assumed to have a dominating effect. In van Heijst (2014) the derivation of the solution of the stream function is represented in short and it is stated that, besides a Stokes flow in a corner region, the apex angle of the corner should be smaller than 146.3° to observe Moffatt eddies. The driving force of the sequence of eddies is a larger-scale recirculation zone outside the corner and the cells become weaker when moving further inward into the corner region.
C  Contour plots of the non-dimensional turbulent kinetic energy for the three turbulence models

To give a better idea about the distribution of \( \sqrt{k/1.1u_0} \) predicted by the three turbulence models along the four lines, contour plots of this quantity are created and depicted here in Figure C.1 overlayed with a few pathlines to visualise the flow field. As already mentioned, the highest levels of turbulence are produced in the shear layer below the wall jet. It is observed that the spread of this shear layer seems to increase faster for the RNG model compared to the other turbulence models. A possible cause of this phenomenon is the prediction of strong shear effects by the RNG model immediately below the inlet slot, leading to higher turbulence values in this area (red colour) and in the nearby regions of the shear layer. Hence, resulting in a larger spread. Other regions of higher turbulence values are the centre of the recirculation cell and the outlet slot. Nearby the left wall, \( \sqrt{k/1.1u_0} \) is the lowest for all three the models compared to the other parts of the room. The SST turbulence model calculates a low level of turbulence over the whole second circulation cell.

Figure C.1: Contour plots of the non-dimensional mean turbulent kinetic energy \( \sqrt{k/1.1u_0} \) for (a) the RNG \( k-\epsilon \) model, (b) the LRN \( k-\epsilon \) model and (c) the SST \( k-\omega \) model.
Additional investigations on the IEA Annex 20 room

Some additional steady-state simulations were performed to investigate the influence of certain parameters. The effect of an inlet and outlet geometry on the flow field is examined since the measurements of Nielsen (1990) were also performed with such a configuration. Also the impact of a change in turbulence parameters at the inlet is verified in order to better match the shape of the turbulent kinetic energy profiles with the measured turbulence data. All simulations are performed with the RNG $k - \epsilon$ model.

Effect of an inlet and outlet geometry on the flow field

A 2D configuration with a straight inlet and outlet geometry as depicted in Figure D.1 is constructed to do additional simulations. The inlet has a length of one meter and the outlet measures 30 centimetres. The room is descretised with grid2 and for the inlet and outlet geometry a sufficiently fine mesh is used. The boundary conditions are the same as described in Section 3.2.2. For the pressure-velocity coupling the SIMPLEC scheme is used and second-order discretisation schemes are chosen for the other variables. The solution was assumed to be converged from the point that the velocity at any position in the room remained unchanged and the scaled residuals have decreased to a satisfactory level ($10^{-9}$). Furthermore, the $y^*$-values at the walls are in the correct range: the minimum value is $2.01 \cdot 10^{-4}$, the maximum equals 2.52 and for the mean an outcome of 0.314 is calculated.

Figure D.1: 2D configuration with inlet and outlet geometry.

An inlet and outlet geometry is more realistic and the measurements of Nielsen (1990) are carried out in such a configuration (see Figure 3.1). Therefore it is expected that these simulation results match the experimental data more closely. A comparison of the velocity profiles is depicted in Figure D.2. Only at the inlet slot ($x/H = 0$) along the line $y = 0.972H$ a rise in velocity is noticed which is a consequence of the no-slip condition together with the maintenance of a constant flow rate as already explained in Section 3.4.2. The simulated profile becomes more accurate and it coincides almost perfectly with the measurements for $x/H < 1$. For the turbulence profiles, depicted in
Figure D.3, the impact is also modest. A small increase in the turbulence peak near the ceiling at \( x = H \) and an overall slight increment of the turbulence level at \( x = 2H \) are observed. In the outlet region at \( y = 0.028H \), the simulated profile is somewhat levelled of and hence in better agreement with the measurements. A longer outlet could possibly yield more improvement in this region. Near the inlet slot along the line \( y = 0.972H \), the largest rise in turbulent kinetic energy is noticed and the simulation profile approaches the measured data more closely. In summary it can be stated that an inlet and outlet channel do have a small positive effect on the simulations, but their influence is mainly limited to the nearby regions of the inlet and outlet slot.

**Influence of a change in turbulent boundary conditions at inlet**

A remarkable feature is observed in Figure D.3 along the line \( y = 0.972H \) at the inlet slot (\( x/H = 0 \)). The predicted turbulence values at this position correspond to the boundary condition \( k = k_0 = 4.97 \times 10^{-4} \text{ m}^2/\text{s}^2 \), according to Nielsen (1990). However, the predictions show a large deviation from the experimental measurements. In addition, the simulations with an inlet and outlet channel are expected to be better than these without, but it is observed that the opposite occurs at \( x/H = 0 \). Apparently, the boundary condition of 4% turbulence intensity at the inlet is not correct. Therefore, other turbulence intensities are used at the inlet to investigate the impact on the flow field. The solutions are obtained with the same discretisation schemes as mentioned above and also the \( y^* \)-values are checked to be in the correct range.

The influence of four different turbulence intensities at the inlet is examined: 2%, 4%, 6% and 7.5%. The other boundary conditions are unchanged. Figure D.4 shows the results along the line \( y = 0.972H \) where it is observed that for a higher turbulence intensity the measurements are better matched. However, only in a small distance from the inlet this effect is observed. This means that the production and dissipation of turbulence in the shear layer are dominant and hence determine the level of turbulent kinetic energy so that the effect of the different inlet values is hardly noticed further away from the inlet (Skovgaard and Nielsen 1991). The profiles along the other lines in the flow domain show no variation with a changing turbulence intensity.
Figure D.2: Comparison of the experimental non-dimensional mean $x$-velocity data (Nielsen, 1990) (○) with the simulation results obtained in a configuration with inlet and outlet geometry (red) and without an inlet and outlet channel (blue) along the four lines.

Figure D.3: Comparison of the experimental non-dimensional mean turbulent kinetic energy data (Nielsen, 1990) (△) with the simulation results obtained in a configuration with inlet and outlet geometry (red) and without an inlet and outlet channel (blue) along the four lines.
**Figure D.4:** Comparison of simulation results corresponding to different values of the turbulence intensity (TI). Four simulations are performed with turbulence intensities of 2% (blue), 4% (bordeaux), 6% (black) and 7.5% (green). The computational domain used is depicted in Figure D.4.