Turbulence Model Validation for Fire Simulation by CFD and Experimental Investigation of a Hot Jet in Crossflow

A dissertation submitted to the
Swiss Federal Institute of Technology, Zurich
for the degree of
Doctor of Sciences

Presented by
Daniel Bernhard Rusch
Dipl. Masch.-Ing. ETH, Zurich
born October 14, 1977
citizen of
Appenzell, AI

Accepted on the recommendation of
Prof. Dr. T. Rösgen, examiner
Prof. Dr. E. Lang, co-examiner
Dr. A. Moser, co-examiner

2007
To my parents
Acknowledgements

The present work was carried out at the Institute of Fluid Dynamics and in the research group Air&Climate of the ETH Zurich. The financial support provided by the European Cost Action C17 and by the Swiss State Secretariat for Education and Research (SER) is gratefully acknowledged.

I would like to thank Prof. Dr. T. Rösgen of the Institute of Fluid Dynamics (IFD), ETH Zurich, for acting as examiner and for supporting the development of my thesis work with his scientific advice and by discussing technical problems. I thank Prof. Dr. E. Lang, professor at the Zurich University of Applied Sciences Winterthur, for his critical review of my work and for being co-examiner of my thesis. I am also indebted to Prof. Dr. L. Kleiser and Prof. Dr. P. Jenny for many fruitful discussions about turbulence modeling.

Special thanks I owe to Dr. A. Moser, head of Air&Climate, for supervising this project, for the fruitful discussions, for the careful proof-reading and for his preparedness to handle every request at any time during this project.

Furthermore, I would like to thank Prof. Dr. D. Poulikakos for lending out the LDA measurement system, R. Holliger for his help in careful design and construction of the experimental setup, P. Stachel for solving every electronic problem, H. Caprez for the IT support, B. Maspero for her secretarial support, W. Müller for the carefully prepared drawings, M. Fehlmann for his assistance during the build-up phase, and especially Dr. L. Blum for his co-work during the measurements. With his profound understanding of the LDA technique, he could always give scientific advice in any LDA concerns. I was also deeply impressed by his skills during the experimental work.

I then would like to include the members of the Air Flow Consulting Company (AFC), Zurich, Dr. A. Schälin, Dr. D. Gubler, Dr. C. Del Taglia, Dr. S. Barp, Dr. P. Lengweiler, Dr. S. Moosberger and H. Hörtl into my acknowledgements. Their technical advice in the field of CFD and friendly support made a large contribution to my thesis. During the whole project they had an open door for me which I strongly appreciate.

I am also indebted to Dr. Y. Egorov and Dr. R. Meier-Staude of ANSYS Germany for letting me run the SAS-case on their cluster and for the friendly support they always gave me.

Besides all members of the Air&Climate group and the Institute of Fluid Dynamics, I would also like to thank my family. Special thanks I owe to my parents who enabled me to study at ETH and gave me such a wonderful support during the whole time at ETH. Furthermore I would like to thank my sisters Denise and Caroline, her husband Albert and my girlfriend Esther Müller and her family for the mental support during this work.

Zurich, December 2006 Daniel Rusch
Abstract

With the increase in computational power and the advances in Computational Fluid Dynamics (CFD) in recent years, it became possible to calculate temperature distributions, smoke spread, and velocity fields for fire scenarios in complex geometries. The CFD technique is becoming a standard tool to provide evidence and certification of fire safety for buildings and tunnels. Both structural considerations as well as the available time for evacuation can be investigated. In addition, emergency ventilation schemes and fire fighting strategies can be tested without the requirement for expensive experiments.

This thesis investigates the influence of the turbulence model choice on the solution of fire scenario simulations. Therefore a generic test case, a hot jet in a confined crossflow configuration, is considered. This configuration is similar to a fire in a tunnel or a long corridor with natural or forced ventilation. Having no chemical reactions, no combustion model has to be applied, and the influence of the turbulence model can be examined separately.

A new experimental setup has been designed and built to obtain experimental data. It consists of a 10m long tunnel with a cross-section of 0.8m × 0.8m which is longitudinally ventilated. The heat source has a diameter of 0.2m and consists of a blower and an electric heater which delivers hot air at about 500°C. The modular and tiltable tunnel has insulated walls and an insulated ceiling. The floor is made of glass which allows for optical access. A two-component Laser Doppler Anemometry system, LDA, is employed for velocity measurements, and thermocouples are used to acquire temperature profiles. Besides the validation measurements in the downstream region of the jet, the inflow boundary conditions of the computational domain are also obtained from measurements. Ordinary cooking salt crystals are successfully employed as seeding material for both the hot and the cold flow. Standard ultrasonic atomizers are used for the seeding particle production at high volume-flow-rate. The seeding salt crystal size distribution has been measured for various salt concentrations and an analytical model is discussed. It shows that the seeding size distribution can be controlled through the salt concentration and tuned for a given experimental requirement.

Two sets of numerical simulations have been performed: Steady computations on a grid with non-resolved walls and transient simulations on a grid with resolved walls. The steady simulations all fail in predicting the flow field accurately and the results depend strongly on the choice of the turbulence model. The ceiling temperatures are higher compared to the transient simulations due to the missing entrainment of the vortical structures near the jet. The transient simulations resolve the vortical structures in the jet vicinity. The transient Shear-Stress Transport (SST) model only resolves the dominant modes of the vortical structures leading to a slightly improved result compared to the steady one. The Detached Eddy Simulation (DES) shows more resolved structures but the applied grid is not fine enough and the model switches back into the SST mode in a large portion of the domain. On the same grid, the new Scale-Adaptive Simulation (SAS) shows very good results for both the velocity and the temperature resolving vortical structures down to a small scale. Only close to the jet discrepancies are still significant. It is the only model which can accurately capture the shedding frequency of the wake vortices and reproduce the backflow at the ceiling close to the hot jet source.
Kurzfassung


Diese Arbeit untersucht, welchen Einfluss die Wahl des Turbulenzmodells auf die Lösung der CFD Berechnung eines Brandszenarios hat. Dazu wird der generische Testfall eines Heissluftstrahls in einer räumlich eingeschränkten Queranströmung untersucht. Diese Konfiguration gleicht dem Szenario eines Brandes in einem Korridor oder in einem Tunnel mit erzwungener oder natürlicher Längslüftung. In dieser Konfiguration treten keine zu modellierenden Verbrennungsprozesse auf, was eine gezielte Untersuchung des TurbulenzmodellEinflusses erlaubt.

Um die für die Validierung notwendigen Messdaten zu erhalten, wurde eigens eine neue Anlage entwickelt und aufgebaut. Sie besteht aus einem mehr als 10m langen, längsgelüfteten Kanal, der einen Querschnitt von 0.8m mal 0.8m aufweist. Eine Wärmequelle, bestehend aus einem Gebläse und einem elektrischen Heizer, liefert 500°C heisse Luft und bläst sie über ein Rohr mit einem Durchmesser von 0.2m senkrecht von unten in den Kanal ein. Die Wände und die Decke des modular aufgebauten und neigbaren Kanals sind isoliert. Die Bodenplatten sind aus Glas gefertigt und erlauben den Einsatz eines Laser Doppler Messverfahrens (LDA) zur berührungslosen Bestimmung der Strömungsgeschwindigkeiten. Die Temperaturmessungen erfolgen mit Thermoelementen. Nebst den Messungen stromab des Heissluftstrahls, die der Validierung dienen, werden auch die zur numerischen Simulation notwendigen Eingangsbedingungen gemessen. Für die LDA Messungen werden sowohl die Längsströmung als auch der Heissluftstrahl mit Salzkristallen beschickt, welche mit einem gewöhnlichen Ultraschallzerstäuber in einer Salzwasserlösung durch Zerstäubungs- und Verdampfungsprozesse erzeugt werden. In einem Vorversuch wurden die Kristallgrössen für verschiedene Salzkonzentrationen der Lösung gemessen. Es zeigt sich, dass die Kristallgrösse über die Salzkonzentration gezielt beeinflusst und auf das jeweilige Experiment abgestimmt werden kann.

Bei den CFD Simulationen werden zwei unterschiedliche Gitter verwendet: Das eine löst die Wandgrenzschichten nicht auf, was zur Konvergenz der stationären Simulationen führt. Das zweite hingegen löst die Wandgrenzschichten auf. Auf diesem Gitter konvergieren die stationären Simulationen nicht mehr, so dass transiente Simulationen durchgeführt werden müssen. Die stationären Simulationen auf dem Gitter ohne aufgelösten Grenzschichten liefern keine befriedigenden Ergebnisse. Sie hängen zudem stark von der Wahl des Turbulenzmodells ab. Die transienten Wirbelstrukturen in der Nähe des Heissluftstrahls werden unterdrückt, was zu einer geringeren Einmischung kalter Luft in den Heissluftstrahl und somit zu höheren Deckentem-
peraturen, im Vergleich zu den transienten Rechnungen, führt. In den transienten Simulationen auf dem Gitter mit aufgelösten Grenzschichten werden diese Strukturen wiedergegeben. Bei der transienten SST Simulation (Shear-Stress Transport) werden nur dominante Strukturen aufgelöst, was zu einer geringen Verbesserung der Lösung im Vergleich zur stationären SST Rechnung führt. Die DES Simulation (Detached Eddy Simulation) liefert detailliertere Strukturen, arbeitet aber auf dem gewählten Gitter in vielen Bereichen im SST Modus. Auf demselben Gitter vermag das SAS Modell (Scale-Adaptive Simulation) beinahe überall im Wirbel auflösenden Modus zu arbeiten und liefert vor allem im Nachlauf sehr gute Ergebnisse. Es ist das einzige Modell, das auf dem gewählten Gitter sowohl die Rückströmung an der Decke in der Nähe des Heissluftstrahls als auch die Wirbelablösefrequenz im Nachlauf des Strahls adäquat wiedergeben kann.
Contents

1 Introduction 1
  1.1 Consequences of fires in buildings ..................... 1
  1.2 Numerical prediction of the flow field in a fire scenario .... 2
  1.3 Validation of fire modeling ................................ 3
  1.4 Aim of this work ........................................ 4

2 Governing equations and turbulence modeling 5
  2.1 Assumptions and conventions ................................ 5
  2.2 Governing instantaneous equations .......................... 6
  2.3 RANS turbulence models .................................... 7
    2.3.1 Reynolds averaging process ............................. 7
    2.3.2 Reynolds-averaged equations ............................ 7
    2.3.3 Eddy viscosity turbulence models ...................... 8
    2.3.4 Reynolds-stress turbulence models .................... 11
  2.4 DES turbulence model ..................................... 12
  2.5 SAS turbulence model ..................................... 13
  2.6 Boundary conditions ...................................... 15
    2.6.1 Wall treatment ....................................... 15
    2.6.2 Turbulent kinetic energy at inlets .................... 16
    2.6.3 Dissipation rate at inlets ............................ 17
  2.7 Discretisation ............................................ 19
    2.7.1 Spatial discretisation ................................. 19
    2.7.2 Temporal discretisation ............................... 19

3 Experiment 21
  3.1 Experimental setup ....................................... 21
    3.1.1 Tunnel geometry ..................................... 21
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1.2</td>
<td>Hot air source</td>
<td>24</td>
</tr>
<tr>
<td>3.1.3</td>
<td>Operational control</td>
<td>25</td>
</tr>
<tr>
<td>3.2</td>
<td>Operating point</td>
<td>26</td>
</tr>
<tr>
<td>3.3</td>
<td>Measurement techniques</td>
<td>28</td>
</tr>
<tr>
<td>3.3.1</td>
<td>Thermocouples</td>
<td>28</td>
</tr>
<tr>
<td>3.3.2</td>
<td>LDA system</td>
<td>30</td>
</tr>
<tr>
<td>3.3.3</td>
<td>Seeding</td>
<td>32</td>
</tr>
<tr>
<td>3.4</td>
<td>Measurements</td>
<td>39</td>
</tr>
<tr>
<td>3.4.1</td>
<td>Hot air source</td>
<td>41</td>
</tr>
<tr>
<td>3.4.2</td>
<td>Crossflow inlet</td>
<td>44</td>
</tr>
<tr>
<td>3.4.3</td>
<td>Downstream measurements</td>
<td>47</td>
</tr>
<tr>
<td>3.4.4</td>
<td>Characteristic parameters</td>
<td>57</td>
</tr>
<tr>
<td>3.4.5</td>
<td>Discussion</td>
<td>58</td>
</tr>
<tr>
<td>4</td>
<td>Numerical simulations</td>
<td>59</td>
</tr>
<tr>
<td>4.1</td>
<td>Boundary conditions</td>
<td>59</td>
</tr>
<tr>
<td>4.2</td>
<td>Grid independence study</td>
<td>60</td>
</tr>
<tr>
<td>4.3</td>
<td>Influence of $P_{kb}$</td>
<td>65</td>
</tr>
<tr>
<td>4.4</td>
<td>Steady RANS simulations</td>
<td>73</td>
</tr>
<tr>
<td>4.5</td>
<td>Transient simulations</td>
<td>81</td>
</tr>
<tr>
<td>4.5.1</td>
<td>Time-averaged quantities</td>
<td>82</td>
</tr>
<tr>
<td>4.5.2</td>
<td>Instantaneous quantities</td>
<td>91</td>
</tr>
<tr>
<td>5</td>
<td>Conclusions</td>
<td>101</td>
</tr>
<tr>
<td>5.1</td>
<td>Experiment</td>
<td>101</td>
</tr>
<tr>
<td>5.2</td>
<td>Numerical simulations</td>
<td>102</td>
</tr>
<tr>
<td>5.3</td>
<td>Outlook</td>
<td>103</td>
</tr>
<tr>
<td>A</td>
<td>Downstream measurements</td>
<td>105</td>
</tr>
<tr>
<td>A.1</td>
<td>LDA measurements</td>
<td>105</td>
</tr>
<tr>
<td>A.2</td>
<td>Temperature measurements</td>
<td>108</td>
</tr>
</tbody>
</table>

Bibliography | 111 |
List of Figures

3.1 Experimental setup. ............................................. 22
3.2 Air intake with and without screens. In Figure 3.2(b), the seeding tubes and the velocity anemometer for the operation control are visible. 23
3.3 Photograph and schematic view of the hot air source. .................. 25
3.4 Schematic view of the operational control. .......................... 26
3.5 Flame height correlations. ............................................ 27
3.6 Schematic view of the tunnel cross-section and the thermocouple rake. At the right upper corner, one enlarged view of the thermocouple probe tip is shown. .................................................. 29
3.7 LDA probe in operation mounted on a traverse. The deflecting front surface mirror is attached at the upper side of the visible black optical support plate. The glass floor plates in the upper part of the picture allow to see inside the tunnel. ............................. 31
3.8 Absorption rate of aerosol particles in the human airway system. . . 33
3.9 Ultrasonic atomizer, mounted on the floating ring, in operation and assembled in a barrel. ............................................. 34
3.10 Properties of sodium chloride aqueous solution and crystal edge length to droplet diameter ratio as a function of $f_m$. .................. 35
3.11 Measured cumulative volumetric distribution function, $Q_3$, and the corresponding probability density function, $q_3$, of the sodium chloride crystal particle size for different concentrations $f_m$ and produced by an ultrasonic atomizer (triangular symbols). In addition, one measurement using a Laskin nozzle is given (circles). ...................... 36
3.12 Microscopy image of sodium chloride crystals produced with the ultrasonic atomizer. The indicated length represents 10µm. .......... 37
3.13 Measurement locations. ............................................. 40
3.14 Measured profiles of axial velocity, circumferential velocity, r.m.s. of the axial velocity and r.m.s. of the circumferential velocity at different angular positions within the hot jet source at $z = -0.3m$. ....... 41
3.15 Measured profiles of turbulence intensity and temperature at different angular positions within the hot air source at $z = -0.3m$. ......... 42
3.16 Computed mean value function $b_1(r_i)$ and amplitude function $b_2(r_i)$ of the applied temperature distribution model $T(r_i)$ given by Eq. (3.13). 43
3.17 Interpolated temperature distribution in the hot air source at $z = -0.3m$. 43
3.18 Circumferential average of the measured axial velocity and of the turbulence intensity at $z = -0.3m$. 44
3.19 Measured $U$ velocity profiles at two different upstream locations. 45
3.20 Measured profiles of $U$ and $V$ velocity, r.m.s. values, and turbulence intensity (based on the $u$ component) at $x = -1.25m$ and $z = 0.25m$. 46
3.21 Measured boundary layer profiles at the upstream position $x = -1.25m$ and a zoom-in. 47
3.22 Overview of the $U$ velocity component of the LDA measurements in the symmetry plane $y = 0m$. Zero values at the walls are inserted for reference. 48
3.23 Velocity profiles at downstream positions. 49
3.24 (a) Measured $V$ component at $x = 1m$ with a data rate higher than 1Hz. (b) Measured spectrum of the $v$ velocity component at $x = 1m$, $y = 0m$ and $z = 0.1m$. The dominant frequency is 1.1Hz. 50
3.25 Measured dimensionless temperature distribution at $x = 0.5m$ made symmetric by mirroring and averaging. 51
3.26 Measured dimensionless temperature distribution made symmetric by mirroring and averaging. 52
3.27 Measured r.m.s. temperature distribution in the cross-section $x = 0.5m$ applying different filter widths. The corresponding cut-off frequencies are: (a) $f_{co} = 1Hz$, (b) $f_{co} = 0.2Hz$, (c) $f_{co} = 0.1Hz$, (d) $f_{co} = 0.05Hz$. Data is made symmetric by mirroring and averaging. 53
3.28 Measured temperature signal at $(x, y, z) = (0.5m, 0m, 0.4m)$. The corresponding low- and high-pass filtered signals are cropped according to the filter width for the given cut-off frequency $f_{co} = 0.22Hz$. In the auto-correlation of the mean-free original signal is given. The corresponding integral time scale is 2.5s. 54
3.29 Measured r.m.s. temperature distribution at $x = 0.5m$. Data is made symmetric by mirroring and averaging. Cut-off frequency: $f_{co} = 0.22Hz$. 55
3.30 Measured r.m.s. temperature distribution. Data is made symmetric by mirroring and averaging. Cut-off frequency: $f_{co} = 0.22Hz$. 56
4.1 Computational domain. 59
4.2 Medium grid with non-resolved walls. 61
List of Figures

4.3 Medium grid with resolved walls. .............................................. 62
4.4 Temperature profiles at downstream positions in the symmetry plane $y = 0\text{m}$ computed with the SST PD model using different grid resolutions and the automatic wall function. .............................................. 63
4.5 Velocity $U$ profiles at downstream positions in the symmetry plane $y = 0\text{m}$ computed with the SST PD model using different grid resolutions and the automatic wall function. .............................................. 64
4.6 Schematic view of the plume within the symmetry plane and the density gradient distribution. .............................................. 65
4.7 Velocity $U$ profiles at downstream positions in the symmetry plane $y = 0\text{m}$ computed with the SST, SST P and SST PD model. .............................................. 67
4.8 Temperature profiles at downstream positions in the symmetry plane $y = 0\text{m}$ computed with the SST, SST P and SST PD model. .............................................. 68
4.9 Turbulent kinetic energy $k$ profiles at downstream positions in the symmetry plane $y = 0\text{m}$ computed with the SST, SST P and SST PD model. .............................................. 69
4.10 Turbulent eddy dissipation $\varepsilon$ profiles at downstream positions in the symmetry plane $y = 0\text{m}$ computed with the SST, SST P and SST PD model. .............................................. 70
4.11 Dimensionless eddy viscosity profiles at downstream positions in the symmetry plane $y = 0\text{m}$ computed with the SST, SST P and SST PD model. .............................................. 71
4.12 In-plane streamlines of the SST PD model at downstream locations colored with the $x$ component of the local vorticity vector which is $\left(\frac{\partial W}{\partial y} - \frac{\partial V}{\partial z}\right) \text{[s}^{-1}\text{]. The view is in negative $x$ direction.} .............................................. 72
4.13 Dimensionless temperature profiles within the plane $x = 0.5\text{m}$ computed with different turbulence models. .............................................. 74
4.14 Dimensionless temperature profiles at downstream positions in the symmetry plane $y = 0\text{m}$ computed with different turbulence models. .............................................. 75
4.15 Dimensionless temperature profiles at downstream positions in the plane $y = 0.2\text{m}$ computed with different turbulence models. .............................................. 76
4.16 Velocity $U$ profiles at downstream positions in the symmetry plane $y = 0\text{m}$ computed with different turbulence models. .............................................. 77
4.17 Velocity $U$ profiles at downstream positions in the plane $y = 0.2\text{m}$ computed with different turbulence models. .............................................. 78
4.18 Dimensionless temperature distribution of the experiment and different turbulence model predictions at the $x$ locations from $x = 0.5\text{m}$ to $x = 6.5\text{m}$ with a spacing of $1\text{m}$. .............................................. 79
4.19 Dimensionless temperature distribution of the RSM PD model at the $x$ locations from $x = 0.5\text{m}$ to $x = 6.5\text{m}$ with a spacing of $1\text{m}$. .............................................. 80
4.20 Dimensionless ceiling temperature distribution of the steady simulations on the grid with unresolved walls. A different scale is used for the $k$-$\varepsilon$ PD model. ......................................................... 80

4.21 Dimensionless ceiling temperature distribution at $y = 0$m of the simulations on the grid with unresolved walls. ........................................... 81

4.22 Time-averaged dimensionless temperature distributions of the experiment and different turbulence model predictions at the $x$ locations from $x = 0.5$m to $x = 6.5$m with a spacing of 1m. ........................................... 84

4.23 Time-averaged dimensionless temperature distributions at $x = 0.5$m. ........................................... 85

4.24 Time-averaged dimensionless temperature profiles within the plane $x = 0.5$m computed with different turbulence models. ............................. 86

4.25 Time-averaged dimensionless temperature profiles at downstream positions in the symmetry plane $y = 0$m computed with different turbulence models. .......................................................... 87

4.26 Time-averaged dimensionless temperature profiles at downstream positions in the plane $y = 0.2$m computed with different turbulence models. .......................................................... 88

4.27 Time-averaged velocity $U$ profiles at downstream positions in the symmetry plane $y = 0$m computed with different turbulence models. ............................. 89

4.28 Time-averaged velocity $U$ profiles at downstream positions in the plane $y = 0.2$m computed with different turbulence models. ............................. 90

4.29 Time-averaged dimensionless ceiling temperature distribution of the transient simulations on the grid with resolved walls. ............................. 91

4.30 Time-averaged dimensionless ceiling temperature distribution at $y = 0$m of the transient simulations on the grid with resolved walls. ............................. 91

4.31 Schematic view of the vortical structures in a jet-in-crossflow configuration as reported by Fric and Roshko (1994). ........................................... 92

4.32 Visualization of the vortical structures by rendering the iso-surfaces of the invariant at $I_v = 75$s$^{-2}$. The iso-surfaces are colored with the instantaneous dimensionless temperature. ........................................... 94

4.33 Blending function between the RANS mode (=1) and the scale-resolved mode (=0) within the symmetry plane $y = 0$m. ........................................... 95

4.34 Scatter plots of the dimensionless temperature and the $V$ velocity component at two different $z$ locations on the line $x = 1$m and $y = 0$m predicted by the DES PD model. ........................................... 97

4.35 Scatter plots and power spectra of the dimensionless temperature and $V$ velocity component at three downstream positions within the symmetry plane, $y = 0$m, at $z = 0.7$m for the SST PD, the DES PD and the SAS models. ........................................... 99
A.1 Measured V velocity and $\sqrt{vv}$ profiles at $x = 1m$. Only measurements with a data rate higher than 1Hz are given. ................................ 105
A.2 Velocity U profiles at downstream positions. ................................ 106
A.3 Measured $\sqrt{uu}$ profiles at downstream positions. ......................... 107
A.4 Measured dimensionless temperature distribution at $x = 0.5m$ and the corresponding r.m.s. temperature distribution at $x = 0.5m$. Cut-off frequency: $f_{co} = 0.22$Hz. ........................................ 108
A.5 Measured dimensionless temperature distribution. ................................. 109
A.6 Measured r.m.s. temperature distribution. Cut-off frequency: $f_{co} = 0.22$Hz. ............................................................... 110
List of Tables

1.1 Limiting values for a healthy person to withstand a fire for about 30 minutes without damage as reported by VdS2827 (2000). 1

2.1 Coefficients of the specific heat capacity polynomial, following Moran and Shapiro (2000). 6

2.2 Constants of the standard $k$-$\varepsilon$ model. 10

2.3 Constants of the $k$-$\omega$ model. 11

2.4 Constants of the SAS model. 14

3.1 Thermal properties of the applied insulation materials as reported by the manufacturers. 24

3.2 Technical data of the applied optics. 30

3.3 Flash point and auto-ignition temperature at the pressure of 1bar for various seeding oils. 32

3.4 Median of the particle size and the measured and theoretical crystal size ratio for different concentrations assuming that the generated solution droplet size does not vary with $f_m$. 36

3.5 Characteristic time, cut-off frequency, and settling velocity for the measured median particle diameters at both 20°C and 500°C. 38

3.6 Boundary layer properties: Displacement thickness $\delta^*$, momentum thickness $\delta$ and shape factor $H$. 47

4.1 Mesh sizes (number of hexahedra). 61

4.2 Dominant frequencies, rounded to one decimal digit, at $(x, y, z) = (0.5\text{m}, 0\text{m}, 0.1\text{m})$ and at $(x, y, z) = (1\text{m}, 0\text{m}, 0.1\text{m})$ for the temperature, $T$, and the crossflow velocity component, $V$. 96
## Nomenclature

### Latin symbols

- $a_1, a_2, a_3, a_4, a_5$: Coefficients of the specific heat capacity polynomial
- $a_{awf}$: Constant of the automatic wall function formulation
- $A_{cf}$: Cross-sectional area of the crossflow tunnel
- $A_j$: Cross-sectional area of the hot jet
- $b_1$: Mean value function, Eq. (3.13)
- $b_2$: Amplitude function, Eq. (3.13)
- $C$: Constant of the log law of the wall
- $C_p$: Specific heat capacity
- $C_{\varepsilon 1}, C_{\varepsilon 2}$: Constants of the $k-\varepsilon$ model
- $C_{\mu}$: Turbulence model constant
- $D$: Hot air source diameter
- $D_p$: Particle diameter
- $D_s$: Salt solution droplet diameter
- $f$: Frequency
- $f_{co}$: Cut-off frequency
- $f_m$: Mass concentration
- $f_{sr}$: Sampling rate
- $g, g_i$: Gravitational acceleration and component in $x_i$ direction
- $g'$: Reduced gravitational acceleration, $= g \frac{\rho_{cf} - \rho_j}{\frac{1}{2}(\rho_{cf} + \rho_j)}$
- $H$: Specific enthalpy
- $H$: Shape factor of the boundary layer
- $I$: Turbulence intensity
- $I_v$: Second invariant of the velocity gradient tensor
- $k$: Turbulent kinetic energy, $= \frac{u_i^2}{2}$
- $k_{M/4}$: Filter kernel
- $L$: Length scale
- $\Delta L$: Local grid size
- $L_c$: Characteristic length
- $L_{cr}$: Salt crystal cube edge length
- $L_{int}$: Integral length scale
- $L_{\varepsilon K}$: Von Karman length scale
- $\dot{m}_{cf}$: Crossflow mass flow rate
- $\dot{m}_{excCV}$: Mass flow rate exiting the control volume
- $m_{H_2O}, m_{NaCl}$: Mass of water and mass of sodium chloride
- $\dot{m}_j$: Jet mass flow rate
Nomenclature

\( N \) Number of experiments

\( P \) Difference of \( P_{stat} \) and hydrostatic reference pressure

\( P_{stat} \) Local static pressure

\( P' \) Modified pressure, \( = P + \frac{2}{3}\rho k \)

\( P_k \) Turbulence production term

\( P_{kb} \) Turbulence production term due to buoyancy

\( Pr_t \) Turbulent Prandtl number

\( q_3 \) Volumetric probability density function

\( Q_3 \) Cumulative volumetric distribution function

\( \dot{Q} \) Heat release rate

\( r \) Radial position

\( r_i, r_j, r_k \) Distance vector

\( R \) Gas constant for air

\( R(\tau) \) Autocorrelation function

\( s \) Relative slip velocity

\( S \) Sutherland’s constant

\( S \) Shear strain rate

\( S_{ij} \) Strain rate tensor

\( t, \Delta t \) Time and time interval

\( T, T_{min}, T_{max}, T_{ref} \) Temperature and minimum, maximum and reference value

\( T_{co} \) Cut-off period, \( = 1/f_{co} \)

\( T^{M'4} \) High-pass filtered temperature signal using \( k_{M'4} \)

\( T_{rms}^{M'4} \) r.m.s. value of the \( T^{M'4} \) signal

\( U \) Velocity parallel to the wall (wall functions)

\( U_c \) Characteristic velocity

\( U_{cf} \) Crossflow velocity

\( U_f \) Fluid velocity

\( U_i, U_j, U_k \) Velocity components in the directions \( i, j \) and \( k \)

\( U, V, W \) Velocity components in the directions \( x, y \) and \( z \)

\( U_p \) Particle velocity

\( U_{pr} \) Propagation velocity

\( U_s \) Settling velocity

\( U_r \) Friction velocity

\( U^+ \) Non-dimensional velocity parallel to the wall

\( U^* \) Velocity scale for wall functions

\( U_x \) Velocity component in circumferential direction

\( \overline{u_i u_i} \) Turbulent heat flux

\( -\overline{u_i u_j}, \overline{u_i u_i} \) Turbulent shear stress and turbulent normal stress

\( V_{CV} \) Volume of the control volume

\( W_j \) Area-averaged jet velocity

\( x \) Coordinate in longitudinal direction

\( x_i, x_j, x_k \) Coordinates in the directions \( i, j \) and \( k \)

\( y \) Coordinate in transversal direction

\( y^+ \) Non-dimensional wall distance

\( y^* \) Wall distance (wall functions and SAS model)

\( z \) Coordinate in vertical direction
Greek symbols

\(\alpha\) Constant of the \(k-\omega\) model
\(\beta, \beta'\) Constants of the \(k-\omega\) model
\(\beta_{as}\) Blending function of the advection scheme
\(\Gamma_t\) Eddy diffusivity
\(\delta\) Boundary layer momentum thickness
\(\delta^*\) Boundary layer displacement thickness
\(\delta_{ij}\) Kronecker delta
\(\Delta\) Filter half width
\(\varepsilon\) Dissipation rate
\(\zeta_1, \zeta_2, \zeta_3\) Model constants of the SAS model
\(\kappa\) Von Karman constant
\(\lambda\) Thermal conductivity
\(\lambda\) Wave length
\(\mu, \mu_{eff}, \mu_{ref}\) Dynamic viscosity, effective and reference value
\(\mu_t\) Dynamic eddy viscosity
\(\nu\) Kinematic viscosity, \(= \mu/\rho\)
\(\nu_t\) Kinematic eddy viscosity
\(\rho\) Density
\(\rho_{ef}\) Density of the crossflow
\(\rho_s\) Density of the sodium chloride crystal
\(\rho_f\) Fluid density
\(\rho_j\) Volume-flow-averaged density of the hot air jet
\(\rho_p\) Particle density
\(\sigma_k, \sigma_\varepsilon, \sigma_\Phi\) Turbulence model constants of the \(k, \varepsilon\) and \(\Phi\) equations
\(\sigma_p\) Model constant of the \(P_{kb}\) term
\(\tau\) Time scale
\(\tau\) Time shift (auto-correlation)
\(\tau_0\) Characteristic time
\(\tau_{int}\) Integral time scale
\(\tau_w\) Wall shear stress
\(\varphi\) Angular coordinate of the polar coordinate system
\(\phi\) Turbulent fluctuation of scalar \(\Phi, = \tilde{\Phi} - \Phi\)
\(\Phi\) Reynolds-averaged value of scalar \(\Phi, = \overline{\Phi}\)
\(\Phi\) SAS model variable, \(= \sqrt{kL}\)
\(\omega\) Vorticity scale, \(= \varepsilon/(C_\mu k)\)
\(\Omega\) Absolute value of the vorticity vector

Subscripts

\(\Phi_{awf}\) Value referring to the automatic wall function
\(\Phi_{avg}\) Spatially averaged value
Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Phi_c$</td>
<td>Characteristic value</td>
</tr>
<tr>
<td>$\Phi_{cf}$</td>
<td>Value referring to the crossflow</td>
</tr>
<tr>
<td>$\Phi_{cr}$</td>
<td>Value referring to salt crystals</td>
</tr>
<tr>
<td>$\Phi_{CV}$</td>
<td>Control volume value</td>
</tr>
<tr>
<td>$\Phi_f$</td>
<td>Fluid value</td>
</tr>
<tr>
<td>$\Phi_{ip}$</td>
<td>Value at the integration point</td>
</tr>
<tr>
<td>$\Phi_j$</td>
<td>Jet value</td>
</tr>
<tr>
<td>$\Phi_p$</td>
<td>Particle value</td>
</tr>
<tr>
<td>$\Phi_s$</td>
<td>Solution value</td>
</tr>
<tr>
<td>$\Phi_{sc}$</td>
<td>Value referring to the scalable wall function</td>
</tr>
<tr>
<td>$\Phi_{up}$</td>
<td>Value at the upwind node</td>
</tr>
<tr>
<td>$\Phi_\infty$</td>
<td>Free-stream value</td>
</tr>
</tbody>
</table>

Superscripts

- $\Phi^+$: Made dimensionless with wall scales
- $\Phi$: Instantaneous value

Non-dimensional numbers

- $CFL$: Courant-Friedrichs-Lewy number, $= U_{pr} \Delta t / \Delta L$
- $Fr$: Froude number, $= \frac{U_{cf}}{\sqrt{g L_c}}$
- $J$: Mean jet-to-crossflow momentum flux ratio, $= \frac{\rho_j U^2_j}{\rho_{cf} U^2_{cf}}$
- $J^*$: Mean jet-to-crossflow momentum ratio, $= J \frac{A_j}{A_{cf}}$
- $Q^*$: Heat release rate, $= \frac{Q}{\rho_{cf} C_p T_{cf} \sqrt{g D B^2}}$
- $Re$: Reynolds number, $= U_{cf} L_c / \nu_{cf}$
- $St$: Strouhal number, $= f L_c / U_{cf}$

Abbreviations

- BSL: Baseline model
- CFD: Computational Fluid Dynamics
- DES: Detached Eddy Simulation
- DNS: Direct Numerical Simulation
- LDA: Laser Doppler Anemometry
- LES: Large Eddy Simulation
- GDP: Gross Domestic Product
- RANS: Reynolds-Averaged Navier-Stokes
- RSM: Reynolds-Stress Model
- SAS: Scale-Adaptive Simulation
- SGS: Sub-Grid Scale
- SST: Shear-Stress Transport
- URANS: Unsteady Reynolds-Averaged Navier-Stokes
Chapter 1

Introduction

1.1 Consequences of fires in buildings

Fires are not only a threat to human lives but also to buildings and cultural heritage. According to Moser (2003), one historic building is being lost each month in Scotland alone on average. As historic buildings are frequently visited by tourists, not only cultural heritage but also people are in danger.

In Switzerland, fires caused 30 deaths in 2002 as reported by Kellenberger (2003). Most fatalities occurred due to late detection of the fire. In the USA, more than 70% of the fire victims died due to the inhalation of toxic smoke gases (Hall and Harwood (1995) and Hall (1996)). Table 1.1 gives an overview of the thresholds a human can sustain for 30 minutes without injury.

Table 1.1: Limiting values for a healthy person to withstand a fire for about 30 minutes without damage as reported by VdS2827 (2000).

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Threshold</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air temperature</td>
<td>&lt; 65°C</td>
</tr>
<tr>
<td>CO concentration</td>
<td>&lt; 1400ppm</td>
</tr>
<tr>
<td>CO$_2$ concentration</td>
<td>&lt; 6 vol.-%</td>
</tr>
<tr>
<td>Oxygen</td>
<td>&gt; 12 vol.-%</td>
</tr>
<tr>
<td>Height of the smoke-free layer</td>
<td>&gt; 1.5m</td>
</tr>
<tr>
<td>View distance</td>
<td>&gt; 10m</td>
</tr>
</tbody>
</table>

In addition to the human consequences, the economic costs are considerable. Cox (1995) reports that direct property losses account for around 0.2% of the gross domestic product (GDP) of many nations each year. Together with the cost of the emergency services, fire protection provision in buildings, fire insurance administration and the subsequent losses in commerce, the total cost amounts to around 1% of GDP.

The European Cost Action C17 project *Built Heritage: Fire Loss to Historic Build-*
ings was initiated to acquire knowledge and to find solutions on how to protect historic buildings and people staying in them. Four working groups were formed:

- Data, loss and evaluating risks
- Available and developing technology
- Cultural and financial value
- Property management strategies

The present work is a contribution to the group dealing with available and developing technology.

1.2 Numerical prediction of the flow field in a fire scenario

The accurate prediction of smoke spread (toxic gases) and temperature distributions as well as of the velocity field is essential for the design of fire detection and protection measures. It is also important in providing evidence for the effectiveness of preventive measures and designed ventilation systems. For the design of buildings, fire protection standards have to be implemented. Especially for large public buildings such as train stations, tunnels, museums, concert halls and atria it is important that in case of emergency, smoke can be removed with the installed devices during the time required for the evacuation. Computational Fluid Dynamics (CFD) in combination with the knowledge of experienced fire protection engineers is a powerful tool to provide such evidence (VKF (2003), Schälin (2005), Gray et al. (2003)).

For simple geometries, zone models (Schneider (2002), Karlsson and Quintiere (2000)) are widely used in fire protection engineering, e.g. Chow (1995). Rooms are vertically divided into two zones, one smoke free and one full of smoke, for which the balance of mass, momentum and energy are formulated individually. These zones can be connected in a network for the case of a multi-room geometry. Having a coarse level of discretisation, these models cannot provide a very accurate description but the corresponding equations can be quickly solved and the results give a good first estimate of what is going on.

Dealing with more and more complex and larger geometries such as shopping malls, underground stations and atria, a need arose for a more general approach. With the increase in computational power, it became possible to discretize the geometries up to a high level of detail and solve the governing equations on those grids. Computational Fluid Dynamics (CFD), Versteeg and Malalasekera (1998), offers the availability of many advanced models. They offer a high level of flexibility to engineers and have become a widely accepted tool for fire protection engineering. As a matter of fact, the flows under consideration are of turbulent nature, Pope (2000). In Direct Numerical Simulation (DNS), all scales of turbulence are resolved, leading
1.3 Validation of fire modeling

As outlined by Casey and Wintergerste (2000), validation is unavoidable to ensure the quality of CFD predictions. Experimental data is required to compare the CFD model predictions to reality. During the last decades, validation of fire modeling has increased in importance. Large-scale experiments of fires in rooms were performed, such as given by Steckler et al. (1982) and Alvares et al. (1984). Multi-room geometries were considered by Luo and Beck (1994). Also fires in tunnels are under investigation, such as the memorial tunnel test, MemReport (1995). A variety of authors did CFD analyses of the experiments and compared the results, e.g. Björkman and Keski-Rahkonen (1996), Liu et al. (2004), McGrattan and Hamins (2003). A comprehensive overview of further validation work is given by Kidger et al. (2002) and Tieszen (2001). Both authors highlight the importance of additional experimental work in the field of fire modeling. In particular, measurements of additional flow properties are required along with temperature measurements as outlined by Kidger et al. (2002).

What is common to the cited studies is the fact that in the experiment, a real fire is used which has to be modeled in the simulations using combustion models or
volumetric heat sources. It is impossible to validate turbulence models with these measurements separately from the combustion model. The new test case developed in this work avoids this complication. In order to have clear boundary conditions, the plume is modeled by a hot gas jet, and a corridor or tunnel-like geometry was chosen with a forced crossflow. With this configuration, the major fluid structures such as a plume, a stably stratified smoke layer, and a crossflow are present. The crossflow appears in real fire cases due to forced or natural ventilation. Similar investigations were performed by Mégrét (1999), where a helium air gas mixture jet in crossflow was used. Heat conduction was simulated with the diffusion of helium species. The major investigations were done for different types of ventilation systems. As the maximum tunnel height was only 0.4 m and low velocities appeared, low Reynolds numbers were achieved. For the validation of turbulence models, high Reynolds numbers are preferable. Hence, it was decided to build a new, larger tunnel with an electrical heat source which makes the operation more safe.

1.4 Aim of this work – experimental measurements and numerical prediction of a buoyant jet in crossflow

The aim of this work is to validate turbulence models for the generic test case of a hot jet in a confined crossflow. This configuration includes the major fluid structures of a fire and has well defined boundary conditions. In addition, the fire heat release rate has not to be modeled which makes it an appropriate test case for turbulence modeling.

Many studies have been devoted to jets in crossflows. A comprehensive literature review is given in Plesniak and Cusano (2005). It is outlined that only little work has been conducted for the case of confined jets. The combination of hot jets and confined crossflows is even more rare in literature. An example are the experiments done by Kamotani and Gerber (1974). The operating points chosen by the authors do not fit the requirements for the validation of fires as the temperature difference is too small and the velocities are too large. Hence, momentum forces would dominate over the buoyancy forces by a large degree.

Finding no suitable measurements, it was decided to build a new experimental facility for this thesis where temperature and velocity distributions can be measured both at defined inlet cross-sections and within the flow field using thermocouples and laser Doppler velocimetry. The inlet boundary conditions are taken from the measurement in order to correctly reproduce the flow into the computational domain. Finally, the calculations, using different types of turbulence model, are compared with the measurements.
Chapter 2

Governing equations and turbulence modeling

In this chapter, the governing instantaneous mass, momentum, and energy equations are presented. The Direct Numerical Simulation (DNS) of fluid flow by these equations is out of reach for industrial flow applications due to the lack of available computational power. For simplification, averaged equations are solved. For these, however, turbulence models have to be used in order to model the effects of the unresolved scales. The models compared in this thesis are presented. In addition, the wall treatment and the discretisation scheme of the commercial solver applied (ANSYS CFX10) are given.

2.1 Assumptions and conventions

In this thesis, instantaneous values are denoted by capital letters with a tilde, e.g. $\tilde{\Phi}$, Reynolds-averaged values are denoted by capital letters, e.g. $\Phi$, and turbulent fluctuations are denoted by the corresponding lower-case characters, e.g. $\phi$. The summation convention applies to all equations. All investigations are based on the following assumptions:

- The fluid is an ideal gas:
  - Density is a function of temperature and static pressure and is computed by the ideal gas law: $\tilde{\rho} = \tilde{P}_{\text{stat}}/(R\tilde{T})$, where $R$ is the Gas constant of air. The Boussinesq approximation does not hold in the flow under consideration as discussed in Gray and Giorgini (1976).
  - Pressure variations are only small with respect to ambient pressure and their effect on the density is negligible (Mach number, $Ma$, near zero).

- Large temperature differences are considered:
  - The specific heat capacity, $C_p$, the dynamic viscosity, $\mu$, and the thermal conductivity, $\lambda$, are assumed to be functions of temperature only. The
specific heat capacity is expressed in polynomial form, Moran and Shapiro (2000):

\[
\frac{C_p}{R} = a_1 + a_2 T + a_3 T^2 + a_4 T^3 + a_5 T^4
\]  \hspace{1cm} (2.1)

The coefficients, \(a_i\), for air at \(T\) in the range of 300K to 1000K are as follows:

Table 2.1: Coefficients of the specific heat capacity polynomial, following Moran and Shapiro (2000).

<table>
<thead>
<tr>
<th>(a_1)</th>
<th>(a_2)</th>
<th>(a_3)</th>
<th>(a_4)</th>
<th>(a_5)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.653</td>
<td>-1.337 \cdot 10^{-3}K^{-1}</td>
<td>3.294 \cdot 10^{-6}K^{-2}</td>
<td>-1.913 \cdot 10^{-9}K^{-3}</td>
<td>0.2763 \cdot 10^{-12}K^{-4}</td>
</tr>
</tbody>
</table>

- The dynamic viscosity, \(\mu\), is described by the Sutherland Formula according to Wilcox (1997):

\[
\frac{\mu}{\mu_{ref}} = \left(\frac{T}{T_{ref}}\right) \left(\frac{T}{T_{ref}}\right)^{3/2},
\]  \hspace{1cm} (2.2)

with the reference temperature \(T_{ref} = 273.15K\), the reference viscosity \(\mu_{ref} = 1.716 \cdot 10^{-5} \text{kg m}^{-1} \text{s}^{-1}\) and the Sutherland constant \(S = 110.6 \text{K}\).

- For the thermal conductivity, \(\lambda\), Sutherland’s Formula of thermal conductivity is used, which is analogous to Eq. (2.2). The reference temperature is set to 273.15K, the reference thermal conductivity to 0.02414 W m\(^{-1}\) K\(^{-1}\) and the Sutherland constant to 194.4K, as reported by White (1974).

### 2.2 Governing instantaneous equations

With these assumptions and conventions, the instantaneous mass and momentum conservation can be written as Eqs. (2.3) and (2.4). Facing low velocities (Mach number, \(Ma\), near zero), the contribution of the kinetic energy, the viscous dissipation, the compressibility as well as of the external forces to the energy equation can be neglected. The simplified energy equation can be written as Eq. (2.5):

\[
\frac{\partial (\tilde{\rho} \tilde{U}_j)}{\partial t} + \frac{\partial (\tilde{\rho} \tilde{U}_i \tilde{U}_j)}{\partial x_j} = 0
\]  \hspace{1cm} (2.3)

\[
\frac{\partial (\tilde{\rho} \tilde{U}_i)}{\partial t} + \frac{\partial (\tilde{\rho} \tilde{U}_i \tilde{U}_j)}{\partial x_j} = -\frac{\partial \tilde{P}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial \tilde{U}_i}{\partial x_j} + \frac{\partial \tilde{U}_j}{\partial x_i} \right) \right] + g_i (\tilde{\rho} - \rho_{ref})
\]  \hspace{1cm} (2.4)

\[
\frac{\partial (\tilde{\rho} \tilde{H})}{\partial t} + \frac{\partial}{\partial x_j} (\tilde{\rho} \tilde{H} \tilde{U}_j) = \frac{\partial}{\partial x_j} \left( \lambda \frac{\partial \tilde{T}}{\partial x_j} \right)
\]  \hspace{1cm} (2.5)
where $\tilde{U}_i$ is the instantaneous velocity, $\tilde{H}$ the specific enthalpy and $g_i$ is the component of the gravitational acceleration in the coordinate direction $x_i$. $\tilde{P}$ is the difference of the local static pressure, $\tilde{P}_{stat}$, and the hydrostatic reference pressure, with $\partial \tilde{P} / \partial x_i = \partial \tilde{P}_{stat} / \partial x_i - \rho_{ref} g_i$. The momentum equation (2.4) is also called the Navier-Stokes equation.

2.3 RANS turbulence models

2.3.1 Reynolds averaging process

In turbulent flow fields, a wide range of time and length scales coexist. Due to the restricted computational power available, not all the scales can be resolved. In order to obtain a set of equations that can be solved numerically, the instantaneous governing transport equations (2.3), (2.4) and (2.5) are Reynolds averaged, Reynolds (1895). A general instantaneous turbulent flow variable $\tilde{\Phi}$ is split into a statistical mean value $\Phi = \overline{\Phi}$ and a stochastic part $\phi$, which fluctuates around the mean value and satisfies $\overline{\phi} = 0$. The averaging process is indicated by the overbar.

$$\tilde{\Phi} = \Phi + \phi, \quad \tilde{H} = H + h, \quad \tilde{U}_i = U_i + u_i, \quad \tilde{P} = P + p, \quad \tilde{T} = T + \theta \quad (2.6)$$

The main Reynolds averaging methods are:

Ensemble averaging If the flow is unsteady, time averaging cannot be used and must be replaced by true ensemble averaging. Ensemble averaging means that the values of the variable of interest at the same time and same location are averaged over all members of an ensemble. For instance, the measurements of $N$ experiments with the same initial and boundary conditions form an ensemble.

Time averaging In a statistically steady flow, the mean value can be obtained by averaging the instantaneous flow at the position over a time interval $\Delta t$, where $\Delta t$ has to be “large” compared to the time scale of the fluctuations.

2.3.2 Reynolds-averaged equations

The averaging process of the governing equations is explained in detail, e.g., by Mossel (1995) and Wilcox (1993). The resulting Reynolds-averaged Navier Stokes,
RANS, equations are:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho U_j)}{\partial x_j} = 0 \quad (2.7)
\]

\[
\frac{\partial (\rho U_i)}{\partial t} + \frac{\partial (\rho U_i U_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \rho u_i u_j \right] + g_i (\rho - \rho_{ref}) \quad (2.8)
\]

\[
\frac{\partial (\rho H)}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho H U_j \right) = \frac{\partial}{\partial x_j} \left[ \lambda \frac{\partial T}{\partial x_j} - \rho u_j h \right] \quad (2.9)
\]

Due to the averaging procedure, new unknown correlations show up in the equations (closure problem). The correlations of the turbulent fluctuations \(u_i u_j\) and \(u_j h\) represent the turbulent transport of momentum and thermal energy. The terms \(-\rho u_i u_j\) act as turbulent stresses and are called Reynolds stresses, while \(\rho u_j h\) act as turbulent heat fluxes. These terms have to be modeled with a so-called turbulence model.

### 2.3.3 Eddy viscosity turbulence models

The eddy viscosity hypothesis assumes that the Reynolds stresses can be related to the mean velocity gradients and a turbulent eddy viscosity \(\mu_t\) by the gradient diffusion hypothesis. In contrast to \(\mu\), the eddy viscosity, \(\mu_t\), is not a material property but rather a local function of the flow field. The hypothesis is in analogy to the relationship between the stress and strain tensors in laminar Newtonian flows. Neglecting the compressibility effects, it can be written as:

\[
-\rho u_i u_j = \frac{2}{3} \rho k \delta_{ij} + \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) = \frac{2}{3} \rho k \delta_{ij} + 2 \mu_t S_{ij} \quad (2.10)
\]

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right), \quad (2.11)
\]

where \(S_{ij}\) is the strain rate tensor. The turbulence kinetic energy, \(k\), is defined as:

\[
k = \frac{1}{2} u_i u_i \quad (2.12)
\]

and has to be added in Eq. (2.10) to avoid a contradiction to the continuity equation. The turbulent eddy viscosity, \(\mu_t\), has still to be modeled. The eddy diffusivity hypothesis states, that the Reynolds fluxes of a scalar, \(\phi\) are linearly related to the mean scalar gradient:

\[
-\rho u_j \phi = \Gamma_t \frac{\partial \phi}{\partial x_j} \quad (2.13)
\]

e.g.

\[
-\rho u_j h = \Gamma_t \frac{\partial H}{\partial x_j} \quad (2.14)
\]
Here, $\Gamma_t$ is the eddy diffusivity and can be written as:

$$\Gamma_t = \frac{\mu_t}{Pr_t},$$

(2.15)

where $Pr_t$ is the turbulent Prandtl number, which can be assumed to be constant for a variety of flows ($Pr_t = 0.9$). Using both the eddy viscosity as well as the eddy diffusivity hypothesis, the Reynolds-averaged equations become:

$$\frac{\partial (\rho U_i)}{\partial t} + \frac{\partial (\rho U_i U_j)}{\partial x_j} = -\frac{\partial P'}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu_{eff} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] + g_i (\rho - \rho_{ref})$$

(2.16)

$$\frac{\partial (\rho H)}{\partial t} + \frac{\partial}{\partial x_j} (\rho H U_j) = \frac{\partial}{\partial x_j} \left[ \lambda \frac{\partial T}{\partial x_j} + \frac{\mu_t}{Pr_t} \frac{\partial H}{\partial x_j} \right],$$

(2.17)

where the effective eddy viscosity is defined as $\mu_{eff} = \mu + \mu_t$ and the $P' = P + \frac{2}{3} \rho k$ is the modified pressure.

The eddy viscosity hypothesis given in Eq. (2.10) uses a scalar turbulent viscosity which forces the principal axis of $-u_i u_j$ to be aligned with the principal axis of the strain rate tensor $S_{ij}$, as outlined by Markatos (1986). Davidson (2004) states that in strongly anisotropic turbulence, such as in stratified flows, where the buoyancy force tends to suppress vertical fluctuations, and in flows with strong rotation, $\mu_t$ should be rather a tensor than a scalar, which leads to the Reynolds-stress transport equations discussed in Section 2.3.4. Due to the underlying assumptions, models based on the eddy viscosity hypothesis have deficiencies in predicting flows with inhomogeneous turbulence such as flows with strong streamline curvature or stratified flows. Nevertheless, they are widely used in industry as the computational requirements are reasonable and there is extended experience for many types of flows.

**k-\(\varepsilon\) model**

In the k-\(\varepsilon\) model, Rodi (1980) and Wilcox (1993), two additional transport equations for the turbulent kinetic energy, $k$, and for the dissipation rate, $\varepsilon = \nu \frac{\partial u_i}{\partial x_j} \frac{\partial u_i}{\partial x_j}$, are solved:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_i k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \varepsilon$$

(2.18)

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho U_i \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{k} \left( C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon \right)$$

(2.19)

The turbulent kinetic energy, $k$, is a measure for the kinetic energy stored in the unresolved velocity fluctuations and $\varepsilon$ is the dissipation rate of the turbulent kinetic energy into heat. With these two variables, a length scale $L$ and a time scale $\tau$ can be derived as discussed in Pope (2000):

$$L = \frac{k^{3/2}}{\varepsilon}$$

(2.20)

$$\tau = \frac{k}{\varepsilon}$$

(2.21)
Using dimensional considerations, the turbulent viscosity can be written as:

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \quad (2.22)$$

The turbulence production term, $P_k$, is modeled using:

$$P_k = \mu_t \frac{\partial U_i}{\partial x_j} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \left( 3 \mu_t \frac{\partial U_i}{\partial x_l} + \rho k \right) + P_{kb} \quad (2.23)$$

with

$$P_{kb} = -\frac{\mu_t}{\rho \sigma} g \frac{\partial \rho}{\partial x_i} \quad (2.24)$$

The term $P_{kb}$ describes the influence of buoyancy on the production and dissipation of turbulent kinetic energy. The model constant is given by $\sigma = 1$. CFX offers different possibilities to control buoyancy effects in turbulent quantities:

1. Without $P_{kb}$.
2. $P_{kb}$ included only in the $k$ equation.
3. $P_{kb}$ included both in the $k$ and $\varepsilon$ equation. In the $\varepsilon$ equation, $P_{kb}$ is only included if $P_{kb} > 0$ to increase numerical robustness.

The model constants of the standard $k$-$\varepsilon$ model are as follows:

<table>
<thead>
<tr>
<th>$C_\mu$</th>
<th>$C_{c1}$</th>
<th>$C_{c2}$</th>
<th>$\sigma_k$</th>
<th>$\sigma_\varepsilon$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.0</td>
<td>1.3</td>
</tr>
</tbody>
</table>

$k$-$\omega$ model

In the $k$-$\omega$ model the time and length scales required for the prediction of $\mu_t$ are obtained from the turbulent kinetic energy, $k$, and the turbulent frequency, $\omega$. For both quantities, a transport equation is solved. The present formulation of the $k$-$\omega$ model was developed by Wilcox (1986):

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \beta' \rho k \omega \quad (2.25)$$

$$\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho U_j \omega)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 \quad (2.26)$$

$$\mu_t = \rho \frac{k}{\omega} \quad (2.27)$$
In this formulation, the buoyancy production term given in Eq. (2.24) is only included, if \( P_{kb} > 0 \). In order to avoid the build-up of turbulent kinetic energy in stagnation regions, a limiter to the production term \( P_k \) is introduced according to Menter (1994). The model constants are as follows:

<table>
<thead>
<tr>
<th>( \alpha )</th>
<th>( \beta )</th>
<th>( \beta' )</th>
<th>( \sigma_k )</th>
<th>( \sigma_\omega )</th>
</tr>
</thead>
<tbody>
<tr>
<td>5/9</td>
<td>0.075</td>
<td>0.09</td>
<td>2</td>
<td>2</td>
</tr>
</tbody>
</table>

One of the advantages of the \( k-\omega \) model is the near-wall treatment for low-Reynolds number simulations. In contrast to the \( k-\varepsilon \) model, no complex non-linear damping functions are required. As a result, the \( k-\omega \) model is more accurate and more robust for near wall modeling. The low Reynolds number \( k-\omega \) model near wall resolution requirements are not that drastic as those for the \( k-\varepsilon \) model.

**SST model**

The Shear-Stress Transport model (SST) was developed by Menter (1994). It is a combination of the \( k-\varepsilon \) and the \( k-\omega \) model. The underlying idea is to combine the advantages of the \( \varepsilon \) and the \( \omega \) equation: As previously discussed, the \( k-\omega \) model is the model of choice in the sublayer of the boundary layer as no damping functions are required near walls and simple Dirichlet boundary conditions can be specified. The deficit of the \( k-\omega \) formulation is the strong freestream sensitivity to the specified \( \omega \) values in the freestream region of the upstream boundaries. In contrast, Menter (1992) shows that the \( k-\varepsilon \) model does not suffer from this deficiency. Hence, in the SST model, the \( \omega \) equation is solved in near wall regions and the \( \varepsilon \) equation in the outer part of the boundary layer. Blending functions are used to switch gradually between the models. A detailed description of the model is given by Menter (1994).

**2.3.4 Reynolds-stress turbulence models**

In contrast to eddy viscosity turbulence models, Reynolds-stress models are not based on the eddy viscosity hypothesis given in Eq. (2.10). Instead, a transport equation for each individual Reynolds-stress is solved. These equations can be derived by multiplying Eq. (2.4) once with \( u_i \) and once with \( u_j \), adding the resulting equations and subsequent Reynolds averaging as discussed by e.g. Wilcox (1993). New unknown triple correlations appear which still have to be modeled using either an \( \varepsilon \) or an \( \omega \) transport equation. The Baseline Reynolds-Stress Model (BSL-RSM) combines the \( \varepsilon \) and \( \omega \) transport equations similar to the SST model. Details of the model are given in ANSYS (2005).
2.4 DES turbulence model

The Detached Eddy Simulation (DES) is a hybrid Large Eddy (LES) and RANS simulation. The model applied in this thesis is described in detail by ANSYS (2005).

The basic idea of the Large Eddy Simulation (LES) is to resolve the largest, energy-containing eddies exactly and to model the effect of the non-resolved small-scale structures on the resolved flow field by a heuristic model as discussed by Davidson (2004). It makes use of the fact that energy and information is mostly transferred from large eddies to smaller ones (energy cascade), but not in the opposite direction. Therefore, the model of the small scales has just to dissipate the energy transferred from the resolved large scales.

In contrast to RANS models, LES models do not average the flow properties in time but in space using filter functions such as e.g. box or Gaussian filter functions. As a result, in large eddy simulations, the grid elements have to be small enough to resolve the energy-containing eddies and the simulation is intrinsically transient which makes it a computationally expensive model. This effect becomes even more important, if walls are present. Within the boundary layer, the scales to be resolved are smaller resulting in a dramatic increase in computational cost.

Common to all turbulence models, also the LES equations have to be closed by a suitable model for the Sub-Grid Scale (SGS) eddy viscosity. Such a model was provided by Smagorinsky in the 1960s. From dimensional considerations, a length and a velocity scale are needed. As the SGS viscosity models the effect of the non-resolved scales, the spatial filter width is used as the length scale. This length scale is typically of the order of the grid cell length. The velocity scale is derived from the strain rate tensor, $S_{ij}$, as defined in Eq. (2.11).

Compared to DNS, where also the energy dissipating scales have to be resolved, LES is much less costly (but much more expensive than RANS) as the effect of these small scales are modeled. In contrast to RANS models, LES models are able to accurately resolve temporal fluctuations and are preferable in flow situations where the large scales dominate e.g. mixing processes. Nevertheless, the LES model is at present still too demanding in the sense of computational requirements for most industrial flow applications.

In order to overcome the wall resolution problems of the LES model, the so-called Detached Eddy Simulation (DES) model approach has been developed. The idea is to combine RANS and LES models such that RANS equations are solved within the boundary layer and the LES equations are used in the free stream region to resolve large scale structures. CFX10 uses a combination of the SST and LES model. Special care has to be taken for the switching condition. In the DES formulation by Strelets (2001), the integral length scale (Eq. 2.20) is computed from the RANS model and compared with the local grid-cell size which is defined to be the maximum edge length in CFX10 to have a restrictive method. As soon as the computed integral length scale is larger as the grid length scale, the large-scale flow structures can be resolved with the applied grid and the LES model is activated. In the other case, the RANS model is used. This formulation suffers from grid-induced separation (Menter
2.5 SAS turbulence model

The Scale-Adaptive Simulation (SAS) model is proposed by Menter and Egorov (2004). It belongs to the group of two-equation models and is based on the eddy viscosity hypothesis, given in Eq. (2.10). The model is derived from the $k$-$kL$ model of Rotta (1972). Rotta derived an exact transport equation for an integral length scale, $L$, based on two-point correlations of the velocity fluctuations which provides a natural length scale of the flow. For the derivation of the $k$-$kL$ model, the Taylor series expansion of an exact term has been applied. Rotta neglected the second spatial derivative of the velocity as its coefficient is zero in homogenous turbulence. Hence the third spacial derivative of the velocity component has been found as the leading term in addition to the production term. In contrast, Menter and Egorov (2004) found that the assumption of homogeneous turbulence is too restrictive and should be avoided. Their formulation keeps the second derivative term and neglects the third derivative term as it is not intuitively clear why the third derivative should be more relevant than the second one in the determination of a natural length scale. In addition the third derivative is a tedious quantity to compute in a general-purpose CFD code. In their new $kL$ equation, the so-called von Karman length scale $L_{vK}$ (Schlichting (1982)) appears which is a further natural length scale of the flow.

Menter et al. (2006) propose both a one-equation and a two-equation SAS model. For the simulations presented within this thesis, the so-called $k$-$\sqrt{kL}$ two-equation formulation is used in combination with a viscous sublayer model which allows to integrate the model through the viscous sublayer down to the wall and avoids the need for wall functions. The transport equations for $k$ and $\Phi = \sqrt{kL}$ are as follows:

$$
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_j k)}{\partial x_j} = P_k - C_{\mu}^{3/4} \frac{k^{3/2}}{\rho L} + \frac{\partial}{\partial x_j} \left( \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right) 
$$

$$
\frac{\partial (\rho \Phi)}{\partial t} + \frac{\partial (\rho U_j \Phi)}{\partial x_j} = \frac{\Phi}{k} P_k \left[ \zeta_1 - \zeta_2 \left( \frac{L}{L_{vK}} \right)^2 \right] - \zeta_3 \rho k + \frac{\partial}{\partial x_j} \left( \frac{\mu_t}{\sigma_\Phi} \frac{\partial \Phi}{\partial x_j} \right)
$$

$$
\nu_t = \frac{\mu_t}{\rho} = C_{\mu}^{1/4} \Phi
$$

$$
P_k = \mu_t S^2 , \ S = \sqrt{2S_{ij} S_{ij}}
$$

The length scale ratio is limited to avoid overly large or small values:

$$
L_{vK} = \kappa \frac{|U'|}{|U''|} , \ |U'| = S , \ |U''| = \sqrt{U_i''U_i''} , \ U_i'' = \frac{\partial^2 U_i}{\partial x_j \partial x_j}
$$

$$
L/c_{t1} < L_{vK} < c_{t2} \kappa y , \ c_{t1} = 10 , \ c_{t2} = 1.3
$$
where \( y \) is the distance to the nearest surface. The model constants are given in Table 2.4:

<table>
<thead>
<tr>
<th></th>
<th>( C'_{\mu} )</th>
<th>( \kappa )</th>
<th>( \zeta_1 )</th>
<th>( \zeta_2 )</th>
<th>( \zeta_3 )</th>
<th>( \sigma_k )</th>
<th>( \sigma_\Phi )</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.09</td>
<td>0.41</td>
<td>0.8</td>
<td>1.47</td>
<td>0.0288</td>
<td>2/3</td>
<td>2/3</td>
</tr>
</tbody>
</table>

For the viscous sublayer model, the following terms are added to the right-hand sides of the \( k \) equation (Eq. (2.28)) and of the \( \Phi \) equation (Eq. (2.29)), respectively:

\[
VSM_k = -2\mu k y^2 \quad (2.34)
\]

\[
VSM_\Phi = -6\mu \Phi y^2 f_\Phi, \quad f_\Phi = \frac{1 + c_{d1} \xi}{1 + \xi^4}, \quad \xi = \sqrt{0.3ky^2} \quad 20\nu, \quad c_{d1} = 4.7 \quad (2.35)
\]

In order to increase the performance of the model in adverse pressure gradient flows, the eddy viscosity is modified according to:

\[
\nu_t = \min \left( \frac{C'^{1/4}_\mu \Phi}{S}, \frac{a_1 k}{S} \right) \quad (2.36)
\]

\[
a_1 = a_1^{SST} f_b + (1 - f_b) a_1^{REAL}, \quad a_1^{SST} = 0.32, \quad a_1^{REAL} = 0.577 \quad (2.37)
\]

\[
f_b = \tanh \left[ \frac{20 \left( C'^{1/4}_\mu \Phi + \nu \right)}{\kappa^2 S y^2 + 0.01\nu} \right]^{2}, \quad (2.38)
\]

where the blending function \( f_b \) is equal to one inside the boundary layer, leading to \( a_1 = a_1^{SST} \), and zero outside, leading to \( a_1 = a_1^{REAL} \). Inside the boundary layer, the eddy viscosity is reduced in regions of high adverse pressure gradients allowing for proper separation-line prediction, similar to the SST model formulation. In the free stream, the realizability constraint ensures the normal-stress components to be always positive avoiding the over-prediction of turbulence in stagnation regions.

The described SAS model is an inherent adaptive model: In regions, where the flow tends to be unstable, the von Karman length scale is reduced, which increases the length scale ratio \( L/L_{vK} \). As the length scale ratio appears in a sink term in Eq. (2.29), \( \Phi \) and also the eddy viscosity \( \nu_t \), which is directly proportional to \( \Phi \) (Eq. (2.30)), are reduced. The flow will become more unstable and hence transient in these regions and vortices down to the scale of the local grid size will be resolved resulting in a LES-like behavior. In stable flow regions, \( L_{vK} \) remains large which leads to high values for the eddy viscosity. In these areas, the model acts like a RANS model. Due to the model ability to resolve the turbulent spectrum it is termed a "scale-adaptive simulation" model. It has similarities to the DES model but has the advantage that it is not based on the local grid size and therefore avoids the grid sensitivity problems.
2.6 Boundary conditions

In order to solve the governing equations, proper boundary conditions have to be applied. Special care has to be employed at the walls and for the turbulent kinetic energy and dissipation rate at inlets as discussed in this section.

2.6.1 Wall treatment

Standard wall function

As discussed e.g. by Schlichting and Gersten (1997), a boundary layer contains the outer layer, where the turbulent portion of the shear stress is dominant and the viscous sublayer near the wall, where the laminar portion of the shear stress is dominant. The asymptotic solutions of the two layers match in the logarithmic layer. The buffer layer connects the logarithmic layer and the viscous sublayer. In the logarithmic layer, the relation between the non-dimensional wall distance \( y^+ \) and the local velocity parallel to the wall, \( U \), nondimensionalized with the friction velocity \( U_\tau \) can be written as:

\[
U^+ = \frac{U}{U_\tau} = \frac{1}{\kappa} \ln(y^+) + C \quad (2.39)
\]

\[
y^+ = \frac{\rho y U_\tau}{\mu} \quad (2.40)
\]

\[
U_\tau = \sqrt{\frac{\tau_w}{\rho}} \quad \text{or} \quad (2.41)
\]

\[
\tau_w = \rho U_\tau^2 \quad (2.42)
\]

were \( y \) is the distance in the wall normal direction, \( \kappa = 0.41 \) is the von Karman constant, \( \tau_w \) the wall shear stress and \( C \) a constant.

Scalable wall function

As outlined by Grotjans and Menter (1998), one of the major drawbacks of the standard wall function is that the prediction of \( U^+ \) depends on the location of the grid point nearest to the wall. Refining a grid or changing the velocity conditions can result in the \( y^+ \) value to lie inside the inertial sublayer or the buffer layer (\( y^+ < 11.06 \)). In this region, Eq. (2.39) does not hold and would provide inaccurate estimates of \( U^+ \). In order to avoid this, the \( y^+ \) value is forced to be larger than or equal to 11.06. A second problem of the standard wall function is that \( U^+ \) approaches zero at separation points. Therefore, an alternative velocity scale \( U^* \) is
introduced. The formulation for the scalable wall-function approach is:

\[ U^* = C_1^{1/4} k^{1/2} \quad (2.43) \]
\[ y^* = \max \left( \frac{\rho y U^*}{\mu}, 11.06 \right) \quad (2.44) \]
\[ U_{\tau,sc} = \frac{U}{\frac{1}{\kappa} \ln(y^*) + C} \quad (2.45) \]
\[ \tau_w = \rho U^* U_{\tau,sc} \quad (2.46) \]

As a result of the definition, the viscous sublayer is “shifted into the wall” if \( \frac{\rho y U^*}{\mu} < 11.06 \) and a slight error in mass flow can result.

**Automatic wall function**

The automatic wall function formulation developed by Vieser et al. (2002) is only available for \( \omega \) based turbulence models as it makes use of the analytic distribution of \( \omega \) near walls. It automatically blends between the viscous sublayer and the log-layer formulation and can therefore be also used for low Reynolds number flows with resolved walls \( (y^+ < 2) \). The analytical expressions for \( \omega \) in the viscous sublayer and in the log-layer region are given in Eq. (2.47). The constants are \( a_{awf} = 0.3 \) and \( \beta = 0.075 \). A smooth blending is used in Eq. (2.48) in order to determine the value of \( \omega \).

\[ \omega_{vis} = \frac{6\nu}{3y^2} ; \omega_{log} = \frac{U_{awf}^*}{a_{awf}k y} = \frac{1}{a_{awf}k y} \frac{U_{awf}^*}{y^+} \quad (2.47) \]
\[ \omega(y^+) = \left( \omega_{vis}^2 + \omega_{log}^2 \right)^{1/2} \quad (2.48) \]

In a similar way, the friction velocity is determined by:

\[ U_{\tau,vis} = \frac{U}{y^+} ; \quad U_{\tau,log} = \frac{U}{\frac{1}{\kappa} \ln(y^*) + C} \quad (2.49) \]
\[ U_{\tau,awf} = \left( U_{\tau,vis}^4 + U_{\tau,log}^4 \right)^{1/4} \quad (2.50) \]

The wall shear stress is computed as:

\[ U_{awf}^* = \max \left( \sqrt{a_{awf}k}, \sqrt{\frac{\nu}{y}} \frac{U}{y} \right) \quad (2.51) \]
\[ \tau_w = \rho U_{awf}^* U_{\tau,awf} \quad (2.52) \]

**2.6.2 Turbulent kinetic energy at inlets**

If not all three appearing correlations in Eq. (2.12) are measured, further assumptions have to be made such as that the third one is estimated to be the average of
the other two (Durbin and Pettersson Reif (2001)):

\[ u_3u_3 = \frac{1}{2}(u_1u_1 + u_2u_2) \]  
\[ k = \frac{3}{4}(u_1u_1 + u_2u_2) \]  

If only one component is available, local isotropy leads to:

\[ u_1u_1 = u_2u_2 = u_3u_3 \]  
\[ k = \frac{3}{2}u_1u_1 \]  

From \( k \), the turbulence intensity \( I \) can be obtained:

\[ I = \sqrt{\frac{2}{3}k} \sqrt{U_jU_j} \]  

It has to be stated that inside the boundary layer, the assumption of local isotropy is not valid.

### 2.6.3 Dissipation rate at inlets

The dissipation rate, \( \varepsilon \), is very hard to measure as it would be necessary to measure gradients of all three velocity components simultaneously. For practical reasons, this cannot be done and other ways to describe \( \varepsilon \) have to be considered:

- By specifying an integral length scale Eq. 2.20, ANSYS (2005):
  - Empirically (Casey and Wintergerste (2000)): For internal flows a constant value of length scale derived from a characteristic geometrical feature can be used. It is typically in the order of 1% to 10% of the hydraulic diameter.
  - From the autocorrelation function as discussed in Tennekes and Lumley (1972) and Borth (1995): The integral length scale \( L_{int} \) is computed with the help of the integral time scale \( \tau_{int} \) which is a measure of how much time it takes for a coherent structure to pass the measurement probe location. \( \tau_{int} \) is computed using the autocorrelation function \( R(\tau) \) of the fluctuation \( u \) of convection velocity \( \tilde{U} \).

\[ \tau_{int} = \int_0^\infty R(\tau) d\tau \]  
\[ R(\tau) = \frac{u(t)u(t+\tau)}{u(t)u(t)} \]  

According to Taylor’s hypothesis (frozen flow assumption), which assumes the flow to be isotropic and the mean convection velocity \( \tilde{U} \) to be much
larger than the disturbance velocity $u$, the integral length scale is obtained from:

$$L_{int} = \tau_{int} U \tag{2.60}$$

In the vicinity of walls, this method is not appropriate as the isotropy assumption does not hold. If the autocorrelation function is weakly damped, the integral in Eq. (2.58) can be even negative. In order to get a good estimate for $\tau_{int}$, the upper limit of the integral is set to the first zero crossing with the $\tau$ axis. The dissipation rate can then be computed from:

$$\varepsilon = \frac{k^{3/2}}{L_{int}} \tag{2.61}$$

- By making use of the mean flow velocity gradient and the assumption that the turbulence is in equilibrium (Esch and Menter (2004)):

$$P_k = \rho \varepsilon \tag{2.62}$$

In addition, if the density along a streamline is assumed to be constant, which causes the divergence of the mean velocity field to vanish, Eq. (2.23) reduces to:

$$P_k = \mu_t \frac{\partial U_i}{\partial x_j} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) + P_{kb} \tag{2.63}$$

If the density gradient is perpendicular to the gravitational acceleration vector or for a flow with vanishing local density gradient, $P_{kb}$ vanishes according to Eq. (2.24). For a velocity field characterized by $U_1(x_2)$ and $U_2 = U_3 = 0$, Eq. (2.62) and Eq. (2.63) in combination with Eq. (2.22) yield:

$$P_k = \rho \varepsilon = C_\mu \rho \frac{k^2 \varepsilon}{\varepsilon} \left( \frac{dU_1}{dx_2} \right)^2 \tag{2.64}$$

$$\varepsilon = \sqrt{C_\mu} k \left| \frac{dU_1}{dx_2} \right| \tag{2.65}$$

Eq. (2.65) also holds for the case of cylindrical coordinate system where $U_1$ denotes the velocity component in axial direction and $x_2$ the radial coordinate (Hughes and Gaylord (1964)). Combining Eq. (2.22) and Eq. (2.27) yields the conversion to the $\omega$ model:

$$C_\mu \rho \frac{k^2 \varepsilon}{\varepsilon} = \frac{k}{\omega} \tag{2.66}$$

$$\varepsilon = C_\mu k \omega \tag{2.67}$$

$$\omega = \frac{1}{\sqrt{C_\mu}} \left| \frac{dU_1}{dx_2} \right| \tag{2.68}$$

- By specifying the non-dimensional eddy viscosity $\mu_t/\mu$ as discussed by Casey and Wintergerste (2000): Away from boundary layers, a value for $\mu_t/\mu$ between 1 and 10 is reasonable with Eq. 2.22 and for a given $k$:

$$\varepsilon = C_\mu \rho \frac{k^2}{\mu_t} \tag{2.69}$$
2.7 Discretisation

The commercial solver ANSYS CFX10 applied here is a finite volume code which uses a coupled solver. A detailed description of the discretisation and the solver algorithms is given in ANSYS (2005).

2.7.1 Spatial discretisation

The discretisation of the advection terms is given by:

$$\phi_{ip} = \phi_{up} + \beta_{as} \frac{\partial \phi}{\partial x_j} r_j,$$

(2.70)

where $\phi_{ip}$ is the value at the integration point, $\phi_{up}$ is the value at the upwind node, and $r_j$ the $j$ component of the vector pointing from the upwind node to the integration point. A value of $\beta_{as} = 0$ leads to the first order upwind difference scheme, which is very robust but leads to numerical diffusion. For the choice $\beta_{as} = 1$, the scheme is second order accurate and less robust leading to numerical dispersion. Applying the so-called high resolution scheme, $\beta_{as}$ is determined based on the recipe of Barth and Jesperson (1989). It tries to compute $\beta_{as}$ to be as close to 1 as possible without violating boundedness principles and reduces to first order near discontinuities and in the free stream where the solution has little variation.

2.7.2 Temporal discretisation

For the discretisation of the transient terms, the second order backward Euler scheme is applied which makes use of the solution field from both the old time level and the new time level. This scheme is robust, implicit and conservative in time. It does not create a time step limitation and is second order accurate in time which can lead to nonphysical overshoots or undershoots.
Chapter 3

Experiment

3.1 Experimental setup

A hot jet in a confined crossflow is chosen as a generic test case for the validation of CFD fire modeling. It includes the major fluid structures of a fire scenario, such as a buoyant plume and natural or forced ventilation. Well defined boundaries and the absence of a combustion process makes it a suitable test case for the study of the influence of the applied turbulence models on the CFD results.

3.1.1 Tunnel geometry

The tunnel, sketched in Figure 3.1, is mounted on a frame made out of Maytec aluminum profiles. The cross-section of the tunnel has a width and a height of 0.8m. The aluminum frame is tiltable around the indicated pivot point which allows to consider also sloped tunnel configurations.

During operation of the tunnel, the hot air jet will stratify along the ceiling and the cold crossflow will be pushed towards to the bottom of the tunnel. Hence, only the side walls and the ceiling have to be insulated as the bottom plates, which are made of glass, face laboratory temperature.

All measurements are taken from the bottom side of the tunnel. The distance between the laboratory floor and the tunnel floor is large enough to mount an LDA traverse and a thermocouple rake on the frame to acquire the measurements through the floor of the tunnel as indicated in Figure 3.1.

The air intake in Figure 3.2 consists of a nozzle, which shows contraction in only one direction, with an area ratio of 1.75. Upstream of the nozzle, three fabric screens and one screen made of stainless steel cause enough pressure loss to damp incoming disturbances from the laboratory. Downstream of the nozzle, an array consisting of vertically and horizontally arranged seeding tubes with an outer diameter of 9mm and a spacing of 10cm is mounted to seed the crossflow. A velocity anemometer is attached to the rods and is used to control the extraction fan. Further downstream, a screen made of aluminum mixes out the wakes of the seeding tubes.
Figure 3.1: Experimental setup.
3.1 Experimental setup

Figure 3.2: Air intake with and without screens. In Figure 3.2(b), the seeding tubes and the velocity anemometer for the operation control are visible.

The cross-sectional view of the tunnel is given in Figure 3.6. The walls and the ceiling of the tunnel consist of an inner supporting structure made of 19.1mm thick Monolux 500 slabs and an insulation layer of aluminum-covered 40mm thick WDS Ultra slabs. The thermal properties of both materials are given in Table 3.1.

The floor is made of nine 0.95m long and 6mm thick glass plates which allow for optical access to the tunnel for LDA measurements. The glass plates are separated by a distance of 5cm in the downstream direction by aluminum plates which in turn can be replaced by a special plate containing ports for the thermocouple traversing rake. The glass plates can be removed from the bottom side of the tunnel for cleaning and maintenance. At the location of the hot air source, the floor is made of a 0.95m long Monalite M1 slab. This slab can sustain the encountered high temperatures and has a port for the hot air source located in the middle of the slab.

The extraction fan drives the crossflow and is attached to the tunnel by a 0.5m long nozzle. This nozzle provides the transition of the rectangular cross-section to a circular one with a diameter of 0.6m. A 40mm thick honeycomb with a pitch of 5mm is mounted between the nozzle and the fan in order to prevent the formation of an upstream vortex driven by the fan.
Table 3.1: Thermal properties of the applied insulation materials as reported by the manufacturers.

<table>
<thead>
<tr>
<th>Material</th>
<th>Classification temperature $T [^\circ C]$</th>
<th>Temperature $T [^\circ C]$</th>
<th>Thermal conductivity $\lambda [\text{W/(m K)}]$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Monolux 500</td>
<td>500</td>
<td>20</td>
<td>0.18</td>
</tr>
<tr>
<td></td>
<td></td>
<td>200</td>
<td>0.19</td>
</tr>
<tr>
<td></td>
<td></td>
<td>400</td>
<td>0.20</td>
</tr>
<tr>
<td></td>
<td></td>
<td>600</td>
<td>0.22</td>
</tr>
<tr>
<td>Monalite M1</td>
<td>850</td>
<td>20</td>
<td>0.21</td>
</tr>
<tr>
<td></td>
<td></td>
<td>200</td>
<td>0.22</td>
</tr>
<tr>
<td></td>
<td></td>
<td>400</td>
<td>0.23</td>
</tr>
<tr>
<td></td>
<td></td>
<td>600</td>
<td>0.25</td>
</tr>
<tr>
<td>WDS Ultra</td>
<td>1000</td>
<td>50</td>
<td>0.018</td>
</tr>
<tr>
<td></td>
<td></td>
<td>200</td>
<td>0.020</td>
</tr>
<tr>
<td></td>
<td></td>
<td>400</td>
<td>0.024</td>
</tr>
<tr>
<td></td>
<td></td>
<td>600</td>
<td>0.031</td>
</tr>
<tr>
<td>FMI 500</td>
<td>750</td>
<td>20</td>
<td>0.035</td>
</tr>
<tr>
<td></td>
<td></td>
<td>200</td>
<td>0.068</td>
</tr>
<tr>
<td></td>
<td></td>
<td>400</td>
<td>0.135</td>
</tr>
<tr>
<td></td>
<td></td>
<td>500</td>
<td>0.192</td>
</tr>
</tbody>
</table>

The coordinate system is chosen as indicated in Figure 3.1. Its origin is located at the center of the exit plane of the hot air source and lies in the symmetry plane of the tunnel. The crossflow direction is denoted by $x$, the span-wise direction by $y$ and the vertical direction by $z$.

### 3.1.2 Hot air source

The hot air source is depicted in Figure 3.3. The electric heater, type Leister 40000, heats up the supplied air from ambient temperature to about 500$^\circ$C. In order to smooth the velocity profiles downstream of the heater, two layers of fine stainless steel screen are inserted in the heater exit plane. A temperature sensor is located between these two layers for the heater control. Downstream of the screens, two seeding tubes with an outer diameter of 6mm are mounted, followed by a diffusor with an opening angle of 4 degree. A further screen with a mesh width of 1mm is located at the diffusor exit plane. It prevents the diffusor from stall and damps spatial velocity variations. Further downstream of the diffusor, a pipe with an inner diameter of 0.2m is attached. This pipe has four access ports located at $z = -0.3$m and spaced by 90 degree in the circumferential direction. These ports can be used for optical access in combination with silica glass windows or for temperature measurements. Both the diffusor and the pipe are insulated with two layers of 30mm thick Flumroc FMI 500 mats. The thermal conductivity of this material is also listed in Table 3.1.
3.1 Experimental setup

(a) Photograph.

(b) Schematic view.

Figure 3.3: Photograph and schematic view of the hot air source.

3.1.3 Operational control

Three PID controllers (Newport Electronics) are employed to assure repeatability and stable operation of the experiment. The schematic of the operational control is given in Figure 3.4. The first PID controller uses the analog signal of a velocity anemometer (Schiltknecht MiniAir6 Mini 20m/s) to control the motor of the extraction fan. This control loop is responsible for a constant total volume flow rate. The second PID control unit controls the blower which supplies the electric heater with air. It receives the input signal from a volumetric flow rate sensor (GWF MessSysteme AG, EQZ Stufe Q250) and keeps the cold supply air volume flow rate constant. The third PID controller reads its signal from a K-type thermocouple which is placed downstream of the electric heater and maintains the jet temperature constant during operation.
3.2 Operating point

The hot jet temperature should be as high as in real fire plumes to have realistic fire plume conditions. However, due to material restrictions, the peak temperature has to be restricted to 500°C which can be achieved by the available electric heater. According to the heater’s manufacturer, the volume flow rate of the air at room temperature at the inlet of the heater should be not less than 2 m³/min, which is taken to be the operating point.

As discussed in Heskestad (2002), the characteristic dimensionless number for a fire plume is the dimensionless heat release rate, defined as:

\[
\dot{Q}^* = \frac{\dot{Q}}{\rho_{cf} C_p T_{cf} \sqrt{gDD^2}},
\]

where \( \dot{Q} \) is the total heat release rate, \( \rho_{cf} \) and \( T_{cf} \) the density and temperature of the crossflow, \( C_p \) the specific heat of air at constant pressure, \( g \) the acceleration of gravity and \( D \) the diameter of the heat source represented as an area source. The dimensionless flame height \( L/D \) is correlated with \( \dot{Q}^* \) as given in Figure 3.5 by the curve denoted by H. Eq. (3.2) is not valid for \( \dot{Q}^* \) in the range of the momentum dominated regime labeled as jet flames in Figure 3.5.

\[
\frac{L}{D} = -1.02 + 3.7\dot{Q}^{*2/5}
\]
Having specified both the temperature difference across the heater and the flow rate, $\dot{Q}$ is already determined. In order to get realistic fire conditions, the source diameter $D$ has to be large enough in order to decrease $\dot{Q}^*$ and to be in the buoyancy driven pool fire region. Given the heater geometry, a diffuser had to be applied. With its outlet diameter of $D = 0.2\text{m}$, the dimensionless heat release rate can be finally reduced to the order of unity. A summary of the actual dimensionless numbers is presented in Section 3.4.4.

Figure 3.5: Flame height correlations found by different authors (capital letters) as reported in Heskestad (2002), Copyright 2002, Society of Fire Protection Engineers. Used with permission.

The crossflow velocity has to be carefully chosen: At the location of $x = -1.25\text{m}$ the velocity distribution is measured and is used as input data for the boundary conditions in the CFD validation cases. Therefore it is desired to have a homogeneous velocity distribution with positive values (inflow) within this plane. Given an insufficient crossflow velocity, the hot air at the tunnel ceiling would move upstream with respect to the jet source position. This phenomenon is called backlayering and would lead to a distortion of the velocity profile at the plane $x = -1.25\text{m}$. It also causes trouble in controlling the tunnel: As the backlayering reaches the cross-section where the velocity anemometer is mounted, the layer causes an acceleration of the cold flow as its effective cross-section is reduced by the hot air layer. The sensor would measure higher velocities and the PID controller would reduce the extract fan speed which would make the system unstable. A too high ventilation velocity in contrast enhances the entrainment of the cold air into the hot jet. The
jet temperature would decrease sooner and the layering in the tunnel would become weaker which is not desired either.

A suitable ventilation velocity is determined by installing two thermocouples at the ceiling \((x, y, z) = (-0.5m, 0.35m, 0.8m)\) and \((x, y, z) = (-0.5m, 0m, 0.8m)\) which monitor backlayering. Starting from a high crossflow velocity, the extract fan speed is continuously reduced until backlayering up to the position \(x = -0.5m\) occurs. A slightly higher ventilation velocity of 0.75m/s at the velocity anemometer, which is shown in Figure 3.4, is chosen as the operating point to be on the safe side. Due to the blockage of the seeding tubes and the growth of the boundary layer, this velocity differs from the one measured at the location \(x = -1.25m\).

### 3.3 Measurement techniques

In this section, the applied measurement technique for both velocity and temperature measurements are described. As discussed in Section 3.1.1, the floor of the tunnel is made of glass and hence accessible from the bottom side of the tunnel for the LDA technique. With this technique, also very low velocities can be measured accurately without disturbing the flow field. The exchangeable aluminium plates separating the bottom glass plates allow to mount a thermocouple rake to acquire temperature data.

#### 3.3.1 Thermocouples

The temperature measurements were conducted using NiCr-Ni Typ K thermocouples (Moser TMT AG, Hombrechtikon). The housing of these probes consist of stainless steel tubes with an outer diameter of 4mm. The tube is responsible for the mechanical stability of the probe and contains a ceramic material which electrically insulates the two thermocouple wires. These wires, with a diameter of 0.5mm each, protrude 2cm out of the housing and are welded together at the tip, as shown in Figure 3.6. The welding point, which is the sensing area, is kept small in order to achieve small thermal inertia and hence good temporal resolution. The 2cm region, where the wires are directly exposed to the flow, is responsible for the good measurement performance in high temperature gradient flow fields. Preliminary measurements with encapsulated thermocouples showed large errors due to the heat conduction along the stainless steel housing and the electrical insulation material of the probe. This effect is reduced in the actual setup by having thin and non-encapsulated wires at the tip.

The signals of the thermocouples are processed (Analog Devices, AD594, including an electronic cold junction compensation) and amplified with a custom-built multichannel amplifier and are then logged with a A/D converter on a data acquisition card (National Instruments, PCI-MIO-16E-4) on a computer. The whole acquisition chain, including cables, was calibrated using a high temperature oven and a calibrator.
Figure 3.6: Schematic view of the tunnel cross-section and the thermocouple rake. At the right upper corner, one enlarged view of the thermocouple probe tip is shown.
In order to measure the temperature distribution within the hot air jet, a single probe is used to traverse the jet. For the measurements inside the tunnel, a traversing rake as indicated in Figure 3.6 is installed. The probe positions are equally spaced by 5cm in the span-wise direction, and the step size in the vertical direction is also chosen to be 5cm. In order to avoid excessive blockage, only eight or seven probes, respectively, are traversed at once. The two configurations of the probe positions are symmetric with respect to the symmetry plane of the setup and are indicated with the black and shaded black probes in Figure 3.6.

For the computation of the temperature r.m.s. value, the original recorded signal is first low-pass filtered using the M’4 filter function as discussed by Monaghan (1985). In a second step, the high-pass filtered signal is computed as the difference of the original signal and the low-pass filtered signal. From the high-pass filtered signal, the r.m.s. values are finally computed. The M’4 filter kernel is defined as follows:

\[
k_{M’4}(t) = \begin{cases} 
1 - \frac{5}{2}(t/\Delta)^2 + \frac{3}{2}|t/\Delta|^3 & |t/\Delta| \leq 1 \\
\frac{1}{2}(2 - |t/\Delta|)^2(1 - |t/\Delta|) & 1 \leq |t/\Delta| \leq 2 \\
0 & |t/\Delta| > 2 
\end{cases}
\]  

(3.3)

\[
f_{co} = \frac{1}{T_{co}} = \frac{f_{sr}}{2\Delta},
\]

(3.4)

where \( \Delta \) is the filter half width. The cut-off frequency \( f_{co} \) is given by the sampling frequency \( f_{sr} \) divided by the filter width. \( T_{co} \) denotes the cut-off periodic time. This filter has the advantage that the Fourier coefficients decay rapidly in the stop-band and it has almost no side-lobes in the time domain.

3.3.2 LDA system

The DANTEC laser Doppler anemometry system used for the velocity measurements is the same apparatus as employed by Blum (2005), Flury (1999) and Dimopoulos (1996). It is a three-component system which is operated in a two-component backscattering mode in the present investigation. The 5W Ar\(^+\)-Laser (Coherent, Inova 90) is run in the multi-line mode at an output power of about 2W. The spectral lines at \( \lambda = 514.5\text{nm} \) and 488nm are used for the two-component transmitting optics in combination with two front lenses of different focal lengths. The properties of both optical setups are given in Table 3.2.

Table 3.2: Technical data of the applied optics. The measurement volume length is measured in the optical axis and normal to the main flow direction.

<table>
<thead>
<tr>
<th>Focal length [mm]</th>
<th>Beam separation [mm]</th>
<th>Measurement volume diameter [( \mu \text{m} )]</th>
<th>Measurement volume length [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>310</td>
<td>73.7</td>
<td>75</td>
<td>0.63</td>
</tr>
<tr>
<td>1200</td>
<td>73.7</td>
<td>292</td>
<td>9.56</td>
</tr>
</tbody>
</table>
The 310mm front lens is used in both the hot air jet and for boundary layer measurements as the measurement volume is reasonably small. For the measurements in the tunnel, the whole height could only be traversed with the 1200mm lens. Therefore the optical head is mounted on a horizontal traverse and a front surface mirror deflects the beams into the tunnel as shown in Figure 3.7.

![LDA probe in operation mounted on a traverse. The deflecting front surface mirror is attached at the upper side of the visible black optical support plate. The glass floor plates in the upper part of the picture allow to see inside the tunnel.](image)

Figure 3.7: LDA probe in operation mounted on a traverse. The deflecting front surface mirror is attached at the upper side of the visible black optical support plate. The glass floor plates in the upper part of the picture allow to see inside the tunnel.

During the measurements, data are processed by means of spectrum analyzers from Dantec, which are known as Burst Spectrum Analyzers (BSA) and recorded using a corresponding Dantec software. The acquired data is further processed with a software written by Blum (2005). Details about the hardware, probes and signal processing as well as LDA theory can be found in Blum (2005).
3.3.3 Seeding

Seeding material

In LDA measurements, particles have to be introduced into the flow to have light-scattering objects. For measurements in air flows, mainly oil droplets produced by atomizers or solid hydrophobic powders, such as metal oxides, are used. For the given application, oil droplets cannot be applied, as the maximum temperature in the flow is of the order of 500°C. Therefore the droplets would evaporate quickly and could even start to burn spontaneously since the temperature is higher than the auto-ignition temperature (Table 3.3).

Table 3.3: Flash point and auto-ignition temperature at the pressure of 1 bar for olive and castor oil given by Sciencelab. The flash point temperature is the minimum temperature at which the vapor pressure of a liquid is sufficient to form an ignitable mixture with air near the surface of the liquid. The auto-ignition temperature is the minimum temperature required for self-sustained combustion in the absence of an external ignition source.

<table>
<thead>
<tr>
<th>Oil</th>
<th>Flash point [°C]</th>
<th>Auto-ignition [°C]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Olive oil</td>
<td>225</td>
<td>343</td>
</tr>
<tr>
<td>Castor oil</td>
<td>230</td>
<td>449</td>
</tr>
<tr>
<td>Dioctyl phthalate (DOP)</td>
<td>207-218</td>
<td>390</td>
</tr>
<tr>
<td>Bis(2-ethylhexyl)sebacate (DEHS)</td>
<td>215</td>
<td>400</td>
</tr>
</tbody>
</table>

As the experiments are performed in an open wind tunnel, metal oxide powders cannot be used for health safety reasons: Rössler et al. (2001) discuss the absorption rate of particles in the human body during breathing. Figure 3.8 shows the total absorption rate and the absorption rate within the lungs. It can be seen that the total absorption rate for particles with a diameter > 10µm is nearly 100% but only less than about 10% reach the lungs as these particles are mainly absorbed in the throat and nose region through sedimentation. Particles with a diameter < 10µm, such as typical seeding particles, reach the lungs and are absorbed by a diffusion process. The smaller the particles the higher the absorption rate within the lungs. Particles with a diameter < 0.1µm do not only reach the lungs but can also enter the bloodstream. The total absorption rate shows a minimum at particle diameters of about 0.3µm. A large portion of particles of this size are not absorbed but rather exhaled.

Depending on particle shape, size and chemical characteristics, the human body reacts in various different ways to those particles: Cough, inflammation, allergic reactions or even cancer can be provoked. In contrast to the hydrophobic metal oxides, sodium chloride (NaCl, “cooking salt”) is believed to be even healthy - or at least not harmful. The effect of inhaling a dry powder of sodium chloride on the airways has been studied by Anderson et al. (1997). In the material safety data
3.3 Measurement techniques

sheet for sodium chloride, the concentration in air that will kill 50% of the test animals (rat) is given to be $> 42 \text{g/m}^3$ for a single exposure of one hour. Hence, sodium chloride particles are considered to be the best choice as a seeding material for this experimental setup because concentrations are way down below $42 \text{g/m}^3$. The melting point of NaCl is at $801^\circ \text{C}$ which is much higher than the maximum temperature in the experiment. Another advantage is the low material cost and its availability.

Figure 3.8: Absorption rate of aerosol particles in the human airway system.

Seeding generation

The salt crystals are produced by an ultrasonic atomizer (Figure 3.9(a)) which is placed in a NaCl water solution. These atomizers are employed both in medical inhalators and in ordinary humidifiers. The membrane of this device oscillates at an ultrasonic frequency and causes cavitation which in turn leads to the formation of small droplets. The size of these droplets is of the order of 1 to 10 $\mu \text{m}$. As the droplets are very small, the water will evaporate quickly and a cube-shaped NaCl crystal is produced from each droplet. Figure 3.9(b) shows the atomizer mounted in the flotation device, which assures a constant water level above the membrane. The fine mesh on top of the atomizer retains splashes. In Figure 3.9(c) the atomizer is in operation.

For continuous seeding, the salt solution is filled into a suitable plastic barrel (Figure 3.9(d)) and the floating atomizer is placed inside. Fittings are attached on the lid in order to supply pressurized air and extract the particle-laden air.
Figure 3.9: Ultrasonic atomizer, mounted on the floating ring, in operation and assembled in a barrel.
3.3 Measurement techniques

Particle size distribution

The NaCl crystal size can be controlled by the NaCl concentration of the solution used. Applying the mass conservation law for one solution droplet with diameter \( D_s \) and density \( \rho_s \), the edge length \( L_{cr} \) of the sodium chloride crystal cube created by evaporation can be expressed as:

\[
\frac{\pi}{6} D_s^3 \rho_s f_m = L_{cr}^3 \rho_{cr} 
\]

\[
\frac{L_{cr}}{D_s} = \left( \frac{\pi \rho_s}{6 \rho_{cr}} f_m \right)^{1/3} 
\]

\[
f_m = \frac{m_{NaCl}}{m_{NaCl} + m_{H_2O}} ,
\]

where \( f_m \) denotes the mass of solute divided by the total mass of solution and \( \rho_{cr} \) the crystal density which is \( \rho_{cr} = 2165 \text{ kg/m}^3 \) at 25°C and 1atm. The density of the solution \( \rho_s \) is a function of \( f_m \). The values at 20°C can be found in Lide (2004) and are given in Figure 3.10 together with the computed length and mass ratios of (Eq. (3.6) and (3.7)).

![Figure 3.10: Properties of sodium chloride aqueous solution and crystal edge length to droplet diameter ratio as a function of \( f_m \).](image)

(a) Density of the solution.

(b) Mass ratio of the solution.

(c) Crystal edge length to droplet diameter ratio.
The particle size distribution for different values of $f_m$ was measured using a laser diffraction based particle size analyzer (Sympatec). In Figure 3.11 the measured distributions are given and the median values are listed in Table 3.4. It can be seen that the crystals are smaller than 5 $\mu$m and typical sizes range between 1 and 4 $\mu$m for the ultrasonic atomizer operating in an almost saturated solution. Producing the crystals with a Laskin nozzle leads to smaller particles which could not be completely resolved with the available analyzer optics.

![Figure 3.11: Measured cumulative volumetric distribution function, $Q_3$, and the corresponding probability density function, $q_3$, of the sodium chloride crystal particle size for different concentrations $f_m$ and produced by an ultrasonic atomizer (triangular symbols). In addition, one measurement using a Laskin nozzle is given (circles).](image)

Table 3.4: Median of the particle size and the measured and theoretical crystal size ratio for different concentrations assuming that the generated solution droplet size does not vary with $f_m$.

<table>
<thead>
<tr>
<th>Generator</th>
<th>$f_m$</th>
<th>Median</th>
<th>Measured</th>
<th>Theory</th>
<th>Theory</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>[-]</td>
<td>[\mu m]</td>
<td>$L_{L/0.05}$</td>
<td>$L_{L/0.05}$</td>
<td>$L_{D/0.5}$</td>
</tr>
<tr>
<td>Ultrasound</td>
<td>0.2</td>
<td>1.98</td>
<td>0.662</td>
<td>0.608</td>
<td>0.381</td>
</tr>
<tr>
<td>Ultrasound</td>
<td>0.1</td>
<td>1.68</td>
<td>0.780</td>
<td>0.785</td>
<td>0.296</td>
</tr>
<tr>
<td>Ultrasound</td>
<td>0.05</td>
<td>1.31</td>
<td>1</td>
<td>1</td>
<td>0.232</td>
</tr>
<tr>
<td>Laskin</td>
<td>0.2</td>
<td>0.90</td>
<td>-</td>
<td>-</td>
<td>0.381</td>
</tr>
</tbody>
</table>

In order to check the theoretical approach for the particle size dependency on $f_m$, the measured and theoretically predicted particle size ratio for the different concen-
trations \( f_m \) are also given in Table 3.4. The size of the generated solution droplet \( D_s \) is assumed to be independent of \( f_m \) for the theoretical prediction and Eq. (3.6) leads to the listed values. Since measurements and theory agree well, the solution droplet size can be determined by Eq. (3.6). The theoretical values for \( L_{cr}/D_s \) are also given in Table 3.4.

A microscopy image of sodium chloride crystals produced is given in Figure 3.12. The depicted crystals are smaller than about 3\( \mu \)m.

Figure 3.12: Microscopy image of sodium chloride crystals produced with the ultrasonic atomizer. The indicated length represents 10\( \mu \)m.

**Resolvable frequency**

In order to resolve flow fluctuations with LDA measurements, the seeding particles have to be able to follow the flow field and its variation in time. Especially measurements in gas flow are critical, as the density ratio of the particle material and the fluid becomes \( \rho_p/\rho_f >> 1 \). In order to avoid significant inertia forces, the seeding particles have to be reasonably small. As discussed in Albrecht et al. (2003), the equation of particle motion of a spherical particle of diameter \( D_p \) in a homogeneous velocity field neglecting particle/particle interaction is known as the Basset-Boussinesq-Oseen (BBO) equation. Considering periodical 1D velocity fluctuations of the flow field and very large density ratios \( \rho_p/\rho_f >> 1 \), the equation can be solved for the cut-off frequency \( f_{co} \) at which the particles still follow the velocity oscillations with a fraction \( (1 - s) \) of the amplitude:
\begin{equation}
    f_{co} = \frac{1}{\tau_0} \frac{1}{2\pi} \sqrt{\frac{1}{(1-s)^2} - 1}
\end{equation}

Here, \( \tau_0 \) is the characteristic time, \( \nu \) the kinematic viscosity of the gas, \( s \) the relative slip velocity and \( U_f \) and \( U_p \) are the fluid and the particle velocity respectively. For a 1\% slip, the cut-off frequency yields \( f_{co} = 0.0227/\tau_0 \). 

Another characteristic number is the settling velocity \( U_s \). A stationary, laminar flow is considered in which the gravitational force acting on the particle causes it to sink with time. The balance of drag force and gravitational force yields

\begin{equation}
    U_s = \frac{D_p^2 \rho_p}{18 \rho_f \nu} g = \tau_0 g,
\end{equation}

where \( g \) is the gravitational acceleration.

The characteristic numbers for the measured median particle diameters are summarized in Table 3.5. Values for both 20\(^\circ\)C and 500\(^\circ\)C are given under the assumption that the crystal density remains the same (cold crystal in hot gas). It can be seen that the variation of the gas density and gas kinematic viscosity with temperature have a large influence on the characteristic numbers.

Table 3.5: Characteristic time, cut-off frequency, and settling velocity for the measured median particle diameters at both 20\(^\circ\)C and 500\(^\circ\)C.

<table>
<thead>
<tr>
<th>Particle diameter [( \mu \text{m} )]</th>
<th>T ([\text{(^{\circ})}\text{C}] )</th>
<th>( \tau_0 ) [( \mu \text{s} )]</th>
<th>( f_{co} ) [Hz]</th>
<th>( U_s ) [mm/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.98</td>
<td>20</td>
<td>26</td>
<td>873</td>
<td>0.255</td>
</tr>
<tr>
<td></td>
<td>500</td>
<td>13.3</td>
<td>1708</td>
<td>0.130</td>
</tr>
<tr>
<td>1.68</td>
<td>20</td>
<td>18.7</td>
<td>1213</td>
<td>0.184</td>
</tr>
<tr>
<td></td>
<td>500</td>
<td>9.57</td>
<td>2372</td>
<td>0.094</td>
</tr>
<tr>
<td>1.31</td>
<td>20</td>
<td>11.4</td>
<td>1995</td>
<td>0.112</td>
</tr>
<tr>
<td></td>
<td>500</td>
<td>5.82</td>
<td>3902</td>
<td>0.057</td>
</tr>
<tr>
<td>0.9</td>
<td>20</td>
<td>5.37</td>
<td>4226</td>
<td>0.053</td>
</tr>
<tr>
<td></td>
<td>500</td>
<td>2.75</td>
<td>8267</td>
<td>0.027</td>
</tr>
</tbody>
</table>
3.4 Measurements

Prior to all measurements, the experiment is run for 1.5 hours. This time is needed to heat up the system and to reach stable control conditions. During the measurements, the doorway and the windows in the vicinity of the extraction fan are left open to the laboratory hall. The air can settle down in the laboratory and also keeps the laboratory temperature stable at about 26°C on average.

In order to have well defined boundary conditions for CFD calculations, the velocity distribution at the plane $x = -1.25\text{m}$ in the tunnel and both the velocity distribution and the temperature distribution in the hot jet source at the plane $z = -0.3\text{m}$ are measured. Preliminary CFD investigations of the secondary flow within the hot jet source, caused by the crossflow within the tunnel (Keimasi and Taeibi-Rahni (2001)), had shown that the secondary velocity at locations $z < -0.2\text{m}$ almost vanishes. Therefore, the location of the measurement plane is set to $z = -0.3\text{m}$ in order to assure non-disturbed velocity profiles.

For the validation of the CFD calculations, temperature profiles are measured downstream of the jet exit at seven planes starting at $x = 0.5\text{m}$ and separated by $1\text{m}$ in the $x$ direction. In addition, LDA measurements are performed in six cross-sections spaced by $1\text{m}$ and starting at $x = 1\text{m}$. All measurement locations in the crossflow tunnel are given in Figure 3.13.
Figure 3.13: Measurement locations. The dashed lines indicate the locations of the LDA measurements and the solid line the locations of the temperature measurements.
3.4 Measurements

3.4.1 Hot air source

In this section, the flow properties inside the hot air jet are presented. All described measurements are taken at $z = -0.3m$. For the velocity measurements, the 310mm lens is used which is described in Table 3.2. The achieved data rates are below 90Hz. The hot jet source is traversed in both $x$ and $y$ direction through the whole pipe diameter. For better comparison, the time averaged data in Figure 3.14 is presented using polar coordinates where $\varphi = 0$ denotes the $x$-axis and $\varphi = \pi/2$ the $y$-axis.

![Figure 3.14: Measured profiles of axial velocity, circumferential velocity, r.m.s. of the axial velocity and r.m.s. of the circumferential velocity at different angular positions within the hot jet source at $z = -0.3m$.](image)
In Figure 3.14(a) the measured time averaged axial velocity component is given. In the center of the pipe, the velocity profiles are very flat and coincide well. Close to the wall, a velocity peak is visible which is present at all four circumferential positions and proves that the diffusor is free of separation due to the applied stainless steel screen. The circumferential velocity distribution is presented in Figure 3.14(b). The magnitudes are smaller than the correspondent r.m.s. values given in Figure 3.14(d). This shows that the jet is free of axial swirl. In addition, the r.m.s. values of the axial velocity component are given in Figure 3.14(c). In the center of the pipe, the r.m.s. values of the circumferential and axial velocity components are of similar magnitude indicating isotropic turbulence.

The turbulence intensity $I$ is shown in Figure 3.15(a) and is computed for isotropic turbulence by combining Eq. (2.56) and Eq. (2.57):

$$I = \frac{\sqrt{w'w}}{\sqrt{W'W}}.$$  \hspace{1cm} (3.12)

The temperature distribution is presented in Figure 3.15(b) and shows a circumferential dependence. It originates from the heater design as it uses 3 pairs of heating coils, one for each electrical phase. They are arranged in three sectors of 120 degree each and are located such that the first one is centered on the $x$-axis. Therefore, the temperature distribution in the $z = -0.3m$ plane is reconstructed using the following model:

$$T(r_i, \varphi) = b_1(r_i) + b_2(r_i) \cos(3\varphi)$$  \hspace{1cm} (3.13)

Having four measurements for each radial position, $r_i$, the coefficients $b_1(r_i)$ and $b_2(r_i)$ are computed by a least-squares minimization algorithm for each radial position $r_i$ and are shown in Figure 3.16. The wall temperature is set constant to
the circumferentially averaged value. The resulting temperature distribution in the $z = -0.3\,\text{m}$ plane is given in Figure 3.17. The r.m.s. value of the errors is $2.2^\circ\text{C}$ and the maximum temperature in the plane is $T_{\text{max}} = 503.5^\circ\text{C}$.

The circumferentially averaged measured axial velocity, $W_{\text{avg}}$, and turbulence intensity, $I_{\text{avg}}$, are given in Figure 3.18 and serve as input data for CFD computations.

Figure 3.16: Computed mean value function $b_1(r_i)$ and amplitude function $b_2(r_i)$ of the applied temperature distribution model $T(r_i)$ given by Eq. (3.13).

Figure 3.17: Interpolated temperature distribution in the hot air source at $z = -0.3\,\text{m}$. 
Figure 3.18: Circumferential average of the measured axial velocity and of the turbulence intensity at $z = -0.3m$. The average turbulence intensity in the interval $0 \leq r \leq 90mm$ is 0.022.

### 3.4.2 Crossflow inlet

In this section, the LDA measurements in the tunnel upstream of the hot air source are described. First, the free stream velocity profiles in the $z$ direction at two different upstream positions are given followed by a transverse profile. In the second part, the boundary layer is analyzed. All profiles in the $z$ direction are recorded using the 1200mm lens as described in Table 3.2. The transverse profiles in the $y$ direction and the boundary layer measurements were measured using the 310mm lens in order to reach higher data rates and better spatial resolution.

**Free stream**

In Figure 3.19 the free stream velocity profiles at the positions $x = -1.25m$ and $x = -0.6m$ are presented. The measurements at the cross-section $x = -1.25m$ are considered as input data for CFD computations as the velocities all over the cross-section are nearly constant. The average velocity over all performed measurements in the $x = -1.25m$ plane amounts to $U_{avg} = 0.73m/s$.

In Figure 3.19(b) the profiles at $x = -0.6m$ are given. They show a slight gradient in the $z$ direction which is caused by the backlayering: The arriving fluid close to the ceiling faces the blockage of the hot air accumulating at the ceiling and is pushed in the negative $z$ direction under the layered hot air. As the blockage due to the hot jet is largest in the center of the tunnel, the velocities for the profile at $y = 0m$ are smallest compared to the location $y = -0.2m$ and $y = 0.2m$. 
3.4 Measurements

Figure 3.19: Measured $U$ velocity profiles at two different upstream locations. The mean value of the measurements at $x = -1.25$ m is found to be $U_{avg} = 0.73$ m/s.

In order to investigate the variations in $y$ direction, a horizontal traverse is conducted at the location of $x = -1.25$ m and $z = 0.25$ m using the 310mm lens. The obtained data is presented in Figure 3.20. The $U$ velocity shows a minimum in the center of the tunnel which can also be found in Figure 3.19(a). The average of the measured $y$ traverse data is 0.73 m/s and corresponds to the mean value of the six $z$ profiles at $x = -1.25$ m.

The r.m.s. values show spatial variations which could originate from the seeding tubes. From the r.m.s. values of the velocity component in the $x$ direction and the $U$ velocity distribution, the turbulence intensity distribution is calculated. The average intensity is 0.026 and shows evidence of low turbulence within the $x = -1.25$ m plane.
Figure 3.20: Measured profiles of $U$ and $V$ velocity, r.m.s. values, and turbulence intensity (based on the $u$ component) at $x = -1.25\text{m}$ and $z = 0.25\text{m}$. The mean value of the turbulence intensity is 0.026.

Boundary layer

Three boundary layer profiles at different $y$ positions on the bottom are measured in the $x = -1.25\text{m}$ plane and presented in Figure 3.21. The positions $y = 0.3\text{m}$ and $y = -0.3\text{m}$ are located downstream of seeding tubes in contrast to the one at $y = 0.05\text{m}$. The profiles are normalized by the free stream velocity $U_\infty$ and a zoom-in is given in Figure 3.21(b). In addition, a polynomial fit of degree 6 is presented.

From the boundary layer profiles, the displacement thickness $\delta^*$, the momentum thickness $\delta$ and the shape factor $H = \delta^*/\delta$ can be computed (Wilcox (1997)). The
values are listed in Table 3.6. For an ideal flat plate boundary layer, Wilcox (1997) gives the shape factor values to be 2.59 for laminar and 1.28 for turbulent flows. The measured values lie in between these two values. Statements, of which type the measured boundary layers are, are questionable as it has to be considered that in the actual setup seeding tubes and several screens are present and influence the boundary layer.

Figure 3.21: Measured boundary layer profiles at the upstream position $x = -1.25m$ and a zoom-in. In Fig. 3.21(b), a polynomial fit of the degree 6 is given.

Table 3.6: Boundary layer properties: Displacement thickness $\delta^*$, momentum thickness $\delta$ and shape factor $H$.

<table>
<thead>
<tr>
<th>Position</th>
<th>$\delta^*$ [mm]</th>
<th>$\delta$ [mm]</th>
<th>$H$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>$y = 0.05m$</td>
<td>4.29</td>
<td>1.87</td>
<td>2.29</td>
</tr>
<tr>
<td>$y = 0.3m$</td>
<td>3.33</td>
<td>1.36</td>
<td>2.44</td>
</tr>
<tr>
<td>$y = -0.3m$</td>
<td>4.05</td>
<td>1.96</td>
<td>2.07</td>
</tr>
<tr>
<td>Polyfit</td>
<td>4.06</td>
<td>1.93</td>
<td>2.11</td>
</tr>
</tbody>
</table>

3.4.3 Downstream measurements

Velocity measurements

The velocity measurements discussed in this section are taken with the 1200mm lens. Data is acquired for 180 seconds in order to produce reliable temporal averages. An overview of the measured time-averaged velocity component in the $x$ direction at $y = 0m$ is presented in Figure 3.22. The detailed profiles are given in Figure 3.23.
The presented data outside of the symmetry plane are made symmetric by averaging the original measurements (Appendix A.1) at corresponding $y$ positions. It can be seen that the velocity profiles at different $y$ locations differ substantially in the planes $x = 1\text{m}$ and $x = 2\text{m}$ whereas the profiles are pretty similar at locations downstream of the $x = 3\text{m}$ plane.

![Figure 3.22: Overview of the $U$ velocity component of the LDA measurements in the symmetry plane $y = 0\text{m}$. Zero values at the walls are inserted for reference.](image)

In Figure 3.24(a), the distributions of the spanwise velocity component in the $y$ direction in the plane $x = 1\text{m}$ are shown. Again, data is made symmetric. The data rate is found to decay going towards the ceiling as discussed in Section 3.4.5. Hence, only measurements with a data rate higher than 1Hz are presented ($z \leq 0.45\text{m}$). It has to be noted that the values at $y = 0\text{m}$ are lower than the r.m.s. values and can be considered to be close to zero due to statistical uncertainties. In the further downstream measurement planes, the velocity components in the $y$ direction are found to be smaller or of the same order as the r.m.s. values. Hence, they are not presented.

In Figure 3.24(b) the power spectrum of the velocity component in the $y$ direction at the location $(x, y, z) = (1\text{m}, 0\text{m}, 0.1\text{m})$ is given. A dominant frequency of 1.1Hz is found within the temporal data indicating the existence of wake vortices as described by Fric and Roshko (1994). From the shedding frequency, the Strouhal number $St$ can be computed:

$$St = \frac{fL_c}{U_c} = 0.3 \ ,$$  \hspace{1cm} (3.14)

where the characteristic length is chosen to be the heat source diameter, $L_c = 0.2\text{m}$, and the characteristic velocity to be the averaged crossflow velocity found at $x = -1.25\text{m}$, $U_c = 0.73\text{m/s}$. 
Figure 3.23: Velocity profiles at downstream positions. The given values are obtained by averaging measured profiles at symmetric $y$ positions.
Figure 3.24: 3.24(a): Measured $V$ component at $x = 1\text{m}$ with a data rate higher than 1Hz. 3.24(b): Measured spectrum of the $v$ velocity component at $x = 1\text{m}$, $y = 0\text{m}$ and $z = 0.1\text{m}$. The dominant frequency is 1.1Hz.
3.4 Measurements

Temperature measurements

Temperatures are recorded with a rate of \( f_s = 50 \text{Hz} \) over 3 minutes for the cross-sections at \( x = 0.5 \text{m} \) and \( x = 1.5 \text{m} \), and over 2 minutes for the other planes. The measured time-averaged temperatures, \( T \), are made dimensionless using the minimum temperature, \( T_{\text{min}} \), which is the inlet temperature of the crossflow, and the maximum temperature, \( T_{\text{max}} = 503.5^\circ \text{C} \) in the hot air source. The minimum temperature is logged during the measurements and is taken at the center of the air intake contraction. Using the non-dimensional temperature, the measurements can be compensated for laboratory temperature fluctuations and a more accurate comparison with CFD data is possible. The original time averaged temperature distributions are given in Appendix A.2. Data presented in this section are made symmetric by folding on the symmetry plane and subsequent averaging. Detailed plots of the temperature measurements are given in Figure 3.25 and 3.26 and an overview can be found in Figure 4.18(a).

In Figure 3.26, the air temperature for the planes downstream of \( x = 2.5 \text{m} \) shows a negative gradient at the ceiling. This could be due to cooling effects by the non-ideal insulation. However, this effect is visually enhanced by the contour plots as the near-wall temperature is recorded about 2mm away from the ceiling and the next \( z \) measurement location is 5cm below the ceiling.

![Dimensionless Temperature Distribution](image)

Figure 3.25: Measured dimensionless temperature distribution at \( x = 0.5 \text{m} \) made symmetric by mirroring and averaging.
Figure 3.26: Measured dimensionless temperature distribution made symmetric by mirroring and averaging.
From the time-resolved temperature signals, the r.m.s. values are computed by high-pass filtering the signal using the M’4 filter as discussed in Section 3.3.1. Thereby, large-scale fluctuations of the flow field, caused e.g. by the control loops, are removed. Figure 3.27 shows the influence of the filter half width \( \Delta \) (Eq. 3.3) on the r.m.s. values for the cross-section at \( x = 0.5 \) m. The global topology remains but values become smaller the shorter the filter width is chosen.

![Figure 3.27: Measured r.m.s. temperature distribution in the cross-section at \( x = 0.5 \) m applying different filter widths.](image)

(a) Cut-off periodic time: \( T_{co} = 1 \) s  
(b) Cut-off periodic time: \( T_{co} = 5 \) s  
(c) Cut-off periodic time: \( T_{co} = 10 \) s  
(d) Cut-off periodic time: \( T_{co} = 20 \) s

Figure 3.27: Measured r.m.s. temperature distribution in the cross-section \( x = 0.5 \) m applying different filter widths. The corresponding cut-off frequencies are: 3.27(a): \( f_{co} = 1 \) Hz, 3.27(b): \( f_{co} = 0.2 \) Hz, 3.27(c): \( f_{co} = 0.1 \) Hz, 3.27(d): \( f_{co} = 0.05 \) Hz. Data is made symmetric by mirroring and averaging.

A suitable filter width has to be chosen in a way that it does not remove the largest coherent flow structures. Therefore, the integral time scale for each temperature signal is computed by integrating the auto-correlation function of the signal from zero up to the first zero crossing. As an example, the original and filtered temperature signals and the auto-correlation function of the mean-free original signal, \( R(\tau) \), at the location \( (x, y, z) = (0.5 \) m, \( 0 \) m, \( 0.4 \) m) are given in Figure 3.28. The auto-correlation function is computed according to Eq. (2.59), where the mean-free
velocity signal is replaced by the mean-free temperature signal. After computing the integral time scale for each temperature signal, an average value is computed for each \( x \) location. The found mean values vary from 3.78 to 5.45 seconds and the median value finally chosen is 4.5 seconds. This value is used as the filter width and the cut-off frequency is given by \( f_{\text{co}} = 1/4.5\text{s} = 0.22\text{Hz} \). The corresponding r.m.s. temperature distributions are given in Figure 3.29 and 3.30. All r.m.s. distributions shown in this section are made symmetric by mirroring and averaging. The not mirrored data filtered using a cut-off frequency of \( f_{\text{co}} = 1/4.5\text{s} = 0.22\text{Hz} \) can be found in Appendix A.2.

![Figure 3.28: Measured temperature signal at \((x, y, z) = (0.5\text{m}, 0\text{m}, 0.4\text{m})\). The corresponding low- and high-pass filtered signals are cropped according to the filter width for the given cut-off frequency \( f_{\text{co}} = 0.22\text{Hz} \). In 3.28(c) the auto-correlation of the mean-free original signal is given. The corresponding integral time scale is 2.5s.](image-url)
The chosen filter width and integral time scale of 4.5s can be transformed into a length scale applying Eq. (2.60). The required mean convection velocity can be estimated from Figure 3.23 in the downstream region and turns out to be of the order of 1m/s. Hence, the chosen filter width corresponds to a length scale of about half the tunnel length. It is therefore smaller than the feedback time of the tunnel control and is able to damp out fluctuations from the tunnel control.

Figure 3.29: Measured r.m.s. temperature distribution at $x = 0.5$m. Data is made symmetric by mirroring and averaging. Cut-off frequency: $f_{co} = 0.22$Hz.
Figure 3.30: Measured r.m.s. temperature distribution. Data is made symmetric by mirroring and averaging. Cut-off frequency: $f_{co} = 0.22\text{Hz}$. 
3.4.4 Characteristic parameters

In this section, the characteristic non-dimensional parameters of the present experiments are given at the operating point (one selected). The index notation is as follows: \( j \) is used as the subscript for jet and \( cf \) as the subscript for crossflow properties.

From the temperature distribution in the hot air source (Figure 3.17) and the axial velocity distribution (Figure 3.18(a)), the volume-flow-averaged density in the hot air source becomes \( \rho_j = 0.462 \text{kg/m}^3 \) and the area-averaged velocity is \( W_j = 2.71 \text{m/s} \). The density of the crossflow is evaluated to be \( \rho_{cf} = 1.18 \text{kg/m}^3 \). Both densities are computed at the ambient pressure of one atmosphere. The density ratio is then given by:

\[
\frac{\rho_j}{\rho_{cf}} = 0.39 \quad (3.15)
\]

Given the geometry, the area ratio is:

\[
\frac{A_j}{A_{cf}} = 0.049 \quad (3.16)
\]

The velocity ratio is computed with the averaged crossflow velocity \( U_{cf} = 0.73 \text{m/s} \) found in the \( x = -1.25 \text{m} \) plane and is:

\[
\frac{W_j}{U_{cf}} = 3.72 \quad (3.17)
\]

From these three independent parameters, the mean jet-to-crossflow momentum flux ratio, \( J \), as defined by Karagozian (2003), and the ratio of momenta, \( J^* \), can be computed:

\[
J = \frac{\rho_j W_j^2}{\rho_{cf} U_{cf}^2} = 5.41 \quad (3.18)
\]

\[
J^* = J \frac{A_j}{A_{cf}} = 0.27 \quad (3.19)
\]

The jet-to-crossflow momentum flux ratio is a pressure ratio. As it does not include the area ratio, it is more suitable for unconfined jet in crossflow configurations, whereas the ratio of momenta represents a force ratio and is only suitable for confined configurations.

The mass flow ratio follows from Eqs. (3.15)-(3.17):

\[
\frac{\dot{m}_j}{\dot{m}_{cf}} = 0.073 \quad (3.20)
\]

and is reasonably small. This is important, as the experiment is designed to be a generic test case of a fire in a confined space. In such a scenario, only the mass from the burnt fuel is added.
Knowing $\dot{m}_j$, the volume flow rate of the cold air which is supplied to the heater can be computed. It turns out to be almost exactly $2m^3/min$ which is the set point of the PID controller. The error is smaller than one percent.

Also the dimensionless heat release rate and the dimensionless flame height can now be computed using the temperature distribution within the hot air source (Figure 3.17) and the axial velocity distribution (Figure 3.18(a)):

\begin{align}
\dot{Q}^* &= 1.05 \\
L/D &= 2.76
\end{align}

The Froude number, $Fr$, is closely related to the dimensionless heat release rate and often found in literature. It is a measure for the ratio of inertial forces and buoyancy forces and is defined according to Woodburn (1995) as:

\begin{align}
Fr &= \frac{U_{cf}}{\sqrt{g' L_c}} \\
g' &= g \frac{\rho_{cf} - \rho_j}{\frac{1}{2}(\rho_{cf} + \rho_j)}
\end{align}

where $g'$ is the reduced gravitational acceleration and $L_c$ a characteristic length scale. In this case the length scale is chosen to be the tunnel height, $L_c = 0.8m$, and the Froude number turns out to be $Fr = 2.07$.

The crossflow Reynolds number represents the ratio of the inertia forces to the viscous forces within the crossflow and is defined as:

\begin{align}
Re &= \frac{U_{cf} L_c}{\nu_{cf}} = 37400
\end{align}

### 3.4.5 Discussion

At this point, a few comments about the LDA measurements are given. It has to be stated, that the applied apparatus has high losses in the optical path of one component due to the ageing of the fiber optics used. This channel is then chosen to measure the $y$ component of the velocity in order to get high data rate for the $x$ component.

Another problem is the decay of the data rate towards the ceiling of the tunnel. At the bottom of the tunnel, data rates up to 150Hz can be achieved for the $x$ component whereas the data rate at the ceiling is on the order of 1Hz. This effect is observed both in the upstream cross-sections at $x = -1.25m$ and at $x = -0.6m$ and at all downstream locations. By eye, the laser beams are clearly visible throughout the whole tunnel cross-section. In order to minimize scattering effects of the ceiling, it has been painted black. Nevertheless, the problem remains and it is believed to be a de-focus effect of the optical paths due to the optical quality of the applied bottom glass plates. A slight angle error induced by the glass plates would cause the beams to de-focus. This effect would also increase with increasing $z$ of the measurement position.

Due to the comparably low data rate, only time-averaged values can be presented. The statistics of the second moments are not sufficient.
Chapter 4

Numerical simulations

In this section, the numerical simulations are discussed and compared with experimental data. Details of the CFD solver and the applied discretisation schemes are given in Section 2.7.

At first, the boundary conditions are described followed by a grid independence study. In a further section, the influence of $P_{kb}$ (Eq. (2.24)) on the solution is discussed. In a next step, different steady RANS computations on a grid with non-resolved walls are presented followed by transient simulations (DES, SAS and SST) on a grid with resolved boundary layers.

4.1 Boundary conditions

The computational domain shown in Figure 4.1 comprises the tunnel and a part of the hot air source. It is extended to the measurement plane $x = -1.25$ in the upstream and $x = 8.05m$ in the downstream direction including the contraction upstream of the extraction fan. The domain is extended into the pipe of the hot air source to the measurement plane $z = -0.3m$. All computations are done with a full 3D grid and no symmetry condition is applied.

Figure 4.1: Computational domain.
All walls are set to be adiabatic and a no-slip condition is applied. The $\omega$-based models use the automatic wall function and the $\varepsilon$-based models the scalable wall function, as discussed in Section 2.6.1.

At the crossflow inlet ($x = -1.25\text{m}$), the inlet velocity is prescribed to be 0.73m/s in the free stream, as found by measurement, and a polynomial fit of the boundary layer profile given in Figure 3.21(b) is applied at the walls. The inlet temperature is set to 26°C and the turbulence intensity to 0.026 as found in Figure 3.20(d). The integral length scale is set to 0.04m which is 5% of the hydraulic diameter of the tunnel.

At the hot air source inlet plane, the temperature distribution given in Figure 3.17 is imposed. The velocity and turbulence intensity distributions are set to be axi-symmetric according to the profiles given in Figure 3.18. The integral length scale is set to 0.01m which is 5% of the inlet pipe diameter.

At the outlet, the average static pressure is set to ambient pressure.

In Section 2.6.3 different options of how to specify the dissipation rate, $\varepsilon$, are given. The method based on the autocorrelation function cannot be applied as the data rate of the LDA-measurements is not high enough. The second method using the velocity gradient is rejected as the velocity gradient almost vanishes over a large portion of the flow field. Finally, the empirical method is used specifying the integral length scale to be 5% of the hydraulic diameter which leads to reasonable non-dimensional eddy viscosity ratios as well.

### 4.2 Grid independence study

In this section, four different hexahedron meshes with non-resolved walls are used in combination with the SST model with inclusion of the $P_{kb}$ term in both the $k$ and the $\varepsilon$ or $\omega$ equations. This model is denoted as the SST PD model within this thesis. In order to save grid elements and to get better resolution of the jet vicinity, general grid interfaces (GGI’s) are used to split the domain in four parts as shown in Figure 4.1: The blue colored inflow domain ($-1.25\text{m} \leq x \leq -0.5\text{m}$), the red colored hot jet domain ($-0.5\text{m} \leq x \leq 2.75\text{m}$), the green colored downstream domain ($2.75\text{m} \leq x \leq 7.55\text{m}$) and the yellow colored nozzle domain ($7.55\text{m} \leq x \leq 8.05\text{m}$). In Table 4.1 the number of hexahedra is given for the different meshes. The nozzle domain mesh size is kept constant for this refinement study. Typical values for the wall distance $y^+$ of the grid nodes adjacent to the walls are in the range of 10 to 30. The hot air inlet plane, a cross-section at $x = -0.5\text{m}$ and a zoom-in of the symmetry plane of the grid labeled *medium* are shown in Figure 4.2.
Table 4.1: Mesh sizes (number of hexahedra).

<table>
<thead>
<tr>
<th>Grid</th>
<th>inflow</th>
<th>hot jet</th>
<th>downstream</th>
<th>nozzle</th>
<th>total</th>
</tr>
</thead>
<tbody>
<tr>
<td>coarser</td>
<td>12988</td>
<td>198180</td>
<td>61292</td>
<td>33696</td>
<td>306156</td>
</tr>
<tr>
<td>coarse</td>
<td>21084</td>
<td>315480</td>
<td>110400</td>
<td>33696</td>
<td>480660</td>
</tr>
<tr>
<td>medium</td>
<td>36348</td>
<td>527154</td>
<td>191828</td>
<td>33696</td>
<td>789026</td>
</tr>
<tr>
<td>fine</td>
<td>67712</td>
<td>1007784</td>
<td>347776</td>
<td>33696</td>
<td>1456968</td>
</tr>
</tbody>
</table>

(a) Hot air pipe. 
(b) Tunnel cross-section. 
(c) Tunnel symmetry plane.

Figure 4.2: Medium grid with non-resolved walls.

All steady simulations are computed using the high resolution scheme for the advection terms as discussed in Section 2.7.1 and converge very well on those meshes. The temperature and velocity distributions at downstream locations, where measurements are available, are given in Figures 4.4 and 4.5, respectively. They show almost no influence of the mesh size on the result with exception of the $U$ velocity component distribution at far downstream locations. It can be stated that the mesh resolution is within the asymptotic range for the applied meshes. For further investigations, the mesh labeled medium is chosen.

In addition, the medium mesh is modified changing only the near wall resolution to resolve the walls as shown in Figure 4.3. This mesh comprises of 1208346 hexahedra.
Numerical simulations

and shows typical $y^+$ values smaller than 2. For this mesh, no steady solution is obtained. The results of the transient simulations on this mesh are discussed in Section 4.5.

Figure 4.3: Medium grid with resolved walls.
4.2 Grid independence study

Figure 4.4: Temperature profiles at downstream positions in the symmetry plane $y = 0m$ computed with the SST PD model using different grid resolutions and the automatic wall function.
Figure 4.5: Velocity $U$ profiles at downstream positions in the symmetry plane $y = 0\text{m}$ computed with the SST PD model using different grid resolutions and the automatic wall function.
4.3 Influence of $P_{kb}$

In this section, the effect of the term $P_{kb}$ on the solution of the CFD simulation is discussed. As described in Section 2.3.3, this term models the influence of the stratification on the production and dissipation of $k$ and $\varepsilon$ or $\omega$. Three different SST simulations have been conducted using the non-resolved medium grid. All simulations are computed using the high resolution scheme for the advection terms as discussed in Section 2.7.1. In the first simulation, $P_{kb}$ is neglected. This simulation is referred to as SST. In the second simulation, labeled as SST P, $P_{kb}$ is included in the $k$ equation only. In the third simulation, SST PD, $P_{kb}$ is included in both the $k$ and $\varepsilon$ or $\omega$ equations with the restriction that it is only taken into account for the $\varepsilon$ or $\omega$ equation if $P_{kb} > 0$.

Figures 4.7 and 4.8 show the resulting $U$ velocity and dimensionless temperature distribution. It can be stated that the difference of the three models increase in the downstream direction and that the SST run shows the highest damping. Its velocity and temperature profiles towards the tunnel outlet are smoother compared to the SST P and SST PD runs. Recalling Eq. (2.24), $P_{kb}$ is almost everywhere negative in the flow field as both the vector of the gravitational acceleration and the gradient of the density distribution point in the negative $z$ direction, as shown in Figure 4.6. Only in the injection region of the hot jet, $P_{kb}$ can be positive. Including $P_{kb}$ in the transport equation Eq. (2.18) for $k$, as it is done for the SST P and SST PD runs, the term represents a sink and the $k$ values will be lowered. This behavior can be found in the $k$ distributions given in Figure 4.9. The models SST P and SST PD show similar $k$ profiles and the values are substantially lower than the ones for the SST model.

![Figure 4.6: Schematic view of the plume within the symmetry plane and the density gradient distribution.](image)

As discussed, $P_{kb}$ is less than zero almost everywhere in the flow field and will be set to zero by the given formulation in the $\varepsilon$ or $\omega$ equation. The $\varepsilon$-distribution is only slightly changed by the limited regions of positive $P_{kb}$ values as can be seen in Figure 4.10. Recalling the definition of the eddy viscosity, Eq. (2.22), it can be
stated that reducing $k$ leads to a reduction in $\mu_t$. Figure 4.11 shows the distribution of the eddy viscosity normalized by the laminar viscosity at the tunnel inlet. The values for the SST P and SST PD model are significantly lower than those for the SST model.

The physical principle behind the $P_{kb}$ term is as follows: Stable stratification leads to damping of the velocity fluctuations and hence of the turbulent kinetic energy. Having less fluctuations, the cross-stream-wise momentum exchange is damped and the fluid shows a smaller turbulent viscosity.

In Figure 4.12 the vertical and lateral motion of the flow is given showing the in-plane streamlines at six downstream $x$ positions. The color indicates the magnitude of the $x$ component of the local vorticity vector. Two different value ranges are employed. At $x = 1m$, the counter-rotating vortex pair can clearly be identified. At $x = 2m$ and $x = 3m$, the vortices induce a velocity in negative $z$ direction in the symmetry plane. This motion is responsible for the transport of fluid with high turbulent kinetic energy, $k$, towards the floor as it can be seen in Figures 4.9(b) and 4.9(c).
4.3 Influence of $P_{kh}$

Figure 4.7: Velocity $U$ profiles at downstream positions in the symmetry plane $y = 0$ m computed with the SST, SST P and SST PD model.
Figure 4.8: Temperature profiles at downstream positions in the symmetry plane \( y = 0 \) m computed with the SST, SST P and SST PD model.
4.3 Influence of $P_{kh}$

Figure 4.9: Turbulent kinetic energy $k$ profiles at downstream positions in the symmetry plane $y = 0$ m computed with the SST, SST P and SST PD model.
Figure 4.10: Turbulent eddy dissipation $\varepsilon$ profiles at downstream positions in the symmetry plane $y = 0$ m computed with the SST, SST P and SST PD model.
4.3 Influence of $P_{kh}$

Figure 4.11: Dimensionless eddy viscosity profiles at downstream positions in the symmetry plane $y = 0$ m computed with the SST, SST P and SST PD model.
Numerical simulations

Figure 4.12: In-plane streamlines of the SST PD model at downstream locations colored with the $x$ component of the local vorticity vector which is $(\frac{\partial V}{\partial y} - \frac{\partial W}{\partial z})$ [s$^{-1}$]. The view is in negative $x$ direction.
4.4 Steady RANS simulations

In this section, the flow is computed on the medium grid (Table 4.1) using the $k$-$\varepsilon$, the $k$-$\omega$, the SST and the BSL-RSM model, which is referred to as RSM model in the plots. The models are described in Section 2.3. The $P_{kb}$ term is taken into account in each simulation which is indicated by PD in the plots. The advection terms are treated with the high resolution scheme as discussed in Section 2.7.1.

In Figure 4.13, the dimensionless temperature distribution at $(x, y) = (0.5\, \text{m}, 0\, \text{m})$ and at $(x, y) = (0.5\, \text{m}, 0.2\, \text{m})$ is given for all simulations. The measurements show a significant hot temperature layer at the ceiling. This layer indicates the existence of a backflow of hot air at the ceiling within this cross-section. It can more clearly be identified in Figure 4.18(a), where the temperature measurements of all planes are plotted in a 3D view. It has to be stated that the first measurement point for the temperature measurement on the bottom is located at $z = 0.05\, \text{m}$. The distance from the side walls of the tunnel is $5\, \text{cm}$ as well. Only at the ceiling the measurement points close to the surface are available. For illustration, edges of the measurement planes are indicated in Figure 4.18(a) by black lines at the walls and the floor for all cross-sections. The first plane is located at $x = 0.5\, \text{m}$ and the hot backflow region is clearly visible. However, none of the simulations is able to reproduce this backflow accurately. The models only predict some hot flow primarily in the ceiling corners of the tunnel. The dimensionless ceiling temperature distribution is given in Figures 4.20. For the $k$-$\varepsilon$ PD model a different scale is used as it predicts the highest ceiling temperatures of the discussed models followed by the RSM PD model. The over-prediction is clearly visible in Figure 4.21, where the dimensionless ceiling temperature distribution at $y = 0\, \text{m}$ is shown. From the backflow point of view, all the models behave in a similar way. The backflow front location in the center of the tunnel is almost the same. Only the $k$-$\omega$ PD model predicts the location at a more downstream position and shows weaker backflow within the ceiling corners compared to the other models.

In Figures 4.14 and 4.15, the dimensionless temperature distributions further downstream at $y = 0\, \text{m}$ and $y = 0.2\, \text{m}$, respectively, are given. Again, none of the models is able to predict the flow accurately. Comparing the 3D views given in Figures 4.18 and 4.19, it can be stated that especially the $k$-$\varepsilon$ and the RSM models significantly overpredict the ceiling temperature for this flow configuration. In contrast, the $k$-$\omega$ model mixes the flow much better in the region between the hot air source and the $x = 1.5\, \text{m}$ cross-section, leading to a better ceiling temperature distribution.

Common to all simulations is the fact that the spatial temperature variation within the cross-sections downstream of $x = 1.5\, \text{m}$ is too high. The measurements show a highly stratified flow and low spatial variation in this part of the flow field.

In Figures 4.16 and 4.17, the $x$ component of the velocity at $y = 0\, \text{m}$ and $y = 0.2\, \text{m}$ is given for both the measurements and the simulations. All simulations fail in accurately predicting the velocity field. As a consequence of conservation of mass and energy and due to damping, the deviation from the measurements becomes smaller further downstream in the tunnel.
Figure 4.13: Dimensionless temperature profiles within the plane $x = 0.5\text{m}$ computed with different turbulence models.
Figure 4.14: Dimensionless temperature profiles at downstream positions in the symmetry plane $y = 0\text{m}$ computed with different turbulence models.
Figure 4.15: Dimensionless temperature profiles at downstream positions in the plane $y = 0.2m$ computed with different turbulence models.
Figure 4.16: Velocity $U$ profiles at downstream positions in the symmetry plane $y = 0m$ computed with different turbulence models.
Figure 4.17: Velocity $U$ profiles at downstream positions in the plane $y = 0.2\text{m}$ computed with different turbulence models.
4.4 Steady RANS simulations

Figure 4.18: Dimensionless temperature distribution of the experiment and different turbulence model predictions at the $x$ locations from $x = 0.5m$ to $x = 6.5m$ with a spacing of 1m.
Figure 4.19: Dimensionless temperature distribution of the RSM PD model at the $x$ locations from $x = 0.5m$ to $x = 6.5m$ with a spacing of 1m.

Figure 4.20: Dimensionless ceiling temperature distribution of the steady simulations on the grid with unresolved walls. A different scale is used for the $k$-$\varepsilon$ PD model.
4.5 Transient simulations

It can be concluded, that the steady-state simulations are not able to accurately describe the flow field within the tunnel. The predicted entrainment or mixing is too low leading to high ceiling temperatures. The highly stratified temperature layers found in the measurements cannot be modeled accurately. This suggests that transient simulations have to be carried out, which are described in the next section.

4.5 Transient simulations

In this section, three transient computations using the SST PD, the DES PD and the SAS models are compared to the experiments. The models are described in the Sections 2.3.3, 2.4 and 2.5. All simulations are computed on the wall-resolved grid given in Figure 4.3. The SST PD and the DES PD simulations use the automatic wall function method as discussed in Section 2.6.1, whereas the SAS model is integrated through the viscous sublayer.

For the advection scheme of the momentum equation, $\beta_{as} = 1$ is used which leads to a second-order scheme as discussed in Section 2.7.1. In order to avoid temperature over- or undershoots, the advection term of the energy equation is discretized using the high resolution method. For the time integration, a second order backward Euler scheme is applied, as discussed in Section 2.7.2.

The ideal gas equation is modified to make the density, $\rho$, a function of local temperature, $T$, only. This is done by using a constant mean (absolute) pressure, $p_{ref}$, in the ideal gas equation. This is a valid assumption for the given flow situation and avoids the build-up of instabilities due to acoustical pressure waves. The Courant number, $CFL = U_{pr} \Delta t / \Delta L$, where $\Delta L$ is the local grid size, $\Delta t$ the time step and $U_{pr}$ the propagation velocity as defined by Fletcher (2000), can now be computed based on the local fluid velocity instead of the sum of local fluid velocity and the speed of sound, which would lead to drastic time-step restrictions. The Courant number,
number in CFX is computed as:

$$CFL_{CFX} = \frac{\dot{m}_{extCV} \Delta t}{\rho V_{CV}}$$

(4.1)

where $\dot{m}_{extCV}$ is the mass flow rate exiting the control volume, $\Delta t$ the time step, $\rho$ the local density and $V_{CV}$ the volume of the control volume. With the chosen time step of $\Delta t = 0.002$ s, the r.m.s. Courant numbers are of the order of 0.24 and maximum values are below 30 for all transient computations.

### 4.5.1 Time-averaged quantities

In this section, the time-averaged dimensionless temperature and the time-averaged $U$ velocity profiles are shown and compared with the experimental data. The time-averaged distributions of the SAS simulation are not completely symmetric and further averaging may be necessary to obtain a better convergence. For comparison of the 1D profiles given in the Figures 4.24 to 4.28, the values are made symmetric by mirroring and subsequent averaging.

In Figure 4.22, an overview of the dimensionless time-averaged temperature distributions is given. The display range is scaled from zero to 0.18. Values higher than 0.18 are cropped as can be seen in the cross-sections at $x = 0.5$m. These cross-sections are plotted on a different scale in Figure 4.23 for comparison. Looking at Figure 4.22, it can be stated that the time-averaged dimensionless temperature predictions from both the transient SST PD and the DES PD simulations are not as much stratified as the experimental data. The DES PD simulation is in this sense slightly superior to the SST PD one. In contrast, the SAS simulation is successful in predicting the stratified flow in the far field. Even the shapes of the iso-lines within the planes fit the ones of the experiment. Only the temperature levels at the ceiling are slightly too high. This discrepancy could stem from the adiabatic wall treatment in the simulations and the non-ideal adiabatic walls in the experiment. The heat flux through the walls in the experiment can be estimated by a 1D heat conduction calculation and the material properties specified in Table 3.1. Assuming a hot layer temperature of 80°C, the heat flux is about 22W/m². Depending on the assumed heat transfer coefficient, the ceiling temperature is of the order of 2°C to 4°C colder compared to an adiabatic wall. For the tunnel section under investigation, the total heat power loss through the walls is of the order of 1% to 2% of the heat release rate $\dot{Q} \approx 20.9$ kW. Hence, the assumption of adiabatic walls is reasonable for a first comparison and the influence of the turbulence model choice will dominate the influence of heat loss. However, better agreement between computations and measurements could be obtained if heat conduction through the walls were included.

Comparing the cross-sections at $x = 0.5$m in Figure 4.22 it can be stated that only the SAS model is capable of predicting the hot temperature layer at the ceiling over the whole tunnel width. The DES PD model shows some hot air in corners only.

Comparing the time-averaged dimensionless temperature distribution in Figure 4.23, it can be stated that all simulations show higher peak temperatures than observed...
transient simulations  

4.5 Transient simulations

in the measurements. It has to be mentioned that the measurement point spacing is 5cm. In high gradient regions, such as close to the vortex cores, this spacing is too coarse to get the peak temperature right. Looking at the vertical location of the cores, the DES PD and SAS predictions are superior to the SST PD ones.

Although the SAS predicts the temperature distribution in the far field of the jet very well, it has problems in the near jet region. A possible cause could be the inertia of the SAS model in switching to the scale-resolved mode in the jet region. Another source of error could be the underlying eddy viscosity assumption of the SAS model which is problematic in the near jet region as there are high temperature gradients and strong streamline curvatures resulting in highly anisotropic turbulence, as discussed in Section 2.3.3.

In Figure 4.24, the time-averaged dimensionless temperature profiles at \((x, y) = (0.5m, 0m)\) and at \((x, y) = (0.5m, 0.2m)\) are shown. Close to the ceiling, a hot air layer can clearly be identified. The SAS model simulation shows the best agreement. Nevertheless, all three models deviate from the exact shape of the profiles.

In Figures 4.25 and 4.26 the time-averaged dimensionless temperature profiles at six downstream positions for \(y = 0m\) and \(y = 0.2m\), respectively, are given. From these plots it can be seen that the SAS model is in good agreement with the experimental data and that the DES PD model is superior to the SST PD one.

The same is found by comparing the time-averaged \(U\) velocity profiles given in Figures 4.27 and 4.28. The deviation from the measurements is largest in the cross-section \(x = 1m\).

In Figures 4.29 and 4.30 the time-averaged dimensionless ceiling temperature distribution is given. Compared to the results of the steady computations given in Figures 4.20 and 4.21, the transient SST PD and DES PD models predict the onset of the hot layer at the ceiling at about the same position. Only the SAS model deviates significantly showing backflow to just upstream of the jet inlet position. Comparing the transient SST PD and the DES PD results with the stationary ones, it can be stated that the backflow region in the ceiling corners is more pronounced in the transient computations. Especially the DES PD simulation shows a significant amount of backflow which reaches about the same position as in the SAS model.
Figure 4.22: Time-averaged dimensionless temperature distributions of the experiment and different turbulence model predictions at the $x$ locations from $x = 0.5\text{m}$ to $x = 6.5\text{m}$ with a spacing of 1m.
4.5 Transient simulations

Figure 4.23: Time-averaged dimensionless temperature distributions at $x = 0.5\text{m}$. 
Figure 4.24: Time-averaged dimensionless temperature profiles within the plane $x = 0.5\text{m}$ computed with different turbulence models.
Figure 4.25: Time-averaged dimensionless temperature profiles at downstream positions in the symmetry plane $y = 0\text{m}$ computed with different turbulence models.
Figure 4.26: Time-averaged dimensionless temperature profiles at downstream positions in the plane $y = 0.2m$ computed with different turbulence models.
Figure 4.27: Time-averaged velocity $U$ profiles at downstream positions in the symmetry plane $y = 0 \text{m}$ computed with different turbulence models.
Figure 4.28: Time-averaged velocity $U$ profiles at downstream positions in the plane $y = 0.2\text{m}$ computed with different turbulence models.
4.5 Transient simulations

Figure 4.29: Time-averaged dimensionless ceiling temperature distribution of the transient simulations on the grid with resolved walls.

Figure 4.30: Time-averaged dimensionless ceiling temperature distribution at \( y = 0 \text{m} \) of the transient simulations on the grid with resolved walls.

4.5.2 Instantaneous quantities

Figure 4.31 shows the main vortical structures of a jet-in-crossflow configuration as reported e.g. by Fric and Roshko (1994): The counter-rotating vortex pair, the
jet shear-layer vortices, the system of horseshoe vortices, and the wake vortices. From flow visualizations, Fric and Roshko (1994) found out that the wake vortices are formed from the boundary layer vorticity of the crossflow wall. The formation process takes place at either side of the jet and just downstream of it. It is caused by a separation process of the crossflow wall boundary layer in the adverse pressure field which is imposed by the external flow. The external flow itself does not separate from the jet but closes around it. The resulting tornado-like vortical structures emerge from the boundary layer at one side and are entrained by the jet at the other side. The associated vortex shedding has been found to be most regular and pronounced at a jet-to-crossflow velocity ratio of $\frac{W_j}{U_{cf}} = 4$, which is close to the value of 3.72 for the flow under investigation. The resulting Strouhal number found by Fric and Roshko (1994) is of the order of 0.13 for a wide range of Reynolds numbers which is well below the measured value of 0.3 (Section 3.4.3). It is also reported that not only the jet-to-crossflow velocity ratio could have an effect on the Strouhal number but also the interaction with, e.g., the jet shear-layer or the horseshoe vortices. Kelso and Smits (1995) investigated the Strouhal numbers of the horseshoe vortex. It turned out to have the same or double the Strouhal number of the wake vortices depending on the Reynolds numbers. In addition, they reported the Strouhal number of the jet shear-layer vortices to be much higher than the Strouhal number of the wake vortices and a dependence on the jet-to-crossflow velocity ratio.

Figure 4.31: Schematic view of the vortical structures in a jet-in-crossflow configuration as reported by Fric and Roshko (1994).

Both investigations were made in a jet-in-crossflow configuration without confinement. The effect of confinement on the Strouhal number is studied by Richter and Naudascher (1976) for a cylinder in crossflow configuration. The shedding phenomenon for this flow configuration differs from the jet-in-crossflow shedding as it is caused by the separation of the cylinder boundary layer. The Strouhal number
is found to increase with increasing blockage. Considering the jet-in-crossflow configuration, the blockage by the hot air jet and the hot air layer at the ceiling force the incoming crossflow to turn towards the floor and to accelerate as the effective cross-sectional area is reduced. Hence, the velocity adjacent to the jet is substantially higher than the averaged inlet velocity. In the SAS simulation, values of about 1 m/s are reached next to the jet. For the shedding phenomenon, the velocity close to the jet is important. Based on this velocity, the Strouhal number would be lower. In addition, the available space for the vortex motion is reduced by both walls and the hot air layer at the ceiling which could also influence the shedding frequency. Lam and Cheung (1988) reported this phenomenon as “narrowing effect” for the flow around three cylinders which are arranged in a triangular position with equidistant spacing. In their configuration, one cylinder was placed upstream and the other two at the same downstream position. The shedding frequency of the upstream cylinder was found to be strongly dependent on the spacing ratio such that a reduction in spacing led to an increase in the shedding frequency.

In Figure 4.32 the instantaneous vortical structures are visualized by rendering isosurfaces of the second invariant $I_v$ of the velocity gradient tensor (Jeong and Hussain (1995)) at the level $75s^{-2}$ and coloring the structures with the instantaneous local dimensionless temperature. The second invariant can be written as:

$$I_v = \Omega^2 - S^2$$

$$S = \sqrt{2S_{ij}S_{ij}}$$

where $\Omega$ is the absolute value of the vorticity vector, $S$ the shear strain rate and $S_{ij}$ the strain rate tensor, as defined in Eq. (2.11).

The SST PD simulation shows some of the dominant structures, as discussed by Fric and Roshko (1994): The counter-rotating vortex pair, the wake vortices and some jet shear-layer vortices. For the specified value of $I_v$, the shear-layer vortices are not closed at the jet upstream position and are staggered on the two counter-rotating vortices. The decay of the vortical structures into smaller ones cannot be observed. Only the dominant modes are present. This behavior is also documented in the literature: According to Menter and Egorov (2004), the unsteady RANS models (URANS) can only predict structures that are of the size of the shear-layer thickness as the turbulence length scale in URANS is proportional to the shear-layer thickness and not to the size of the resolved scales, which has been demonstrated for a cylinder in crossflow configuration. Spalart (2000) states that the URANS methods tend to provide solutions with only one dominating mode whereas the experiments show a spectrum of scales. Therefore, the URANS results are often unphysical. In addition, the $k-\varepsilon$, $k-\omega$, and combined models are based on the eddy viscosity hypothesis which is problematic in flows with high streamline curvature as discussed in Section 2.3.3.

In contrast to the SST PD model, the DES PD model shows some detailed structures and the SAS model even more. They both form shear-layer vortices which are closed at the jet upstream position. In the SAS result, Figure 4.32(c), the vortices in the hot air layer at the ceiling can even be identified which form the backlayering with its recirculation zone.
Figure 4.32: Visualization of the vortical structures by rendering the iso-surfaces of the invariant at $I_v = 75s^{-2}$. The iso-surfaces are colored with the instantaneous dimensionless temperature. Values higher than 0.2 are colored red.
In Figure 4.33 the instantaneous blending function values in the symmetry plane are given for the DES PD and the SAS model. In regions with a value of 1, the models operate in the RANS mode and for values below 1, the scale-resolved mode is blended in. The DES PD model works in the LES mode only close to the hot jet source and switches back to the SST PD formulation further downstream. Also in the wall vicinity, the model operates in the SST PD mode. This region extends deeply into the core flow. In contrast, the SAS model operates in the scale-resolved mode almost everywhere in the flow field. Only a thin layer close to the wall is treated with the RANS formulation. Hence, the vortical structures predicted by the DES PD model are not as detailed as the ones predicted by the SAS model. The grid is too coarse to activate the DES PD model to work in the LES mode in many parts of the flow domain. On the same grid, the SAS model is able to resolve the vortical structures.

Figure 4.33: Blending function between the RANS mode (=1) and the scale-resolved mode (=0) within the symmetry plane \( y = 0 \text{ m} \).

As the DES PD model works in the SST PD mode in a large part of the flow domain, the results show similar tendencies as the pure SST PD calculation. This can be seen by looking at the shedding frequencies of the wake vortices given in Table 4.2. In Figure 4.34 two scatter plots of the dimensionless temperature and the velocity component \( V \) predicted by the DES PD model are given for two different \( z \) locations at \( x = 1 \text{ m} \) and \( y = 0 \text{ m} \). The locations are downstream of the hot air
source in a region where shedding of vortices occurs. Hence, both the dimensionless
temperature and the velocity component \( V \) are periodic in time. Close to the floor,
at \( z = 0.1 \text{m} \), the temperature fluctuations are much smaller compared to those at
\( z = 0.3 \text{m} \). In both plots it can be seen that for one cycle through the scatter plot,
the dimensionless temperature shows two oscillations and the velocity component \( V \)
only one. Hence, the shedding frequency of the temperature signal is twice the one
of the \( V \) velocity component.

<table>
<thead>
<tr>
<th>((x, y, z))</th>
<th>((0.5 \text{m}, 0 \text{m}, 0.1 \text{m}))</th>
<th>((1 \text{m}, 0 \text{m}, 0.1 \text{m}))</th>
</tr>
</thead>
<tbody>
<tr>
<td>Variable</td>
<td>(T)  (V)</td>
<td>(T)  (V)</td>
</tr>
<tr>
<td>SST PD</td>
<td>2.8Hz  5.5Hz</td>
<td>2.8Hz  5.5Hz</td>
</tr>
<tr>
<td>DES PD</td>
<td>2.5Hz  5.0Hz</td>
<td>2.5Hz  5.0Hz</td>
</tr>
<tr>
<td>SAS</td>
<td>1.0Hz  2.0Hz</td>
<td>n.a.   2.0Hz</td>
</tr>
<tr>
<td>Experiment</td>
<td>n.a.   2.2Hz</td>
<td>1.1Hz  n.a.</td>
</tr>
</tbody>
</table>

This frequency doubling can also be seen in Table 4.2 where the dominant frequen-
cies of the temperature and \( V \) velocity for different turbulence models and for the
experiment are listed at two different downstream positions. Values denoted by
"n.a." are not available. For the SST PD and DES PD simulations the dominant
frequency of the \( V \) velocity component is the same at both locations and the corre-
spanding frequency of the temperature signal is doubled at a given location as in
the scatter plots in Figure 4.34. The frequencies predicted by the SST PD and the
DES PD models are more than twice as high as those measured whereas the SAS
model predicts the frequency within the error-band of the measurements.
4.5 Transient simulations

\[
\frac{(T - T_{\text{min}})}{(T_{\text{max}} - T_{\text{min}})} \times 10^{-4}
\]

Figure 4.34: Scatter plots of the dimensionless temperature and the V velocity component at two different z locations on the line x = 1m and y = 0m predicted by the DES PD model.

The DES PD model predicts a lower frequency compared to the SST PD model indicating the correct trend. A grid refinement could make the DES PD model working in the DES mode rather than in the SST mode and could lead to better results but would also lead to a dramatic increase in computational demands. However, the SAS model is able to accurately predict the frequencies and the temperature profiles on the same grid.

In order to further investigate the behavior of the three applied models, data of three monitor points close to the ceiling are compared. Figure 4.35 shows the scatter plots and spectra of the dimensionless temperature and the V velocity component at the three downstream positions \(x = 1\)m, \(x = 2\)m and \(x = 3\)m within the symmetry plane \(y = 0\)m and at \(z = 0.7\)m. Data of the SST PD model are given in blue, the ones of the DES PD model in green, and the SAS data in red. For the spectra of the dimensionless temperature, a line without symbols is used and for the V velocity component, triangle-shaped symbols are chosen.

From both the scatter plots and the spectra, it can be seen that the SST PD model shows smallest variations at all locations. The dominant frequency is close to 3Hz for both the temperature and the velocity and it persists at all locations. The amplitudes are damped further downstream and the higher harmonics are damped out at \(x = 3\)m.

The DES PD model predicts stronger variations than the SST PD model but still more moderate ones compared to the SAS model. The amplitudes of the variations decay faster in \(x\) direction than in the SAS model. The dominant frequency is again the same for both the temperature and the velocity and is slightly lower than the one found for the SST PD model.
The SAS model shows the largest amplitudes for the given models at all three locations. The damping of the amplitudes is smaller than for the SST PD and DES PD cases. The dominant frequency is close to 1Hz for the $V$ velocity component and close to 2Hz for the temperature. Higher harmonics are not as pronounced as compared to the other models and are damped out in the downstream direction.

The SST PD model shows one pronounced dominant structure with higher harmonics in the spectral plots. This structure remains in the downstream direction and only the higher harmonics are damped out. The SAS model shows also one dominant frequency but with less pronounced higher harmonics. In contrast to the SST PD model, the SAS model shows a broader spectrum.

The behavior of the DES PD model is strongly influenced by the SST PD model as it operates in the SST mode in a large part of the flow field. For a better comparison, a simulation on a refined grid would be necessary.
Figure 4.35: Scatter plots and power spectra of the dimensionless temperature (line without symbols) and $V$ velocity component (triangle symbols) at three downstream positions within the symmetry plane, $y = 0\text{m}$, at $z = 0.7\text{m}$ for the SST PD (blue), the DES PD (green) and the SAS (red) models.
Chapter 5

Conclusions

This thesis investigates the degree of fidelity with which different turbulence models can reproduce the flow field in a fire scenario. Therefore, a hot-jet-in-crossflow experiment was built. The performed measurements are compared to CFD simulations with different turbulence models.

5.1 Experiment

For the turbulence model validation of CFD fire simulations a hot-jet-in-crossflow configuration is chosen. It is a generic test case which avoids chemical reactions within the flow but still produces the major fluid structures such as a hot plume with entrainment and interaction with ventilation. Moreover, well defined boundary conditions for the numerical simulations can be obtained from measurements. Having no chemical reactions, the heat release does not have to be modeled and the turbulence model aspect can be studied independently.

The experimental setup is designed in a modular way and the tunnel is also tiltable allowing for a variety of further experimental configurations. The ceiling and the walls are made of a highly insulating material and the bottom glass plates allow for optical access which is needed for LDA measurements. The bottom plates do not have to insulate as the hot fluid spreads along the upper part of the tunnel due to buoyancy. Aluminum plates separate the glass plates and can be replaced by an insert that holds a thermocouple rake for temperature measurements.

The hot air source consists of a blower and an electric heater which is safely and easily controllable compared to a flame heater. Several screens made of stainless steel are installed to damp out spatial velocity variations in the hot gas stream. The measurement plane within the hot jet source is 1.5 diameters upstream of the hot jet outlet as the flow field in the exit cross-section is largely influenced by the crossflow leading to secondary flow. At the upstream position, this is not the case.

The sensing junctions of the applied K-type thermocouples are directly exposed to the flow. This is critical as heat conduction along the probe shaft can be minimized to make reliable measurements in regions with high temperature gradients.
In addition, it minimizes thermal inertia which assures a high frequency response for accurate temporal data acquisition.

Facing temperatures up to 500°C, oil seeding could not be applied for the LDA measurements and also ceramic powders could not be used due to health considerations. Ordinary cooking salt crystals are successfully applied instead. They are both cheap in production and safe for the operators. The self-designed ultrasonic atomizer system allows for continuous large volume flow rate seeding. The crystal sizes are analyzed by a laser diffraction based particle size analyzer and it has been shown that the particle size can be controlled by the salt concentration. Hence, the particle size can be tuned to the experimental requirements.

From the measurements, a generic data set is derived for validation containing velocity components and temperature readings. LDA measurements were performed both in the inlet cross-section of the longitudinal tunnel and in the measurement plane of the hot jet. Besides the mean velocity distributions, also r.m.s. values are obtained. Together with the temperature measurements in the hot jet, they define the boundary conditions for CFD calculations. For validation, the temperature distributions in seven vertical planes downstream of the jet were measured. In addition, several velocity profiles were recorded at six downstream positions. From these measurements, the Strouhal number of the wake vortices, based on the jet diameter and the tunnel inlet velocity, is found to be 0.3, which is more than twice the value reported in the literature for an unconfined isothermal jet in crossflow. The highly confined nature of the flow is believed to increase the Strouhal number as it reduces the effective cross-sectional area and increases the crossflow velocity. In addition, the lateral motion of the vortices is restricted by the tunnel walls. Similar effects are also reported in the case of cylinders in confined crossflows.

5.2 Numerical simulations

In this thesis, both steady computations on a grid without resolved wall boundary layers and transient simulations on a grid with resolved wall boundary layers were performed. In the case of the steady computations, the \( k-\varepsilon \), the \( k-\omega \), the SST and BSL Reynolds-stress models were applied. All models converged very well in steady simulations on the non-resolved grid, whereas they do not converge on the resolved grid. Having comparatively low Reynolds numbers, the log-layer of the boundary layer is thin and the non-resolved grid has only a few grid points within this layer. According to Fric and Roshko (1994), the incoming boundary layer is responsible for the formation of the wake vortices. Suppressing these vortices leads, amongst other effects, to a more stable flow field and to a converged steady solution.

The results of the steady simulations show too high ceiling temperatures, and the spatial temperature distributions within the measurement planes are not sufficiently stratified compared to the time-averaged measurements. The \( x \) position of the backflow region at the ceiling is too far downstream and similar for all turbulence models. Comparing the temperature distribution in the cross-section at \( x = 1.5 \)m, the \( k-\varepsilon \) model shows the highest temperatures. The entrainment of cold air from the cross-
flow into the hot jet is underestimated in all the applied models leading to too high temperatures. A reason for this might be the - in reality existing - vortical structures close to the jet, such as the shear-layer vortices, which are not resolved in the steady simulations. They would largely contribute to the entrainment as is seen in the unsteady simulations.

The influence of the production term $P_{kb}$ is discussed for the SST model in a steady simulation. It has been shown that it models the effect of stable stratification which leads to weaker momentum exchange in the direction of the gravitational acceleration vector and thus to lower eddy viscosity. This, in turn, leads to even larger temperature variations in the downstream region which are not observed in the measurements. It has to be stated that the inclusion of the $P_{kb}$ term is physically correct but leads - in the case of the steady simulations - to even worse results. But in these simulations, the major problem is that the large-scale vortices are not resolved. The inclusion of $P_{kb}$ just amplifies the effect.

The SST, the DES, and the SAS models are applied with a wall-resolved grid. These simulations are transient. The SST model resolves fewer turbulent structures compared to the DES and SAS simulations, leading to only a slight improvement compared to the steady SST simulation on the non-resolved grid. The vortical structures of the shear-layer vortices are staggered upstream of the jet. In contrast, the DES and the SAS models predict them to be closed. From the blending function it can be seen that the applied grid is not fine enough for the DES model. It switches back to the SST mode in a large part of the flow domain. Refining the grid is assumed to be necessary for a better performance of the DES model. As an example, the Strouhal number of the wake vortices is more than twice the measured value in DES but is correctly obtained by the SAS simulation on the same grid. The SAS model is found to be the only one that shows backflow at the ceiling at $x = 0.5\text{m}$ across the whole tunnel width which is consistent with the measurements. Also, the spatial temperature and velocity distributions compare very well with the measured ones. Only close to the jet do the results deviate significantly from the measurements.

The accurate wall treatment combined with resolved large-scale vortices has been found to be critical for accurate simulations. The large-scale vortices contribute to the entrainment leading to a cooling of the hot gas jet. For fire safety simulations, the accurate prediction of entrainment is crucial as it affects both the volume flow rate of the smoke contaminated air and also the ceiling temperature distribution which is important for both sprinkler activation and structural integrity.

### 5.3 Outlook

It has been found that the simulations done with the SAS model performed well in comparison with the presented experiment. In a next step it would be desirable to investigate whether the good performance of the SAS model is independent of the flow parameters, such as the slope of the tunnel or the jet-to-crossflow momentum ratio. Therefore, new measurements and CFD simulations have to be carried out at altered flow conditions.
Sloped configurations are of particular interest as inclined geometries are common in real structures. They do not only appear in road and rail tunnels but also in buildings, e.g., in staircases. In subway stations, escalator shafts are part of the access and evacuation routes, and smoke control in this geometry is crucial during fire incidents.

For all experimental investigations, the development of a suitable flow visualization technique in hot air would be desirable. Fast visualization would also help in finding interesting operating conditions. A permanent monitoring system of the backlayering position would help to find specific operating points empirically.

From the simulation point of view, it would be useful to further investigate the performance of the DES model on a refined grid. A comparison of the performance of the DES and SAS model in terms of both the quality of the results and the computational cost would be illustrative.
Appendix A

Downstream measurements

A.1 LDA measurements

In this appendix the LDA measurements at various downstream locations are given. The data presented in Section 3.4.3 are obtained by averaging measurements at symmetric $y$ positions.

![Graph](image)

Figure A.1: Measured $V$ velocity and $\sqrt{\overline{\nu v}}$ profiles at $x = 1$m. Only measurements with a data rate higher than 1Hz are given.
Downstream measurements

Figure A.2: Velocity $U$ profiles at downstream positions.
Figure A.3: Measured $\sqrt{uu}$ profiles at downstream positions.
A.2 Temperature measurements

The measured time-averaged temperature distribution is made dimensionless using the minimum temperature, \( T_{\text{min}} \), which is the inlet temperature of the crossflow, and the maximum temperature, \( T_{\text{max}} = 503.5^\circ \text{C} \) in the hot air source. The non-dimensional temperature distribution is shown in Figure A.4(a) and Figure A.5.

From the time-resolved temperature signals, the r.m.s. values are computed by high-pass filtering the signal using the \( M'4 \) filter as discussed in Section 3.3.1. The cut-off frequency is chosen to be \( f_{\text{co}} = 1/4.5s = 0.22\text{Hz} \). The corresponding r.m.s. temperature distributions are given in Figure A.4(b) and A.6.

Figure A.4: Measured dimensionless temperature distribution at \( x = 0.5\text{m} \) and the corresponding r.m.s. temperature distribution at \( x = 0.5\text{m} \). Cut-off frequency: \( f_{\text{co}} = 0.22\text{Hz} \).
Figure A.5: Measured dimensionless temperature distribution.
Figure A.6: Measured r.m.s. temperature distribution. Cut-off frequency: \( f_{co} = 0.22 \text{Hz} \).
Bibliography


Björkman, J. and O. Keski-Rahkonen: 1996, Simulation of the Steckler room fire experiment by using SOFIE CFD-model. VTT.


Hall, J. and B. Harwood: 1995, ‘Smoke or burns - which is deadlier’. *NFPA Journal*.


Menter, F. and Y. Egorov: 2004, ‘Revisiting the turbulent scale equation’. In: *Proc. IUTAM Symposium in Göttingen; One hundred years of boundary layer research*. 


Curriculum Vitae

Personal Data
Date of birth October 14, 1977
Native place Appenzell, Switzerland

Education
1984 - 1993 Elementary schools in Wettingen
1993 - 1997 Gymnasium in Baden
July 1997 Matura Typus C

Practical experience
November 1997 Organ builder Goll, Luzern
December 1997 Trainee at hospital, Baden
Jan. - March 1998 School for engineering workshop, Brugg-Windisch
April 1998 Bookkeeping assistant at Schmid Transport AG, Wettingen
May - June 1998 Organ builder Felsberg, Felsberg GR

Studies
Oct. 1998 - Apr. 2003 Study of mechanical engineering at ETH Zurich, Switzerland
Majoring in turbomachinery and fluid dynamics
April 2003 Final diploma with diploma thesis:
Experimental investigation of the unsteady flow around a
blunt body in a narrow channel

Research
Since May 2003 Ph.D.-student at ETH Zurich
Institute of Fluid Dynamics, Prof. Dr. T. Rösgen
in collaboration with Air & Climate, Dr. A. Moser