Plane turbulent impinging jets (PTIJ) at moderate Reynolds numbers are used in a wide range of applications. A typical example in building engineering is an air curtain (AC), which can be used to separate two environments in terms of heat and mass transfer, for example, at entrances of buildings to reduce heat losses, in clean rooms and hospitals to prevent contamination and inside buildings and tunnels to prevent smoke propagation between two or more zones.

This thesis provides high-quality experimental data for a detailed analysis of the fluid dynamics of PTIJ. The experimental data are also used to validate steady Reynolds-averaged Navier-Stokes (RANS) computational fluid dynamics (CFD) simulations of PTIJ flows. The validated model is applied in a generic study in which an isothermal jet separates two environments subjected to a cross-jet pressure gradient. The results of this study provide insights into the separation efficiency of PTIJ and the sensitivity of the separation efficiency to jet height-to-width ratio, jet angle and the ratio of jet discharge momentum flux to jet cross-flow momentum flux (momentum flux ratio). Finally, the separation efficiency of ACs is analyzed for the situation where an AC is subjected to the influence of a vertical wall above the door opening.

Recommendations are provided with respect to the performance of the different RANS turbulence models for predicting PTIJ flows. It is shown that the separation efficiency of PTIJ can be improved by adjusting certain parameters of PTIJ. These recommendations and findings can be relevant in practical applications of ACs and can be used for both future research and AC design and implementation.
Dynamics of plane impinging jets at moderate Reynolds numbers
– with applications to air curtains

PROEFSCHRIFT

ter verkrijging van de graad van doctor aan de Technische Universiteit Eindhoven, op gezag van de rector magnificus, prof.dr.ir. F.P.T. Baaijens, voor een commissie aangewezen door het College voor Promoties in het openbaar te verdedigen op donderdag 12 december 2019 om 11.00 uur

door

Adelya Afgatovna Khayrullina-Doudart de la Grée

geboren te Belebey, Russische Federatie (USSR)
Dit proefschrift is goedgekeurd door de promotoren en de samenstelling van de promotiecommissie is als volgt:

voorzitter: prof.dr.ir. T.A.M. Salet
1e promotor: prof.dr.ir. B. Blocken
2e promotor: prof.dr.ir. G.J.F. van Heijst
copromotor: dr.ir. T. van Hooff
leden: prof.dr.ir. J.L.M. Hensen
        prof.dr.ir. D.M.J. Smeulders
        prof.dr. A.E.P. Veldman (Rijksuniversiteit Groningen)
adviseur(s): dr. J. Moureh (IRSTEA, Institut National de Recherche en Sciences et Technologies pour l'Environnement et l'Agriculture)

Het onderzoek of ontwerp dat in dit proefschrift wordt beschreven is uitgevoerd in overeenstemming met de TU/e Gedragscode Wetenschapsbeoefening.
Reading maketh a full man; conference a ready man; and writing an exact man.

Francis Bacon 1601
# Table of contents

## Table of contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Acknowledgements</td>
<td>iii</td>
</tr>
<tr>
<td>Summary</td>
<td>vii</td>
</tr>
</tbody>
</table>

## Chapter 1 – Introduction

1.1 Plane turbulent impinging jet and air curtain                         1
1.2 Literature study and problem statement                                 5
1.3 Objectives and methodology                                            11
1.4 Dissertation outline                                                  17

## Chapter 2 – PIV measurements of isothermal plane turbulent impinging jets at moderate Reynolds numbers

2.1 Introduction                                                          28
2.2 Experimental set-up                                                   31
2.3 Flow visualization results                                            34
2.4 PIV measurement results                                               35
2.5 Discussion                                                            51
2.6 Conclusions                                                           52

## Chapter 3 – Validation of steady RANS modelling of isothermal plane turbulent impinging jets at moderate Reynolds numbers

3.1 Introduction                                                          60
3.2 Reduced-scale PIV measurements                                        63
3.3 CFD simulations                                                       64
3.4 Results                                                               71
| 3.5 Discussion and future work | 90 |
| 3.6 Conclusions and recommendations | 93 |

**Chapter 4 – Minimum momentum flux ratio required to prevent air curtain breakthrough in case of cross-curtain pressure gradients:**

| 4.1 Introduction | 106 |
| 4.2 Validation study | 111 |
| 4.3 Case studies | 119 |
| 4.4 Results | 123 |
| 4.5 Discussion | 135 |
| 4.6 Conclusions | 137 |

**Chapter 5 – Impact of a wall downstream of an air curtain nozzle on air curtain separation efficiency**

| 5.1 Introduction | 144 |
| 5.2 CFD validation studies | 146 |
| 5.3 Case studies: computational settings and parameters | 152 |
| 5.4 CFD simulations case studies: results | 156 |
| 5.5 Limitations and further work | 170 |
| 5.6 Conclusions | 170 |

**Chapter 6 – Limitations, future work and discussion**

| 6.1 Limitations and future work | 175 |
| 6.2 Discussion | 177 |

**Chapter 7 – Summary and conclusions**

| 7.1 Summary and conclusions | 179 |

**Curriculum vitae**

| 183 |

**List of publications**

| 185 |
Almost 7 years passed between starting my PhD research and writing the first words of this chapter. Almost 9 years ago I arrived to the Netherlands for my master studies in Building Physics and Services. Those were the years that enriched my life with relevant knowledge and discoveries, and a diversity of friends and colleagues from many different countries. I would like to express my gratitude to the many people I was fortunate to meet and to spend my days with during my PhD study.

I owe a debt of gratitude to my supervisors. Dear Bert (prof.dr.ir. Bert Blocken), thank you for the invitation to start PhD research in your group. I still remember that great feeling of excitement at the moment I received this offer, at the same time doubting whether I would manage to perform this task. Thanks for the knowledge and ideas you shared with me and the help you provided me with during my master graduation project and through the whole PhD research. Thank you very much for this great opportunity you created for me! Dear GertJan (prof.dr.ir. GertJan van Heijst), thanks for sharing your knowledge of fluid dynamics and enthusiasm for the complexity of fluid flows. Since I know you, I pay more attention to and admire many natural wonders, like diverse shapes of clouds and vortices of cyclones shown in weather forecasts. Thanks for a nice talk from time to time, your support and sharing your life wisdom. Dear Twan (dr.ir. Twan van Hooff), thanks for the guidance and help throughout my years at university, thanks for your professional feedback and for knowledge and experience you shared with me. Special thanks for your patience and all the invested time with writing the articles and this thesis. Thanks for your commitment and overall responsibility in being my daily supervisor.

Big thanks to the members of the thesis committee, prof.dr.ir. J.L.M. Hensen, dr. J. Moureh, prof.dr.ir. D.M.J. Smeulders, prof.dr. A.E.P. Veldman, for, first, accepting to be a part of this committee and, second, for your valuable advice and comments on the thesis.

I am very thankful to the staff of laboratory of Building Physics and Services: Geert-Jan, Stan, Jan, Harrie and Wout, and also to Ad Holten from the laboratory of Turbulence and Vortex Dynamics. Thanks for making my PIV measurements work and for letting the CFD simulations run on the lab servers. This research will not be possible without you and your hard work.
Thanks to the secretaries at Building Physics and Services: Renée, Yeliz, Janette, Ginny Moniek, Léontine, and Nathaly. Thanks for all the provided administrative support and all nice brief talks we had these years. Thanks to the Communication Expertise Centre of TU/e, and especially to Ingrid and Tina, for involving me in the Virtual Visit TU/e campaign. It was a very nice extracurricular experience during my PhD studies.

I was very lucky to meet and to work with my colleagues at the Urban Physics research group, intelligent and talented researchers and very nice persons from all over the world: Alessandro, Alessio, Anto, Christof, Claudio, Fabio, Feiyu, Gerson, Hamid, Ivo, Jo-Hendrik, Jorge, José Antonio, Marlies, Mattia, Nestoras, Olga, Pierre, Raffaele, Rahim, Rick, Rob, Rubina, Samy, Thijs, Zara, Xing. It was a pleasure to learn you all and to spend nice time together during conferences and dinners! I am always happy to meet you on my way. Special thanks to Karin, Katarina Kosutova, and Wendy, for our friendship and exchange of our opinions all these years – we will stay in touch for sure! I would like to thank colleagues from other research groups for nice memories created at the Department of the Built Environment: Benedetto, Bruno, Pieter-Jan, Fotis, Roel, Raúl, Parisa, Juliëtte, Christel, Sanae, Jikke, Isabella, Hemshikha, Zahra, Lyui, John, Rajesh, Mike, Vojtech, Stepan, Perry, Christof, Qingliang, Azee, Alberto, Przemek, Rui, George, Pei, Miruna, Katarina Katic, Basar. Veronica and Qadeer, you are an important part of Guillaume’s life and you also became my dear friends. I am very happy to meet you and hope we see each other and keep in touch in the future. Kate Loginova and Dmitry Surugin, I was very happy to meet you at floor 6 and to have our talks in Russian and share our experiences of life in Noord-Brabant.

During my PhD research I had a chance to supervise the master students Jochem, Karin, and Odete. Thank you all for your hard work and for the given experience in supervision.

Thanks to the PhD Network for spending nice times at organizing our events and enjoying these events in a great company. Special thanks to my friends Widi and Blago, thanks for nice afternoons and dinners we spent together and for treating me with most special cups of coffee!

After I moved to the Netherlands I met a number of people from Russia and two of them have become a very special part of my life. Anna and Lidia, thanks for being all these years with me, thanks for sharing your great Russian sense of humor and for our nice get-togethers. Though we are all very busy all the time I still wish we can meet more often. Also, thank you both for being part of this special day, being my seconds at the PhD defense.

In particular, I wish to thank my international friends Argyrios and Angeliki, Ning and Xin, Yasin and Sanem. We explored Dutch culture and education, shared time-to-time our encouragements and frustrations, and eventually gained invaluable experiences
together. Thanks for our friendship and all pleasant memories, there are more to come in the future, for sure!

I also would like to thank my friends in Russia. It might be hard sometimes to keep our friendship warm due to distances of thousand kilometers. Thanks for your understanding and keeping in touch, it is always a pleasure to meet with you, always with a feeling that we know each other forever and never split apart. Aleksandra, Lyaysan, Kate, Kseniya, Lena, Dasha, I miss you enormously and wish we have much more holidays to meet each other. Thanks to my Dutch friends, Wieke and Juan, Laura and Kevin, Britt and Martijn, Ingrid and Bjorn, for our warm gatherings and making life fun!

Also thanks to my colleagues at Cauberg Huygen and our director Louis van Brakel for your support in this adventure and your trust in me. Special thanks to my close colleagues Femke, Iris, Larissa, Liselotte, Adonia for your interest in the progress of my PhD work and asking me from time to time when I am eventually going to finish.

At last I would like to express my thanks to my family. Joke, Hennie, Odessa, Geert-Jan, Casimir, Noor, Jente, Joran, Mirne, Marik, Anne and Teun. Thanks for all your support and help with everything, thanks for having fun times with Nathan when I was busy. Thanks for your interest in the progress of this thesis. You can probably not imagine how great your contribution to this thesis is! My heartfelt thanks to my parents, Afgat and Liza, and my sister Lyalya, for your support, love and care you always give me even at this great distance we have between us. Thanks for keeping my spirit high to successfully finish this research. Thanks for always being there for me, I love you with all my heart and I look forward to meet again. Thanks to my uncles, aunts and cousins – Askat, Irina, Alina, Guzel, Ivan, Lyalya, Olga, Slava, Irek, Raushaniya, Adeliya – we do not see each other that often but I always know you are there for me and every time we meet, in Russia or in the Netherlands, I realize what a happy, rich and charmed life I do have, thanks to you!

And finally my sincere thanks to the three beautiful wonders of my life, Guillaume, Nathan and Evelien, for making my life balanced and complete, for our love, for our home and for the experiences we gain together. I look forward to our new adventures!
Plane turbulent impinging jets (PTIJs) at moderate to high Reynolds numbers are used in a wide range of applications. A typical example in building engineering is an air curtain (AC), which can be used to separate two environments in terms of heat and mass transfer, for example, at entrances of buildings and refrigeration rooms to reduce heat losses, in cleanrooms and hospitals to prevent pollutant spreading and inside buildings and tunnels to prevent smoke propagation between two or more zones.

The existing literature on PTIJs and ACs can be divided into two clear categories: (1) basic studies on PTIJs; and (2) application-oriented studies on ACs; both of which can be performed experimentally and numerically (e.g. using computational fluid dynamics; CFD). With regard to basic studies, in spite of the rather large number of experimental and numerical studies on PTIJs performed in the past, there is a scarcity of detailed and high-quality experimental and numerical data on PTIJs at moderate Reynolds numbers (7,000 – 14,000) and for a jet heights larger than 10 jet widths that are representative for AC applications. Moreover, most previous basic numerical studies on PTIJs only focused on the jet impingement region. The majority of the application studies are case-specific and lack generality in conclusions. Finally, the vast majority of previous studies focused on ACs discharged from nozzles just above the door opening. However, in some cases ACs have to be installed close to the ceiling, i.e. at a substantial distance from the top of the door opening. In these cases the AC jet characteristics first resemble a plane wall jet flowing along a vertical wall and after reaching the top of the door opening the jet flow gradually obtains the characteristics of a plane free jet.

The objectives of the thesis are: (1) to obtain high-quality experimental data for a detailed analysis of the fluid dynamics of PTIJs at moderate Reynolds numbers; (2) to validate steady Reynolds-averaged Navier-Stokes (RANS) CFD simulations of PTIJ flows based on the obtained experimental data; (3) to provide insight into the performance in terms of separation efficiency of ACs positioned in the opening between two environments and to analyze the sensitivity of the separation efficiency to different jet discharge widths, jet discharge angles and jet discharge velocities; (4) to investigate the
behavior and separation efficiency of an AC when it first flows along a wall before reaching the door opening.

The research is performed using: (1) 2D particle image velocimetry (PIV) measurements as an experimental method for a basic study on PTIJs at moderate Reynolds numbers; and (2) validated CFD simulations as a suitable numerical method to model PTIJs and to assess the influence of AC jet parameters on jet separation efficiency.

Chapter 2 of this thesis presents 2D PIV measurements for the whole vertical centerplane of a PTIJ. The jet Reynolds number ranges from 7,200 to 13,500 and two different jet height-to-width ratios are studied. The profiles of centerline and cross-jet mean velocity and turbulence intensity show that PTIJs behave as a free plane turbulent jet until 70-75% of the total jet height. In addition, the jet issued from the nozzle with higher aspect ratio shows more intensive entrainment and a faster decay of the centerline velocity compared to the jet with lower aspect ratio for identical jet Reynolds numbers.

Chapter 3 presents a detailed validation study for 3D steady RANS CFD simulations of PTIJ flows with a jet height-to-width ratio of 22.5 at two moderate Reynolds numbers, i.e. $Re = 8,000$ and $Re = 13,000$ based on the 2D PIV data presented in Chapter 2. The results show that the differences in validation metrics between the considered turbulence models are very small. Nonetheless, the results do show that the best overall performance for mean velocity and turbulent kinetic energy is provided by the realizable $k$-$\varepsilon$ and RNG $k$-$\varepsilon$ models. It is concluded that 3D steady RANS simulations are suitable to assess general flow characteristics such as the jet spreading rate, the jet decay rate, and the extent of the potential core region. These quantities are relevant, for example, in predictions of heat and mass transfer through ACs.

In Chapter 4, the generic situation of an isothermal jet separating two environments subjected to a cross-jet pressure gradient is studied. The study is performed using 2D steady RANS simulations with the RNG $k$-$\varepsilon$ turbulence model for $5,000 < Re < 30,000$. First, the computational model is validated based on available data from particle image velocimetry (PIV) measurements. Second, the influence of several jet parameters, such as the ratio of jet discharge momentum flux to jet cross-flow momentum flux (momentum flux ratio), jet height-to-width ratio and jet discharge angle, on jet separation efficiency is evaluated and quantified. Finally, the minimum deflection modulus to prevent breakthrough of the jet and the corresponding minimum momentum flux ratios are defined based on conservation of momentum and are
compared to the values of the optimal momentum flux ratios as obtained from the CFD simulations.

The results show that for this particular case: (1) the jets with the smallest height-to-width ratios ($h_{jet}/w_{jet} = 18$) provide the highest modified separation efficiency; and (2) inclined jets with angles $\alpha_0 = 5^\circ$ and $10^\circ$ result in a slightly higher modified separation efficiency than straight jets ($\alpha_0 = 0^\circ$) and jets with angle $\alpha_0 = 20^\circ$. It is also shown that the optimal separation efficiency is reached at lower momentum flux ratios for jets with smaller height-to-width ratios and for inclined jets. The values of optimal momentum flux ratio calculated for jets at angles $\alpha_0 = 0^\circ$, $5^\circ$, $10^\circ$ and $20^\circ$ determined based on CFD simulations deviate by up to 31.2% from the values obtained from the analytical equation based on conservation of momentum.

Chapter 5 presents a numerical study to predict the influence of an upstream wall on AC separation efficiencies. The analysis is performed by steady isothermal 2D RANS simulations after validation with experimental data from literature for both a plane wall jet and a plane free jet. The effect of different wall heights is analyzed: no wall (reference case), 0.5 m, 1.0 m and 2.0 m wall. For every wall height, a configuration with the AC unit installed against the ceiling (Case A) and a configuration with the AC unit installed near the bottom of the wall (Case B) are considered. The numerical results indicate that for both Cases A and B the presence of the wall leads to reduced jet bending and reduced jet decay. For Case A this is attributed to (1) the Coanda effect, (2) the fact that jet expansion is partly constrained by the wall, and (3) the fact that the static pressure difference acts over a lower (free) jet height. For Case B, this is attributed to the lower jet height. The results show that the presence of a wall in both Cases A and B increases the separation efficiency $\eta$ (based on infiltration of pollutant) of the air curtain. However, the presence of a wall in both Cases A and B decreases the modified separation efficiency $\eta^*$ (based on infiltration and exfiltration).

Chapter 6 provides the limitations and recommendations for future work and a general discussion. Finally, Chapter 7 summarizes the results of the previous chapters and provides the conclusions.
Chapter 1

Introduction

1.1 Plane turbulent impinging jet and air curtain

The development of air curtains started in 1904 when Theophilus Van Kannel applied for a patent (Van Kannel 1904) on means for excluding drafts from open doorways. The patent presented a device preventing the entrance of dust or wind through an open doorway by blowing air from the top, the bottom or from both sides of the entrance or by blowing air horizontally from the opposite side of the doorway, from the inside straight to the outside. The created air stream would counteract the wind pressure across a doorway. The device developed by Van Kannel (1904) would allow an open doorway for the public.

The patent was granted, but the door was never built and never put into practical use (Hayes and Stoecker 1969a). Van Kannel’s device was the actual predecessor of the modern air curtain (AC). However, there are several clear distinctions between these two devices. Firstly, the invention of Van Kannel required a doorway with a length of several meter to fit all the necessary equipment (Fig. 1a), while modern ACs are compact and require a depth of 0.7 m to 1.0 m (e.g. Biddle, Fig. 1b). Secondly, the invention of Van Kannel consisted of multiple separate elements such as multiple blasts, regulating wings and exhausts, while a modern air curtain is composed of a single complete unit with all elements necessary for proper functioning. Finally, the first AC by Van Kannel had mechanical regulating wings to adjust the blowing speed according to the pressure gradient across the doorway and channels to direct the flow. Advanced modern ACs have control sensors that measure pressure gradients across the doorway and adjust the velocity at the nozzle exit according to these gradients. The AC technique aims to provide an optimal aerodynamic sealing between two environments separated by an AC. In addition, there is a need to avoid excessive energy consumption of the AC due to unnecessarily high velocities at the nozzle exit, which cause thermal and kinetic energy
dissipation from the jet to the ambient environment. Moreover, too high velocities at the nozzle exit result in unnecessary high electricity consumption of the operating fan (e.g. Gil-Lopez et al. 2013).

Figure 1: a) Van Kannel’s invention, the configuration with air blowing from the sides: plan view (adapted from US patent 1904). b) Modern air curtain (biddle.co.uk)

In general, ACs are plane turbulent impinging jets (PTIJs), as schematically shown in Figure 2. A PTIJ consists of the following three distinctive flow regions (e.g. Guyonnaud et al. 2000, Maurel and Solliere 2001, Koched et al. 2011): (1) the potential core region with a centerline velocity more than or equal to 98% of the jet discharge velocity $V_0$, where the length of the potential core region $h_p$ generally satisfies $3w_{jet} \leq h_p \leq 8w_{jet}$, with $w_{jet}$ the jet discharge width (the width of the nozzle from which the jet is issued); (2) the intermediate region with decaying centerline velocity, including the transition region from the potential core to the developed region, with self-similar profiles for mean velocity and turbulence intensity; and (3) the impingement region, where the flow is influenced by the pressure gradient created by the opposing impingement plate, where the height of the impingement region $h_i$ satisfies $0.10h_{jet} \leq h_i \leq 0.13h_{jet}$, with $h_{jet}$ the distance from the nozzle exit to the floor.

In order to specify the jet geometry the following conventional parameters are used: nozzle aspect ratio ($d_{jet}/w_{jet}$, the ratio of nozzle depth to width), jet height-to-width ratio ($h_{jet}/w_{jet}$), and jet discharge angle ($\alpha_0$, angle between jet centerline at the nozzle exit and line perpendicular to the impingement plate, Fig. 2b). The spreading of the jet along the jet centerline is characterized by the jet half-width. The jet half-width $\delta_{0.5}$ at a certain
downstream position $y$ (Fig. 2a) is the horizontal distance between the jet centerline and the point where the mean streamwise velocity is equal to half the local jet mean centerline velocity $V_{0,y}$. The jet spreading rate $K_y$ can be defined as the variation of the jet half-width $\delta_{0.5}$ along the jet centerline (Pope 2000):

$$K_y = d\delta_{0.5} / dy$$

(1)

Within the jet intermediate region the distribution of jet centerline velocities can be described as (Flora and Goldschmidt 1969):

$$(V_0 / V_{0,y})^2 = K_u (y - y_0) / w_{jet}$$

(2)

with $K_u$ the jet decay rate and $y_0$ the vertical distance between the nozzle exit and the kinematic virtual jet origin. The jet decay rate is a measure of mixing of the jet with the ambient environment. It can be quantified as the slope of the velocity distribution
in Fig. 3, where near-linear behavior is observed. Also, the kinematic virtual jet origin can be estimated from the distribution of \((V_0/V_{0,y})^2\) along the jet centerline. It is defined as the distance from the nozzle exit \((y = 0)\) to the extrapolated intersection of the near-linear distribution \((V_0/V_{0,y})^2\) with the abscissa \(y/w_{jet}\). The virtual origin for the velocity distribution shown in Fig. 3 is the point with coordinates \((x; y) = (-y_0; 0)\). The virtual origin is generally located upstream the actual nozzle exit \((y < 0)\), however, in some cases it is located downstream of the nozzle exit \((y > 0)\) (Rajaratnam 1976). The position of the virtual origin is found to be dependent on the turbulence intensity at the nozzle exit (Flora and Goldschmidt 1969).

The physical properties of PTIJs have a lot in common with those of plane turbulent free jets (PTFJs). Similar to PTIJs, PTFJs consist of three regions, the first two of those three regions are the potential core and the intermediate region (Fig. 4a). As in the case of PTIJs, the intermediate region consists of the transition region, where the potential core is totally consumed by the shear layer and the centerline velocity starts to decay due to entrainment of ambient air towards the jet centerline, and the self-similar region zone, where the normalized cross-jet velocity profiles are self-similar at different distances from the discharge nozzle and show a Gaussian shape. The intermediate region is followed by the termination region, which is the zone where the jet centerline velocity

![Figure 3: Plot to define jet decay rate \(K_u\) and kinematic virtual jet origin \(y_0\) based on measurement results from Khayrullina et al. (2017). The plot includes indications of the virtual origin \((y_0)\), the potential core \((h_p)\) and the start of the intermediate and developed regions.](image)
decreases to zero due to diffusion and cannot be distinguished from the surrounding fluid (e.g. Awbi 1991). PTIJ subjected to a cross-flow can become similar to PTFJ in a cross-flow (Fig. 4b). The structure of the PTFJ in a cross-flow comprises three regions (Fig. 4b): the potential core region, the zone of the maximum deflection and the vortex zone (e.g. Rajaratnam 1976). From the end of the potential core region (first region) the zone of the maximum deflection (second region) starts where the jet undergoes the largest deflection and develops a turbulent mixing layer. In the third region, the vortex zone, the jet axis approaches the cross-flow direction asymptotically and the jet velocity magnitude and direction become close to those of the cross-flow stream.

Figure 4: a) Structure of a free jet. b) Free jet in a cross-flow

1.2 Literature study and problem statement

As mentioned before, ACs can be represented by PTIJs and partially by PTFJs, the properties of which are similar to PTIJs. This section first provides a non-exhaustive overview of experimental studies on PTFJs and PTIJs. Then, available numerical studies on both PTFJs and PTIJs are briefly discussed. Finally, a concise overview of studies on ACs concludes this section.
1.2.1 Experimental studies on plane turbulent free jets

An overview of the first experimental studies on PTFJs was provided in a publication by Gutmark and Wygnanski (1976). The first investigation on PTFJs (to the best knowledge of the author) was conducted by Forthmann (1934). Later, among others, Abramovitch (1963), Rajaratnam (1976), Chen and Rodi (1980) and Ramaprian and Chandrasekhar (1985) conducted experiments on the flow structure of PTFJs. These studies showed that due to the large velocity gradients between the jet and the ambient environment, turbulent eddy structures are generated. These eddy structures are responsible for the entrainment of ambient fluid into the jet through the jet boundaries. Due to this entrainment, the jet is spreading with a growing shear layer on both jet sides and the jet centerline velocity decays with increasing distance from the discharge nozzle.

Recent experimental studies on PTFJs considered the influence of different parameters on the jet structure (e.g. Deo et al. 2007a-c, 2008; Mi and Nathan 2010). The investigated parameters included the jet Reynolds number, the presence of sidewalls, the shape of the nozzle and the nozzle aspect ratio. For the influence of sidewalls, Deo et al. (2007a) showed that for jets at $Re = 7,000$ and $10,000$ (at large aspect ratio equal to 60) the length of the potential core was greater for the case with sidewalls, which implied a lower rate of near-field entrainment. Also, the near-field decay and spreading rates were lower for the case with sidewalls than for the jet without sidewalls. This was attributed to the fact that jets with sidewalls cause less three-dimensional flow behavior than jets which are not constrained by sidewalls. Deo et al. (2007b) studied the influence of nozzle shape on the flow and highlighted that the downstream development of any PTFJ is dependent upon its exit boundary conditions (i.e. nozzle-exit profiles) and upstream conditions (i.e. contraction and nozzle-exit shapes). Deo et al. (2007c) analyzed the influence of nozzle aspect ratio. In line with the earlier study by van der Hegge Zijnen (1958), they found that an increase in aspect ratio leads to increasing rates of jet spreading and centerline velocity decay. With regard to the jet Reynolds number, Deo et al. (2008) found that an increase of $Re$ causes the length of the potential core to decrease and the near-field spreading rate to increase. Mi and Nathan (2010) investigated various cross-sectional nozzle-exit geometries, such as circular, rectangular, triangular, star-shaped and ellipsoidal. They found that the near-field flow (close to the nozzle exit) for jets with non-axisymmetric (non-circular) nozzles had a higher decay rate of mean velocity and higher entrainment rates than these for axisymmetric jets. The greatest decay rate was observed for a triangular nozzle. In addition they indicated that a change in the nozzle-exit geometry did not change the far-field properties of the jet.
Deo et al. (2007b) and Deo (2013) showed that the decay, jet spreading rate and entrainment rate are enhanced for the case of a jet issued from a sharp-edged nozzle (Fig. 5a) due to higher turbulence intensity at the nozzle exit compared to the smoothly-shaped nozzle (Fig. 5b).

![Image of jet nozzles](image)

**Figure 5**: a) Sharp-edged nozzle. b) Smoothly-shaped nozzle (Adapted from Deo et al. (2007b))

The influence of a cross-flow on jet behavior was studied by e.g. Haniu (1985), Huang et al. (2005), and Plesniak and Cusano (2005). For the potential core region it was found that jets in a cross-flow have in general a smaller length of the potential core than jets without a cross-flow (Rajaratnam 1976). Also, the jet had a slightly higher spreading rate than a vertical jet that was not subjected to a cross-flow (e.g. Haniu and Ramaprian 1979, Huang et al. 2005).

### 1.2.2 Experimental studies on plane turbulent impinging jets

A range of basic experimental studies on PTIJs has been published since the 1960s. These studies generally analyzed the structure of the jet and the behavior of turbulent vortical structures by means of different measurement and visualization techniques. Part of these studies focused on the jet dynamics in the impingement region of the jet. For example, they investigated the influence of jet parameters, such as the jet properties at the nozzle exit, the presence of sidewalls and the jet height on heat transfer at the impingement plate. The influence of turbulence parameters at the nozzle exit on heat
transfer at the impingement plate was highlighted by Gardon and Akfirat (1965), Cartwright and Russell (1967), Beitelmal et al. (2000) and Zhou and Lee (2007), who showed that higher turbulence intensities at the nozzle exit resulted in enhanced heat transfer at the impingement plate. They also showed that the influence of turbulence intensity decreased for jets with a height-to-width ratio beyond eight. Regarding jet heights, Gardon and Akfirat (1965) and Korger and Krizek (1966) showed that the maximum mass transfer coefficients at the stagnation point occurred for jets with height-to-width ratios of 8 to 10.

The effect of the jet discharge angle was studied by e.g. Beitelmal al. (2000), Eren and Celik (2006) and Akansu et al. (2008). They concluded that the maximum heat transfer region in inclined slot jets shifted towards the direction of the jet nozzle and heat transfer at the impingement plate increased as the inclination decreased.

The influence of cross-flow on impingement heat transfer was discussed by Qi et al. (2001) and Zuckerman and Lior (2006). They mentioned that the stagnation point was shifted due to the cross-flow and the heat transfer rates were reduced compared to the situation without cross-flow.

Another group of studies examined the whole jet flow field, i.e. over the entire height of the jet. These studies focused on the structure of the jet (e.g. Maurel and Solliec 2001, Koched et al. 2011), the existence of vortical structures in the jet flow (e.g. Sakakibara et al. 2001, Loubière and Pavageau 2008) and the role of these vortical structures in the dissipation of the jet energy to the ambient environment. The existence of 3D vortices in the impingement region was also highlighted, in which heat can be exchanged between the jet and the impingement surface. In line with studies on PTFJs, Ashforth-Frost et al. (1997) showed a decrease in entrainment and jet spreading and an increase of the potential core length for the configuration with sidewalls compared to the situation without sidewalls.

1.2.3 Numerical studies on plane turbulent jets

In addition to all aforementioned experimental studies, a range of numerical studies on both PTFJs and PTIJs have been published. The vast majority of these studies employed computational fluid dynamics (CFD) simulations, consisting of either the (steady)
Reynolds-averaged Navier-Stokes (RANS) or the large eddy simulation (LES) approach. Many RANS studies, e.g. Seyedein et al. (1994), Heyerichs and Pollard (1996) and Shi et al. (2002) questioned the performance of the standard $k$-$\varepsilon$ turbulence model (SKE) for PTIJs, while a number of validation studies for PTFJs (i.e. without impingement) showed a sufficient performance of steady RANS in combination with the SKE turbulence model (e.g. Berg et al. 2006, Aziz et al. 2008, Isman et al. 2008). Moreover, it was shown that the performance of turbulence models can differ per characteristic zone of a PTIJ (e.g. Zuckerman and Lior 2006) and per predicted flow variable (e.g. mean velocity, turbulent kinetic energy). For example, Aziz et al. (2008) found a better performance of the SKE turbulence model than the renormalization group $k$-$\varepsilon$ turbulence (RNG) model in predicting the jet spreading rate, jet decay rate and velocity profiles of a PTFJ, while the turbulent kinetic energy was only accurately predicted by the RNG model with a mean deviation of 14% from the experimental values. Finally, Angioletti et al. (2005), Jaramillo et al. (2008) and Dutta et al. (2013) indicated the absence of generality in the results of different studied cases of PTIJs due to different boundary conditions and computational settings.

1.2.4 Studies on air curtains

Air curtain studies have generally been focused on the dimensioning of air curtains, determination of the air-curtain efficiency by development of empirical formulae, and defining the most influential parameters with respect to the air curtain separation efficiency. The separation efficiency is the most widely used performance indicator of an air curtain. It is defined as the rate of heat or mass transfer through the opening with an AC compared to that of the same opening without an AC (e.g. Frank and Linden 2014):

$$\eta = 1 - \frac{Q_{ac}}{Q_0}$$  \hfill (3)

with $Q_{ac}$ and $Q_0$ the heat or pollutant mass transfer through the opening with an AC, and without an AC, respectively.
Along with environmental parameters, e.g. temperature and pressure differences between the two separated environments, the separation efficiency of an AC depends on a wide range of jet parameters. Similar to basic studies on PTFJs and PTIJs, these parameters include the nozzle shape, the nozzle aspect ratio, the jet height-to-width ratio, the jet Reynolds number, the jet turbulence intensity at the nozzle exit and the jet discharge angle. Regarding the nozzle shape, to the best knowledge of the author no studies on ACs are available for different nozzle shapes, however, conclusions of studies on PTFJs might be relevant and applicable to ACs. For example, Deo et al. (2007b) showed that PTFJs with sharp-edged nozzles resulted in higher jet spreading rates than smoothly-shaped nozzles. Hayes and Stoecker (1969b) stated in their AC study that a smaller aspect ratio resulted in a higher separation efficiency and Shih et al. (2011) and Moureh and Yataghene (2016) showed that for increasing height-to-width ratios the separation efficiency of an AC decreased. Many studies investigated the influence of jet discharge velocity and resulting jet Reynolds number on the performance of ACs. For example, Hayes and Stoecker (1969a), Sirén (2003), and Frank and Linden (2014) specified the minimum jet momentum flux for ensuring the aerodynamic sealing of a doorway. The influence of turbulence intensity at the nozzle exit was discussed in Hayes and Stoecker (1969b), who recommended to reduce the turbulence intensity at the nozzle exit to decrease the entrainment of the ambient fluid into the jet. The influence of the jet discharge angle on AC performance was studied by, among others, Hayes and Stoecker (1969b), Costa et al. (2006), and Valkeapää et al. (2010), who showed that by inclining the jet to the exterior environment the separation efficiency can be increased due to the counterbalancing of the pressure differences across the doorway. The defined optimum discharge angles of the air curtain jet vary from 15° (Costa et al. 2006) to 30° (Hayes and Stoecker 1969b).

1.2.5 Knowledge gap and problem statement

With regard to basic studies, in spite of a rather large number of experimental and numerical studies on PTFJs and PTIJs performed in the past, there is a clear scarcity of detailed and high-quality experimental and numerical data on PTIJs at moderate Reynolds numbers (7,000 – 14,000) and for a jet height larger than 10 jet widths (both of which are representative for air curtain applications). Moreover, most previous basic numerical studies on PTIJs only focused on the jet impingement region.
Secondly, modeling ACs as well as modeling PTIJs is complex as the numerical model should accurately solve and/or model both the free shear layer development, and the wall jet near the impingement surface. Properly solving these two complex flows requires certain grid resolutions and computational settings (e.g. turbulence model, near-wall treatment). Due to limitations in computational resources, available numerical studies on AC applications do not always comply with best-practice guidelines for the modeling of PTIJs as defined for basic studies. Therefore, a large gap exists between basic numerical studies on PTIJ and application studies on AC using numerical models. Moreover, the majority of the AC studies are application studies that are very case-specific and lack generality in conclusions.

Finally, the vast majority of previous studies focused on air curtains discharged from nozzles just above the door opening. However, in some cases, air curtains have to be installed close to the ceiling, i.e. at a substantial distance from the top of the door opening. In these cases the air curtain jet characteristics first resemble a plane wall jet flowing along a vertical wall and after reaching the top of the door opening the jet flow gradually obtains the characteristics of a PTFJ (see Section 1.1).

### 1.3 Objectives and methodology

#### 1.3.1 Objectives

This research aims to provide insight in and knowledge of the relationship between a range of jet parameters on the one hand and jet separation efficiency on the other hand for basic PTIJs. The findings regarding dynamics of PTIJs should be relevant for improving AC performance in different AC applications.

The objectives of the thesis are:

1) To obtain high-quality experimental data for a detailed analysis of the fluid dynamics of PTIJs at moderate Reynolds numbers (Chapter 2).

2) To validate steady Reynolds-averaged Navier-Stokes (RANS) CFD simulations of PTIJ flows based on the obtained experimental data (Chapter 3).
3) To provide insights into the performance of ACs (separation efficiency) positioned in the opening between two environments and to analyze the sensitivity of the separation efficiency to different jet discharge widths, jet discharge angles, and jet discharge velocities (Chapter 4).

4) To investigate the behavior and separation efficiency of an air curtain when it first flows along a wall before reaching the door opening (Chapter 5).

1.3.2 Methodology

The research is performed using: (1) 2D particle image velocimetry (PIV) measurements as an experimental method for a basic study on PTIJs at moderate Reynolds numbers; and (2) validated CFD simulations as a suitable numerical method to model PTIJs and to assess the influence of AC jet parameters on jet separation efficiency.

Particle image velocimetry (PIV)

PIV is an optical measurement method that can provide the instantaneous spatial flow field description and quantitative results of flow dynamics in most common fluids, such as air and water. The principle of PIV is based on the measurement of the displacement of small tracer particles, which are carried by the fluid, during a short time interval.

The experiments with a PIV system typically require an optically transparent test facility (e.g. water channel or wind tunnel), seeding particles, a pulsed laser system with light sheet optics as a light source, a charge-coupled device (CCD) digital camera system for recordings and a computer with suitable software for data acquisition and processing (e.g. Prasad 2000, Atkins 2016). As an example, Figure 6 provides a schematic illustration of the set-up for PIV recording of two velocity components in the vertical centerplane in the water channel as performed in this research.

A thin laser light sheet is created from the laser beam using laser light sheet optics (e.g. mirrors and lenses). The seeding particles, e.g. polyamide particles, following the flow are illuminated within the plane of the light sheet. The laser light is scattered by these particles and is captured by a CCD camera focused on the laser sheet. This short-duration pulsed laser illumination “freezes” the particle motion on the two successive images (frames) of the PIV recordings separated by the time interval $\Delta t$ between the two
laser illuminations. The vectors of the local flow velocity into the plane of the light sheet are estimated based on the averaged displacement of the particles and the time interval $\Delta t$.

![Figure 6: Experimental arrangement for planar 2D PIV in a water channel (adapted from Khayrullina et al. 2017)](image)

For data-processing the digital PIV recording is divided in small subregions called “interrogation windows” I and I’ (Fig. 7). The size of the interrogation window should be chosen together with the time interval between two successive frames $\Delta t$ and the seeding concentration. The number of particles (so-called image density) within the interrogation window should be $N_i > 7$ to ensure high measurement reliability (Keane and Adrian 1993). In addition, the size of the time interval $\Delta t$ should ensure that the mean in-plane particle displacement $D_i$ in an interrogation window does not exceed one-fourth of the size of the interrogation window, i.e. $\Delta x \leq D_i/4$ and $\Delta y \leq D_i/4$ (Westerweel 1997). Moreover, the seeding particles should be small enough to accurately follow the fluid motion and should not alter the fluid properties or flow characteristics (Keane and Adrian 1992, Westerweel 2000). The local displacement vector of the tracer particles of the first and second frames is determined for each interrogation window of the entire image domain by means of statistical methods (cross-correlation). The relation between the particle image displacement in the image plane and the actual
tracer particle displacement in the flow can be determined by a calibration with, for example, a black-and-white calibration target.

![Figure 7: Schematic illustration of estimation of velocity components u and v within interrogation windows I and I’ by PIV measurements from a pair of frames at time $t_1$ (a) and $t_2$ (b). Adapted from Atkins (2016)](image)

The accuracy and reliability of PIV measurements depends – among other aspects – on the visibility of the particles (Atkins 2016). The experimental setup has to be constructed from transparent materials, e.g. glass or polymethyl methacrylate, to allow access of both the laser light sheet to illuminate the seeded particles and the camera to record the scattered light. Moreover, these transparent surfaces should be thin to reduce optical distortion of the laser light. The background surfaces of the setup should be covered by, for example, a matte black surface to improve the visibility of the seeding particles and to avoid reflections from the background surfaces. The laser sheet thickness should be adjusted considering the reduction of the out-of-the-plane motion of the particles as well as depth limitations of the optics of the CCD camera.

One of the main advantages of PIV compared to other measurement techniques is that – apart from the seeding particles which however are assumed not to distort the flow – PIV is a non-intrusive optical technique in contrast to the measurement with probes (hot wires, pressure probes) that can disturb the flow field. In addition, PIV, in contrast to single-point measurements, is a technique that allows to record velocity vectors at a
large number of points simultaneously. PIV can thus provide physical flow information of a large area in a limited amount of time. Moreover, PIV provides instantaneous spatial flow information that is relevant for turbulent flows.

In this thesis 2D PIV measurements are used to study PTIJs at moderate Reynolds numbers and to provide the required validation data for the CFD simulations.

**Computational fluid dynamics (CFD)**

Fluid flows are governed by the following three fundamental principles: 1) the mass of fluid is conserved (conservation of mass), 2) the rate of change of momentum equals the sum of the forces on a fluid particle (Newton’s second law), and 3) the rate of change of energy is equal to the sum of the rate of heat addition to and the rate of work done on a fluid particle (first law of thermodynamics). These fundamental principles can be expressed in terms of mathematical equations, which in their most general form are usually partial differential equations. With computational fluid dynamics (CFD), these equations or their approximate forms can be numerically solved in space and/or time to obtain a numerical description of the complete flow field of interest. Three main approaches can be distinguished for CFD simulations based on the extent of the turbulence resolved in the equations: direct numerical simulation (DNS), large eddy simulation (LES) and Reynolds-averaged Navier-Stokes (RANS) simulations.

DNS is the most accurate approach with all scales of turbulence being explicitly solved (Fig. 8a). However, it requires very fine computational grids and very fine time steps and the computational demand of DNS is very high, even at low Reynolds numbers. An alternative approach is LES, in which the Navier-Stokes equations are filtered, often with the computational grid size as filter. With this approach the mean flow and the eddies that are larger than the size of the filter are resolved, while eddies that are smaller than the size of the filter are modeled (Fig. 8b). Albeit less computationally demanding than DNS, also for LES large computer resources can be required, since the mesh size and the time step size to be used are very small if a sufficient amount of turbulence needs to be resolved, and since LES simulations, in contrast to RANS, should normally only be performed in 3D. The RANS approach is generally adopted, both in science and for practical engineering, because of the strongly reduced computational requirements compared to LES and DNS methods (e.g. for building engineering, see Franke et al. 2011, Blocken 2018). This approach only solves the mean flow quantities, with all the scales of
turbulence (Reynolds stresses) being modeled (Fig. 8c). Turbulence modeling can generally be subdivided in first-order closure models, based on the Boussinesq eddy viscosity hypothesis (Boussinesq 1877), and second-order closure models, based on Reynolds stress modeling (RSM). The first category of turbulence models represents the most common approach in turbulence modeling. However, an important limitation is the common assumption that the eddy viscosity is a scalar, while in reality it will generally be a tensor. In addition, in the vicinity of the walls the viscous forces dominate and many turbulence models fail to accurately predict these wall-bounded flows. In these regions, wall functions (high-Reynolds number modeling) or low-Reynolds number modeling can be used to overcome this issue. Wall functions are used to correlate the region close to the wall and the fully turbulent region by semi-empirical formulae, while for low-Reynolds number modeling the turbulence model, with or without modifications, solves the inner layer near the wall. The second main category of turbulence models is RSM, in which six additional transport equations are solved for each of the individual Reynolds stresses as well as one transport equation for the turbulence dissipation rate, and this category is therefore considered to provide a more accurate prediction of the Reynolds stresses. However, literature shows that RSM does not always lead to more accurate predictions, while the computational demand is larger than for first-order closure models (e.g. in building engineering: Ramponi and Blocken 2012, van Hooff et al. 2017, Kosutova et al. 2018). For a more detailed description of the aforementioned methods and models the reader is referred to e.g. Versteeg and Malalasekera (2007) and Anderson (2009).

Figure 8: Schematic illustration of a) DNS, b) LES and c) RANS turbulence modeling of PTJs, with ○ - modeled scales of turbulence, ◦ - resolved scales of turbulence (Adapted from ANSYS, Inc., 2011)
Although CFD is capable of providing detailed flow information in the whole calculation domain, it is prone to errors and uncertainties. In order to reduce these errors and uncertainties, CFD best practice guidelines have been developed in a concerted attempt towards more accuracy and reliability in CFD. These include both general best practice guidelines (e.g., Casey and Wintergerste 2000, Tucker and Mosquera 2001, Oberkampf et al. 2004, Roy and Oberkampf 2011) as specific best practice guidelines for specific subdomains of engineering such as wind engineering and building engineering (e.g., Franke et al. 2004, Nielsen et al. 2007, Chen 2009, Blocken 2015). Care is required in the geometrical implementation of the model, in grid generation, in selection of the proper boundary conditions, turbulence models, numerical schemes and solution strategies, and in interpretation of the results. Numerical and physical modeling errors need to be assessed by solution verification and validation studies.

This thesis presents results of validated 3D and 2D steady RANS simulations to assess the influence of jet parameters on jet separation efficiency.

1.4 Dissertation outline

This thesis is structured as follows.

Chapter 2 contains a detailed experimental analysis of an isothermal plane turbulent impinging jet (PTIJ) at moderate Reynolds numbers (7,200 – 13,500) with two height-to-width ratios ($h_{jet}/w_{jet} = 22.5$ and 45) issued on a horizontal plane. More specifically, mean and instantaneous velocity, instantaneous vorticity, turbulence intensity and Reynolds shear stresses are obtained and analyzed by means of 2D PIV measurements. In addition, flow visualizations with fluorescent dye are performed. Alongside the information obtained on the jet dynamics, the data can be used for the validation of numerical simulations of PTIJs.

This chapter has been published as a peer-reviewed journal paper:

Chapter 3 presents results of a validation study of 3D steady RANS turbulence models for predicting PTIJs at two different slot Reynolds numbers, i.e. $Re = 8,000$ and $13,000$, based on the 2D PIV measurements described in Chapter 2. In addition, an in-depth analysis of the results provided by the five commonly used turbulence models is performed. Numerically predicted distributions of mean velocity and turbulent kinetic energy are compared with experimental data along both the jet centerline and several cross-jet lines in lateral direction. Furthermore, the production of turbulent kinetic energy in the jet potential core region near the nozzle exit is analyzed. Finally, recommendations are provided with respect to the performance of the different RANS turbulence models for predicting PTIJs.

This chapter has been published as a peer-reviewed journal paper:


Chapter 4 focuses on the generic situation of an isothermal PTIJ (representing an air curtain) at $5,000 < Re < 30,000$ separating two environments subjected to cross-jet pressure gradient. 2D steady RANS simulations with the RNG $k$-$\varepsilon$ turbulence model are employed. The chapter provides a parametric analysis of the impact of several jet parameters (momentum flux ratio, jet height-to-width ratio, jet discharge angle) on the jet separation efficiency. For each jet configuration (depending on jet height-to-width ratio and jet discharge angle), the ratio of jet discharge momentum flux to jet cross-flow momentum flux (momentum flux ratio) at which the separation efficiency of an air curtain reaches its optimal value is defined. Moreover, the minimum deflection modulus to prevent breakthrough of the jet and the corresponding minimum momentum flux ratios are obtained based on the conservation of momentum and are subsequently compared to the values of optimal momentum flux ratios obtained from CFD simulations.

This chapter has been submitted for publication in a peer-reviewed journal:
Chapter 5 provides the results of a numerical study with 2D steady RANS to predict the influence of the wall downstream of an air curtain on the separation efficiency under isothermal conditions and without air recirculation. The effect of different wall heights is analyzed: no wall (i.e. a free jet), 0.5 m, 1.0 m and 2.0 m. A 2D computational geometry representing two spaces is applied, with clean and polluted air, respectively, separated by an air curtain installed above a permanently opened door opening. Based on the results of the CFD simulations, distributions of jet velocity, development of jet half-width and distribution of pollutant mass fraction are analyzed. The findings of this chapter provide conclusions regarding the influence of the wall presence on jet dynamics and the separation efficiency of the air curtain.

This chapter has been submitted for publication in a peer-reviewed journal:


Chapter 6 provides the limitations and recommendations for future work and a general discussion. Finally, Chapter 7 summarizes the results of the previous chapters and provides the conclusions.
References


Ansys, Inc. (2011) The most accurate and advanced turbulence capabilities [presentation slides]. Confidence by Design Workshop, Chicago, delivered 14 June 2012


group on "quality and trust in industrial CFD" (Version 1). European Research Community on Flow, Turbulence and Combustion, London


Deo RC (2013) Comparative analysis of turbulent plane jets from a sharp-edged orifice, a beveled-edge orifice and a radially contoured nozzle. Mechanical and Mechatronics Engineering 7(12): 2584-2593


Gardon R, Akfirat JC (1965) The role of turbulence in determining the heat-transfer characteristics


Haniu H (1985) LDV measurements in two-dimensional bent-over jets (Part 1). Mean flow properties and curvature effects on turbulence structures of nonbuoyant jets. Memoirs of the Kitami Institute of Technology 17(1):24

Hayes FC, Stoecker WF (1969a) Heat transfer characteristics of the air curtain. ASHRAE J 75(2120):153–167

Hayes FC, Stoecker WF (1969b) Design data for air curtains. ASHRAE Tran 2121:168–180


Ramaprian BR, Chandrasekhara MS (1985) LDA measurements in plane turbulent jets. J Fluids Eng


Tucker PG, Mosquera A (2001) NAFEMS introduction to grid and mesh generation for CFD (R0079). NAFEMS CFD Working Group


Chapter 2

PIV measurements of isothermal plane turbulent impinging jets at moderate Reynolds numbers

This chapter has been published as a peer-reviewed journal paper:


Abstract

This paper contains a detailed experimental analysis of an isothermal plane turbulent impinging jet (PTIJ) for two jet widths at moderate Reynolds numbers (7,200 – 13,500) issued on a horizontal plane at fixed relative distances equal to 22.5 and 45 jet widths. The available literature on such flows is scarce. Previous studies on plane turbulent jets mainly focused on free jets, while most studies on impinging jets focused on the heat transfer between the jet and an impingement plane, disregarding jet development. The present study focuses on isothermal PTIJs at moderate Reynolds numbers characteristic of air curtains. Flow visualisations with fluorescent dye and 2D particle image velocimetry (PIV) measurements have been performed. A comparison is made with previous studies of isothermal free turbulent jets at moderate Reynolds numbers. Mean and instantaneous velocity and vorticity, turbulence intensity and Reynolds shear stress are analysed. The jet issued from the nozzle with higher aspect ratio shows more intensive entrainment and a faster decay of the centreline velocity compared to the jet of lower aspect ratio for the same value of jet Reynolds number. The profiles of centreline and cross-jet velocity and turbulence intensity show that the PTIJs behave as a free plane turbulent jet until 70 - 75% of the total jet height. Alongside the information obtained on the jet dynamics, the data will be useful for the validation of numerical simulations.
Chapter 2

List of symbols and acronyms

Roman symbols

\[D_s\] [m] diameter of PIV seeding particles

\[h_i\] [m] height of impingement region

\[h_{jet}\] [m] jet height

\[h'_{jet}\] [m] part of the jet height visible during PIV measurements

\[h_p\] [m] potential core length

\[H_{BL}\] [-] boundary layer shape factor

\[I_{uv}\] [-] turbulence intensity based on both streamwise and lateral velocity fluctuations

\[I_v\] [-] turbulence intensity based on only streamwise velocity fluctuations

\[K_u\] [-] jet decay rate

\[K_y\] [-] spreading rate of the jet

\[Q\] [-] Okubo-Weiss function

\[Re\] [-] Reynolds number

\[t\] [s] time

\[u\] [m/s] instantaneous lateral (x-direction) velocity component

\[u'\] [m/s] fluctuating lateral (x-direction) velocity component

\[u_{RMS}\] [m/s] root mean square value of lateral (x-direction) velocity component

\[U\] [m/s] mean lateral (x-direction) velocity component

\[v\] [m/s] instantaneous streamwise (y-direction) velocity component

\[v'\] [m/s] fluctuating streamwise (y-direction) velocity component

\[|v|\] [m/s] instantaneous 3D velocity magnitude

\[v_{RMS}\] [m/s] root mean square value of the streamwise (y-direction) velocity component

\ [|v|_{RMS} \] [m/s] root mean square value of 3D velocity magnitude

\[|V|\] [m/s] mean 3D velocity magnitude

\[V\] [m/s] mean streamwise (y-direction) velocity component

\[V_0\] [m/s] mean jet velocity at the nozzle exit

\[V_{0,y}\] [m/s] mean jet centerline velocity
PIV measurements

\( w \) \([\text{m/s}]\)  instantaneous spanwise (z-direction) velocity component

\( w_{\text{jet}} \) \([\text{m}]\)  jet width at the nozzle exit

\( w_{\text{RMS}} \) \([\text{m/s}]\)  root mean square value of the spanwise (z-direction) velocity component

\( W \) \([\text{m/s}]\)  mean spanwise (z-direction) velocity component

\( x \) \([\text{m}]\)  Cartesian coordinate

\( y \) \([\text{m}]\)  Cartesian coordinate

\( y_0 \) \([\text{m}]\)  vertical distance calculated starting from \( y = 0 \)

\( y' \) \([\text{m}]\)  vertical distance calculated starting from the kinematic virtual jet origin

\( z \) \([\text{m}]\)  Cartesian coordinate

**Greek symbols**

\( \delta^* \) \([\text{m}]\)  boundary layer displacement thickness

\( \delta_{0.5} \) \([\text{m}]\)  jet half-width

\( \theta \) \([\text{m}]\)  boundary layer momentum thickness

\( \rho_s \) \([\text{kg/m}^3]\)  density of the PIV seeding particles

\( \gamma \) \([-\text{-}]\)  jet height-to-width ratio

\( v \) \([\text{m}^2/\text{s}]\)  kinematic viscosity

\( \omega_z \) \([1/\text{s}]\)  instantaneous z-vorticity

\( \Omega_z \) \([1/\text{s}]\)  mean z-vorticity

2D  two-dimensional

3D  three-dimensional

AR  aspect ratio, i.e. ratio of nozzle depth \( d_{\text{jet}} \) to width \( w_{\text{jet}} \)

CCD  charge coupled device (camera)

CFD  computational fluid dynamics

LDA  laser Doppler anemometry

PIV  particle image velocimetry

PTIJ  plane turbulent impinging jet

ROI  region of interest
2.1 Introduction

Plane impinging jets at moderate to high Reynolds numbers (i.e. turbulent jets) are used in a wide range of applications. A typical example in building engineering is an air curtain, which can be used to separate two environments in terms of heat and mass transfer, for example in laboratories to reduce contamination hazard or in operating theatres. The separation efficiency of plane turbulent impinging jets (PTIJs) when used as an air curtain depends on a wide range of parameters. On the one hand, these include the environmental parameters such as air temperature differences and pressure differences over the air curtain. On the other hand, they include the jet parameters that influence the vortex structures in the jet and the entrainment process, such as the jet Reynolds number, the turbulence intensity and the jet temperature relative to the ambient temperature.

The existing literature on PTIJs and air curtains can be divided in two clear categories: (1) basic studies on PTIJs; and (2) application-oriented studies on air curtains, both of which can be performed experimentally and numerically (e.g. using computational fluid dynamics (CFD)). Basic studies on PTIJs are important to gain a better basic understanding of the jet dynamics, which can then be utilised to improve their performance in applications such as air curtains.

A range of basic experimental studies on PTIJs has been published in the past decades. These studies generally analyse the structure of the jet and the behaviour of turbulent vortical structures by means of different measurement and visualisation techniques. Below, an overview of previous experimental studies is provided, from which the objectives of the present study are derived.

An extensive body of literature exists on the jet dynamics in the impingement region of the jet. Gardon and Akfirat (1965) applied a transducer based on the operating principle of thermocouples to investigate heat transfer at a plane subjected to a plane impinging jet. They also provided velocities and turbulence intensities along the centreline of the jet for $Re = 5,500$ obtained by hot-wire anemometry. The focus of their work was on characterising the Nusselt numbers and pressure coefficients (measured by pressure taps) at the impingement plane for $Re$ ranging from 450 to 28,000 and jet height-to-width ratios $\gamma = h_{\text{jet}}/w_{\text{jet}}$ ranging from $\gamma = 2$ to 80, with $h_{\text{jet}}$ the jet height and $w_{\text{jet}}$ the jet width. They showed the influence of inlet jet velocity and turbulence intensity on heat
transfer at the impingement plane. Gardon and Akfirat (1965) suggested that the nozzle configuration becomes less influential for jet heights larger than 8 jet widths. Korger and Krizek (1966) used naphthalene sublimation to study jet impingement for \( Re \) between 6,000 and 37,800 and \( \gamma = 0.25 \text{ - } 40 \). Their experiments provided values for the mass transfer coefficients at the impingement plane. The maximum mass transfer coefficients on the jet centreline occurred for jets with \( \gamma = 8 \text{ - } 9 \). Cartwright and Russell (1967) employed thermocouples to analyse the evolution of the wall jet along the impingement plane and provided pressure distributions and heat transfer characteristics on the impingement plane for jets with \( Re \) between 25,000 and 110,000 and \( \gamma = 8 \text{ - } 47 \). They also showed that maximum heat transfer for jets with \( Re \leq 51,000 \) occurred at the stagnation point, while for \( Re > 51,000 \) the maximum heat transfer lied at some distance from stagnation point. They analysed existing correlations for wall jet heat transfer and assessed their applicability based on the measurement results. This analysis showed that these correlations were accurate for the developed wall jet formed after jet impingement. However, it failed to accurately predict heat transfer near the stagnation point due to the high level of turbulence in this region. They also emphasised the influence of jet turbulence on augmented heat transfer in the impingement region. Ashforth-Frost et al. (1997) used hot-wire anemometry to analyse PTIJs with \( Re = 20,000 \) and \( \gamma \) up to 9.2 with focus on the near-wall region. They compared the results for an unconfined configuration with a semi-confined configuration and revealed the influence of the horizontal confining plane adjacent to the jet nozzle. The results showed a decrease in entrainment and jet spreading, and an increase of the potential core length for the semi-confined configuration. Beitelmal et al. (2000; 2006) employed thermocouples to study the heat transfer in the impingement region for \( Re \) between 4,000 and 12,000 and \( \gamma = 2 \text{ - } 12 \). By comparing their experimental data with other studies on heat transfer at the impingement plane they noted the importance of inlet turbulence parameters at the jet nozzle on the results. Zhou and Lee (2007) used thermocouples for PTIJ at \( Re \) ranging from 2,715 to 25,005 and \( \gamma = 1 \text{ - } 30 \) to analyse the influence of \( Re \) and \( h_{jet} \) on heat transfer at the impingement plane. They showed that for situations where the impingement plane is situated beyond the potential core region, the heat transfer between the impinging jet and the impingement plane appeared to be mainly governed by the turbulence intensity.

Another group of studies examined the whole jet flow field, i.e. over the entire height of the jet. Gutmark et al. (1978) applied hot-wire anemometry at the centreline of a PTIJ at \( Re = 30,000 \) and \( \gamma = 100 \). They observed an influence of the impingement plane on the jet development starting at a distance of \( 0.75 h_{jet} \) from the nozzle. Yoshida et al. (1990)
examined the turbulence characteristics and heat transfer of a PTIJ with $Re = 10,000$ using laser Doppler anemometry (LDA) and thermocouples for $\gamma = 8$. They observed a steep increase in turbulence intensity and heat transfer near the stagnation point. Guyonnaud et al. (2000) used hot-wire anemometers, pressure sensors and smoke visualisations for an analysis of a bending PTIJ ($33,000 \leq Re \leq 66,000$, $\gamma = 12$) separating environments with different ambient pressures. The authors provided a detailed description of the physics of the jet and its general structure. Maurel and Solliec (2001) performed experiments on PTIJs with $Re = 13,500$ and $Re = 27,000$ and $\gamma = 5 - 50$, impinging perpendicularly on a flat plate from a variable distance. The structure of the jet was analysed by LDA along the whole jet height and by particle image velocimetry (PIV) in the impingement region. The study provided centreline velocity distributions, instantaneous velocity and vorticity fields and Reynolds stresses along the jet centreline and along vertical lines at some distance from the centreline. They concluded that the height of the impingement region constitutes 12 - 13% of the total jet height, regardless of Reynolds number, nozzle width or jet height. They defined velocity laws for each of the three characteristic jet regions: the potential core, with length of about 3 to 4 jet widths, the intermediate region and the impingement region. The study of Sakakibara et al. (2001) provided an extensive analysis of vortical structures occurring in a periodically forced impinging jet at $Re = 2,000$ and $\gamma = 8$ using digital PIV (DPIV). The study showed the presence of ‘wall ribs’, organised vortices along the impingement plane in the lateral direction of the jet. They questioned whether these wall ribs are formed in the boundary layer close to the impingement plane or already upstream of the impingement plane. They highlighted that these wall ribs are enhanced and might be sustained by vorticity upstream of the impingement region. Moreover, the authors observed so-called ‘cross ribs’, existing only in forced impinging jets, that are responsible for an increase of the intensity of wall ribs. Loubière and Pavageau (2008) investigated the vortical structures occurring in a double PTIJ formed by a splitting plate at the nozzle ($Re = 20,000 - 140,000$, $\gamma = 10$) using PIV with focus on the impingement region. They showed that these vortical structures are responsible for significant mass transfer across the jet stream. It was found that most of the vortical structures were concentrated in the impingement region, and the authors suggested that shape and size of these structures are primarily influenced by flow curvature caused by the impingement plane, and that this curvature is mainly determined by the jet height. The study by Koched et al. (2011) focused on wall ribs in the impingement region for water jets with $3,000 \leq Re \leq 16,000$ and $\gamma = 10$. They found that the length of the potential core depends on $Re$ and reaches a maximum of 3 jet widths, which complies with
previous studies. The impingement region height is found to be slightly dependent on $Re$ and equal to 10% and 13% of the total jet height for $Re = 3,000$ and $Re = 16,000$, respectively. The amount and size of the observed vortical structures is also slightly dependent on $Re$; they occur in a wide range of length scales and their shape is mainly elliptical, and the amount of clockwise vortices is proportional to the amount of counterclockwise vortices.

The aforementioned studies all highlighted the existence of vortical structures in the jet flow, their role in the dissipation of jet energy to the ambient environment, and the existence of 3D vortices in the impingement region, in which thermal energy can be exchanged between the jet and the impingement surface.

In spite of this rather large number of experimental studies on PTIJs performed in the past, there is still a scarcity of detailed and high-quality experimental data on PTIJs at moderate Reynolds numbers and a jet height larger than 10 jet widths, including an evaluation of mean and instantaneous velocity, turbulence intensity, Reynolds shear stress and vorticity. Although Maurel and Sollicic (2001) performed a similar study for a large extent of jet heights (5 to 50 jet widths), their experiments only provided point measurements of velocities and Reynolds stresses for $Re = 13,500$ and 27,000; while their PIV measurements were only conducted near the impingement region and were used to describe the vorticity field in this region. Therefore, the aim of the present study is to provide experimental data based on 2D PIV for the whole vertical centreplane of the PTIJ. The jet Reynolds number ranges from 7,200 to 13,500 and two different jet widths are studied: (1) jet width of 16 mm and $\gamma = h_{jet}/w_{jet} = 22.5$; and (2) jet width of 8 mm and $\gamma = 45$. Both instantaneous and time-averaged results are presented. Section 2.2 provides a short description of the experimental set-up, Section 2.3 contains results of the flow visualisations and the results of the PIV measurements are presented in Section 2.4. Section 2.5 (discussion) and Section 2.6 (conclusions) conclude this paper.

### 2.2 Experimental set-up

The isothermal PTIJ is generated in a water channel by issuing from a smoothly-shaped nozzle with a rectangular section upstream of the jet inlet (Fig. 1). The hydrostatic
pressure driving the flow is exerted by a water column with a conditioning section consisting of one honeycomb and two screens in order to reduce lateral and longitudinal turbulence intensity. Water was chosen as working fluid because it allowed to reach the moderate Reynolds numbers under study without a too large setup and the related required large field of view for the PIV measurements. The water jet is dynamically similar (equal Reynolds number) with air curtains in full-scale. The streamwise inlet velocity $V_0$ at the end of the rectangular section downstream of the smoothly-shaped contraction and the jet width $w_{\text{jet}}$ at the inlet determine the jet Reynolds number ($Re = V_0 w_{\text{jet}} / \nu$, with $\nu$ the kinematic viscosity). In this experiment, the jet inlet velocity corresponds to jet Reynolds numbers $Re = 7,200$ to $13,500$. The jet is issued vertically on a horizontal plate at a distance of $h_{\text{jet}} = 360$ mm. The jet width $w_{\text{jet}}$ is equal to $8$ mm and $16$ mm and the nozzle depth (dimension in spanwise direction) is $300$ mm, resulting in nozzle aspect ratios ($AR$, which is the ratio of nozzle depth to width $w_{\text{jet}}$) of $37.50$ and $18.75$, respectively. Due to the obstructing horizontal beam of the water channel the visible part of the jet is limited to $h'_{\text{jet}} = 320$ mm, with the missing part close to the jet inlet.

Figure 1: a) PIV experimental set-up. The measurement planes are indicated with ROI1, ROI2 and ROI3. b) Schematic representation of impinging jet
The following notations are used for the geometric parameters: $x$, $y$, $z$ for the lateral, the streamwise and the spanwise (i.e. out-of-plane) directions, respectively. The flow parameters are $u$, $v$, $w$ – the instantaneous lateral, streamwise and spanwise velocity components; $U$, $V$, $W$ – the mean lateral, streamwise and spanwise velocity components, $|v|$ and $|V|$ – the instantaneous and the mean velocity magnitude, $V_0$ – the mean jet inlet velocity; $v_{RMS}$, $u_{RMS}$, $v_{RMS}$ – root mean square values of the measured velocity magnitude and velocity components; $\omega_z$ and $\Omega_z$ – the instantaneous and the mean $z$-vorticity; $I_v$ and $I_{uv}$ – turbulence intensity based on only streamwise and both streamwise and lateral velocity fluctuations, respectively. The jet half-width $\delta_{0.5}$ at a certain downstream position $y$ (Fig. 1b) is the horizontal distance between the jet centreline and the point where the mean streamwise velocity is equal to half the local jet mean centreline velocity $V_{0,y}$.

Prior to the PIV measurements, flow visualisations with fluorescent dye are performed to reveal the flow pattern. The dye is injected in the flow field at $y = -35$ mm using an injection needle with an outer diameter of 0.8 mm and a length of 80 mm, enabling injection in the centre of the jet. It is illuminated in the centreplane of the test section by a slide projector mounted below the water channel. The obtained flow patterns are recorded with a digital photo and video camera positioned perpendicular to the measurement plane.

For the PIV measurements, a 2D PIV system is used consisting of a solid-state frequency-doubled Nd:Yag laser (wavelength 532 nm and repetition rate 15 Hz) as a light source, and a charge coupled device (CCD) camera (1600 x 1200 pixels resolution, up to 30 frames/s) for image recordings. The light sheet is delivered at the bottom of the water channel by a set of mirrors and cylindrical and spherical lenses. Seeding is provided by polyamide particles ($D_s = 50 \mu m$, density $\rho_s = 1,030 \text{ kg/m}^3$) added to the water. Each measurement set consists of 5,000 uncorrelated samples.

Three sets of measurements are performed, which focus on (Fig. 1a): (1) ROI1 – the entire measurement plane of width $W \times$ height $H = 192 \times 360 \text{ mm}^2$; (2) ROI2 – the region close to the jet inlet ($W \times H = 128 \times 192 \text{ mm}^2$); and (3) ROI3 – the impingement region ($W \times H = 128 \times 176 \text{ mm}^2$). The two latter sets provide measurement data at a higher resolution. The instantaneous PIV images are processed in five passes from 32 x 24 pixels

---

1 coordinates of the vertices in mm $(x,y,z) = (-96,0,150); (96,0,150); (96,360,150); (-96,360,150)$
2 coordinates of the vertices in mm $(x,y,z) = (-64,0,150); (64,0,150); (64,192,150); (-64,192,150)$
3 coordinates of the vertices in mm $(x,y,z) = (-64,184,150); (64,184,150); (64,360,150); (-64,360,150)$
to 16 x 12 pixels (for ROI1 and ROI2) and 48 x 32 pixels to 24 x 16 pixels (ROI3) with 50% overlap. Correlation peaks are fitted by a 3-point Gauss fit algorithm advised by Forliti et al. (2000). Filters for outlier detection are applied for maximum and normalised median displacements. The correlation peak ratio is set to 1.3 to disable erroneous vectors resulting from insufficient seeding and background noise. The time separation between pulses is chosen according to the general guideline for particle displacement, which states that the particle displacement should be limited to 25% of the interrogation area (Prasad 2000).

2.3 Flow visualisation results

Fig. 2 shows snapshots of the dye-visualised jet for $Re = 7,200$ and $w_{jet} = 8$ mm, corresponding to an inlet velocity $V_0$ of 0.9 m/s. The image indicates a growing shear layer and spreading of the jet after injection into the ambient fluid starting at a certain distance from the jet inlet. The indistinct jet boundaries exhibit turbulent behaviour with spanwise vortices developing in the outer region of the jet (Fig. 2a). The visualisation also indicates jet flapping as the jet is moving from side-to-side in the region close to the impingement plane: at time $T_1$ the jet is perpendicular to the impingement plane (Fig. 2a), however, 2 s later at $T_2$ it is slightly bent to the left (Fig. 2b), and 4 s later at $T_4$ it is bent to the right (Fig. 2d). At time $T_6$ (Fig. 2f), the jet again impinges perpendicularly on the surface.

The jet flapping was also reported and analysed in earlier studies, such as Goldschmidt and Bradshaw (1973) and Antonia et al. (1983). It plays an important role in heat transfer between the jet and the impingement plane (e.g. Camci and Herr 2002; Chiriac and Ortega 2002; Varieras et al. 2007).
Figure 2: Visualisation of the jet in the vertical centreplane with $Re = 7,200$ and $w_{jet} = 8$ mm. Time interval between consecutive images is 2 s

2.4 PIV measurement results

2.4.1 Main jet flow characteristics

Fig. 3a,b shows cross-jet profiles of the dimensionless time-averaged velocity magnitude ($|V|/V_0$) and of the turbulence intensity ($I_{uv} = |v_{RMS}|/V_0 = \sqrt{u_{RMS}^2 + v_{RMS}^2}/V_0$) obtained from the PIV measurements in ROI2. The values are obtained in the vertical centreplane for $Re = 7,200$ to 13,500 at two different downstream locations: $y = 4w_{jet}$ for $w_{jet} = 8$ mm and $y = 2w_{jet}$ for $w_{jet} = 16$ mm. These locations are chosen to provide information on flow conditions as close as possible to the inlet, while avoiding erroneous
data caused by reflections and visual obstruction by the top front beam of the water channel.

Fig. 3a shows that the time-averaged profiles for the velocity magnitude clearly resemble a top-hat distribution. The velocity profiles for different \( Re \) values for \( w_{jet} = 16 \text{ mm} \) show a good agreement among each other, which is also the case for the profiles of \( w_{jet} = 8 \text{ mm} \). However, the velocity profiles for different jet widths do show some differences near the edges of the jet \( (|x|/w_{jet} = 0.5) \), with larger gradients \( \partial(|V|/V_0)/\partial(x/w_{jet}) \) for \( w_{jet} = 16 \text{ mm} \), while the actual gradients \( \partial(|V|)/\partial x \) are larger for \( w_{jet} = 8 \text{ mm} \). Fig. 3b shows that the turbulence intensity is highest in the shear layer due to the high velocity gradients (high turbulence production) in this area. The maximum values are equal to \( I_{uv} \approx 0.4 \) for \( w_{jet} = 16 \text{ mm} \) and \( I_{uv} \approx 0.44 \) for \( w_{jet} = 8 \text{ mm} \). The turbulence intensities in the core of the jet reach \( l_{uv} \approx 0.05 \) for \( w_{jet} = 16 \text{ mm} \) and \( l_{uv} \approx 0.06 \) for \( w_{jet} = 8 \text{ mm} \). There is no clear tendency visible in terms of different \( Re \); however, the turbulence intensities in the central and shear layer regions are slightly higher for \( w_{jet} = 8 \text{ mm} \) \((AR = 37.50)\) than for \( w_{jet} = 16 \text{ mm} \) \((AR = 18.75)\), which is partly attributed to the higher actual gradients for \( w_{jet} = 8 \text{ mm} \).

To indicate whether the flow has laminar, turbulent or transitional conditions near the jet inlet at the beginning of the potential core, the boundary layer displacement thickness \( \delta^* \) and momentum thickness \( \theta \) are calculated as follows (Deo et al. 2007a):

\[
\delta^* = \int_0^{0.5w_{jet}} \left(1 - \frac{V}{V_{0,y}}\right) \, dx
\]
\[
\theta = \int_0^{0.5w_{jet}} \left(1 - \frac{V}{V_{0,y}}\right) \frac{V}{V_{0,y}} \, dx
\]

The displacement thickness \( \delta^* \) for \( w_{jet} = 16 \text{ mm} \) is about 0.05\( w_{jet} \) to 0.06\( w_{jet} \), while the momentum thickness \( \theta \approx 0.03w_{jet} \). This results in a shape factor \( H_{BL} = \delta^*/\theta \approx 1.6 - 2 \), indicative of turbulent flow. Note that the characteristic values of the shape factor for laminar and turbulent boundary layers are \( H_{BL} = 2.3 - 3.5 \) and \( H_{BL} = 1.3 - 2.2 \), respectively (Schlichting 1979). For \( w_{jet} = 8 \text{ mm} \) the displacement and momentum thicknesses are \( \delta^* \approx 0.06w_{jet} \) to 0.07\( w_{jet} \) and \( \theta \approx 0.03w_{jet} \) to 0.04\( w_{jet} \), respectively, yielding a shape factor of around \( H_{BL} \approx 1.7 - 1.8 \). That means that the measured flow conditions near the inlet are turbulent.
Fig. 3c,d compares the measured profiles of mean velocity and turbulence intensity \( (w_{\text{jet}} = 16 \text{ mm}; AR = 18.75, Re = 13,000; w_{\text{jet}} = 8 \text{ mm}; AR = 37.50, Re = 13,500) \) with those by Deo et al. (2007a) (\( AR = 20 \) and \( 30; Re = 18,000 \)), Maurel and Solliec (2001) (\( AR = 18; Re = 27,000 \)) and Koched et al. (2011) (\( AR = 20; Re = 11,000 \)). Note that Deo et al. (2007a) provided the inlet distributions of the mean velocity and the turbulence intensity at a distance \( y = 0.2w_{\text{jet}} \) from the jet inlet; Maurel and Solliec (2001) and Koched et al. (2011) measured the profiles at the jet inlet \( (y = 0) \). Fig. 3c shows that the mean velocity profiles of Deo et al. (2007a) and Maurel and Solliec (2001) have a more pronounced top-hat distribution than our experimental data. This can be explained by the fact that our profiles were taken at a certain distance downstream of the nozzle, where the jet shear layer has already grown to some extent and consumed part of the jet potential core. In addition, the geometrical configurations of the contractions in the aforementioned studies differ from each other. Deo et al. (2007a) used a radially-shaped contraction, Maurel and Solliec (2001) used a bell-shaped contraction with a long converging section, and Koched et al. (2011) used a long sharp-edged nozzle. This results in variations of the boundary layer development upstream of the inlet, causing dissimilarity of the inlet conditions, which can explain the differences found here. Fig. 3d shows a comparison of the profiles of the streamwise turbulence intensity \( (I_v) \) with those from literature. Maurel and Solliec (2001) did not provide turbulence intensity profiles near the inlet, but only its range from 1.6 to 2.8\% for different \( Re \), assuming a uniform distribution. Note that in the remainder of the paper \( I_{uv} \) is used, based on the turbulent fluctuations in both the streamwise and lateral directions. The values reported in the mentioned studies lie below \( I_v = 0.2 \) and are lower than the values measured in our experiments. This is again attributed to differences in nozzle geometries, \( Re \) at the jet inlet and different downstream sampling positions.

The centreline velocity of plane jets varies as a function of \( 1/\sqrt{y} \), where \( y \) is the downstream distance from the jet inlet (e.g. Scorer 1987; Pope 2000). Based on these centreline velocity distributions in the vertical centreplane, the vertical distances \( y_0 \) between jet inlet \( (y = 0) \) and the kinematic virtual jet origins \( (y' = 0) \) are defined. The virtual origin lies at \( y_0 \approx 2.3w_{\text{jet}} \) for \( w_{\text{jet}} = 8 \text{ mm} \), and \( y_0 \approx -0.8w_{\text{jet}} \) for \( w_{\text{jet}} = 16 \text{ mm} \). That means that the values for the profiles presented in Fig. 3a-d are taken at \( y' \approx 1.7w_{\text{jet}} \) for \( w_{\text{jet}} = 8 \text{ mm} \), and \( y' \approx 2.8w_{\text{jet}} \) for \( w_{\text{jet}} = 16 \text{ mm} \), where \( y' \) is the downstream distance from the kinematic virtual jet origin. To provide a clear comparison of the flow development at the beginning of the potential core between two different \( ARs \), Fig. 3e,f compares cross-jet profiles of the mean velocity and the turbulence intensity near the jet inlet.
Figure 3: a) Dimensionless mean velocity magnitude (|V|/V_0) and b) turbulence intensity $I_{uv}$ near jet inlet (at y = 2w_{jet} for w_{jet} = 16 mm, at y = 4w_{jet} for w_{jet} = 8 mm). c) Dimensionless mean velocity magnitude (|V|/V_0) and d) streamwise turbulence intensity $I_v$ near jet inlet (at y = 2w_{jet} for Re = 13,000, at y = 4w_{jet} for Re = 13,500) compared to other studies. e) Dimensionless mean velocity magnitude (|V|/V_0) and f) turbulence intensity near jet inlet at y' = 2.8w_{jet}.
taken at the same distance from the virtual jet origin, i.e. \( y' \approx 2.8w_{jet} \). The mean velocity profiles for \( w_{jet} = 8 \text{ mm} \) resemble rather parabolic than top-hat profiles, meaning that the jet spreads faster laterally and that ambient fluid has entrained the jet to a larger extent resulting in more developed shear layers than for \( w_{jet} = 16 \text{ mm} \).

Fig. 4a shows the dimensionless mean streamwise velocity along the jet centreline. From the mean velocity decay the length of the potential core \( (h_p) \) is defined as the distance from the jet nozzle where the local centreline mean velocity remains more than or equal to 98% of the inlet mean jet velocity, i.e. \( V_{0,y}/V_0 \geq 0.98 \) (Deo et al. 2007a). The length of the potential core measured starting from the jet inlet is \( h_p \approx 4.8w_{jet} \) for \( w_{jet} = 16 \text{ mm} \), whereas it is \( h_p \approx (6.1 - 7.0)w_{jet} \) for \( w_{jet} = 8 \text{ mm} \). This is caused by differences in the jet inlet conditions, i.e. nozzle \( AR \) and velocity at the jet inlet. Note that these distances do not take into account the positions of the virtual jet origins. Deo et al. (2007a) showed that the ratio \( h_p/w_{jet} \) for jets with equal \( Re \) increases with \( AR \) of the nozzle, which is in line with the present results. The centreline velocity decreases to the half of its inlet value \( V_0 \) at \( y \approx 0.85h_{jet} (=19w_{jet}) \) for \( w_{jet} = 16 \text{ mm} \) and at \( y \approx 0.55h_{jet} (=25w_{jet}) \) for \( w_{jet} = 8 \text{ mm} \).

Fig. 4b shows the decay of the inverse of the dimensionless mean centreline velocity squared as a function of \( y \). The jet decay rate \( K_u \) is defined as \( K_u = (V_0/V_{0,y})^2/(y - y_0)/w_{jet} \), with \( y_0 \) the vertical distance between jet inlet and the kinematic virtual jet origin. It can be quantified as the slope of the velocity distribution in Fig. 4b where near-linear behaviour is observed. The jet with \( w_{jet} = 8 \text{ mm} \) (\( AR = 37.50 \)) has a higher decay rate than the jet with \( w_{jet} = 16 \text{ mm} \) (\( AR = 18.75 \)), which is also in line with the findings of Deo et al. (2007a) regarding the influence of nozzle \( AR \). The values for \( w_{jet} = 16 \text{ mm} \) are about \( K_u \approx 0.14 - 0.15 \), and for \( w_{jet} = 8 \text{ mm} \) \( K_u \approx 0.17 - 0.18 \) meaning that entrainment rate of the ambient fluid within this region is higher for \( w_{jet} = 8 \text{ mm} \) (i.e. for higher \( AR \)). In previous experimental studies of free plane turbulent jets with sidewalls in the range of \( Re = 6,000 - 16,000 \), e.g. Jenkins and Goldschmidt (1973), Browne et al. (1983), Thomas and Chu (1989), Sakai et al. (2004), Deo et al. (2007a), Suresh et al. (2008), Alnahhal and Panidis (2009), a range of values of \( K_u \) from 0.11 to 0.22 was reported, which encompasses the values found in the present study.

The impingement region can be identified as the region in the vicinity of the impingement plane with a linear decrease of the centreline velocity (see Fig. 4a), the height of which can be denoted with \( h_i \). Fig. 4a shows that \( h_i \approx 0.12h_{jet} \) for \( w_{jet} = 16 \text{ mm} \) and \( h_i \approx 0.13h_{jet} \) for \( w_{jet} = 8 \text{ mm} \). These heights correspond well with the values by Maurel and Solliec (2001) and Koched et al. (2011), who report impingement region heights of \( h_i = (0.12 - 0.13)h_{jet} \) and \( h_i = (0.10 - 0.13)h_{jet} \), respectively. For a better estimation of the
impingement height additional flow quantities, such as centreline distribution of turbulence intensity and Reynolds shear stress in the vertical centreplane will be analysed and described later.

The linear distribution in Fig. 4b indicates the self-similarity of normalised mean velocity in the region 0.35 ≤ \( \frac{y}{h_{\text{jet}}} \) ≤ 0.75 for \( w_{\text{jet}} = 16 \text{ mm} \) and 0.20 ≤ \( \frac{y}{h_{\text{jet}}} \) ≤ 0.75 for \( w_{\text{jet}} = 8 \text{ mm} \). When converted to \( w_{\text{jet}} \) these values result in 7.5 ≤ \( \frac{y}{w_{\text{jet}}} \) ≤ 17 for \( w_{\text{jet}} = 16 \text{ mm} \) and 9 ≤ \( \frac{y}{w_{\text{jet}}} \) ≤ 33 for \( w_{\text{jet}} = 8 \text{ mm} \). Beyond \( \frac{y}{h_{\text{jet}}} > 0.75 \) the jet is influenced by strong three-dimensional effects which occur due to the presence of the impingement wall and sidewalls that prevent the jet to spread in the spanwise direction.

To further investigate the existence of self-similarity in the flow, the cross-jet profiles along the jet height are examined and compared with a Gaussian distribution: \( \frac{V_y}{V_0,y} = \exp(a(x/\delta)^2) \), where \( a = -0.693 \) (Rajaratnam 1976). This equation was used by, among others, Namer and Otugen (1988), Deo et al. (2007a) and Suresh et al. (2008). As jets with the same \( w_{\text{jet}} \) result in identical distributions for the investigated range of Reynolds numbers, only results for cases I and II, the parameters of which are specified in Table 1, are reported in Fig. 5. Note that the profiles are provided at distances \( y' \) from the virtual jet origins.

Figure 4: a) Dimensionless mean streamwise velocity along jet centreline \((V_{0,y}/V_0)\). Orange dashed lines indicate impingement region. b) Ratio \((V_0/V_{0,y})^2\) along jet centreline. Orange dashed lines indicate regions with close-to-linear distribution to define the jet decay rate. Note the double scale for \( y/w_{\text{jet}} \) on the right of Fig. 4a, 4b: in black for \( w_{\text{jet}} = 8 \text{ mm} \) and in cyan for \( w_{\text{jet}} = 16 \text{ mm} \).
From Fig. 5 it can be concluded that the velocity profiles resemble a Gaussian distribution for \( 5.9 \leq y'/w_{\text{jet}} \leq 20.1 \) for \( w_{\text{jet}} = 16 \text{ mm} \) and \( 4.8 \leq y'/w_{\text{jet}} \leq 35.1 \) for \( w_{\text{jet}} = 8 \text{ mm} \).

<table>
<thead>
<tr>
<th>Case</th>
<th>( \text{Re} )</th>
<th>( w_{\text{jet}} ) [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case I</td>
<td>8,000</td>
<td>16</td>
</tr>
<tr>
<td>Case II</td>
<td>7,200</td>
<td>8</td>
</tr>
</tbody>
</table>

Fig. 5 presents the turbulence intensity along the jet centreline for the two jet widths and for different \( \text{Re} \) values. The data from measurement sets at ROI2 and ROI3 are combined. A slight mismatch is observed around \( y/h_{\text{jet}} \approx 0.5 - 0.65 \) for \( w_{\text{jet}} = 16 \text{ mm} \) and \( y/h_{\text{jet}} \approx 0.5 - 0.6 \) for \( w_{\text{jet}} = 8 \text{ mm} \). This relatively small mismatch (6%) is likely to be caused by differences in interrogation area size between ROI2 and ROI3 due to different velocity magnitudes in these regions. Interrogation area size acts as a different spatial filter on the PIV data possibly leading to different turbulence statistics (e.g. Keane and Adrian 1990; Forliti et al. 2000). The shapes of the turbulence intensity profiles for the two jet widths are different due to different nozzle ARs and velocities at the jet inlet. The
maximum turbulence intensity at the centreline reaches $I_{uv} \approx 0.21$ at $y/h_{jet} \approx 0.55$ (equivalent to $y/w_{jet} \approx 12.4$) and $I_{uv} \approx 0.19$ at $y/h_{jet} \approx 0.25$ (equivalent to $y/w_{jet} \approx 11.3$) for $w_{jet} = 16$ mm and 8 mm, respectively. This gradual increase of the turbulence intensity downstream of the jet inlet can be attributed to the growth of vorticity in the shear layer, which consumes the potential core and causes higher velocity fluctuations along the centreline of the jet. At the same time, the local jet velocity downstream of the jet inlet is decreasing, resulting in higher turbulence intensities.

The same behaviour of centreline turbulence intensity was observed by e.g. Browne et al. (1983) and Deo et al. (2007a). Slightly higher values of turbulence intensity for $w_{jet} = 16$ mm can be explained by smaller $AR$ compared to $w_{jet} = 8$ mm that causes stronger influence of sidewalls and enhances velocity fluctuations (Deo et al. 2007a). Downstream of the locations of maximum turbulence intensities a linear decrease is present until $I_{uv} \approx 0.16$ and $I_{uv} \approx 0.13$ for $w_{jet} = 16$ mm and 8 mm, respectively. This linear decrease is present until the jet impingement region. In the impingement region, the shape of the turbulence intensity profiles for the two different jet widths is similar, and the turbulence intensities show an increase to $I_{uv} \approx 0.21$ and $I_{uv} \approx 0.16$ for $w_{jet} = 16$ mm and 8 mm, respectively. Maximum values are reached at $y/h_{jet} \approx 0.99$ for $w_{jet} = 16$ mm and $y/h_{jet} \approx 0.97$ for $w_{jet} = 8$ mm. This increase is caused by the compression of the
vortices in the streamwise direction due to the presence of the impingement plane causing stretching of the vortices in the lateral direction. The jet flow near the impingement plane diverges to two opposite lateral directions and forms so-called stretching wall vortices (e.g. Sakakibara et al. 2001) enhanced by vorticity from upstream of the impinging jet and by the high pressure at the stagnation zone (e.g. Tsubokura et al. 2003).

Fig. 7 shows the cross-jet profiles of turbulence intensity for case I (Fig. 7a) and case II (Fig. 7b). The distribution of $I_{uv}$ near the jet centreline downstream of $y'/w_{jet} = 5.9$ smoothens out to about 0.2 for case I and to about 0.15 for case II due to entrainment of the ambient fluid and shear layer growth, which also results in a decrease of bulk jet velocity. Within $2.8 \leq y'/w_{jet} \leq 7.9$, the turbulence intensities at the jet centreline and shear layer for case II are slightly higher than for case I at the same relative downstream distances. The opposite is valid when the flow approaches the impingement plane ($y'/w_{jet} \geq 12.1$), where turbulence intensities for case I are slightly higher. To determine at which regions of the jet the turbulence intensity achieves self-similarity, the cross-jet profiles of turbulence intensity along the jet have been analysed at more lines than depicted in Fig. 7 (not all profiles are included in Fig. 7 for clarity). It appears that the profiles of the turbulence intensity tend to exhibit self-similar behaviour within regions of $12.1 \leq y'/w_{jet} \leq 18.4$ for case I and $12.1 \leq y'/w_{jet} \leq 20.1$ for case II, which partially comprise the regions of self-similar behaviour found for cross-jet velocity profiles.

![Figure 7: Cross-jet profiles of the turbulence intensity ($I_{uv}$) for a) case I; b) case II](image-url)
2.4.2 *Velocity contours and velocity vectors*

Fig. 8 shows contours of the dimensionless mean streamwise velocity and the mean velocity vectors in the vertical centreplane for case I (Fig. 8a,c) and case II (Fig. 8b,d). The white areas at the top and bottom parts of the figure indicate zones where erroneous measurement data due to obstructions by the structural beam of the water channel and reflections from the surfaces of the water channel are excluded.

The jet half-width $\delta_{0.5}$ and the nominal jet boundaries $2\delta_{0.5}$ (e.g., Fan 1967; Jirka 2004; Socolofsky et al. 2013) are indicated by the dashed and dotted white lines, respectively. The near-linear variation of the jet half-width $\delta_{0.5}$ along jet centreline within the intermediate jet region characterises the spreading rate of the jet $K_y = d\delta_{0.5}/dy$ (Pope 2000), which is equal to $K_y = 0.10$ for case I and $K_y = 0.12$ for case II. Experimental studies for free plane turbulent jets with sidewalls in the range of $Re = 6,000 - 16,000$, e.g., Jenkins and Goldschmidt (1973), Browne et al. (1983), Thomas and Chu (1989), Sakai et al. (2004), Deo et al. (2007a,b), Suresh et al. (2008), reported values of $K_y$ in the range from 0.05 to 0.11. The jet spreading rate for case I lies within the range from literature, while the spreading rate for case II falls just outside this range. A possible explanation could be that although the jets considered in the aforementioned studies have a $Re$ range (6,000 - 16,000) and an $AR$ range (20 - 35) similar to the values in our experiments, there are some differences in the contractions used in various studies. Jenkins and Goldschmidt (1973), Browne et al. (1983), Thomas and Chu (1989) and Suresh et al. (2008) conducted experiments with a gradually converging contraction; Sakai et al. (2004) applied a gradually converging contraction but without a rectangular section upstream of the jet inlet; and Deo et al. (2007a,b) used a contraction with radially-shaped edges but without a conditioning section.

The nominal jet boundaries are determined based on the distance of two jet half-widths $\delta_{0.5}$ from the jet centreline ($2\delta_{0.5}$) along the jet height. The angles between the jet boundaries and jet axis in the intermediate jet region are about $11 - 12^\circ$ for case I and about $12 - 12.5^\circ$ for case II. The slope of the jet boundaries indicates the jet entrainment rate. Thus, entrainment of ambient fluid is slightly more intensive for case II than for case I. The larger opening angle, and thus the more intensive entrainment and jet spreading can be caused by the higher turbulence intensities in the shear layer close to the jet inlet for $w_{jet} = 8$ mm (case II) compared to $w_{jet} = 16$ mm (case I) (as shown
Figure 8: Contours of dimensionless mean streamwise velocity ($V/V_0$) and mean velocity vectors in the vertical centreplane for a, c) case I; b, d) case II
in Fig. 3b). Note that the development of the jet half-width $\delta_{0.5}$ and the nominal jet boundaries $2\delta_{0.5}$ is identical for jets with the same $w_{jet}$ but different Reynolds numbers.

Fig. 9 provides contours of the dimensionless absolute mean lateral velocity ($abs(U)/V_0$) in the vertical centreplane and indicates the lines where the lateral velocity component $U = 0$ and changes its direction. The plots indicate the growth of the lateral component of velocity near the jet centreline at the height $y/h_{jet} \approx 0.7 - 0.75$ due to the influence of the impingement plane. Recirculation occurs at the top part of the water channel, where the lateral velocity component reaches $abs(U) = 0.08V_0$.

**Figure 9: Contours of dimensionless absolute mean lateral velocity (abs(U)/V_0) with mean velocity vectors in the vertical centreplane for a) case I; b) case II**

### 2.4.3 Instantaneous velocity, vorticity $\omega_{xy}$, and Okubo-Weiss function $Q$

Contours of the dimensionless instantaneous velocity magnitude ($|\mathbf{V}|/V_0$) from the measurements in ROI2 for cases I and II are depicted in Fig. 10 and indicate widening of the jet and growing of its shear layer due to entrainment of ambient fluid on either side of the jet. This entrainment is caused by vorticity induced in the jet shear layer due to...
high velocity gradients. At this Re value the flow is highly turbulent and it is difficult to clearly distinguish individual vortices in the jet with the present measurement resolution of 16 x 12 pixels per interrogation area of 1.8 x 1.4 mm².

![Figure 10: Contours of dimensionless instantaneous velocity magnitude and instantaneous velocity vectors in the vertical centreplane for a) case I; b) case II. Note that the Fig.10b provides only part of the data for the measurement region ROI2](image)

The distributions of the instantaneous vorticity for cases I and II (Fig. 11a and Fig. 11c) indicate that the highest levels of vorticity occur close to the inlet induced by the shear layer. The absolute maximum values of vorticity reach \( \Omega_z \approx 160 \, \mu \text{s}^{-1} \) for case I and \( \Omega_z \approx 400 \, \mu \text{s}^{-1} \) for case II. Although not presented here, it is worth mentioning that for the nozzle with \( w_{\text{jet}} = 16 \, \text{mm} \) and \( Re = 13,000 \), the vorticity reaches \( \Omega_z \approx 250 \, \mu \text{s}^{-1} \), while for the nozzle with \( w_{\text{jet}} = 8 \, \text{mm} \) and \( Re = 13,500 \) the value reaches \( \Omega_z \approx 650 \, \mu \text{s}^{-1} \). So, for a similar range of Reynolds numbers the vorticity for the jet width \( w_{\text{jet}} = 8 \, \text{mm} \) is at least two times higher than for the jet width \( w_{\text{jet}} = 16 \, \text{mm} \), which can be attributed to the higher velocities (same Re and smaller width) and higher mean velocity gradients at the jet inlet for \( w_{\text{jet}} = 8 \, \text{mm} \). Note that the gradients \( \partial (|V|/V_0) / \partial (x/w_{\text{jet}}) \) at the jet inlet are larger for \( w_{\text{jet}} = 16 \, \text{mm} \), while the actual gradients \( \partial |V| / \partial x \) are larger for \( w_{\text{jet}} = 8 \, \text{mm} \).
Figure 11: Contours of a) instantaneous vorticity; b) Okubo-Weiss function in vertical centreplane for case I. c, d) Same for case II. Note that the figure provides only part of the data for the measurement region ROI2.

To analyse the presence of vortical structures in the flow domain the Okubo–Weiss function is used. The Okubo-Weiss function was independently defined by Okubo (1970) and Weiss (1991). Based on the Okubo–Weiss function, the $Q$ criterion for the vertical centreplane ($z = 0$) can be defined as follows:
\[
Q(x,y,z=0) = \left( \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 - \left( \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right)^2
\]  

In this equation, the first two terms denote the normal and shear strain component, respectively, and the last term represents the vorticity \( \omega_z \). The strain-dominated regions can be identified by \( Q > 0 \), while rotation-dominated regions can be identified by \( Q < 0 \). The Okubo–Weiss function is valid for two-dimensional flow and has been successfully used by, among others, Vosbeek et al. (1997), Isern-Fontanet et al. (2004), Molenaar et al. (2004), Cieślik et al. (2010) and van Hooff et al. (2012). Fig. 11b and Fig. 11d show distributions of \( Q \) for cases I and II, respectively, which reveal that along the jet centreline within the potential core region the \( Q \) values lie around 0, which is also the case in the background flow on both sides of the jet. The values \( Q < 0 \) indicate turbulent eddy cores surrounded by circulation areas, while the regions with \( Q > 0 \) indicate the strain dominated regions.

### 2.4.4 Reynolds shear stress

Fig. 12 shows the distribution of the dimensionless Reynolds shear stress \( (u'v'/V_0^2) \) in the vertical centreplane for cases I and II. These data are analysed for each measurement set and are used to define the impingement region height. The Reynolds shear stress reaches its maximum value in the shear layer where velocity gradients and turbulence intensity are high. The change in sign of the Reynolds stresses occurs at the centreline of the jet due to symmetry of the time-averaged flow, and at the transition region from the jet to a wall jet, which defines the impingement region. For both jet widths, the impingement height is defined based on the Reynolds shear stress and is about 0.1\( h_{\text{jet}} \). This value corresponds with the impingement region height based on the dimensionless centreline velocity magnitudes in Fig. 4 of this paper, and also with the values reported in other studies (Maurel and Solliec 2001; Koched et al. 2011).
Figure 12: Contours of dimensionless Reynolds shear stress ($u'v'/V_0^2$) in the vertical centreplane for a, b) case I (Re = 8,000 and $w_{jet} = 16$ mm): a) ROI1; b) ROI3. c, d) Same for case II (Re = 7,200 and $w_{jet} = 8$ mm)
2.5 Discussion

In this study 2D particle image velocimetry (PIV) measurements are performed for the analysis of plane turbulent impinging jets (PTIJs) at moderate Reynolds numbers ($Re = 7,200 – 13,500$). Two jet configurations are considered: (1) $w_{jet} = 8 \text{ mm (AR} = 37.50)$ and jet height $h_{jet} = 45w_{jet}$, (2) $w_{jet} = 16 \text{ mm (AR} = 18.75)$ and jet height $h_{jet} = 22.5w_{jet}$. Note that the absolute jet height for both configurations is $h_{jet} = 360 \text{ mm}$. In addition, flow visualisations are performed to analyse the flow pattern, which shows cyclic jet flapping.

Based on the centreline and cross-jet velocity and the turbulence intensity profiles the present study shows that PTIJs behave as a free plane turbulent jet up to at least $y/h_{jet} \approx 0.75$ for the two jet widths, which is in line with the study of Gutmark et al. (1978).

The jet decay rate $K_u$ was quantified by analysing the centreline velocity distribution and the following values were obtained: $K_u \approx 0.14 - 0.15$ for $w_{jet} = 16 \text{ mm}$ and $K_u \approx 0.17 - 0.18$ for $w_{jet} = 8 \text{ mm}$. These values from the current experimental study of PTIJs are within the range of jet decay rates for free jets as reported in the literature. In addition, the spreading rate of the jet $K_y$ was defined by examining the development of the jet half-width $\delta_{0.5}$ along the jet height: $K_y \approx 0.10$ for $w_{jet} = 16 \text{ mm}$ and $K_y \approx 0.12$ for $w_{jet} = 8 \text{ mm}$. These comparisons indicate that for jet flows with the same Reynolds number decay of the centreline velocities and entrainment of ambient fluid by the jet is more intensive for $w_{jet} = 8 \text{ mm (AR} = 37.50)$ than for $w_{jet} = 16 \text{ mm (AR} = 18.75)$. A comparison with values from previous experimental studies shows that the jet spreading rate for $w_{jet} = 16 \text{ mm}$ lies within the range reported in the literature, whereas the spreading rate for $w_{jet} = 8 \text{ mm}$ falls just outside this range. This difference can be explained by the variation of the jet inlet conditions between these experiments, namely the configurations of the contractions used.

Future research\(^4\) of the authors will include experiments for an extended Reynolds number range, i.e. higher $Re$ values (higher velocities). Moreover, 3D PIV measurements

---

\(^4\) Future research can be focused on the analysis of instantaneous fields of velocity and vorticity in order to identify and to analyze the Kelvin-Helmholtz instabilities that dominate the early stages of the jet transition process (from the potential core to intermediate region)
will be performed to address the influence of sidewalls on the flow development within the potential, intermediate and impingement regions of the current jet configuration, as was done in studies of Alnahhal and Panidis (2009) and Deo et al. (2007b). Finally, the experimental results will be applied for turbulence model validation in CFD simulations of PTIJs, including Reynolds-averaged Navier-Stokes and large eddy simulations.

2.6 Conclusions

This paper provides a detailed experimental analysis of plane turbulent impinging jets (PTIJs) with two jet widths (8 mm and 16 mm) at moderate Reynolds numbers (7,200 – 13,500) issued on a horizontal plane at a fixed relative distance of 22.5 and 45 jet widths. There is a clear scarcity on similar experimental studies that aim at understanding the flow and to provide a database for comparison with numerical models. Prior to the PIV measurements, flow visualisations are performed to analyse the flow pattern.

For each jet width and for a range of Reynolds numbers, three sets of measurement data are obtained and examined: (1) the entire measurement region (ROI1), (2) the region close to the inlet (ROI2), and (3) the region close to the impingement plane (ROI3) to increase the measurement resolution in this area. The measurement data include time-averaged quantities of velocity, turbulence intensity and Reynolds shear stress measured in the vertical centreplane. In addition, instantaneous fields of velocity and vorticity have been analysed.

From the obtained results the following conclusions can be drawn:

- The flow visualisations show the presence of cyclic flapping of the impinging jet.
- The following regions of the jet are distinguished: (1) the potential core region with the centreline velocity more than or equal to 98% of the inlet jet velocity \( h_p = 4.8 w_{jet} \) for \( w_{jet} = 16 \text{ mm} \), \( h_p = (6.1 - 7.0) w_{jet} \) for \( w_{jet} = 8 \text{ mm} \), (2) the intermediate region with decaying centreline velocity comprising the transition region from the potential core to the developed region with largely self-similar profiles for mean velocity and turbulence intensity, (3) the impingement region, where the flow is influenced by the pressure created by the opposing impingement plane \( h_i \approx (0.10 - 0.13) h_{jet} \).
– Calculation of the shape factor \((H_{BL} = \delta'/\Theta)\) near the inlet indicated that the flow near the jet inlet was turbulent.
– The profiles of centreline mean velocity, cross-jet mean velocity and turbulence intensity show that the PTJJs considered in the present study behave as a free plane turbulent jet until \(y/h_{jet} \approx 0.7 - 0.75\).
– The self-similarity and self-preservation of the jet is shown by fitting the actual velocity profiles with theoretical profiles of a Gaussian distribution and analysing the cross-jet profiles of the turbulence intensity depending on the jet half-width. The investigated jet configurations exhibit self-similar behaviour of cross-jet velocity profiles between \(5.9 \leq y'/w_{jet} \leq 20.1\) for \(w_{jet} = 16\) mm, and between \(4.8 \leq y'/w_{jet} \leq 35.1\) for \(w_{jet} = 8\) mm. The profiles of turbulence intensity tend to achieve self-similarity between \(12.1 \leq y'/w_{jet} \leq 18.4\) for \(w_{jet} = 16\) mm, and between \(12.1 \leq y'/w_{jet} \leq 20.1\) for \(w_{jet} = 8\) mm.
– The jet decay rate \(K_u\) is quantified by analysing the centreline velocity distribution and the following values are obtained: \(K_u \approx 0.14 - 0.15\) for \(w_{jet} = 16\) mm and \(K_u \approx 0.17 - 0.18\) for \(w_{jet} = 8\) mm.
– The spreading rate \(K_y\) of the jet is defined by examining the development of the jet half-width \(\delta_{0.5}\) along the jet height: \(K_y \approx 0.10\) for \(w_{jet} = 16\) mm and \(K_y \approx 0.12\) for \(w_{jet} = 8\) mm.
– A distinct influence of the nozzle aspect ratio on the development of the profiles of the centreline mean velocity and the turbulence intensity in the streamwise direction was found. Jet nozzles with a higher aspect ratio showed more intensive entrainment and a faster decay of the centreline velocity for the same value of the jet Reynolds number. This is caused by the higher inlet jet velocity, resulting in higher mean velocity gradients, higher turbulence intensity and higher vorticity in the shear layer.

**Acknowledgements**

The authors are thankful to Geert-Jan Maas and Stan van Asten (Laboratory of the unit Building Physics and Services) and Ad Holten (Fluid Dynamics Laboratory) at Eindhoven University of Technology for their assistance in the development and construction of the experimental set-up and their support during the measurements.
Twan van Hooff is currently a postdoctoral fellow of the Research Foundation – Flanders (FWO) and acknowledges its financial support (project FWO 12R9715N).
References


Fan L-N (1967) Turbulent buoyant jets into stratified or flowing ambient fluids. W. M. Keck Laboratory of Hydraulics and Water Resources Report, 15. California Institute of Technology, Pasadena


Jenkins PE, Goldschmidt VW (1973) Mean temperature and velocity measurements in a plane turbulent jet J Fluid Eng-T ASME 95:81-584


Scorer RS (1978) Environmental aerodynamics. Ellis Horwood, Chichester


Chapter 3

Validation of steady RANS modelling of isothermal plane turbulent impinging jets at moderate Reynolds numbers

This chapter has been published as a peer-reviewed journal paper:


Abstract

The numerical modelling of impinging jet flows is not straightforward as it should not only solve the shear layer development in the free jet region, but also the near-wall behaviour (streamline curvature) and the resulting wall jets after impingement. This study presents a validation study of steady Reynolds-averaged Navier-Stokes turbulence models for predicting isothermal plane turbulent impinging jets at two different slot Reynolds numbers, i.e. \( Re = 8,000 \) (case I) and \( Re = 13,000 \) (case II), based on 2D particle image velocimetry measurements. In addition, an in-depth analysis of the results provided by the five different turbulence models: standard \( k-\epsilon \) (SKE), realizable \( k-\epsilon \) (RKE), RNG \( k-\epsilon \), SST \( k-\omega \), and a Reynolds stress model (RSM), is performed. The results show that: (1) for both Reynolds numbers the best agreement with measured velocities and turbulent kinetic energy in the region near the jet nozzle is achieved with SST; (2) the best predictions of potential core length are provided by RNG (case I) and RKE (case II); (3) centreline distributions of velocities and turbulent kinetic energy are most accurately predicted by RNG and RKE for case I, while for case II the best agreement with experimental data is obtained by SKE and RNG; (4) the best overall performance for both cases in predicting velocities is provided by RKE, and by RKE and RNG when considering turbulent kinetic energy; (5) all models more accurately predict the jet spreading rate in the intermediate region than in the potential core region; (6) for both Reynolds numbers SKE provides the most accurate estimation of jet decay rate.
3.1 Introduction

Air curtains are used in many practical applications to separate two environments in terms of heat and mass transfer, for example, at entrances of buildings to reduce heat losses (e.g. Siren 2003, Foster et al. 2006, Gil-Lopez et al. 2014), in cleanrooms and hospitals to prevent pollutant spreading (e.g. Shih et al. 2011, Zhai and Osborne 2013), inside buildings (e.g. Luo et al. 2013) and tunnels (e.g. Rydock et al. 2000) to prevent smoke propagation between two or more zones.

An air curtain can be represented by a plane turbulent impinging jet (PTIJ) as schematically shown in Figure 1. The jet consists of the following three distinctive flow regions (e.g. Guyonnaud et al. 2000, Maurel and Sollic 2001, Koched et al. 2011): (1) the potential core region with a centreline velocity more than or equal to 98% of the inlet jet velocity (length of the potential core region \( h_p \) is generally \( 3w_{jet} \leq h_p \leq 8w_{jet} \), with \( w_{jet} \) the width of the nozzle from which the jet is issued), (2) the intermediate region with decaying centreline velocity, including the transition region from the potential core to the developed region, with self-similar profiles for mean velocity and turbulence intensity, (3) the impingement region, where the flow is influenced by the pressure gradient created by the opposing impingement plate (height of the impingement region \( h_i \approx (0.10 - 0.13)h_{jet} \) with \( h_{jet} \) the distance from the nozzle to the floor). The length of the potential core \( h_p \) is influenced by the turbulence intensity at the jet nozzle, jet Reynolds number \( Re \), the shape of the jet nozzle and its aspect ratio \( AR \), which is the ratio of nozzle depth to width \( w_{jet} \) (e.g. Maurel and Sollic 2001, Koched et al. 2011). The height of the impingement region \( h_i \) is found to be slightly dependent on jet Reynolds number (e.g. Koched et al. 2011). However, Maurel and Sollic (2001) stated that the jet behaviour did not vary with Reynolds number or jet height. A number of studies provided correlations for jet velocity distributions within the intermediate region (e.g. Rajaratnam 1976, Awbi 1991) and near the impingement plate (e.g. Maurel and Sollic 2001).

The scientific literature available on air curtains and plane impinging jets can be divided into two categories: (1) application-oriented studies on air curtains and (2) basic studies on plane impinging air jets. Applied and basic studies can both be performed experimentally (at full or reduced scale, on-site or in a laboratory) and numerically, i.e. using computational fluid dynamics (CFD). Application-oriented studies are generally focused on the dimensioning of air curtains, determination of the air-curtain efficiency
by development of empirical formulae, and defining the most influential parameters with respect to the air curtain separation efficiency. Basic studies generally analyse the structure of the jet, turbulence levels and vortical structures by means of different measurement, visualisation and numerical techniques. This paper focuses on the capability of steady Reynolds-averaged Navier-Stokes (RANS) CFD simulations to predict plane turbulent impinging jet flows. Table 1 provides a non-exhaustive overview of previous numerical validation studies with the steady RANS approach on PTIJs. This overview is used to define the novel contribution of the present paper.

Modelling PTIJ involves solving the free shear layer within the jet potential core and intermediate region, and the jet near-wall behaviour in the impingement region. Seyedein et al. (1994), Heyerichs and Pollard (1996), Shi et al. (2002), Habli et al. (2014), Achari and Das (2015), Moureh and Yataghene (2016) and others questioned the performance of the standard $k$-$\varepsilon$ model (SKE) for PTIJ, while a number of validation studies for free plane turbulent jets (i.e. without impingement) showed a sufficient performance of steady RANS in combination with SKE (e.g. Berg et al. 2006,
<table>
<thead>
<tr>
<th>Authors</th>
<th>$h_{jet} / w_{jet}$</th>
<th>$Re$</th>
<th>Turb. model</th>
<th>Inlet boundary condition</th>
<th>Validation parameters</th>
<th>Provided results</th>
</tr>
</thead>
<tbody>
<tr>
<td>Seyedein et al. 1994</td>
<td>2.5-7.5</td>
<td>5,000-10,000</td>
<td>LRKE$<em>{LS}$, LRKE$</em>{C}$</td>
<td>Vel, $k$, $\varepsilon$</td>
<td>$P$, $Nu$</td>
<td>Vel, $P$, $Nu$</td>
</tr>
<tr>
<td>Heyerichs and Pollard 1996</td>
<td>2.6</td>
<td>10,000</td>
<td>SKE, LRKE, SKO</td>
<td>Vel, $k$, $\varepsilon$</td>
<td>$Nu$</td>
<td>$Nu$, $C_f$</td>
</tr>
<tr>
<td>Chen et al. 1999</td>
<td>2-8</td>
<td>250-20,000</td>
<td>SKO</td>
<td>Vel, $k$, $\varepsilon$</td>
<td>$Nu$, Vel</td>
<td>Vel, $Nu$, $Sh$</td>
</tr>
<tr>
<td>Shi et al. 2002</td>
<td>2, 6, 12</td>
<td>1,500-71,300</td>
<td>SKE, RSM$_L$</td>
<td>Vel, $I$, $l$</td>
<td>$Nu$</td>
<td>Vel, $Nu$</td>
</tr>
<tr>
<td>Park et al. 2003</td>
<td>0.5-4</td>
<td>9,800; 16,400; 25,100</td>
<td>SKO</td>
<td>Vel, $k$, $\omega$</td>
<td>$Nu$, Vel</td>
<td>Vel, $Nu$, $C_f$</td>
</tr>
<tr>
<td>Angioletti et al. 2005</td>
<td>4.5</td>
<td>1,000-4,000</td>
<td>RNG, RSM$_L$, SST</td>
<td>Vel, $I$</td>
<td>Vel, $Nu$</td>
<td>Vel, $Nu$</td>
</tr>
<tr>
<td>Le Song and Prud’homme 2007</td>
<td>10</td>
<td>6,000</td>
<td>URANS LRKE</td>
<td>Vel, $k$, $\varepsilon$</td>
<td>Vel, $RMS$</td>
<td>Vel, $k$, Vel, $k$</td>
</tr>
<tr>
<td>Isman et al. 2008</td>
<td>4-10</td>
<td>4,000-12,000</td>
<td>SKE, RNG</td>
<td>Vel, $k$, $\varepsilon$</td>
<td>$Nu$</td>
<td>$k$, $\varepsilon$, $Nu$</td>
</tr>
<tr>
<td>Jaramillo et al. 2008</td>
<td>4, 9.2</td>
<td>20,000</td>
<td>SKO, SKE</td>
<td>Vel, $k$, $\varepsilon$</td>
<td>Vel, $RMS$, $Nu$</td>
<td>Vel, $RMS$, $Nu$</td>
</tr>
<tr>
<td>Rhea et al. 2009</td>
<td>8</td>
<td>10,000</td>
<td>RSM$_D$</td>
<td>Vel, $l$</td>
<td>Vel, $RMS$, $Re$ stresses</td>
<td>Vel, $RMS$, $Re$ stresses</td>
</tr>
<tr>
<td>Koseoglu and Baskaya 2010</td>
<td>2, 6, 12</td>
<td>10,000</td>
<td>LRKE</td>
<td>Vel, $k$, $\varepsilon$</td>
<td>$Nu$</td>
<td>$Nu$</td>
</tr>
<tr>
<td>Kubacki and Dick 2010</td>
<td>4, 9.2, 10</td>
<td>13,500; 20,000</td>
<td>WKO</td>
<td>Vel, $k$, $\omega$</td>
<td>Vel, $RMS$, $Re$ stresses, $C_i$, $Nu$</td>
<td>Vel, $I$, $Re$ stresses, $C_i$, $Nu$, Vel, $k$</td>
</tr>
<tr>
<td>Dutta et al. 2013</td>
<td>4, 9.2</td>
<td>20,000</td>
<td>LRKE, RNG, RKE, RKE$_{SKE}$, SKO, SST, $v^2f$</td>
<td>Vel, $k$, $\varepsilon$, $\omega$, $RMS$</td>
<td>Vel, $RMS$, $Nu$</td>
<td>Vel, $I$, $Nu$, Vel, $I$</td>
</tr>
<tr>
<td>Achari and Das 2015</td>
<td>4, 9.2</td>
<td>20,000</td>
<td>LRKE, SKE, SKO</td>
<td>Vel, $k$, $\varepsilon$, $\omega$</td>
<td>Vel, $RMS$, $I$, $P$, $C_f$, $Nu$, $T$</td>
<td>Vel, $RMS$, $P$, $C_f$, $Nu$</td>
</tr>
<tr>
<td>Benmouhoub and Mataoui 2015</td>
<td>8</td>
<td>10,000-25,000</td>
<td>RSM$_L$</td>
<td>Vel, $k$, $\varepsilon$</td>
<td>Vel, $Nu$</td>
<td>Vel, $T$</td>
</tr>
<tr>
<td>Moureh and Yataghene 2016</td>
<td>10</td>
<td>8,000</td>
<td>SKE, RSM$_{LR}$</td>
<td>Vel, $I$, $Re$ stresses</td>
<td>Vel, $RMS$</td>
<td>Vel, $I$, $T$</td>
</tr>
</tbody>
</table>

Table 1: Overview of CFD studies with RANS on PTIJ

Table legend: WKO - $k$-$\omega$ by Wilcox, LRKE - Chien Low-Re $k$-$\varepsilon$, LRKE$_{LS}$ - Launder and Sharma Low-Re $k$-$\varepsilon$, LRKE$_{LB}$ - Lam-Brethorst Low-Re $k$-$\varepsilon$, RSM$_L$ – Launder Reynolds stress model (RSM), RSM$_{LR}$ – Wilcox RSM, RSM$_{D}$ – Dianat RSM, $v^2f$ – normal velocity relaxation model of Durbin, $C_i$ – skin friction coefficient, $I$ – turbulence intensity, $l$ – turbulence length scale, $k$ – turbulent kinetic energy, $Nu$ – Nusselt number, $P$ – surface pressure, $RMS$ – RMS velocity distributions, $Sh$ – Sherwood number, $T$ – temperature distributions, Vel – velocity distributions, $\varepsilon$ – turbulence dissipation rate, $\omega$ – specific turbulence dissipation rate.
Aziz et al. 2008, Isman et al. 2008). According to Le Song and Prud’homme (2007), Rhea et al. (2009) and Kubacki et al. (2013), the Boussinesq relationship that uses the turbulent viscosity to relate turbulent shear stresses and mean strain rate is responsible for the inability of RANS models to accurately predict the turbulence quantities in the free jet region and specifically close to the wall. However, the performance of turbulence models can differ per characteristic zone of a PTIJ (e.g. Zuckerman and Lior 2006). In addition, one turbulence model can provide accurate predictions for one flow variable while lacking accuracy in predicting another. For example, Aziz et al. (2008) found a better performance of SKE than RNG in predicting the jet spreading rate, jet decay rate and velocity profiles of a free plane turbulent jet, while the turbulent kinetic energy was only accurately predicted by RNG. Angioletti et al. (2005), Jaramillo et al. (2008) and Dutta et al. (2013) indicated the absence of generality in results for different studied cases of PTIJ due to different boundary conditions and computational settings.

In spite of the rather large number of numerical studies on PTIJs performed in the past, there is still a scarcity of validation studies for PTIJ at moderate Reynolds numbers (i.e. $5,000 < \text{Re} < 15,000$) for a jet height larger than 10 jet nozzle widths, including an evaluation of both mean velocity and turbulence intensity (see Table 1). Moreover, most of the numerical studies on PTIJs only focused on the jet impingement region. Furthermore, there is an evident inconsistency in conclusions of validation studies on PTIJs. Therefore, the aim of the present study is to validate steady RANS CFD simulations of PTIJ flows with a jet height to width ratio of $\frac{h_{\text{jet}}}{w_{\text{jet}}} = 22.5$ at two moderate Reynolds numbers, i.e. $\text{Re} = 8,000$ and $\text{Re} = 13,000$, based on 2D particle image velocimetry (PIV) data for the whole vertical centreplane of the PTIJ. Section 3.2 provides a short description of the experimental set-up. Section 3.3 contains an overview of the computational model. The results of the CFD simulations are presented in Section 3.4. Section 3.5 (discussion) and Section 3.6 (conclusions) conclude this paper.

### 3.2 Reduced-scale PIV measurements

For validation purposes, isothermal reduced-scale experiments of PTIJ flow are performed in a water channel, where a plane jet is issued from a smoothly-shaped nozzle (Fig. 2). Hydrostatic pressure is created by a water column and upstream of the nozzle,
water flows through a conditioning section consisting of one honeycomb and two screens in order to reduce lateral and longitudinal turbulence intensity and the mean velocity gradients (more uniform flow). The Reynolds number is defined based on the mean streamwise velocity at the jet exit $V_0$ and the jet nozzle width $w_{jet}$; $Re = (V_0 w_{jet}) / \nu$, with $\nu$ the kinematic viscosity. In the experiment, $V_0$ is equal to 0.50 m/s and 0.81 m/s, corresponding to $Re = 8,000$ and $Re = 13,000$, respectively. The jet is issued vertically on a horizontal plate at a distance of $h_{jet}$ = 360 mm from the jet inlet. The value of $w_{jet}$ is equal to 16 mm resulting in $h_{jet}/w_{jet} = 22.5$ and the nozzle depth (dimension in spanwise direction; z direction) is 300 mm, resulting in a nozzle aspect ratio $AR$ of 18.75.

For the PIV measurements, a 2D PIV system is used consisting of a solid-state frequency-doubled Nd:Yag laser (wavelength 532 nm and repetition rate 15 Hz) as a light source, and a charge coupled device (CCD) camera (1600 x 1200 pixels resolution, up to 30 frames/s) for image recordings. The light sheet is delivered at the bottom of the water channel by a set of mirrors and cylindrical and spherical lenses. Seeding is provided by polyamide particles ($D = 50 \mu$m, density $\rho_s = 1,030$ kg/m$^3$) added to the water. Three sets of measurements are performed, which focus on (Fig. 2): (1) ROI1 – the entire measurement plane of ($W \times H = 192 \times 360$ mm$^2$), (2) ROI2 – the region close to the jet nozzle ($W \times H = 128 \times 192$ mm$^2$), and (3) ROI3 – the impingement region ($W \times H = 128 \times 176$ mm$^2$). The PIV measurements provide information on the mean velocities and turbulence intensities in the vertical centreplane. More information on the experimental set-up and measurement results can be found in Khayrullina et al. (2017).

### 3.3 CFD simulations

#### 3.3.1 Computational geometry and grid

The computational geometry ($L \times H \times D = 1.0 \times 0.36 \times 0.15$ m$^3$) replicates one-fourth of the experimental setup described in Khayrullina et al. (2017) (Fig. 3). A symmetry boundary condition with zero normal velocities and zero normal gradients of all variables is used to limit the computational costs, as the geometry and time-averaged flow pattern are regarded to be symmetrical. This assumption is successfully verified by comparison of the results with those obtained from a simulation with the full domain.
Figure 2: Experimental set-up with indication of measurement regions of interest (ROI) (Khayrullina et al. 2017)

(dimensions equal to experimental set-up). In addition, a 3D computational geometry is chosen for this validation study, since small but noticeable differences are observed between the results from 3D and 2D simulations. The computational grid is created by the surface-grid extrusion technique (e.g. van Hooff and Blocken 2010) resulting in a structured hexahedral mesh for the water channel section and an unstructured hexahedral mesh for the smoothly shaped nozzle. A high spatial resolution is applied for the boundary layers (near the nozzle surfaces and the impingement plate), in regions with high velocity gradients at the jet nozzle \((y = 0)\), and at the outlet of the domain. The grid resolution is obtained from a grid-sensitivity analysis using three different grids (see Section 3.3.4 for results) with a grid-refinement factor of \(\sqrt{2}\) in each direction; coarse grid \((1,003,696\) cells\), basic grid \((2,809,800\) cells\) and fine grid \((7,915,275\) cells\). Across the computational inlet (half of the nozzle width) 14, 20 and 28 cells are used for coarse, basic and fine grids, respectively (Fig. 4). The dimensionless wall distances \((y^*,\text{average values})\) for the coarse, basic and fine grids in the jet nozzle are equal to 4, 2.5 and 1,
respectively, while at the impingement plate they are equal to 5, 1.25 and 0.5, respectively. The value of \( y^* \) is defined as \( y^* = \frac{\rho C_\mu^{1/4} k_P^{3/4} y_P^{1/2}}{\mu} \) with \( \rho \) the density of the fluid (water), \( \mu \) the dynamic viscosity, \( k_P \) the turbulent kinetic energy in the centre point \( P \) of the wall-adjacent cell, \( y_P \) the distance from point \( P \) to the wall, \( C_\mu \) the model constant generally equal to 0.09 (exact value depends on the used turbulence model). The low values of \( y^* \) (below 5) enable the use of low-Reynolds number modelling, which implies resolving the flow in the immediate vicinity of the wall including the thin viscous sublayer.

### 3.3.2 Boundary conditions

At the inlet of the computational domain a uniform jet exit velocity in vertical direction of \( V_0 = 0.085 \) m/s and \( V_0 = 0.138 \) m/s is imposed, which lead to \( Re = 8,000 \) (case I) and \( Re = 13,000 \) (case II) at the nozzle exit, respectively. The turbulence intensity is set to 15\%, which resulted in a turbulence intensity on the jet centreline near the jet nozzle that corresponded to the measured values, and the turbulence length scale \( l (= 0.07 L_{gap}/C_\mu^{3/4}) \) is calculated based on the opening size (\( L_{gap} = 1 \) mm) in the finest screen in the conditioning section with \( C_\mu \) equal to 0.09 (ANSYS Inc. 2013). At the outlet, zero static gauge pressure is applied, whereas zero normal velocities and zero normal gradients of all the variables are imposed at the two symmetry planes. The remaining surfaces are modelled as no-slip walls.

### 3.3.3 Solver settings

The 3D steady RANS equations are solved with ANSYS Fluent 16 (ANSYS Inc. 2013) using five different turbulence models to provide closure: (1) standard \( k-\varepsilon \) model (SKE) (Jones and Launder 1972); (2) realizable \( k-\varepsilon \) model (RKE) (Shih et al. 1995); (3) renormalization group \( k-\varepsilon \) model (RNG) (Yakhot et al. 1992); (4) shear stress transport \( k-\omega \) model (SST) (Menter 1994) with low-Re corrections and a limiter applied to the production of turbulent kinetic energy (Menter 1993); (5) linear pressure-strain Reynolds stress model (RSM) (Launder 1989). These models are chosen due to their good performance in previous PTIJ studies (see Section 3.1) and their common use in engineering applications due to their good balance between computational demand and accuracy (e.g. Blocken 2018). Moreover, these turbulence models represent models from two different groups: 1) first-order closure models based on the Boussinesq eddy-viscosity hypothesis (\( k-\varepsilon \)
and \( k-\omega \) models); and 2) second-order closure models; i.e. Reynolds stress models (RSM). All models utilise low-Re number modelling for near-wall regions which is possible due to the sufficiently low dimensionless wall distances that are present (\( y^* < 5 \), see Section 3.3.1).

Second-order discretisation schemes are used for both the convection and viscous terms of the governing equations, pressure-velocity coupling is performed by the SIMPLEC algorithm, and pressure interpolation is second order. Oscillatory convergence of the scaled residuals is observed for RKE, RNG, SST and RSM. Due to oscillations of the velocity magnitude observed at a reference point on the jet centreline at \( y/h_{\text{jet}} = 0.9 \) (see
Figure 4: Configurations of the computational grid with indication of the boundary conditions and the number of cells over the half of the contraction width: coarse (14 cells per inlet), middle (20 cells) and fine (28 cells)

Fig. 5a), the results of RKE, RNG, SST presented in the remainder of the paper are averaged over a sufficient number of iterations (70,000–100,000 iterations). The values of the cumulative moving average (CMA) of velocity magnitude in Figure 5b show a steady behaviour for all models meaning that the results are averaged over a sufficient number of iterations.

3.3.4 Grid-sensitivity analysis

The grid-sensitivity analysis is conducted with RKE and SST for case I ($Re = 8,000$) with coarse (1,003,696 cells), basic (2,809,800 cells) and fine (7,915,275 cells) grids. The results of the grid-sensitivity analysis are provided in Figure 6; i.e. the centreline profiles ($x = 0$, $z = 0$) of normalised jet velocity in the vertical direction ($V/V_0$). The grid-convergence index (GCI) by Roache (1994, 1997) is calculated for the basic grid, which is shown by shaded bounds in Figure 6, using:
Figure 5: a) Velocity magnitude |V| monitored over number of iterations on the jet centreline at y/h_{jet} = 0.9 for different turbulence models. b) Cumulative moving average of velocity magnitude at the same point. Note that the zeroth iteration corresponds to the iteration where the scaled residuals levelled off and started to oscillate.

\[
GCI'_{\text{basic}} = F_s \frac{r^p \left( \frac{V_{\text{basic}}}{V_0} - \frac{V_{\text{fine}}}{V_0} \right)}{1 - r^p}
\]

with \( F_s \) the safety factor equal to the recommended value of 1.25 when at least three grids are analysed, \( r \) the linear grid refinement factor equal to \( \sqrt{2} \), \( p \) the formal order of accuracy, which is assigned the value of 2 as second-order discretisation schemes are used for the simulations (Roache 1997). The requirement regarding the minimum grid refinement factor is not straightforward. Guidelines by Casey and Wintergerste (2000) stated the grid should be doubled twice in each direction for grid-sensitivity analyses, resulting in \( r = 2 \). However, Roache (1998) and Celik et al. (2008) stated that for engineering applications, an appropriate grid refinement factor should be at least 1.3 and 1.1, respectively. In order to limit the increase in computational demand per grid.
refinement, and to be in line with the different aforementioned guidelines, the grid refinement factor in this study is $r = \sqrt{2}$.

Figure 6 shows that the results of the grid-sensitivity study vary per turbulence model. The prediction of velocities by RKE is sensitive to the grid resolution within the intermediate ($0.2 < y/h_{jet} < 0.87$) and the impingement ($y/h_{jet} \geq 0.87$) regions. However, the results obtained with the basic and the fine grids are very close to each other. The solution provided by SST shows a higher sensitivity to the grid resolution for basic and fine grids in the intermediate ($0.35 < y/h_{jet} < 0.87$) jet region. The average values of $GCI'_{\text{basic}}$ along the jet centreline are 1.0% and 4.3% for RKE and SST, respectively. The maximum values of $GCI'_{\text{basic}}$ are 2.2% at $y/h_{jet} = 0.55$ for RKE and 9.7% at $y/h_{jet} = 0.89$ for SST. Based on this analysis it is concluded that the basic grid provides nearly grid-independent results and it is therefore used in the remainder of this study.

Figure 6: Results of grid-sensitivity analysis: $V/V_0$ along jet centreline. a) RKE; b) SST. Grey bounds indicate $GCI$ index calculated for the basic grid.
3.4 Results

3.4.1 Validation metrics

In order to provide a quantitative assessment of the performance of different turbulence models the following validation metrics are applied: the factor of 1.1 and 1.5 of the observations ($\text{FAC}_{1.1}$ and $\text{FAC}_{1.5}$, respectively) and the correlation factor ($R$). These metrics are calculated as follows:

\[
\text{FAC}_{1.1} = \frac{1}{n} \sum_{i=1}^{n} N_i \quad \text{with} \quad N_i = \begin{cases} 
1 & \text{for} \quad 0.91 \leq \frac{P_i}{O_i} \leq 1.1 \\
0 & \text{else}
\end{cases}
\]  

(2)

\[
\text{FAC}_{1.5} = \frac{1}{n} \sum_{i=1}^{n} N_i \quad \text{with} \quad N_i = \begin{cases} 
1 & \text{for} \quad 0.67 \leq \frac{P_i}{O_i} \leq 1.5 \\
0 & \text{else}
\end{cases}
\]  

(3)

\[
R = \frac{1}{n} \sum_{i=1}^{n} (O_i - \overline{O})(P_i - \overline{P})
\]  

\[= \frac{\sigma_P \sigma_O}{\sigma_P^2 + \sigma_O^2}
\]  

(4)

with $P_i$ and $O_i$ the time-averaged values obtained from CFD simulations and PIV experiments, respectively, $n$ the number of measurement data points, $\sigma_P$ and $\sigma_O$ the standard deviation of $P_i$ and $O_i$, respectively. The square brackets indicate averaging over all data points. This study provides an analysis of 150 data points along the jet centreline and 50 cross-jet values per line (three lines), resulting in a total number of 300 data points used in the analysis, which is considered to be sufficient for this particular case. The number of required data points depends on the flow pattern, flow complexity, and aim of the study.

The values of $\text{FAC}_{1.1}$ and $\text{FAC}_{1.5}$ show the fraction of considered data points, where the simulation results fall within a factor of 1.1 and 1.5 of the experimentally obtained values, respectively. $\text{FAC}_{1.1}$ and $\text{FAC}_{1.5}$ provide a clear picture of randomly occurring high or low differences between measured and predicted values of characteristic flow quantities.
(Schatzmann et al. 2010). The values of $R$ reflect the linear relationship between experimental and numerical results and provide an overall assessment of model performance. However, it can be influenced by a few either low or high outliers. To the best knowledge of the authors, the quantitative requirements regarding model validation are available only for prediction of pollution dispersion cases, e.g. Chang and Hanna (2004), Schatzmann et al. (2010), who used a threshold of $\text{FAC2} > 0.5$ in the model assessment for pollution dispersion predictions. In the present study, $\text{FAC1.1}$ and $\text{FAC1.5}$ are used in order to clearly illustrate the performance differences between different RANS models.

3.4.2 Mean velocity and turbulent kinetic energy distributions near the jet nozzle

Figure 7 shows normalised time-averaged velocity magnitude ($|V|/V_0$) and turbulent kinetic energy (TKE; $k/V_0^2$) profiles near the jet nozzle from the PIV measurements (Khayrullina et al. 2017) and CFD simulations of case I (Fig. 7a,b; $Re = 8,000$) and case II (Fig. 7c,d; $Re = 13,000$). The distributions are taken at a distance of $y = 2w_{\text{jet}}$ from the jet inlet$^1$. Note that the simulations are performed with symmetry boundary conditions, therefore the results are reflected in the jet centreline ($x/w_{\text{jet}} = 0$).

In the remainder of the paper, all results will consist of the time-averaged measurement data from PIV and the mean values from the steady RANS CFD simulations. A quantitative comparison is made in Table 2 using the validation metrics from Section 3.4.1.

Based on the velocity distributions provided in Figure 7a,c and the validation metrics provided in Table 2 the following observations are made:

- For both cases, the distributions of velocities predicted by the numerical simulations generally show a less clear top-hat shape profile compared to the measured profiles, except for the results from SST that most clearly represent the top-hat profile. The results of the validation studies by Berg et al. (2006) and Rhea et al. (2009), who investigated SKE and RSM models, respectively, are in line with this finding. The best agreement with the measured velocities in this initial region based on $R$ is achieved with SST ($R = 0.947$ and $0.951$ for cases I and II, respectively) and the worst with SKE ($R = 0.909$ and $0.923$ for cases I and II, respectively). When considering $\text{FAC1.1}$, the

$^1$ Note that the boundary conditions at the computational inlet are identical for all tested turbulence models. However, the predicted cross-jet profiles of flow quantities at the nozzle exit ($y = 0$) are different for all tested turbulence models.
**Table 2: Validation metrics defined for cross-jet velocity distributions near the jet inlet at y = 2wjet**

<table>
<thead>
<tr>
<th>Model</th>
<th>Case I</th>
<th>Case II</th>
<th>Case I</th>
<th>Case II</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>R FAC1.1</td>
<td>R FAC1.1</td>
<td>R FAC1.5</td>
<td>R FAC1.5</td>
</tr>
<tr>
<td>SKE</td>
<td>0.909 0.32</td>
<td>0.923 0.40</td>
<td>0.634 0.08</td>
<td>0.710 0.16</td>
</tr>
<tr>
<td>RKE</td>
<td>0.918 0.40</td>
<td>0.929 0.44</td>
<td>0.659 0.08</td>
<td>0.724 0.32</td>
</tr>
<tr>
<td>RNG</td>
<td>0.919 0.40</td>
<td>0.930 0.44</td>
<td>0.636 0.08</td>
<td>0.701 0.28</td>
</tr>
<tr>
<td>SST</td>
<td>0.947 0.36</td>
<td>0.951 0.36</td>
<td>0.741 0.40</td>
<td>0.800 0.44</td>
</tr>
<tr>
<td>RSM</td>
<td>0.919 0.32</td>
<td>0.929 0.32</td>
<td>0.662 0.08</td>
<td>0.720 0.20</td>
</tr>
</tbody>
</table>

Perfect performance

---

**Figure 7:** Comparison of PIV results with results of steady RANS CFD simulations. Cross-jet profiles of: a,c) normalised jet velocity magnitude (|V|/V₀); b,d) normalised turbulent kinetic energy (k/V₀²).

a,b) Case I c,d) Case II. x/w_{jet} = 0 corresponds to the jet centreline.
best agreement is obtained with RKE and RNG; \( \text{FAC}1.1 = 0.40 \) and \( 0.44 \) for case I and case II, respectively, while the worst agreement is for case I is obtained with SKE and RSM, and for case II with RSM.

- For both cases the velocities in the outer jet region \( (0.5 < |x|/w_{\text{jet}} < 1) \) are overestimated by all the turbulence models.
- The ambient fluid starts to entrain the jet potential core at a distance \( |x|/w_{\text{jet}} \) from the jet centreline, where \( |V|/V_0 = 0.98 \) (Deo et al., 2007a). In case I, \( |x|/w_{\text{jet}} = 0.15 \) for SKE, \( |x|/w_{\text{jet}} = 0.20 \) for RKE and RNG, \( |x|/w_{\text{jet}} = 0.30 \) for SST and \( |x|/w_{\text{jet}} = 0.18 \) for RSM. Note that in Figure 7a the line for SKE overlaps with that of RKE and RSM for \( |x|/w_{\text{jet}} < 0.5 \). In case II, \( |x|/w_{\text{jet}} = 0.23 \) for RKE and RNG, \( |x|/w_{\text{jet}} = 0.20 \) for SKE, \( |x|/w_{\text{jet}} = 0.30 \) for SST and \( |x|/w_{\text{jet}} = 0.23 \) for RSM. The value obtained from PIV is equal to \( |x|/w_{\text{jet}} = 0.33 \) and \( 0.32 \) for cases I and II, respectively.

Figure 7b,d shows three profiles of \( k/V_0^2 \) (PIV-1, PIV-2, PIV-3) calculated using:

\[
k/V_0^2 = 0.5(u_{\text{RMS}}^2 + v_{\text{RMS}}^2 + w_{\text{RMS}}^2)/V_0^2
\]  

(5)

with \( u_{\text{RMS}}, v_{\text{RMS}}, w_{\text{RMS}} \) the lateral, streamwise and spanwise RMS velocities, respectively. Due to the absence of the spanwise component \( (w_{\text{RMS}}^2) \) in the 2D PIV experiments by Khayrullina et al. (2017), its value needs to be estimated, which is done using the assumptions described in Table 3. The profile of PIV-1 uses the relation between the centreline profiles of \( u_{\text{RMS}}^2 \) and \( w_{\text{RMS}}^2 \) provided by Gutmark et al. (1978); \( u_{\text{RMS}}^2 = w_{\text{RMS}}^2 \) on the jet centreline for \( y/h_{\text{jet}} \leq 0.85 \). For \( y/h_{\text{jet}} > 0.85 \) the values of \( w_{\text{RMS}}^2 \) become slightly higher than \( u_{\text{RMS}}^2 \) (factor \( a \) is larger than 1), which can be explained by vortex stretching along the impingement plate, which enhances spanwise \( (w_{\text{RMS}}) \) and streamwise \( (v_{\text{RMS}}) \) velocity fluctuations. For PIV-2, the spanwise component is equal to zero; for PIV-3 \( w_{\text{RMS}}^2 \) is assumed to be equal to the average of the lateral and streamwise RMS velocities. Figure 7b,d shows that:

- The CFD profiles of \( k/V_0^2 \) near the jet inlet correspond best to the profile with the assumption that \( w_{\text{RMS}}^2 \) is zero (PIV-2). Therefore, only the results of PIV-2 are provided in the remainder of the paper.
- SST provides the closest agreement \( (R = 0.741 \) and 0.8, and \( \text{FAC}1.5 = 0.40 \) and \( 0.44 \) for cases I and II, respectively), especially within the jet potential core \( |x|/w_{\text{jet}} < 0.3 \).
Steady RANS CFD simulations

SKE, RKE, RNG and RSM underestimate $k/V_0^2$ in the shear layer ($0.3 < |x|/w_{jet} < 0.75$) and overestimate $k/V_0^2$ close to the jet centreline ($x/w_{jet} = 0$).

- The values of $k/V_0^2$ near the jet nozzle in the shear layer are underestimated by CFD simulation compared to that of experiments for both cases. For example, for case I (Fig. 7b) SST provides a maximum value of $k/V_0^2 = 0.028$, while experimental data (PIV-2) shows a maximum of $k/V_0^2 = 0.068$. Similarly, Rhea et al. (2009) and Achari and Das (2015) observed underestimations of turbulent kinetic energy by $k$-$\varepsilon$ models in the shear layer, especially close to the inlet. Achari and Das (2015) demonstrated that the standard $k$-$\omega$ model provided better predictions of turbulent kinetic energy near the inlet; however, this turbulence model is not tested in the present study.

- In both cases SKE shows the largest overestimation of $k/V_0^2$ within the jet potential core. For example, for case I (Fig. 7b), SKE provides a maximum value of $k/V_0^2 = 0.012$, while the experimental data show that $k/V_0^2$ approaches zero ($k/V_0^2 \approx 0.0005$) near the jet centreline ($x/w_{jet} = 0$).

**Table 3: Spanwise component $w_{RMS}^2$ for TKE**

<table>
<thead>
<tr>
<th>Profile name</th>
<th>Spanwise component $w_{RMS}^2$</th>
</tr>
</thead>
</table>
| PIV-1        | $y/h_{jet} > 0.85$: $w_{RMS}^2 = a(y) \cdot u_{RMS}^2$  
               | $y/h_{jet} \leq 0.85$: $w_{RMS}^2 = u_{RMS}^2$  
               | $a(y)$ – variable defined from Gutmark et al. 1978 |
| PIV-2        | $w_{RMS}^2 = 0$ |
| PIV-3        | $w_{RMS}^2 = 0.5(u_{RMS}^2 + v_{RMS}^2)$ |

3.4.3 Centreline mean velocity and turbulent kinetic energy distributions

Figure 8 shows $V/V_0$ and $k/V_0^2$ along the jet centreline obtained with different turbulence models.

For case I (Fig. 8a,b) the following observations are made:

- RNG and RKE accurately predict $V/V_0$ along the jet centreline. SKE underestimates $V/V_0$ in the potential core and the beginning of the intermediate jet region ($0 < y/h_{jet} < 0.5$), and accurately predicts $V/V_0$ for $y/h_{jet} > 0.5$. RSM is consistent with the experimental data within $0 < y/h_{jet} < 0.4$, while it overestimates $V/V_0$ further downstream ($y/h_{jet} > 0.4$). SST overestimates $V/V_0$ for $y/h_{jet} > 0.2$. 

- The values of $k/V_0^2$ near the jet nozzle in the shear layer are underestimated by CFD simulation compared to that of experiments for both cases. For example, for case I (Fig. 7b) SST provides a maximum value of $k/V_0^2 = 0.028$, while experimental data (PIV-2) shows a maximum of $k/V_0^2 = 0.068$. Similarly, Rhea et al. (2009) and Achari and Das (2015) observed underestimations of turbulent kinetic energy by $k$-$\varepsilon$ models in the shear layer, especially close to the inlet. Achari and Das (2015) demonstrated that the standard $k$-$\omega$ model provided better predictions of turbulent kinetic energy near the inlet; however, this turbulence model is not tested in the present study.

- In both cases SKE shows the largest overestimation of $k/V_0^2$ within the jet potential core. For example, for case I (Fig. 7b), SKE provides a maximum value of $k/V_0^2 = 0.012$, while the experimental data show that $k/V_0^2$ approaches zero ($k/V_0^2 \approx 0.0005$) near the jet centreline ($x/w_{jet} = 0$).
- Values of $k/V_0^2$ within the jet potential core ($0 < y/h_{jet} < 0.2$) are overestimated by SKE, RNG and RKE compared to the experimental values.
- At the beginning of the intermediate jet region ($0.2 < y/h_{jet} < 0.4$), profiles of $k/V_0^2$ by RKE and RSM correspond well to the experimental data.
- Within $0.60 < y/h_{jet} < 1$ all turbulence models overestimate values of $k/V_0^2$. In contrast, Rhea et al. (2009) revealed some underpredictions, particularly by RSM, of turbulence quantities in the free and wall jet regions.
- The maximum experimentally obtained value of $k/V_0^2$ at the centreline within the intermediate jet region occurs at $y/h_{jet} \approx 0.53$ and reaches $k/V_0^2 \approx 0.023$. The location of this peak is accurately predicted by RKE and RSM ($y/h_{jet} \approx 0.52$ and 0.53, respectively), and their predicted values ($k/V_0^2 = 0.019$ and $0.018$, respectively) provide the closest agreement with the experimental data.
- The maximum experimentally obtained TKE in the jet impingement region occurs at $y/h_{jet} \approx 0.98$ and is equal to $k/V_0^2 \approx 0.020$. SST estimates the maximum TKE near the impingement region to be $k/V_0^2 \approx 0.024$ at $y/h_{jet} \approx 0.98$, which is close to the experimental data. The other models overestimate the maximum value of TKE in the impingement region.

A slightly different performance of the turbulence models is observed for case II (Fig. 8c,d):

- RNG and SKE predict mean velocities that are consistent with the experimental data, while RKE, SST and RSM overestimate $V/V_0$ in the intermediate jet region ($0.3 < y/h_{jet} < 0.9$).
- The maximum measured value of $k/V_0^2$ at the centreline in the intermediate region reaches $k/V_0^2 \approx 0.021$ at $y/h_{jet} \approx 0.55$. The location and magnitude of this peak is quite accurately predicted by RNG, showing $k/V_0^2 \approx 0.021$ at $y/h_{jet} \approx 0.53$.
- The maximum measured value of $k/V_0^2$ in the jet impingement region occurs at $y/h_{jet} \approx 0.99$ and is equal to $k/V_0^2 \approx 0.022$. The maximum value of $k/V_0^2$ near the impingement region by SST is $k/V_0^2 \approx 0.024$ at $y/h_{jet} \approx 0.98$, which corresponds well to the value from PIV. The other models again overpredict $k/V_0^2$ in the impingement region. Heyerichs and Pollard (1996) and Chen at al. (1999) found that the standard $k-\omega$ model could adequately predict turbulence levels in the stagnation zone. Moreover, Angioletti et al. (2005) and Dutta et al. (2013) stated that SST provided more reliable predictions of turbulent kinetic energy in the stagnation region in their particular case.
Figure 8: Comparison of PIV results with results of steady RANS CFD simulations obtained along the jet centreline ($x/w_{jet} = 0$): a,c) normalised centreline velocity ($V/V_0$); b,d) normalised turbulent kinetic energy ($k/V_0^2$). a,b) Case I. c,d) Case II. $y/w_{jet} = 0$ corresponds to the jet nozzle.

Table 4 shows the validation metrics for velocities and TKE along the jet centreline, from which it can be concluded that RKE and RNG show the best performance for predicting centreline velocities for case I, while SKE and RNG show the best performance for case II. RNG and RKE show the best agreement of centreline distributions of $k/V_0^2$ for case I and case II.
Table 4: Validation metrics for centreline distributions of velocity and turbulent kinetic energy

<table>
<thead>
<tr>
<th>Model</th>
<th>Centreline velocity</th>
<th>Turbulent kinetic energy</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Case I</td>
<td>Case II</td>
</tr>
<tr>
<td></td>
<td>R</td>
<td>FAC1.1</td>
</tr>
<tr>
<td>SKE</td>
<td>0.98</td>
<td>0.93</td>
</tr>
<tr>
<td>RKE</td>
<td>0.98</td>
<td>0.96</td>
</tr>
<tr>
<td>RNG</td>
<td>0.98</td>
<td>0.91</td>
</tr>
<tr>
<td>SST</td>
<td>0.96</td>
<td>0.37</td>
</tr>
<tr>
<td>RSM</td>
<td>0.97</td>
<td>0.72</td>
</tr>
<tr>
<td>Perfect performance</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

The measured and predicted normalised length of the jet potential core ($h_p/w_{jet}$) for both cases are provided in Table 5. For case I the best agreement with experiments is provided by RNG and RSM, resulting in an underestimation by 8% and 10%, respectively. For case II, RKE underestimates the jet potential core length by 8%. For both cases, SKE fails to predict the length of the potential core due to excessive levels of TKE along the jet centreline and in the shear layer near the jet nozzle, as shown in Figure 7b,d and Figure 8b,d. The opposite behaviour is shown by SST, which underestimates values of TKE on the jet centreline resulting in a substantially longer potential core (+80% and +73% for case I and case II, respectively). This finding is in line with the results from the study of Achari and Das (2015). In contrast, Achari and Das (2015) and Moureh and Yataghene (2016) revealed an overestimation of jet potential core by SKE model, while the present study indicates an underestimation of the potential core length (-43% and -24% for case I and case II, respectively).
Table 5: Normalised length of the jet potential core $h_p/w_{jet}$

<table>
<thead>
<tr>
<th>Model</th>
<th>Case I</th>
<th>Case II</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$h_p/w_{jet}$</td>
<td>$h_p/w_{jet}$</td>
</tr>
<tr>
<td>PIV</td>
<td>4.85</td>
<td>4.85</td>
</tr>
<tr>
<td>SKE</td>
<td>2.76</td>
<td>3.67</td>
</tr>
<tr>
<td>RKE</td>
<td>3.80</td>
<td>4.48</td>
</tr>
<tr>
<td>RNG</td>
<td>4.48</td>
<td>5.33</td>
</tr>
<tr>
<td>SST</td>
<td>8.74</td>
<td>8.41</td>
</tr>
<tr>
<td>RSM</td>
<td>4.34</td>
<td>5.91</td>
</tr>
</tbody>
</table>

3.4.4 Cross-jet mean velocity and turbulent kinetic energy distributions

3.4.4.1 Case I

The cross-jet distributions of $V/V_0$ and $k/V_0^2$ for case I are compared at three different distances from the jet nozzle: $y/w_{jet} = 4$, $y/w_{jet} = 11.3$ and $y/w_{jet} = 19.3$. The validation metrics averaged over these three distances are provided in Table 6 that indicates that the results of RKE show overall the closest agreement with the experimental data ($R = 0.98$ and $FAC1.1 = 0.62$ for velocities; $R = 0.95$ and $FAC1.5 = 0.75$ for TKE).

Table 6: Validation metrics for cross-jet distributions of velocity and TKE for case I

<table>
<thead>
<tr>
<th>Model</th>
<th>$V/V_0$</th>
<th>$k/V_0^2$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$R$</td>
<td>$FAC1.1$</td>
</tr>
<tr>
<td>SKE</td>
<td>0.97</td>
<td>0.56</td>
</tr>
<tr>
<td>RKE</td>
<td>0.98</td>
<td>0.62</td>
</tr>
<tr>
<td>RNG</td>
<td>0.97</td>
<td>0.56</td>
</tr>
<tr>
<td>SST</td>
<td>0.98</td>
<td>0.62</td>
</tr>
<tr>
<td>RSM</td>
<td>0.98</td>
<td>0.51</td>
</tr>
<tr>
<td>Perfect performance</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>
Figure 9a,b shows the cross-jet velocity and TKE profiles at $y/w_{jet} = 4$. The velocity profile of SST, which provides a close correspondence to experimental data ($R = 0.98$ and FAC1.1 = 0.62), shows less spreading of the jet than the other models (Fig. 9a). This is due to the lower predicted levels of TKE by SST near the jet centreline and in the shear layer, and therefore less intensive entrainment of ambient fluid compared to the other models. The velocity profiles of RKE, RNG and RSM are nearly identical. The jet potential core predicted by SST runs until $|x|/w_{jet} = 0.2$. RKE, RNG and RSM provide a fairly accurate prediction of TKE within $|x|/w_{jet} < 0.3$ (Fig. 9b), with a total average deviation of 38%, 22% and 37%, respectively. Figure 9b also shows that SKE predicts the highest turbulence levels among all the models within $|x|/w_{jet} < 1$. The maximum values of TKE in the shear layer reach $k/V_0^2 \approx 0.040$ for SKE, which corresponds well to the experimental value ($k/V_0^2 \approx 0.044$). Figure 9c,d indicates that at $y/w_{jet} = 11.3$ the cross-jet velocity profiles of RKE nearly overlap with the measurement data for $|x|/w_{jet} < 1$. Figure 9d shows that SKE provides the best correspondence to the measured TKE profile for $|x|/w_{jet} < 1$, while RKE provides the best agreement for $|x|/w_{jet} > 1$. The highest values in jet shear layer ($k/V_0^2 \approx 0.032$) are predicted by SST. Finally, Figure 9e,f shows the profiles at $y/w_{jet} = 19.3$, which is at the end of the intermediate jet region and at the beginning of the impingement region. The cross-jet velocity profiles show a good overall correspondence to the PIV measurements. The best agreement appears to be present for SKE and RNG, followed by RKE (Fig. 9e). All models overestimate the values of TKE, with a maximum value of $k/V_0^2 \approx 0.021$ (SST), while the measured maximum value is $k/V_0^2 \approx 0.016$ (Fig. 9f).

3.4.4.2 Case II

The cross-jet distributions of $V/V_0$ and $k/V_0^2$ for case II are compared in this section. The results in Table 7 show that SST provides the best agreement with experimental data for velocity distributions ($R = 0.97$ and FAC1.1 = 0.45), while RKE shows the best agreement with experimental data for TKE ($R = 0.94$ and FAC1.5 = 0.68).
Figure 9: Cross-jet profiles for case I of normalised streamwise velocity $V/V_0$ a,c,e) and turbulent kinetic energy $k/V_0^2$ b,d,f) at: a,b) $y/w_{jet} = 4$; c,d) $y/w_{jet} = 11.3$; e,f) $y/w_{jet} = 19.3$.
Table 7: Validation metrics for cross-jet distributions of velocity and TKE for case II

<table>
<thead>
<tr>
<th>Model</th>
<th>( \frac{V}{V_0} )</th>
<th>( \frac{k}{V_0^2} )</th>
<th>( \frac{V}{V_0} )</th>
<th>( \frac{k}{V_0^2} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>SKE</td>
<td>0.97 0.42</td>
<td>0.92 0.64</td>
<td>0.97 0.43</td>
<td>0.94 0.68</td>
</tr>
<tr>
<td>RKE</td>
<td>0.97 0.43</td>
<td>0.94 0.68</td>
<td>0.97 0.42</td>
<td>0.91 0.71</td>
</tr>
<tr>
<td>RNG</td>
<td>0.97 0.42</td>
<td>0.91 0.71</td>
<td>0.97 0.45</td>
<td>0.93 0.67</td>
</tr>
<tr>
<td>SST</td>
<td>0.97 0.45</td>
<td>0.93 0.67</td>
<td>0.96 0.40</td>
<td>0.92 0.66</td>
</tr>
<tr>
<td>RSM</td>
<td>0.96 0.40</td>
<td>0.92 0.66</td>
<td>0.96 0.40</td>
<td>0.92 0.66</td>
</tr>
<tr>
<td>Perfect performance</td>
<td>1 1</td>
<td>1 1</td>
<td>1 1</td>
<td>1 1</td>
</tr>
</tbody>
</table>

Figure 10a,b shows the cross-jet profiles at \( y/w_{jet} = 4 \). Again, SST provides the best agreement with PIV experiments for the cross-jet velocity distribution close to the inlet. All models overestimate the values of \( \frac{k}{V_0^2} \), with maximum differences of 21% (SST) to 28% (SKE). The maximum values of TKE in the shear layer reach \( \frac{k}{V_0^2} \approx 0.036 \) for SKE, which predicts the highest turbulence level among all the models and deviates most from the experimental data (maximum \( \frac{k}{V_0^2} \approx 0.028 \)). Figure 10c,d depicts the cross-jet profiles of \( \frac{V}{V_0^2} \) and \( \frac{k}{V_0^2} \) at \( y/w_{jet} = 11.3 \). The numerically obtained cross-jet profiles of velocity do not show very large differences. As for TKE, SST again provides the highest values in the shear layer (maximum \( \frac{k}{V_0^2} \approx 0.030 \)) and overestimates the experimentally obtained values (maximum \( \frac{k}{V_0^2} \approx 0.025 \)) by 20% in this region.

Finally, cross-jet profiles at \( y/w_{jet} = 19.3 \) are shown in Figure 10e,f. SKE and RNG appear to provide the best agreement with the measured velocity profile (Fig. 10e). SST and RNG provide the highest values for TKE (maximum \( \frac{k}{V_0^2} \approx 0.02 \)) and overestimate the experimentally obtained values (maximum \( \frac{k}{V_0^2} \approx 0.018 \)) by 11%, while SKE, RKE and RSM show a very good agreement (within 3%) (Fig. 10f).
Figure 10: Cross-jet profiles for case II of normalised streamwise velocity $V/V_0$ a,c,e) and turbulent kinetic energy $k/V_0^2$ b,d,f) at: a,b) $y/w_{jet} = 4$; c,d) $y/w_{jet} = 11.3$; e,f) $y/w_{jet} = 19.3$
3.4.5 Overall performance

As shown in previous sections, no single model shows the best performance in predicting both mean velocities and TKE at different distances from the jet inlet. The overall performance (along jet centreline and three cross-jet profiles; in total 300 data points) is provided in Table 8. The values provided are calculated as mean values of $R$, $FAC1.1$ and $FAC1.5$ as provided in Tables 2, 4, 6 and 7. The values for case I and II are combined in these mean values, however, separate values for mean velocities and TKE are calculated, resulting in four values per model ($R$ and $FAC1.1$ for velocity and $R$ and $FAC1.5$ for TKE).

The results show that SST provides the best agreement with experimental data for mean velocity distributions when considering the $R$ value ($R = 0.966$), although differences in $R$ values with the other models are small. However, SST predicts only 46.6% of the considered mean velocities within a factor of 1.1 ($FAC1.1 = 0.468$). Hence, based on both $R$ and $FAC1.1$, RKE provides the best agreement with experimental data for mean velocities ($R = 0.964$ and $FAC1.1 = 0.567$). RKE and RNG provide the best agreement with the measured TKE ($R = 0.883$ and $FAC1.1 = 0.554$ for RKE, $R = 0.873$ and $FAC1.1 = 0.612$ for RNG). Note that the differences in performance between the models are small.

Table 8: Overall validation metrics for distributions of velocity and TKE (cases I and II combined)

<table>
<thead>
<tr>
<th>Model</th>
<th>$R$</th>
<th>$FAC1.1$</th>
<th>$R$</th>
<th>$FAC1.5$</th>
</tr>
</thead>
<tbody>
<tr>
<td>SKE</td>
<td>0.961</td>
<td>0.553</td>
<td>0.833</td>
<td>0.502</td>
</tr>
<tr>
<td>RKE</td>
<td>0.964</td>
<td>0.567</td>
<td>0.883</td>
<td>0.554</td>
</tr>
<tr>
<td>RNG</td>
<td>0.963</td>
<td>0.564</td>
<td>0.873</td>
<td>0.612</td>
</tr>
<tr>
<td>SST</td>
<td>0.966</td>
<td>0.468</td>
<td>0.863</td>
<td>0.545</td>
</tr>
<tr>
<td>RSM</td>
<td>0.961</td>
<td>0.464</td>
<td>0.851</td>
<td>0.486</td>
</tr>
<tr>
<td>Perfect performance</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>
3.4.6 Spatial distribution of mean velocity and vector plots

Figure 11 and Figure 12 show spatial distribution of $|V/V_0|$ and the mean velocity vectors in the vertical centreplane for case I and case II, respectively. The white areas at the top and bottom parts of Figure 11a and Figure 12a indicate zones where erroneous measurement data due to obstructions and reflections are omitted. The development of the jet half-width is indicated by dashed white lines. The slope of these lines is used to characterise the jet spreading rate $K_y = d\delta_{0.5}/dy$ (Pope 2000). The values of $K_y$ are defined separately for the potential core and the intermediate jet regions and are listed in Table 9. Figure 10e clearly shows less intensive jet spreading for SST compared to the other models. However, the jet spreading rate within the intermediate region for SST differs only by 2% and 7% from the value from the PIV experiments for cases I and II, respectively (Table 9). However, note that the spreading rate in the intermediate region can be similar while the local width of the jet at the beginning of the intermediate region is different due to different jet spreading rates in the potential core region. The substantial difference between the values of jet spreading rate for the potential core region and the intermediate jet region for SST (Table 9) is caused by a steep growth of TKE at $y/h_{jet} > 0.35$, which is depicted in Figure 8b,d, due to the entrainment of ambient fluid into the jet. The experimentally obtained values for the potential core region are $K_y = 0.05$ and $K_y = 0.04$, for case I and case II, respectively. SKE provides the closest agreement for jet spreading rate within the jet potential core for both cases ($K_y = 0.04$ and 0.03, corresponding to a 22% and 20% difference, for case I and II, respectively). Similarly, Aziz et al. (2008) showed that SKE outperformed RNG in predicting jet spreading rate. The spreading rate within the jet potential core for case II predicted by the other models differs with 33% to 58% from the experimental data. All the models provide a good agreement with the experiments within the intermediate jet region for both cases ($K_y = 0.10$). Largest differences are present for RSM and RNG for case I, with predictions that deviate by 16% and 21% (Table 9).

Experimental studies for free plane turbulent jets with sidewalls in the range of $Re = 6,000$–$16,000$ (e.g. Jenkins and Goldschmidt 1973, Browne et al. 1983, Thomas and Chu 1989, Sakai et al. 2006, Deo et al. 2007a,b, Suresh et al. 2008) reported values of $K_y$ in the range from 0.05 to 0.11 for the potential core and intermediate jet regions. The jet spreading rates in the potential core region as calculated by the steady RANS CFD simulations are lower than those from the PIV measurements (case I: 0.05, case II: 0.04), and thus also lower than the values reported in other experimental studies. Achari and Das (2015) attributed the underestimation of jet spreading rate as predicted by $k-\varepsilon$
models to a lower entrainment from surrounding stagnant fluid towards the jet centreline compared to what was found in the experiments.

Table 9: Spreading rates $K_y$ for the potential core region and intermediate region

<table>
<thead>
<tr>
<th></th>
<th>Case I</th>
<th></th>
<th></th>
<th>Case II</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Potential core region</td>
<td>% difference</td>
<td>Intermediate region</td>
<td>% difference</td>
<td>Potential core region</td>
<td>% difference</td>
</tr>
<tr>
<td>PIV</td>
<td>0.05</td>
<td>0.10</td>
<td>0.04</td>
<td>0.10</td>
<td>0.05</td>
<td>0.10</td>
</tr>
<tr>
<td>SKE</td>
<td>0.04</td>
<td>-22</td>
<td>0.11</td>
<td>+9</td>
<td>0.03</td>
<td>-20</td>
</tr>
<tr>
<td>RKE</td>
<td>0.03</td>
<td>-46</td>
<td>0.09</td>
<td>-8</td>
<td>0.02</td>
<td>-40</td>
</tr>
<tr>
<td>RNG</td>
<td>0.03</td>
<td>-48</td>
<td>0.12</td>
<td>+16</td>
<td>0.02</td>
<td>-43</td>
</tr>
<tr>
<td>SST</td>
<td>0.01</td>
<td>-72</td>
<td>0.10</td>
<td>-2</td>
<td>0.02</td>
<td>-58</td>
</tr>
<tr>
<td>RSM</td>
<td>0.03</td>
<td>-40</td>
<td>0.08</td>
<td>-21</td>
<td>0.03</td>
<td>-33</td>
</tr>
</tbody>
</table>

Table 10 shows the values of the jet decay rate $K_u$ defined as $K_u = (V_0/V)^2/((y - y_0)/w_{jet})$, with $y_0$ the vertical distance between jet nozzle and the kinematic virtual jet origin. The experimentally obtained values are $K_u = 0.15$ and $K_u = 0.14$, for case I and case II, respectively. For case I, SKE and RNG accurately predict the jet decay rate within the intermediate jet region ($K_u = 0.15$), and for case II SKE provides an accurate prediction as well ($K_u = 0.14$). This observation is in line with Aziz et al. (2008), who found that SKE predicted jet decay rate more accurately than RNG. For both cases the values of jet decay rate are significantly overestimated by SST ($K_u = 0.18$), and underestimated by RSM ($K_u = 0.12$ and $K_u = 0.10$ for case I and case II, respectively). The overestimation of $K_u$ by SST can also be observed in Figure 8, in which large velocity gradients in the streamwise direction are present for $y/h_{jet} > 0.4$. In previous experimental studies of free plane turbulent jets with sidewalls, with $Re = 6,000-16,000$ (e.g. Jenkins and Goldschmidt 1973, Browne et al. 1983, Thomas and Chu 1989, Sakai et al. 2006, Deo et al. 2007a, Suresh et al. 2008, Alnahhal and Panidis 2009), a range of values of $K_u$ from 0.11 to 0.22 was reported, which encompasses both the experimental and numerical values obtained in the present study.
Figure 11: Spatial distributions of dimensionless velocity magnitude ($|V|/V_0$) for case I ($Re = 8,000$):
a) PIV measurements; b-f) steady RANS CFD simulations: b) SKE; c) RKE; d) RNG; e) SST; f) RSM
Figure 12: Spatial distributions of dimensionless velocity magnitude ($|V|/V_0$) for case II ($Re = 13,000$): a) PIV measurements; b-f) steady RANS CFD simulations: b) SKE; c) RKE; d) RNG; e) SST; f) RSM.
Table 10: The jet decay rate $K_u$ for intermediate jet region

<table>
<thead>
<tr>
<th></th>
<th>Case I</th>
<th></th>
<th>Case II</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Intermediate</td>
<td>% difference</td>
<td>Intermediate</td>
<td>% difference</td>
</tr>
<tr>
<td>PIV</td>
<td>0.15</td>
<td></td>
<td>0.14</td>
<td></td>
</tr>
<tr>
<td>SKE</td>
<td>0.15</td>
<td>0</td>
<td>0.14</td>
<td>-2</td>
</tr>
<tr>
<td>RKE</td>
<td>0.18</td>
<td>+19</td>
<td>0.13</td>
<td>-9</td>
</tr>
<tr>
<td>RNG</td>
<td>0.15</td>
<td>+1</td>
<td>0.17</td>
<td>+20</td>
</tr>
<tr>
<td>SST</td>
<td>0.18</td>
<td>+23</td>
<td>0.18</td>
<td>+30</td>
</tr>
<tr>
<td>RSM</td>
<td>0.12</td>
<td>-21</td>
<td>0.10</td>
<td>-31</td>
</tr>
</tbody>
</table>

3.4.7 Production of turbulent kinetic energy

In order to support the discussion on differences between results provided by the different turbulence models, cross-jet profiles of the production of turbulent kinetic energy per unit volume ($G_k = \rho P_k$, with $\rho$ the density and $P_k$ the production of turbulent kinetic energy per unit mass) are depicted at two distances from the jet nozzle; i.e. at $y/w_{jet} = 0.1$ and $y/w_{jet} = 2$, for cases I and II in Figure 13. At $y/w_{jet} = 0.1$, there is a clear difference in predicted values of $G_k$ in the shear layer of the jet; the highest levels are predicted by SKE, RKE and RNG and the lowest levels are present for SST. The maximum values of $G_k$ for case I are located in the shear layer and range from 250 kg/ms$^3$ for SST to 2,510 kg/ms$^3$ for SKE (Fig. 13a) at $y/w_{jet} = 0.1$, while they are reduced to about 210 kg/ms$^3$ for SST and around 190 kg/ms$^3$ for the other models at $y/w_{jet} = 2$ (Fig. 13b). For case II, the maximum values in the shear layer range from 1,160 kg/ms$^3$ for SST to 10,340 kg/ms$^3$ for SKE (Fig. 13c). At $y/w_{jet} = 2$, the values of $G_k$ for all models decreased to around 800 kg/ms$^3$ in the shear layer with the highest values for SST (980 kg/ms$^3$) (Fig. 13d).

The excessive production of TKE in the initial shear layer by SKE explains the highest entrainment and largest jet spreading rate within the jet potential core, resulting in a shorter potential core region (see Table 4). On the contrary, near the jet nozzle, the lowest levels of $G_k$ are present for SST, resulting in a lower spreading rate within the jet potential core region and an extended potential core length (see Table 4). The production term can be regarded as an important factor in the prediction of the mean velocities and turbulence levels in impinging jets by steady RANS, as studied in this paper.
Figure 13: Production of turbulent kinetic energy $G_k$ at: a,c) $y/w_{jet} = 0.1$; b,d) $y/w_{jet} = 2$. a,b) Case I c,d) Case II

3.5 Discussion and future work

This paper presented the results of a validation study of a PTIJ at $Re = 8,000$ and $Re = 13,000$. The results obtained from steady RANS CFD simulations with the SKE, RKE, RNG, SST and RSM turbulence models were compared with experimental data of Khayrullina et al. (2017).

3.5.1 Differences in turbulence model performance

The substantial differences in turbulence model performance as described in Section 3.4 are – among others – caused by different formulations for the eddy viscosity $\mu_t$. The $k-\varepsilon$ models utilise the equation relating $\mu_t$ to turbulent kinetic energy $k$ and its dissipation rate $\varepsilon$, and applying a model constant or variable $C_\mu$ representing an empirical ratio of
Reynolds shear stresses to turbulent kinetic energy (e.g. Durbin and Pettersson Reif 2001). While SKE and RNG use a model constant $C_{\mu}$, RKE uses a model variable $C_{\mu}$, which depends on the deformation of mean flow (strain) and turbulent kinetic energy and its dissipation in the flow (terms $k$ and $\varepsilon$). The SST model employed uses a modified eddy-viscosity formulation to account for transport effects of the turbulent shear stress near walls by applying a limiter to the eddy-viscosity formulation. In addition, a turbulence damping term is introduced in the $\omega$ equation. Please note that SST also utilises a limiter applied to the production of turbulent kinetic energy, as mentioned in Section 3.3.3.

Furthermore, the production term $G_k$ is treated differently in the transport equations of different turbulence models. The values of $G_k$ near the jet nozzle are presented in Section 3.4.7. SKE, RKE and RNG utilise a simplified production term based on the Boussinesq relationship, which relates the Reynolds (turbulent shear) stresses to the mean strain rate using the turbulent viscosity, resulting in the inability to accurately predict an anisotropic distribution of Reynolds stresses due to the assumption that the turbulence viscosity is an isotropic scalar quantity, which is not always the case. The Boussinesq relationship is known for overestimation of $k$ in the impingement jet region (e.g. Craft et al. 1993, Murakami et al. 1992, Durbin and Pettersson Reif 2001, Wright and Easom 2003). In the current study, too high levels of turbulence are exhibited in the impingement region by the $k$-$\varepsilon$ models, as expected based on literature. The lower values of turbulent kinetic energy in the impingement region obtained with SST, compared to the results of other eddy-viscosity models, can be explained by the limiting term applied in the equation for the production of turbulent kinetic energy. As for RSM, the linear pressure-strain model used in this study is known for the overestimation of $k$ in the impingement region due to the erroneous redistribution of normal stresses in the impingement regions by the wall-reflection term applied to the pressure-strain equation (e.g. Craft et al. 1993, Murakami et al. 1992).

3.5.2 Applicability of considered RANS models in application studies

As mentioned before, RANS models are commonly used in engineering applications due to their good balance between computational demand and accuracy. The current paper showed that RKE and RNG models accurately predicted the general jet flow patterns and characteristics. Therefore, it is expected that moderate errors from these predictions will not have a significant impact on application case results. However, if the focus of engineering studies lies in the impingement region (e.g. heat or mass transfer
particularly in this region) some discrepancies near the impingement plate are expected. Furthermore, realistic cases involve a large number of external processes influencing the jet flow, e.g. pressure gradient across the jet due to, for example, wind and temperature differences between two sides of the jet. Here, turbulence models should be able to accurately predict these physical processes as well as the jet flow.

3.5.3 Future work

The current study showed that RANS was not able to accurately predict turbulence levels in the shear layer, especially near the inlet region\(^2\) and in the impingement region. The latter is known to be a complex flow region, where the effect of anisotropy of turbulent fluctuations and streamline curvature are pronounced (e.g. Moureh and Yataghene 2016). The accuracy of numerical predictions in these regions will benefit from LES (e.g. Kubacki et al. 2013). Moreover, temporal flow features, including jet flapping, development of vortical structures in the shear layers, etc. can be solved by LES (e.g. Le Song and Prud'homme 2007, van Hooff et al. 2017). However, LES will require high-resolution grids throughout the flow domain, which would increase the computational demand substantially; a hybrid unsteady RANS-LES (URANS-LES) approach might therefore be a good compromise between computational demand and increased prediction accuracy. Hybrid URANS-LES can couple a URANS model in the near-wall region with LES for the outer flow, reducing the need for a very fine near-wall grid to resolve the small-scale eddies near the wall when a full LES would be performed (e.g. Fröhlich and von Terzi 2008). Therefore, future work will include an elaboration of hybrid URANS-LES models.

In addition, the observed oscillatory convergence of the residuals and resolved parameters near the impingement plate could suggest the application of URANS, LES, and hybrid URANS-LES models. Furthermore, the present study can be extended by analysing the development of the wall jet near the impingement plate. Finally, the application of non-linear eddy-viscosity models and, for example, the \(v^2\)-f model, which utilises a dependency of turbulent viscosity on the velocity scale \(v^2\) and overcomes the problem of inaccurate prediction of near-wall turbulence anisotropy by the linear eddy-viscosity models (i.e. \(k-\varepsilon\) and \(k-\omega\) models), can be worthwhile to improve the prediction

\(^2\) Future research should include a sensitivity study to evaluate the influence of turbulence intensity at the computational inlet on the distributions of turbulent kinetic energy at the nozzle exit (\(y = 0\))
accuracy in the impingement region (Durbin 1996, Behnia et al. 1998), without having to resort to LES or hybrid RANS-LES methods.

### 3.6 Conclusions and recommendations

The aim of the present study is the validation of steady RANS CFD simulations of PTIJ flows with a jet height to width ratio of \( h_{\text{jet}}/w_{\text{jet}} = 22.5 \) at \( Re = 8,000 \) and 13,000 based on 2D PIV experimental data for the whole vertical centreplane of the PTIJ. Previous numerical studies on PTIJ showed inconsistencies in conclusions regarding the performance of turbulence models. Moreover, there is still a scarcity of basic studies on PTIJ at moderate Reynolds numbers for a jet height larger than 10 jet nozzle widths, including an evaluation of mean velocity and turbulence intensity over the entire height of the impinging jet.

Numerically predicted distributions of mean velocity and turbulent kinetic energy were compared with experimental data along both the jet centreline and several cross-jet lines in lateral direction. Furthermore, the production of turbulent kinetic energy in the jet potential core region near the jet nozzle was analysed.

The most important findings of this study regarding the applicability of steady RANS models to predict isothermal plane turbulent jets at \( Re = 8,000 \) (case I) and 13,000 (case II) are:

- For both cases the best agreement with measured mean velocity and turbulent kinetic energy in the region near the jet nozzle is achieved with SST.
- For case I, the best correspondence in potential core length is provided by RNG, which underestimates the experimental value by 8%. For case II, RKE underestimates the experimental jet potential core length by 8%, providing the closest agreement with experimental data.
- Validation metrics show that centreline distributions of mean velocity and turbulent kinetic energy are most accurately predicted by RNG and RKE for case I, while for case II the best agreement with experimental data is obtained by SKE and RNG. For both cases, the peaks of TKE in the jet impingement region are most accurately predicted by SST.
Based on the overall validation metrics the best performance in predicting mean velocity for both cases is provided by RKE and in predicting turbulent kinetic energy by RKE and RNG.

- All the models, except RSM, provide a good agreement with the experimental values of jet spreading rate within the intermediate jet region for both cases.
- For case I, SKE and RNG accurately predict the jet decay rate within the intermediate region and for case II SKE provides the most accurate estimation as well.
- For several of the lines of analysis considered, the differences in validation metrics between the considered turbulence models are negligibly small. Therefore, based on these metrics, it is difficult to draw general straightforward conclusions regarding the best performing RANS turbulence model for predicting PTIJs.

Based on the results of this study the following recommendations can be made with respect to the performance of the different RANS turbulence models:

- RKE and RNG are suitable for the prediction of jet flow in general. SKE should not be used due to erroneous profiles of turbulent kinetic energy on the jet centreline, especially near the jet nozzle.
- SKE is able to quantify the jet decay rate.
- SST is able to quantify jet spreading rate in the jet intermediate region.
- RNG and RKE are able to quantify the potential core lengths. SST should not be used due to overestimation of this length.
- Only SST is able to fairly accurately predict turbulent kinetic energy in the impingement zone.

Overall, RANS models are suitable to assess general flow characteristics such as jet spreading rate, jet decay rate and estimating of potential core region. These quantities are relevant, for example, in prediction of heat and mass transfer through air curtains. However, steady RANS models are not suitable for basic research focusing on the effect of transient flow features (jet flapping, development and advection of vertical structures, etc.), and are in general less suitable for a detailed analysis of the impingement region.

**Acknowledgements**

Twan van Hooff is currently a postdoctoral fellow of the Research Foundation – Flanders (FWO) and acknowledges its financial support (project FWO 12R9718N). The authors gratefully acknowledge the partnership with ANSYS CFD.
References


Craft TJ, Graham LJW and Launder BE (1993) Impinging jet studies for turbulence model


Thomas FO, Chu HC (1989) An experimental investigation of the transition of a planar jet:


Chapter 4

Minimum momentum flux ratio required to prevent air curtain breakthrough in case of cross-curtain pressure gradients: CFD versus analytical equation

This chapter has been submitted for publication in a peer-reviewed journal:


Abstract

This paper presents a numerical study on the required momentum flux ratio to prevent air curtain breakthrough in case of cross-curtain (i.e. cross-jet) pressure gradients. 2D steady Reynolds-averaged Navier-Stokes (RANS) CFD simulations with the RNG $k$-$\varepsilon$ turbulence model are employed for jet Reynolds numbers ranging from 5,000 to 30,000. First, the computational model is validated based on particle image velocimetry (PIV) measurements. Second, the influence of several jet parameters on the separation efficiency is evaluated for a moderate cross-jet pressure difference of 10 Pa. These are the ratio of the jet discharge momentum flux to the jet cross-flow momentum flux (momentum flux ratio), the jet height-to-width ratio and the jet discharge angle. Finally, the minimum deflection modulus to prevent jet breakthrough and the corresponding momentum flux ratio by an analytical equation and by CFD are compared. The results show that, for the configuration under study: (1) jets with the smallest height-to-width ratios ($\beta = 18$) provide the highest separation efficiency; (2) inclined jets with discharge angles $\alpha_0 = 5^\circ$ and $10^\circ$ provide slightly higher separation efficiency than straight jets ($\alpha_0 = 0^\circ$) and jets with $\alpha_0 = 20^\circ$; (3) the maximum modified separation efficiency is reached at lower momentum flux ratios for jets with smaller height-to-width ratios and for inclined jets; (4) the analytical and CFD values of the optimal momentum flux ratio differ with up to 31.2%. This study shows how the separation efficiency of air curtains can be improved by adjusting certain jet parameters.
List of symbols and acronyms

Roman symbols

\( d_{jet} \) \[m\] nozzle depth
\( D \) \[m\] depth
\( D_m \) [-] deflection modulus
\( D_{m,min} \) [-] minimum deflection modulus
\( D_s \) \[m\] diameter of PIV seeding particles
\( f_{middle} \) \[m/s\] solution for mean velocity obtained on the middle grid
\( \left[ m^2/s^2 \right] \) solution for turbulent kinetic energy obtained on the middle grid
\( f_{coarse} \) \[m/s\] solution for mean velocity obtained on the coarse grid
\( \left[ m^2/s^2 \right] \) solution for turbulent kinetic energy obtained on the coarse grid

\( FAC1.1 \) [-] factor of 1.1 of the observations
\( FAC1.5 \) [-] factor of 1.5 of the observations
\( F_S \) [-] safety factor used for grid convergence index
\( g \) \[m/s^2\] gravitational acceleration
\( h_c \) \[m\] height of contraction
\( h_{jet} \) \[m\] jet height
\( H \) \[m\] height
\( H_d \) \[m\] doorway height
\( k \) \[m^2/s^2\] turbulent kinetic energy
\( L \) \[m\] length
\( M_{cf} \) \[kg/s^2\] cross-flow momentum flux
\( M_{jet} \) \[kg/s^2\] jet discharge momentum flux
\( M_{jet,min} \) \[kg/s^2\] jet discharge momentum flux corresponding to \( D_{m,min} \)
\( M_{jet,sf} \) \[kg/s^2\] jet discharge momentum flux \( M_{jet,min} \) corrected by a safety factor of 2
\( n \) [-] number of data points
\( \vec{n}_i \) [-] outward normal vector
\( O_i \) \[m/s\] time-averaged values of mean velocity obtained from PIV experiments (observations)
Minimum momentum flux ratio: CFD vs. analytical equation

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \Delta P )</td>
<td>[Pa]</td>
<td>cross-jet static pressure gradient</td>
</tr>
<tr>
<td>( P_i )</td>
<td>[m/s]</td>
<td>time-averaged values (predictions) of mean velocity obtained from CFD simulations</td>
</tr>
<tr>
<td>( P_l )</td>
<td>[Pa]</td>
<td>mean static pressure in left side of enclosure</td>
</tr>
<tr>
<td>( P_r )</td>
<td>[Pa]</td>
<td>mean static pressure in right side of enclosure</td>
</tr>
<tr>
<td>( Q_0 )</td>
<td>[kg/s]</td>
<td>heat or pollutant mass transfer rate by transport of outdoor air through the opening to the indoor environment without AC in operation (infiltration)</td>
</tr>
<tr>
<td>( Q'_0 )</td>
<td>[kg/s]</td>
<td>heat or mass transfer rate by transport of indoor air through the opening to the outdoor environment (exfiltration) without AC in operation</td>
</tr>
<tr>
<td>( Q_{ac} )</td>
<td>[kg/s]</td>
<td>heat or pollutant mass transfer rate by transport of outdoor air through the opening to the indoor environment (infiltration) with AC in operation</td>
</tr>
<tr>
<td>( Q'_{ac} )</td>
<td>[kg/s]</td>
<td>heat or mass transfer rate by transport of indoor air through the opening to the outdoor environment (exfiltration) and by transport of air originating from the AC to the outdoor environment with AC in operation</td>
</tr>
<tr>
<td>( p )</td>
<td>[-]</td>
<td>formal order of accuracy used for grid convergence index</td>
</tr>
<tr>
<td>( r )</td>
<td>[-]</td>
<td>linear grid refinement factor for grid sensitivity analysis</td>
</tr>
<tr>
<td>( Re )</td>
<td>[-]</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>( Re_y )</td>
<td>[-]</td>
<td>wall-distance-based Reynolds number</td>
</tr>
<tr>
<td>( Sc_t )</td>
<td>[-]</td>
<td>turbulent Schmidt number</td>
</tr>
<tr>
<td>( U )</td>
<td>[m/s]</td>
<td>mean lateral (x-direction) velocity component</td>
</tr>
<tr>
<td>( U_{cf} )</td>
<td>[m/s]</td>
<td>the average velocity of the cross-flow through the enclosure created by the pressure gradient for the case without AC in operation</td>
</tr>
<tr>
<td>(</td>
<td>V</td>
<td>)</td>
</tr>
<tr>
<td>( \vec{V} )</td>
<td>[m/s]</td>
<td>mean velocity vector</td>
</tr>
<tr>
<td>(</td>
<td>V_0</td>
<td>)</td>
</tr>
<tr>
<td>( V )</td>
<td>[m/s]</td>
<td>mean streamwise (y-direction) velocity component</td>
</tr>
<tr>
<td>( V_0 )</td>
<td>[m/s]</td>
<td>mean streamwise jet velocity at the nozzle exit</td>
</tr>
</tbody>
</table>

[m²/s²] time-averaged values of turbulent kinetic energy obtained from PIV experiments (observations)
\( w_{\text{jet}} \) [m] jet width at the nozzle exit
\( w'_{\text{jet}} \) [m] jet width downstream the nozzle exit at \( y = h_{\text{jet}} \)
\( x \) [m] Cartesian coordinate
\( y \) [m] Cartesian coordinate
\( y' \) [-] dimensionless wall distance
\( Y_{cl} \) [-] clean water mass fraction
\( Y_{pol} \) [-] pollutant mass fraction

Greek symbols
\( \alpha \) [°] angle of the jet centerline downstream of the nozzle exit at \( y = h_{\text{jet}} \)
\( \alpha_0 \) [°] jet discharge angle
\( \beta \) [-] jet height-to-width ratio
\( \gamma \) [-] momentum flux ratio
\( \gamma_{\text{CFD}} \) [-] optimal momentum flux ratio as obtained from the CFD simulations
\( \gamma_{\text{min}} \) [-] minimum momentum flux ratio as obtained from the analytical equation
\( \gamma_{sf} \) [-] momentum flux ratio as obtained from the analytical equation and corrected by a safety factor
\( \delta \gamma \) [-] relative difference between momentum flux ratios
\( \varepsilon \) [m\(^2\)/s\(^3\)] turbulence dissipation rate
\( \eta \) [-] separation efficiency
\( \eta' \) [-] modified separation efficiency
\( \rho \) [kg/m\(^3\)] fluid density
\( \rho_i \) [kg/m\(^3\)] density of indoor air
\( \rho_o \) [kg/m\(^3\)] density of outdoor air
\( \rho_s \) [kg/m\(^3\)] density of the PIV seeding particles
\( \rho_w \) [kg/m\(^3\)] density of water
\( \nu \) [m\(^2\)/s] kinematic viscosity

2D two-dimensional
3D three-dimensional
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>AC</td>
<td>air curtain</td>
</tr>
<tr>
<td>AR</td>
<td>aspect ratio, i.e. the ratio of nozzle depth $d_{jet}$ to width $w_{jet}$</td>
</tr>
<tr>
<td>CFD</td>
<td>computational fluid dynamics</td>
</tr>
<tr>
<td>GCI</td>
<td>grid convergence index</td>
</tr>
<tr>
<td>LES</td>
<td>large eddy simulation</td>
</tr>
<tr>
<td>LRNM</td>
<td>low-Reynolds number modeling</td>
</tr>
<tr>
<td>PIV</td>
<td>particle image velocimetry</td>
</tr>
<tr>
<td>PTIJ</td>
<td>plane turbulent impinging jet</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-averaged Navier-Stokes</td>
</tr>
<tr>
<td>RNG</td>
<td>renormalization group $k$-$\varepsilon$ turbulence model</td>
</tr>
</tbody>
</table>
4.1 Introduction

Air curtains (ACs) are plane turbulent impinging jets (PTIJs) at moderate to high Reynolds numbers that are used to separate two environments in terms of heat and mass transfer. ACs are applied in many practical applications, for example, at entrances of buildings and refrigerated rooms to reduce heat losses (e.g. Wang and Zhong 2014, Goubran et al. 2016, Gonçalves et al. 2019, Yang et al. 2019) and in laboratories or operating theatres (e.g. Nino et al. 2011, Chen et al. 2013, Zhai and Osborne 2013) to reduce contamination hazard. The existing literature on PTIJs and ACs can be divided in two clear categories: (1) basic studies on PTIJs; and (2) application-oriented studies on ACs, both of which can be performed experimentally and numerically (e.g. using computational fluid dynamics (CFD)). Note that an extensive literature review on the topic of AC is not the purpose of this paper and only some relevant studies are discussed below.

With regard to basic studies, a non-exhaustive overview of experimental studies on PTIJs up to 2016 is provided in Khayrullina et al. (2017). These experimental studies highlighted the existence of vortical structures in the jet flow, their role in the dissipation of jet energy to the ambient environment, and the existence of 3D vortices in the impingement region, in which thermal energy can be exchanged between the jet and the impingement surface. A non-exhaustive overview of numerical studies on PTIJs using steady RANS is provided in Khayrullina et al. (2019). This overview concluded that the majority of the numerical studies on PTIJ only focused on heat transfer within the jet impingement region. In line with commonly used general guidelines for CFD studies in building simulation (e.g. Casey and Wintergerste 2000, Nielsen et al. 2007, Blocken 2015), several numerical studies on PTIJs indicated the importance of grid-independent results (e.g. Jaramillo et al. 2008, Rhea et al. 2009), the use of at least second-order discretization schemes (e.g. Craft et al. 1993, Park et al. 2003, Dutta et al. 2013) and the use of low-Reynolds number modeling (LRNM) as a near-wall modeling approach (e.g. Isman et al. 2008). Furthermore, Isman et al. (2008), Jaramillo et al. (2008), Rhea et al. (2009), Kozeoglu and Baskaya (2010) and others highlighted the importance of accurately reproducing the geometry of the jet nozzle and representative boundary conditions of the nozzle exit in CFD simulations of air curtains.

Application-oriented studies generally focused on the dimensioning of the AC, determining the AC efficiency by empirical formulae, and defining the most influential
parameters with respect to the AC efficiency. The efficiency of ACs ($\eta$) can be defined based on the rate of heat or mass transfer through the opening with an AC compared to that of the same opening without an AC (e.g. Frank and Linden 2014, Gonçalves et al. 2019):

$$\eta = 1 - \frac{Q_{ac}}{Q_0}$$  \hspace{1cm} (1)

with $Q_{ac}$ the heat or mass transfer rate by transport of outdoor air through an opening with AC to the indoor environment (infiltration), and $Q_0$ the heat or mass transfer rate by transport of outdoor air through an opening without AC to the indoor environment. It is important to provide aerodynamic sealing along the entire height and width of the opening between the two environments separated by the AC, while also limiting excessive mixing between the jet and the ambient air. Alanis Ruiz et al. (2018) expanded the definition of separation efficiency to the modified separation efficiency ($\eta^*$) that includes mass transfer due to both infiltration and exfiltration and due to the airflow from the AC:

$$\eta^* = 1 - \frac{Q_{ac} + Q^{*}_{ac}}{Q_0 + Q^{*}_{0}}$$  \hspace{1cm} (2)

with $Q^{*}_{ac}$ the heat or mass transfer rate with AC, calculated as the sum of heat or mass transfer rate by transport of indoor air through the opening with an AC to the outdoor environment (exfiltration), and the heat or mass transfer rate by the transport of air originating from the AC to the outdoor environment; and $Q^{*}_{0}$ the heat or mass transfer rate by transport of indoor air through the opening without an AC to the outdoor environment (exfiltration).

Along with environmental parameters, e.g. temperature and pressure differences between the two environments, the separation efficiency of an AC depends on a wide range of jet parameters. The jet nozzle shape has a direct impact on the profiles of jet
velocity and turbulence intensity when the jet exits from the nozzle. For example, Deo et al. (2007a) showed that plane free jets with sharp-edged nozzles result in a higher jet spreading rate than those from smoothly-shaped nozzles, which can negatively influence the AC separation efficiency. This was explained by the higher turbulence intensity levels at the nozzle exit and the faster shear layer growth compared to jets issued by a smoothly-edged nozzle. To the best knowledge of the authors, no studies on ACs are available for different nozzle shapes, although, conclusions of studies on plane free jets (e.g. Deo et al 2007a) could be applied to ACs as well, as long as the other jet parameters in those studies, such as jet aspect ratio and Reynolds number, are similar to those of ACs. The jet aspect ratio (AR; ratio of nozzle depth to width at the nozzle exit \( \frac{d_{\text{jet}}}{w_{\text{jet}}} \), where \( w_{\text{jet}} < d_{\text{jet}} \)) is another important parameter with respect to the jet flow and thus the separation efficiency. Deo et al. (2007b) found that jets at certain Re and smaller AR experience a stronger influence of the sidewalls on the jet flow near the nozzle exit, which enhances velocity fluctuations and results in higher turbulence intensities and higher entrainment rates of ambient fluid compared to jets with larger AR. On the other hand, they reported that jets with larger AR have higher jet spreading and jet decay rates in the jet far field (jet intermediate region). Finally, the jet height-to-width ratio (\( \frac{h_{\text{jet}}}{w_{\text{jet}}} \)) was studied by – among others – Shih et al. (2011) and Moureh and Yataghene (2016), who showed that for increasing height-to-width ratio the AC separation efficiency decreases. Jets with increased height-to-width ratio have a shorter potential core length in relation to \( h_{\text{jet}} \) and experience larger entrainment of the ambient fluid compared to a jet with a smaller height-to-width ratio.

Several studies (e.g. Hayes and Stoecker 1969a, Sirén 2003, Frank and Linden 2014) specified the minimum ratio of jet momentum flux to cross-jet forces acting on AC to ensure the optimal separation efficiency for a doorway. At a certain jet velocity the separation efficiency \( \eta \) (Eq.(1)) of the jet reaches its maximum value. With further increase of the jet velocity the separation efficiency of the air curtain decreases, which is caused by increased mixing with the ambient fluid along the jet and near the floor due to the higher discharge momentum flux of the jet. The jet turbulence intensity also has an effect on jet spreading and entrainment rate. Hayes and Stoecker (1969b) recommended to reduce the turbulence intensity at the nozzle exit to decrease the entrainment rate of the ambient fluid into the jet. The turbulence intensity can be reduced by equipping the discharge nozzle with fine mesh screens and by avoiding sharp-edged nozzles. The separation efficiency of the jet can also be increased by inclining the jet to the exterior environment, as this counterbalances pressure
Minimum momentum flux ratio: CFD vs. analytical equation

Hayes and Stoecker (1969a) introduced the so-called ‘deflection modulus’ to determine the minimum jet momentum flux required for sealing a doorway with an air curtain and preventing breakthrough of the jet. Jet breakthrough occurs if the jet bends at a distance from the impingement plate (floor) and does not impinge on the floor due to insufficient jet momentum flux to counteract a pressure gradient across the doorway (e.g. Hayes and Stoecker 1969a, Frank and Linden 2014). Subsequently, unwanted heat and/or mass transfer occurs through the lowest part of the opening (below the bended jet) between the two environments that needed to be aerodynamically separated. The deflection modulus is defined as the ratio of the momentum flux of the air curtain jet at the nozzle exit to the transverse forces acting across the jet due to stack pressure across the doorway. Based on the conservation of momentum, the authors provided an equation to estimate the required minimum value of the deflection modulus ($D_{m,min}$) of an air curtain at the doorway of a heated room:

$$D_{m,min} = \frac{\rho w_{\text{jet}} |V_0|^2}{g H_d^2 (\rho_o - \rho_i)}$$

with $\rho$ the density of the jet at the nozzle exit, $w_{\text{jet}}$ the jet width at the nozzle exit, $g$ the gravitational acceleration ($\approx 9.8 \text{ m/s}^2$), $|V_0|$ the jet velocity magnitude (i.e. the magnitude of the 3D velocity vector) at the nozzle exit, $H_d$ the height of the doorway, and $\rho_o$ and $\rho_i$ the density of air outside and inside the room, respectively. Moreover, it was suggested to multiply the velocity at the nozzle exit by a safety factor of 1.3 to 2.0 (Hayes and Stoecker 1969b), resulting in a safety factor for the deflection modulus between 1.7 to 4.0, in order to ensure a stable air curtain in real situations.

Recently some experimental and numerical studies were performed that used the deflection modulus to estimate the optimal jet momentum flux. The CFD study of Costa et al. (2006) considered an air curtain restricting heat transfer between two environments at different temperatures. They showed that a deflection modulus $D_{m,min}$
multiplied with a safety factor of approximately 2.6 prevented air curtains from breakthrough. Foster et al. (2006) performed a CFD study on an air curtain installed in a doorway of a building and showed that a safety factor of 1.6 applied to $D_{m,min}$ provided the highest separation efficiency. Van Belleghem et al. (2012) conducted a CFD study on an air curtain flow in a sealed cold room application. They showed that a safety factor of 2 was required for the deflection modulus ($D_{m,min}$) to be able to provide a stable air curtain without breakthrough, which is in line with the initial recommendation by Hayes and Stoecker (1969b). Frank and Linden (2014) investigated the performance of an air curtain in a doorway of a ventilated building both with analytical and experimental models. They considered a number of situations with different positions of the neutral pressure level height, which is the height at which indoor and outdoor pressures are equal. For situations where the neutral pressure level height was located slightly above the mid-height of the door opening, the defined $D_{m,min}$ had to be multiplied by a safety factor of 2.8 to prevent breakthrough. However, a safety factor of 7 had to be applied in case the neutral pressure level height was far above the mid-height of the door opening. This higher safety factor can be explained by the ventilation losses through the additional top opening introduced by Frank and Linden (2014), whereas the study of Hayes and Stoecker (1969b) considered only transverse forces acting on the doorway.

From the literature it can be concluded that application studies on AC are often very case-specific and in spite of their value to solve specific real-life problems, lack generality in their conclusions. On the other hand, basic and generic studies on PTIJs generally focused on heat transfer within the jet impingement region. As a result, there is a scarcity with respect to detailed generic studies on ACs and PTIJs that assess the influence of the jet parameters on the separation efficiency. The primary focus of this paper is therefore (1) to provide a parametric analysis of the impact of several jet parameters (momentum flux ratio, jet height-to-width ratio, jet discharge angle\textsuperscript{1}) on jet separation efficiency; and (2) to perform a comparison between values of the momentum flux ratio based on $D_{m,min}$ as obtained from the analytical equation by Hayes and Stoecker (1969b) and values of the momentum flux ratio required for a maximum modified separation efficiency as obtained from CFD simulations, and this for a wide range of parameters. This paper focuses on the generic situation of an isothermal AC separating two environments subjected to a moderate cross-jet pressure difference (in this study fixed at 10 Pa). 2D steady RANS CFD simulations are performed for jet

\textsuperscript{1} Jet discharge angle $\alpha_0$ is taken at the nozzle exit ($y = 0$ m)
Reynolds numbers ranging from 5,000 to 30,000. First, the computational model is validated based on particle image velocimetry (PIV) data by Khayrullina et al. (2017). Subsequently, a parametric study is performed, in which the influence of several jet parameters on jet separation efficiency is evaluated. Finally, for each jet configuration (depending on jet height-to-width ratio and jet discharge angle), the ratio of jet discharge momentum flux to jet cross-flow momentum flux (momentum flux ratio) at which the modified separation efficiency of AC reaches its maximum value is obtained. These values are compared with the values of the momentum flux ratio obtained from the deflection modulus as defined by Hayes and Stoecker (1969b). The paper is structured as follows. Section 4.2 presents the experimental set-up, the PIV measurements, the computational model and the computational settings/parameters used in the CFD simulations. Finally, it presents the results of the validation study. Section 4.3 presents the computational settings and parameters for the case study. Section 4.4 presents the results of the case study. Finally, Sections 4.5 (Discussion) and Section 4.6 (Conclusions) conclude this paper.

4.2 Validation study

4.2.1. Reduced-scale PIV measurements

The PIV measurements of PTIJ flow were conducted in a water channel with dimensions $L \times H \times D = 2.00 \times 0.36 \times 0.30$ m$^3$. The reduced-scale experiments (scale 1:8) were performed with water as working fluid for purposes of dynamic similarity (equal Reynolds number) between the water jet at the reduced scale (i.e. the size of the water channel) and an actual air curtain jet in full scale ($H \times D = 2.88 \times 2.40$ m$^2$). The experimental setup is shown in Figure 1. A water column (not shown in the figure) created hydrostatic pressure and drove the flow through a conditioning section consisting of one honeycomb and two screens in order to provide a uniform flow and to reduce the turbulence intensity. Subsequently, a plane jet was issued vertically from the smoothly-shaped nozzle. In the experiments, the mean streamwise velocity at the nozzle exit $V_0$ was equal to 0.5 m/s, corresponding to $Re = 8,000$. The Reynolds number was defined as $Re = (V_0w_{jet})/\nu$, with $w_{jet} = 16$ mm the jet width at the nozzle exit and $\nu$ the kinematic viscosity at 20 °C. The jet height ($h_{jet}$, distance from the nozzle exit to the impingement plate) was 360 mm and the nozzle depth (dimension in spanwise direction;
The nozzle aspect ratio was $AR = 18.75$ and the jet height-to-width ratio was $\beta = 22.5$.

![Figure 1: Experimental set-up (adapted from Khayrullina et al. 2017)](image)

The PIV measurements were conducted with a 2D PIV system consisting of a solid-state frequency-doubled Nd:Yag laser (wavelength 532 nm and repetition rate 15 Hz) as an illuminating source and a charge coupled device (CCD) camera (1600 x 1200 pixels resolution, up to 30 frames/s) for image acquisition. Seeding was provided by polyamide particles ($D_s = 50 \mu m$, density $\rho_s = 1,030 \text{ kg/m}^3$) added to the water. The particles were illuminated by means of a light sheet delivered at the bottom of the water channel. The PIV measurements provided information on the mean velocities and turbulence intensities in the vertical centerplane. More information on the experimental set-up and measurement results can be found in Khayrullina et al. (2017). The experimental results will be shown together with the CFD results in the following subsections.
4.2.2. **Computational model**

4.2.2.1 **Computational geometry and grid**

The 2D computational geometry \((L \times (h_{\text{jet}} + h_j)) = (2.0 \times (0.36 + 0.15)) \text{ m}^2\) replicates the experimental setup described in Section 4.2.1 (Fig. 2). A 2D instead of a 3D geometry is used to limit the computational costs for the parametric study. The simulation results for a 2D geometry have been compared to the results for a 3D geometry and only negligible differences have been observed between their results in the vertical centerplane, which can be explained by the limited influence of sidewalls at a jet aspect ratio of \(AR = 18.75\). The maximum difference between the results obtained along the jet centerline by 2D and 3D CFD simulations is less than 7% for predictions of mean velocity and less than 5% for predictions of turbulent kinetic energy.

![Figure 2](image-url)

*Figure 2: (a) 2D computational geometry with indication of boundary conditions. (b-c) Computational grid at: (b) smoothly-shaped nozzle; (c) impingement plate*
A structured grid with a high spatial resolution is applied within the smoothly-shaped nozzle, within the zone outlined by the jet spreading rate and within the jet impingement zone. For the remainder of the computational domain an unstructured quadrilateral mesh is applied. The grid resolution is determined by means of a grid sensitivity analysis using three different grids (see Section 4.2.3 for results) with a grid-refinement factor of $\sqrt{2}$ in each direction: coarse grid (23,553 cells), middle grid (41,209 cells) and fine grid (85,110 cells). Across the computational inlet, i.e. the inlet of the nozzle, 28, 40 and 56 cells are used for the coarse, middle and fine grids, respectively (Fig. 3). The dimensionless wall distances ($y^*$, average values) for the coarse, middle and fine grids within the smoothly-shaped jet nozzle are equal to 4, 2.5 and 1, respectively, while at the impingement plate the average $y^*$ values are 5, 1.5 and 0.5, respectively. These three grids with $y^* < 5$ at the walls enable low-Reynolds number modeling for the wall-adjacent flow, i.e. resolving this flow all the way down the viscous sublayer, rather than resorting to wall function modeling.

![Figure 3: Computational grid in and near the nozzle with indication of number of cells over the width of the nozzle: (a) coarse grid (23,553 cells); (b) middle grid (41,209 cells); (c) fine grid (85,110 cells)](image-url)
4.2.2.2 Boundary conditions

At the inlet of the computational domain a uniform velocity in vertical direction corresponding to \( V_{0,\text{CFD,inlet}} = 0.085 \text{ m/s} \) is imposed, which results in \( V_0 = 0.5 \text{ m/s} \) and \( Re = 8,000 \) at the nozzle exit. The turbulence intensity at the domain inlet is set to 15\%, which results in a turbulence intensity on the jet centerline near the nozzle exit corresponding to the measured value (= 4\%). At the outlets zero static gauge pressure is applied. The remaining surfaces are modeled as no-slip walls.

4.2.2.3 Turbulence model and solver settings

The 2D steady RANS equations are solved with the commercial CFD code ANSYS Fluent 16 (ANSYS Inc. 2013) using the renormalization group (RNG) \( k-\varepsilon \) turbulence model (Yakhot et al. 1992) to provide closure to the governing equations. This model is chosen due to its reported good overall performance in previous PTIJ studies (e.g. Isman et al. 2008, Sharif and Mothe 2009, Khayrullina et al. 2019). Low-Reynolds number modeling (LRNM) is used to solve the near-wall flow. The LRNM option used here consists of the two-layer model. The domain is divided in a viscosity-affected region and a fully turbulent region, based on the wall-distance-based Reynolds number \( Re_y \). The border between both regions is defined as \( Re_y = 200 \); if \( Re_y < 200 \), the Wolfshtein model is used (Wolfshtein 1969), while for \( Re_y > 200 \) the original transport equations of the RNG model are solved. An Eulerian advection-diffusion equation is used to model pollutant dispersion, with the turbulent Schmidt number taken equal to \( Sc_t = 0.7 \) (Tominaga and Stathopoulos 2007). Second-order discretization schemes are used for both the convective and viscous terms of the governing equations and for the turbulence model equations, pressure-velocity coupling is performed by the coupled algorithm, and pressure interpolation is second order. Convergence is considered to be achieved when the scaled residuals (ANSYS Inc. 2013) reach the following minimum values: \( 10^{-14} \) for \( x \) and \( y \) velocity, \( 10^{-13} \) for \( k \) and \( \varepsilon \), and \( 10^{-12} \) for continuity.

4.2.3 Grid sensitivity analysis

The grid sensitivity analysis is conducted using a coarse, middle and fine grid (see Section 4.2.2.1). The results of the grid sensitivity analysis in terms of the centerline profiles \( (x/w_{\text{jet}} = 0) \) of normalized jet velocity in the vertical direction \( (V/V_0) \) and
normalized turbulent kinetic energy \( (k/V_0^2) \) are shown in Figure 4. The grid-convergence index (GCI) by Roache (1994, 1997) is calculated for the coarse grid, which is superimposed on the results for the coarse grid by shaded bounds in Figure 4, using:

\[
GCI_{\text{coarse}} = F_S \left| r^p \left( f_{\text{coarse}} - f_{\text{middle}} \right) \right| \left/ \left(1 - r^p\right) \right|
\]

with \( F_S \) the safety factor that is equal to the recommended value of 1.25 when at least three grids are analyzed, \( r \) the linear grid refinement factor equal to \( \sqrt{2} \), \( p \) the formal order of accuracy, which is assigned the value of 2 as second-order discretization schemes are used for the simulations (Roache 1997) and \( f_{\text{coarse}} \) and \( f_{\text{middle}} \) the solutions obtained on the coarse and middle grid, respectively, which correspond to \( V/V_0 \) and \( k/V_0^2 \). The average values of \( GCI_{\text{coarse}} \) along the jet centerline are 1% and 8% for distributions of mean velocity and turbulent kinetic energy, respectively. The largest discrepancy occurs at the transition between the potential and intermediate jet regions.

Figure 4: Results of grid-sensitivity analysis along jet centerline: (a) normalized velocity \( (V/V_0) \); (b) normalized turbulent kinetic energy \( (k/V_0^2) \). Blue bounds indicate the GCI index calculated for the coarse grid, \( y/w_{\text{jet}} = 0 \) corresponds to the jet nozzle exit. Note that the three lines almost overlap in both graphs and are therefore less visible.
(\(y/h_{\text{jet}} \approx 0.2\)). This is due to the increasing size of the cells further downstream from the nozzle exit along the jet height in combination with the large gradients of mean velocity and turbulent kinetic energy that occur around \(y/h_{\text{jet}} \approx 0.2\). Based on this analysis it is concluded that the coarse grid provides nearly grid-independent results and it is therefore used in the remainder of this study.

4.2.4. Validation

Figure 5 shows profiles of \(V/V_0\) and \(k/V_0^2\) along the jet centerline obtained from the PIV measurements and the CFD simulations with the RNG \(k-\varepsilon\) turbulence model. In general, a very good agreement is observed for \(V/V_0\), while an overestimation of \(k/V_0^2\) in CFD is present along the entire vertical centerline. RNG utilizes a simplified production term of turbulent kinetic energy based on the Boussinesq relationship with the assumption that the turbulence viscosity is an isotropic scalar quantity, which is expected to result in the deviations observed in Figure 5b.

Figure 5: Comparison of time-averaged PIV results with results of steady RANS CFD simulations obtained along the jet centerline (\(x/w_{\text{jet}} = 0\)): (a) \(V/V_0\); (b) \(k/V_0^2\)
In order to provide a quantitative assessment of the agreement between experimental and numerical results, the factor of 1.1 and 1.5 of the observations (FAC1.1 and FAC1.5) are applied for predictions of mean velocity and turbulent kinetic energy, respectively:

\[
FAC1.1 = \frac{1}{n} \sum_{i=1}^{n} N_i \quad \text{with} \quad N_i = \begin{cases} 
1 & \text{for} \quad 0.91 \leq \frac{P_i}{O_i} \leq 1.1 \\
0 & \text{else} 
\end{cases} 
\]

\[
FAC1.5 = \frac{1}{n} \sum_{i=1}^{n} N_i \quad \text{with} \quad N_i = \begin{cases} 
1 & \text{for} \quad 0.67 \leq \frac{P_i}{O_i} \leq 1.5 \\
0 & \text{else} 
\end{cases} 
\]

with \( P_i \) and \( O_i \) the values obtained from CFD simulations and PIV experiments, respectively, and \( n \) the number of data points. The values of \( FAC1.1 \) and \( FAC1.5 \) indicate the fraction of the considered data points, where the simulation results fall within a factor of 1.1 and 1.5 of the experimentally obtained values, respectively. As indicated by Schatzmann et al. (2010), \( FAC1.1 \) and \( FAC1.5 \) provide a clear picture of randomly occurring high or low differences between measured and predicted values of characteristic flow quantities. The predictions of the centerline mean velocity result in \( FAC1.1 = 0.91 \), while the predictions of the centerline turbulent kinetic energy result in \( FAC1.5 = 0.64 \). Compared with the results from 3D RANS simulations with the RNG \( k-\varepsilon \) turbulence model (see Khayrullina et al. 2019), the current 2D predicted values of the centerline mean velocity show a very similar agreement (i.e. \( FAC1.1 = 0.91 \) for 3D RANS), while the 2D values of the centerline turbulent kinetic energy show a worse agreement with the experimental data (i.e. \( FAC1.5 = 0.80 \) for 3D RANS), which can – at least partly – be explained by the fact that turbulent kinetic energy is based on the two modeled normal Reynolds stress terms in 2D simulation (instead of three terms as in the 3D simulation). The values of \( FAC1.1 \) and \( FAC1.5 \) obtained from the 2D simulations are
considered to be within an acceptable range and allow a reduction of the computational demand for the large amount of simulations conducted in this study.\(^2\)

### 4.3 Case studies

A parametric study is performed for the case of an isothermal jet representing an air curtain that separates two environments subjected to a cross-jet pressure gradient, in which the influence of the momentum flux ratio, the jet height-to-width ratio and the jet discharge angle on the jet separation efficiency is evaluated. Similar to the validation study described in Section 4.2, this case study is performed with a water jet in a reduced-scale domain, representative of an air jet in a full-scale domain (geometric scaling factor = 1:8).

#### 4.3.1 Computational geometry and grid

Figure 6a shows the 2D computational geometry for the case study replicating to a large extent the computational geometry used in the validation study (Section 4.2). The jet widths at the nozzle exit \(w_{\text{jet}} = 8, 12, 16, 20\) mm (where 16 mm corresponds to the validation study) are studied resulting in height-to-width ratios of \(\beta = 45.0, 30.0, 22.5\) and 18.0, respectively. Additional jet discharge angles are considered in this case study: i.e. \(\alpha_0 = 5^\circ, 10^\circ, \text{ and } 20^\circ\) (Fig. 6c – 6e, respectively). This resulted in 16 different computational geometries. Note that \(w_{\text{jet}}\) does not change when varying the jet discharge angle. As in the validation study, the coarse grid is used based on a grid sensitivity analysis.

\(^2\) Note that it is important that turbulence models are able to predict the length of the potential core due to its influence on the separation efficiency of an air curtain. The potential core is airtight and provides a perfect sealing against heat or mass transfer through an air curtain.
Figure 6: (a) Computational geometry with indication of boundary conditions, (b-e) nozzle with: (b) $\alpha_0 = 0^\circ$; (c) $\alpha_0 = 5^\circ$; (d) $\alpha_0 = 10^\circ$; (e) $\alpha_0 = 20^\circ$. The dashed lines on both sides of the jet indicate the lines used for the analysis of the results.

4.3.2 Boundary conditions

In order to assess the AC separation efficiency, a cross-jet static pressure difference of 10 Pa is applied by imposing a 10 Pa static gauge pressure at the left side of the enclosure (along the height $h_{jet}$) by means of a pressure inlet boundary condition and zero static gauge pressure at the right side of the enclosure by means of a pressure outlet boundary condition. The water in the computational domain is defined as a mixture of two species: clean water and a passive pollutant. The condition $Y_{pol} + Y_{cl} = 1$, with $Y_{pol}$ the pollutant mass fraction and $Y_{cl}$ the clean water mass fraction, applies throughout the computational domain. The pollutant mass fraction imposed at the pressure inlet of the enclosure is $Y_{pol} = 1$, while the pollutant mass fraction at the velocity inlet (nozzle inlet) and the pressure outlet on the right side of the enclosure is $Y_{pol} = 0$. The pollutant mass fraction is also specified at the pressure outlet as backflow can occur at this boundary. A range of mean velocities at the velocity inlet is considered, resulting in 240 cases with jet $Re (= |V_0|w_{jet}/\nu$, with $|V_0|$ the jet velocity magnitude at the nozzle exit) varying from 5,000 to 30,000 with a uniform step ($\Delta Re = 1,000$).

4.3.3 Turbulence model and solver settings

The settings are identical to those in the validation study (see Section 4.2.2.3). Convergence is considered to be achieved when the scaled residuals reach the following minimum values: $10^{-14}$ for $x$ and $y$ velocity, $10^{-13}$ for $k$ and $\epsilon$, $10^{-12}$ for continuity, and $10^{-8}$ for species transport.
4.3.4 Evaluation parameters

In this study the ratio of jet discharge momentum flux \( M_{\text{jet}} \) to cross-flow momentum flux \( M_{\text{cf}} \), called the momentum flux ratio \( \gamma \), is used to characterize the jet:

\[
\gamma = \frac{M_{\text{jet}}}{M_{\text{cf}}} = \frac{\rho_w w_{\text{jet}} |V_0|^2}{\rho_w h_{\text{jet}} U_{\text{cf}}^2} = \frac{w_{\text{jet}} |V_0|^2}{h_{\text{jet}} U_{\text{cf}}^2}
\]

(7)

with \( |V_0| \) the mean jet velocity magnitude at the nozzle exit (\( y = 0 \) m) and \( U_{\text{cf}} \) the average mean velocity of the cross-flow through the enclosure created by the pressure gradient for the case without an AC (\( U_{\text{cf}} \) is equal to \( \approx 0.13 \) m/s\(^3\)). Figure 7 shows the quadratic relationship between the jet Reynolds number and \( \gamma \).

![Figure 7: Relation between jet Reynolds number and \( \gamma = M_{\text{jet}}/M_{\text{cf}} \) for four jet widths](image)

\(^3\) Representative of cross-flow velocity (air) in full scale equal to 0.25 m/s
The jet separation efficiencies \( \eta \) and \( \eta^* \) are calculated according to Eq. (1) and Eq. (2), respectively. However, since there is no exfiltration through the opening in the case without an AC due to the imposed pressure gradient, Eq. (2) can, in this particular case, be simplified to:

\[
\eta^* = 1 - \frac{Q_{ac} + Q^*_{ac}}{Q_0}
\]  

(8)

The transfer of the pollutant to the right side of the enclosure (infiltration) is computed along line-2 at a distance \( x_2 = 0.05 \) m from the middle of the nozzle exit \( (x = 0) \) (see Fig. 6a), with:

\[
Q_i = \int_0^{h_{jet}} Y_{pol} \rho_w (\vec{V} \cdot \vec{n}_2) \, dy
\]  

(9)

with \( Q_i \) the pollutant mass transfer rate from the left side to the right side (with or without air curtain, i.e. \( Q_{ac} \) or \( Q_0 \), respectively), \( Y_{pol} \) the mass fraction of the pollutant, \( \rho_w \) the water density, \( \vec{V} \) the velocity vector, and \( \vec{n}_2 \) the outward vector normal to line-2, i.e. the vector corresponding to the direction of the pollutant mass transfer \( Q_i \) as indicated in Figure 6a. The transfer of clean water from the nozzle exit and from the right side of the enclosure to the left side (exfiltration) is computed along line-1 at a distance \( x_1 = -0.20 \) m from the middle of the nozzle exit \( (x = 0) \) (see Fig. 6a) with:

\[
Q^*_{ac} = \int_0^{h_{jet}} Y_{cl} \rho_w (\vec{V} \cdot \vec{n}_1) \, dy
\]  

(10)
with \( Q'_{ac} \) the clean water mass transfer rate with air curtain to the left side of the enclosure, \( Y_{cl} \) the clean water mass fraction, and \( n_1 \) the outward vector normal to the line-1, i.e. the vector corresponding to the direction of the clean water mass transfer \( Q'_i \) as indicated in Figure 6a. The distances of \( x_1 = -0.20 \) m and \( x_2 = 0.05 \) m are based on a sensitivity analysis with the aim to minimize the influence of jet flow on the results for species mass transfer.

4.4 Results

This section analyzes the influence of jet height-to-width ratio (\( \beta \)), jet discharge angle (\( \alpha_0 \)) and jet momentum flux ratio (\( \gamma \)) on jet separation efficiency. We define the “optimal momentum flux ratio” (optimal \( \gamma \)) as the ratio for which the modified separation efficiency \( \eta^* \) (see Eq. (2)) reaches its highest value and we provide the values of the separation efficiencies \( \eta \) (see Eq. (1)) that correspond to this optimal \( \gamma \). In order to provide a more accurate estimation of the optimal \( \gamma \) for different geometries, additional simulations have been performed for each computational geometry. These simulations are performed for a range of \( \gamma \), for which the defined separation efficiencies \( \eta^* \) reach the highest values, with a refined step of \( \Delta Re = 100 \) for the jet.

4.4.1 Influence of jet height-to-width ratio on jet separation efficiency

Figure 8 provides the separation efficiencies \( \eta \) and \( \eta^* \) as a function of \( \gamma \) for two different jet discharge angles: \( \alpha_0 = 0^\circ \) and \( 20^\circ \), presented separately for each considered jet height-to-width ratio: \( \beta = 18 \) (Fig. 8a,b), \( \beta = 22.5 \) (Fig. 8c,d), \( \beta = 30 \) (Fig. 8e,f) and \( \beta = 45 \) (Fig. 8g,h). Each data point in the figure corresponds to one case with a certain momentum flux ratio of the jet and a certain jet discharge angle. The data for jets with discharge angles \( \alpha_0 = 5^\circ \) and \( 10^\circ \) are excluded from the figure to ensure its readability.

For jets with discharge angle \( \alpha_0 = 0^\circ \), Figure 8 shows that starting from \( \gamma = 0.2 \) until \( \gamma \approx 1.5 \) the separation efficiency \( \eta \) (Fig. 8a,c,e,g) steeply increases from 18% to 97.2% for jets with \( \beta = 18 \), to 96.4% for jets with \( \beta = 22.5 \), to 95.3% for jets with \( \beta = 30 \), and to 93.9% for jets with \( \beta = 45 \). The modified separation efficiency \( \eta^* \) (Fig. 8b,d,f,h) reaches
96.0% for jets with $\beta = 18$, 94.3% for jets with $\beta = 22.5$, 93.7% for jets with $\beta = 30$, and 92.8% for jets with $\beta = 45$. For $\gamma > 1.5$ the separation efficiency $\eta$ gradually decreases to 96.2%, 94.5%, 92.0% and 87.6% for jets with $\beta = 18$, 22.5, 30 and 45, respectively. The modified separation efficiency $\eta^*$ shows a steeper decrease to 66.1%, 62.4%, 57.1% and 47.9% for jets with $\beta = 18$, 22.5, 30 and 45, respectively, within the considered range of values of $\gamma$. Note that for each considered computational geometry the highest value of the modified separation efficiency $\eta^*$ is obtained at slightly different values of $\gamma$.

Similarly, for jets with discharge angle $\alpha_0 = 20^\circ$, the separation efficiency $\eta$ (Fig. 8a,c,e,g) steeply increases from 29% to 97.8% for jets with $\beta = 18$, to 97.2% for jets with $\beta = 22.5$, to 96.6% for jets with $\beta = 30$, and to 96.2% for jets with $\beta = 45$. The modified separation efficiency $\eta^*$ (Fig. 8b,d,f,h) reaches 95.9% for jets with $\beta = 18$, 95.2% for jets with $\beta = 22.5$, 94.2% for jets with $\beta = 30$, and 91.5% for jets with $\beta = 45$. After reaching the maximum value, the separation efficiency $\eta$ gradually decreases to 97.5%, 96.6%, 95.4% and 94.3% for jets with $\beta = 18$, 22.5, 30 and 45, respectively. The modified separation efficiency $\eta^*$ shows a decrease to 54.5%, 49.0%, 41.1% and 28.2% for jets with $\beta = 18$, 22.5, 30 and 45, respectively, within the considered range of values of $\gamma$.

Table 1 provides the values of optimal $\gamma$ for each considered computational geometry ($\beta$ and $\alpha_0$) and the corresponding values of the separation efficiencies $\eta$ and $\eta^*$. The results for jet discharge angle $\alpha_0 = 0^\circ$ show that the highest value of the modified separation efficiency $\eta^* = 0.960$ and the corresponding value of the separation efficiency $\eta = 0.961$ are obtained for a jet with $\beta = 18$, i.e. for larger jet widths. In addition, for $\alpha_0 = 0^\circ$, the lowest modified separation efficiency $\eta^* = 0.928$ and the corresponding value of the separation efficiency $\eta = 0.929$ are obtained for jets with $\beta = 45$ (i.e. the smallest jet width). Likewise, for jets with jet discharge angle $\alpha_0 = 20^\circ$ the highest values of the separation efficiency $\eta^* = 0.959$ and corresponding $\eta = 0.961$ are obtained for a jet with $\beta = 18$, and the lowest value of the modified separation efficiency $\eta^* = 0.915$ and corresponding $\eta = 0.936$ are obtained for a jet with $\beta = 45$. Similar findings are obtained for jet discharge angles $\alpha_0 = 5^\circ$ and $10^\circ$. For each considered jet discharge angle, the optimal $\gamma$ for jets with $\beta = 18$ is lower than for jets with larger $\beta$ (i.e. smaller jet widths).
4.4.2 Influence of jet discharge angle on jet separation efficiency

Figure 9 provides the distributions of $\eta$ and $\eta^*$ as a function of $\gamma$ for two different jet height-to-width ratios: $\beta = 18$ and $\beta = 45$, which are presented separately for each considered jet discharge angle: $\alpha_0 = 0^\circ$ (Fig. 9a,b), $5^\circ$ (Fig. 9c,d), $10^\circ$ (Fig. 9e,f), and $20^\circ$ (Fig. 9g,h). Similar to the previous subsection, the data for jets with $\beta = 22.5$ and $30$ are excluded from the figure to ensure its readability.

The results for jets with $\beta = 18$ show that the separation efficiency $\eta$ (Fig. 9a,c,e,g) reaches $97.2\%$ for jets with $\alpha_0 = 0^\circ$, $97.1\%$ for jets with $\alpha_0 = 5^\circ$, $97.3\%$ for jets with $\alpha_0 = 10^\circ$, and $97.8\%$ for jets with $\alpha_0 = 20^\circ$. The modified separation efficiency $\eta^*$ (Fig. 9b,d,f,h) reaches $96.0\%$ for jets with $\alpha_0 = 0^\circ$, $96.3\%$ for jets with $\alpha_0 = 5^\circ$ and $10^\circ$, and $95.9\%$ for jets with $\alpha_0 = 20^\circ$.

With regard to jets with $\beta = 45$, the separation efficiency $\eta$ (Fig. 9a,c,e,g) reaches $93.9\%$ for jets with $\alpha_0 = 0^\circ$, $94.1\%$ for jets with $\alpha_0 = 5^\circ$, $94.4\%$ for jets with $\alpha_0 = 10^\circ$, and $96.2\%$ for jets with $\alpha_0 = 20^\circ$. The modified separation efficiency $\eta^*$ (Fig. 9b,d,f,h) reaches $92.8\%$ for jets with $\alpha_0 = 0^\circ$, $93.6\%$ for jets with $\alpha_0 = 5^\circ$ and $10^\circ$, and $91.5\%$ for jets with $\alpha_0 = 20^\circ$.

The results in Table 1 show that for jets with $\beta = 18$ the highest value of the modified separation efficiency $\eta^* = 0.963$ and the corresponding value of the separation efficiency $\eta = 0.963$ are obtained for jets with $\alpha_0 = 5^\circ$ and $10^\circ$. Likewise, for jets with $\beta = 45$ the highest value of $\eta^* = 0.936$ and corresponding value of $\eta = 0.937$ are obtained for jets with $\alpha_0 = 5^\circ$ and $10^\circ$. Similar findings are obtained for with $\beta = 22.5$ and $30$. Overall, inclined jets with $\alpha_0 = 5^\circ$ and $10^\circ$ provide slightly higher separation efficiencies than straight ($\alpha_0 = 0^\circ$) jets and jets with $\alpha_0 = 20^\circ$. Note that optimal $\gamma$ for jets with $\alpha_0 = 20^\circ$ are lower than for jets with smaller discharge angles. The optimal $\gamma$ for straight jets ($\alpha_0 = 0^\circ$) are highest compared to those of jets with discharge angles $\alpha_0 = 5^\circ$, $10^\circ$, $20^\circ$.

4.4.3 Influence of momentum flux ratio on jet separation efficiency

Figure 10 shows the distributions of $\eta$ and $\eta^*$ as a function of $\gamma$. Each data point in the figure corresponds to one case with a jet at a certain $Re$, $\beta$ and $\alpha_0$. For each case two data points are shown that present $\eta$ and $\eta^*$, respectively. The shape of the distribution of $\eta$ resembles those provided by e.g. Costa et al. (2006), Foster et al. (2006) and Frank and Linden (2015). Figures 11a-11c show the distributions of dimensionless mean velo-
Figure 8: Influence of jet height-to-width ratio $\beta$ on jet separation efficiencies $\eta$ and $\eta^*$ for jets with $\alpha_0 = 0^\circ$ and $20^\circ$: (a,b) $\beta = h_{\text{jet}}/w_{\text{jet}} = 18$; (c,d) $\beta = 22.5$; (e,f) $\beta = 30$; (g,h) $\beta = 45$. 
city magnitude ($|V|/|V_0|$) for the jet with $\beta = 22.5$, $\alpha_0 = 0^\circ$ and for three $\gamma = 0.25, 1.22$ and 9.07, respectively. For $\gamma = 0.25$, the jet momentum flux cannot compensate the cross-jet pressure gradient and does not reach the floor, i.e. so-called breakthrough of the jet occurs (Fig. 11a). The separation efficiencies here are $\eta = \eta^* = 0.178$. From $\gamma = 0.25$ to $\gamma = 1.22$ the separation efficiencies $\eta$ and $\eta^*$ are steeply increasing from 0.18 to 0.955 and 0.943, respectively. At $\gamma = 1.22$, the jet reaches the ground providing an aerodynamic sealing (Fig. 11b). The separation efficiency $\eta^*$ starts to decrease for $\gamma > 1.22$, while the separation efficiency $\eta$ starts to decrease for $\gamma > 1.97$, with a steeper decrease for $\eta^*$ compared to that for $\eta$. At $\gamma = 9.07$ (Fig. 11c) the jet provides an aerodynamic sealing with $\eta = 0.945$ and $\eta^* = 0.624$. The explanation for this steep decrease of the separation efficiency $\eta^*$ is that at $\gamma > 1.22$ a significant part of the supplied water from the nozzle exit flows to the left side of the enclosure, resulting in a lower modified separation efficiency. Moreover, higher momentum flux ratios cause more entrainment of the ambient fluid into the jet. In the considered case the pollutant mass transfer from the left part of the enclosure into the right part was 18% lower for $\gamma = 1.22$ ($Q_{ac} = 2.26$ kg/s) than that for $\gamma = 9.07$ ($Q_{ac} = 2.76$ kg/s).

Table 1: Optimal values of momentum flux ratio $\gamma = M_{jet}/M_{cf}$ and corresponding separation efficiencies $\eta$ and $\eta^*$ for the considered cases

<table>
<thead>
<tr>
<th>$\beta$ [-]</th>
<th>$\alpha_0$ [°]</th>
<th>$\gamma$, optimal [-]</th>
<th>$\eta$ [-]</th>
<th>$\eta^*$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>18.0</td>
<td>0</td>
<td>1.07</td>
<td>0.961</td>
<td>0.960</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>0.93</td>
<td>0.963</td>
<td>0.963</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>0.82</td>
<td>0.963</td>
<td>0.963</td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>0.66</td>
<td>0.961</td>
<td>0.959</td>
</tr>
<tr>
<td>22.5</td>
<td>0</td>
<td>1.22</td>
<td>0.955</td>
<td>0.943</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>1.01</td>
<td>0.958</td>
<td>0.956</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>0.87</td>
<td>0.957</td>
<td>0.956</td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>0.69</td>
<td>0.953</td>
<td>0.952</td>
</tr>
<tr>
<td>30.0</td>
<td>0</td>
<td>1.34</td>
<td>0.946</td>
<td>0.937</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>1.10</td>
<td>0.948</td>
<td>0.947</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>0.95</td>
<td>0.948</td>
<td>0.947</td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>0.74</td>
<td>0.945</td>
<td>0.942</td>
</tr>
<tr>
<td>45.0</td>
<td>0</td>
<td>1.46</td>
<td>0.929</td>
<td>0.928</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>1.21</td>
<td>0.937</td>
<td>0.936</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>1.03</td>
<td>0.937</td>
<td>0.936</td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>0.85</td>
<td>0.936</td>
<td>0.915</td>
</tr>
</tbody>
</table>
Figure 9: Influence of jet discharge angle $\alpha_0$ on separation efficiencies $\eta$ and $\eta^*$ for jets with $\beta = 18$ and 45: (a,b) $\alpha_0 = 0^\circ$; (c,d) $\alpha_0 = 5^\circ$; (e,f) $\alpha_0 = 10^\circ$; (g,h) $\alpha_0 = 20^\circ$
Minimum momentum flux ratio: CFD vs. analytical equation

Figure 10: Separation efficiencies $\eta (■)$ and $\eta^* (▲)$ as a function of momentum flux ratio $\gamma$

Figure 11: Contours of dimensionless mean velocity magnitude ($|V|/|V_0|$) for $\beta = 22.5$ for three momentum flux ratios: (a) $\gamma = 0.25$; (b) $\gamma = 1.22$; (c) $\gamma = 9.07$
The results in Table 1 show that:

- The optimal $\gamma$ ranges from 0.66 to 1.46. The highest value ($\gamma = 1.46$) occurs for a jet with $\beta = 45$ at $\alpha_0 = 0^\circ$. The lowest value ($\gamma = 0.66$) occurs for a jet with $\beta = 18$ at $\alpha_0 = 20^\circ$.

- The modified separation efficiencies $\eta^*$, i.e. the values corresponding to the optimal $\gamma$ in Table 1, range from 0.915 to 0.963. The highest value ($\eta^* = 0.963$) occurs for $\beta = 18$ at $\alpha_0 = 5^\circ$ and $10^\circ$. The lowest value ($\eta^* = 0.915$) occurs for $\beta = 45$ at $\alpha_0 = 0^\circ$.

- The separation efficiency $\eta$ corresponding to the optimal $\gamma$ ranges from 0.929 to 0.963. The highest value ($\eta = 0.963$) occurs for $\beta = 18$ at $\alpha_0 = 5^\circ$ and $10^\circ$. The lowest value ($\eta = 0.929$) occurs for $\beta = 45$ at $\alpha_0 = 0^\circ$.

The value of the optimal $\gamma$ for a jet with $\beta = 18$ at $\alpha_0 = 20^\circ$ is lower than for jets with other $\beta$ at the same discharge angle $\alpha_0 = 20^\circ$. Figure 12 provides the distributions of pollutant mass fractions for jets with $\gamma = 0.66$: $\beta = 18$ at $\alpha_0 = 0^\circ$ and $\alpha_0 = 20^\circ$ and $\beta = 45$ at $\alpha_0 = 20^\circ$. Figure 12 shows that a significant amount of pollutant is transferred through the jet to the right side of the enclosure for the cases with jet $\beta = 18$ at $\alpha_0 = 0^\circ$ ($Q_{ac} = 19.13$ kg/s, $\eta = 0.618$) and $\beta = 45$ at $\alpha_0 = 20^\circ$ ($Q_{ac} = 9.61$ kg/s, $\eta = 0.808$) compared to the case with $\beta = 18$ at $\alpha_0 = 20^\circ$ ($Q_{ac} = 2.06$ kg/s, $\eta = 0.961$). Therefore, the combination of two factors, such as smaller jet height-to-width ratio and larger jet discharge angle, helps a jet to counteract the cross-jet pressure gradient. Figure 12a also shows that, for the case with $\beta = 18$ at $\alpha_0 = 20^\circ$, the pollutant concentration on the left side of the enclosure remains much lower compared to the cases with $\beta = 18$ at $\alpha_0 = 0^\circ$ and $\beta = 45$ at $\alpha_0 = 20^\circ$, where $Y_{pol} \approx 1$. Thus, mixing occurs between “polluted” fluid in the left part of the enclosure with the fluid from the nozzle exit and the ambient fluid from the right part of the enclosure. For the case with $\beta = 18$ and $\alpha_0 = 20^\circ$, the transfer of clean water from the nozzle exit and from the right side of the enclosure to the left side $Q_{ac}^* = 0.025$ kg/s, while for the cases with $\beta = 18$ and $\alpha_0 = 0^\circ$, and $\beta = 45$ and $\alpha_0 = 20^\circ$ the values of $Q_{ac}^*$ are equal to 0. In combination with the mentioned above values of pollutant mass transfer $Q_{ac}$ this results in the higher value of the modified separation efficiency ($\eta^* = 0.959$) for the case with $\beta = 18$ and $\alpha_0 = 20^\circ$ than for the cases with $\beta = 18$ and $\alpha_0 = 0^\circ$ ($\eta^* = 0.618$), and $\beta = 45$ and $\alpha_0 = 20^\circ$ ($\eta^* = 0.808$).
Figure 12: Distributions of pollutant mass fraction $Y_{pol}$ for the cases with $\gamma = 0.66$: (a) $\beta = 18$ at $\alpha_0 = 20^\circ$; (b) $\beta = 18$ at $\alpha_0 = 0^\circ$; (c) $\beta = 45$ at $\alpha_0 = 20^\circ$

4.4.4 Deflection modulus

The deflection modulus can also be defined as the dimensionless ratio of the magnitude of the discharge momentum flux of the jet to the transverse forces acting on the jet due to a cross-jet pressure gradient across the doorway (e.g. Howell and Shibata 1980, Sirén 2003, Frank and Linden 2015):

$$D_m = \frac{\rho W_{jet} |V_0|^2}{\Delta P h_{jet}}$$  \hspace{1cm} (11)
with $\Delta P$ the static pressure difference over the jet. In this section, the focus is on the minimum deflection modulus $D_{m,min}$ that is reached at the conditions where jet breakthrough just do not occur yet. For this, the approach also applied by Sirén (2003) is adopted that is based on the momentum balance. Assuming that the flow is steady-state and neglecting gravity and viscous forces, the force associated with the change in jet momentum flux in the $x$-direction (i.e. horizontal direction) is equal to the force associated with the cross-jet pressure gradients in $x$-direction (Fig. 13):

$$
\int_{0}^{h_{jet}} \rho \mathbf{U} \cdot \mathbf{n} \, dy = \Delta P h_{jet} = (P_t - P_r) h_{jet}
$$

(12)

The left-hand side of Eq. (12) can be written as:

$$
\int_{0}^{h_{jet}} \rho \mathbf{U} \cdot \mathbf{n} \, dy = \rho w_{jet} \left| V_0 \right|^2 \sin \alpha_0 - \rho w'_{jet} \left| V \right|^2 \sin \alpha
$$

(13)

with $U$ the mean lateral ($x$-direction) velocity component, $w'_{jet}$ and $\alpha$ the width and the angle of the jet at the impingement region near the floor. Eq. (13) is the $x$-component of the momentum balance applied to the contour indicated by the orange dashed rectangle in Figure 13. In addition, it is assumed that the jet momentum flux along the jet centerline is conserved (Rajaratnam 1976, Frank and Linden 2015):

$$
\rho w_{jet} \left| V_0 \right|^2 = \rho w'_{jet} \left| V \right|^2
$$

(14)
The value of $D_{m,min}$ is found when jet breakthrough occurs at $y = h_{jet}$ assuming that the jet angle $\alpha = -90^\circ$, i.e. when the jet is bent to the right side of the enclosure. Combining Eqs. (11-14) yields:

$$D_{m,min} = \frac{1}{1 + \sin \alpha_0}$$

Table 2 lists the values of $D_{m,min}$ calculated based on the simplified Eq. (15) for the four jet discharge angles ($\alpha_0 = 0^\circ$, 5°, 10°, 20°) and the corresponding values of $\gamma_{min} = M_{jet,min}/M_{cf}$. In addition, the values of $\gamma_{sf} = M_{jet,sf}/M_{cf}$ are provided, which are the values of $\gamma_{min}$ corrected by a safety factor of 2, as recommended in previous studies (e.g. Hayes and Stoecker 1969a, Foster et al. 2006, Van Belleghem et al. 2012). Moreover, a comparison is provided between these values of $\gamma_{min}$ and the values of the optimal momentum flux ratios $\gamma_{CFD} = M_{jet,CFD}/M_{cf}$ as obtained from the CFD simulations (see Table 1 with the values of $\gamma$ for four jet height-to-width ratios and for four jet discharge angles). The relative difference between these two quantities is calculated as:

$$\delta \gamma = \frac{\gamma_{CFD} - \gamma_{sf}}{\gamma_{CFD}}$$
Table 2: Values of deflection modulus $D_{m, \text{min}}$ and corresponding momentum flux ratios as determined by Eq. (15) in combination with a safety factor and by CFD, and deviations between the latter two parameters

<table>
<thead>
<tr>
<th>$\alpha_0$ [°]</th>
<th>$D_{m, \text{min}}$ [-] (Eq.15)</th>
<th>$\gamma_{\text{min}}$ [-]</th>
<th>$\gamma_{\text{sf}}$ [-]</th>
<th>$\gamma_{\text{CFD}}$ [-]</th>
<th>$\delta \gamma$ [%]</th>
<th>$\beta$</th>
<th>$\beta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>1.00</td>
<td>0.58</td>
<td>1.16</td>
<td>1.07</td>
<td>1.22</td>
<td>1.34</td>
<td>1.46</td>
</tr>
<tr>
<td>5</td>
<td>0.92</td>
<td>0.53</td>
<td>1.07</td>
<td>0.93</td>
<td>1.01</td>
<td>1.10</td>
<td>1.21</td>
</tr>
<tr>
<td>10</td>
<td>0.85</td>
<td>0.50</td>
<td>0.99</td>
<td>0.82</td>
<td>0.87</td>
<td>0.95</td>
<td>1.03</td>
</tr>
<tr>
<td>20</td>
<td>0.75</td>
<td>0.43</td>
<td>0.87</td>
<td>0.66</td>
<td>0.69</td>
<td>0.74</td>
<td>0.85</td>
</tr>
</tbody>
</table>

Figure 14 shows the values of the optimal $\gamma$ for four different $\alpha_0$ and four different $\beta$ together with distribution of $\gamma_{\text{sf}}$.

The results in Table 2 show that the highest minimum value of the momentum flux ratio required to prevent a breakthrough of the jet and defined analytically in combination with a safety factor of 2 ($\gamma_{\text{sf}} = 1.16$) is obtained for jets with $\alpha_0 = 0°$. The lowest minimum value of $\gamma_{\text{sf}} = 0.87$ is obtained for a jet with $\alpha_0 = 20°$. Note that the analytical method to define the minimum required momentum flux ratio considered in this study does not depend on the jet height-to-width ratio $\gamma$, but solely on the jet discharge angle $\alpha_0$.

In contrast, the values of the optimal $\gamma_{\text{CFD}}$ depend on both $\alpha_0$ and $\beta$. The values of $\gamma_{\text{CFD}}$ range from 1.07 to 1.46 for jets with $\alpha_0 = 0°$, from 0.93 to 1.21 for jets with $\alpha_0 = 5°$, from 0.82 to 1.03 for jets with $\alpha_0 = 10°$, and from 0.66 to 0.85 for jets with $\alpha_0 = 20°$. The highest value of $\gamma_{\text{CFD}} = 1.46$ occurs for a jet with $\beta = 45$ and $\alpha_0 = 0°$ and the lowest value ($\gamma_{\text{CFD}} = 0.66$) occurs for a jet with $\beta = 18$ and $\alpha_0 = 20°$. The results also show that the difference between the momentum flux ratios $\gamma_{\text{sf}}$ obtained by Eq. (15) and corrected with a safety factor of 2, and the values of optimal $\gamma_{\text{CFD}}$ reaches $\delta \gamma = -31.2\%$ for the jet with $\beta = 18$ and $\alpha_0 = 20°$. These deviations are related to the complex flow dynamics of the jet in a cross-flow and to the assumptions made in defining the minimum jet deflection modulus (Eq. 15). As mentioned before, Eq. (15) does not take into account the jet height-to-width ratio. It also does not take into account the mixing of the jet with the ambient fluid, while optimal momentum flux ratios $\gamma_{\text{CFD}}$ defined with CFD simulations consider the potential transfer of the jet flow to the left part of the enclosure and clean water transfer from the right to the left part of the enclosure.

Figure 14 illustrates the common trend that both $\gamma_{\text{sf}}$ and $\gamma_{\text{CFD}}$ decrease with increasing jet discharge angle $\alpha_0$. 
Discussion

This paper presents the results of a numerical study of a PTIJ (representing an air curtain) at Reynolds numbers ranging from 5,000 to 30,000. The influence of several jet parameters, i.e. the momentum flux ratio, the jet height-to-width ratio, and the jet discharge angle, on the jet separation efficiency is evaluated with 2D steady RANS CFD simulations. Based on this analysis optimal values of the momentum flux ratio are defined when the jet reaches its maximum modified separation efficiency. This modified separation efficiency does not only take infiltration into account but also exfiltration. Finally, the minimum deflection modulus required to prevent breakthrough of the jet and the corresponding minimum momentum flux ratios are defined based on a simple analytical equation and by applying a safety factor according to Hayes and Stoecker (1969a). These minimum momentum flux ratios are compared to the values of the optimal momentum flux ratios obtained from the CFD simulations.
4.5.1 Applicability of steady RANS CFD simulations

Although large eddy simulations (LES) are intrinsically superior to steady RANS simulations, a review of the literature on CFD in building simulation for outdoor and indoor applications shows that the latter are frequently used in both science and engineering and in both basic and applied research (Blocken 2018). Although future research will include LES as well, the validation study in the present paper has indicated that steady RANS was a suitable choice.

4.5.2 Applicability of the analytical equation to define the minimum deflection modulus

The analytical equation to determine the minimum deflection modulus is based on the principle of momentum conservation. It might be used in the initial estimation of the AC settings to prevent breakthrough of the jet. The analytical equation is simple and very easy to use but it has the following limitations:

- Mixing processes between the jet and the ambient environment are not taken into account.
- The specific influence of the jet height-to-width ratio is not taken into account.
- The assumption that jet breakthrough occurs at \( y = h_{\text{jet}} \) when the jet angle \( \alpha = -90^\circ \) does not take into account realistic jet behavior.
- The equation will be less reliable in more complex situations, e.g. involving pollutant mass transfer at densities different than density of the ambient fluid or with localized sources.

4.5.3 Future work

Based on this study the following recommendations for future work can be made:

- Future studies can consider angles larger than \( \alpha_0 = 20^\circ \) and investigate their influence on the jet separation efficiencies and the corresponding optimal momentum flux ratios. Moreover, the optimum jet discharge angle can be defined to obtain the highest separation efficiency.
- This study considered a given constant cross-jet pressure that was uniform along the height of the opening. Future studies should consider a range of pressure gradients and their influence on the optimal momentum flux ratios. In real situations, pronounced pressure gradients can be present along the jet height and the jet can
be imposed to highly dynamic cross-jet wind pressure (i.e. when ACs separate outdoor and indoor environment). Such more case-specific situations should be taken into account for actual/real case studies with AC implementation.

- This study considered an isothermal situation. Real situations often involve differences in air temperature between the indoor and outdoor environments, which also induces cross-jet pressure differences. Moreover, the presence of natural or mechanical ventilation in the building has its influence on the flow inside the building.

- The optimal momentum flux ratios were determined with CFD simulations for a steady-state situation. Future studies can consider transient environmental conditions such as moving public, sliding doors and changing wind speeds, which can affect the actual separation efficiency of the AC. Such effects could be included by means of safety factors that can be determined by transient CFD simulations or experiments.

4.6 Conclusions

The generic situation of an isothermal PTIJ (air curtain) at $5,000 < Re < 30,000$ separating two environments subjected to a cross-jet pressure gradient is studied using 2D steady RANS CFD simulations. The study involves a parametric analysis of the influence of several jet parameters (momentum flux ratio, jet height-to-width ratio, jet discharge angle) on the jet separation efficiency. Four jet height-to-width ratios ($\beta = 18, 22.5, 30, 45$) and four jet discharge angles ($\alpha_0 = 0^\circ, 5^\circ, 10^\circ, 20^\circ$) are considered. For each jet height-to-width ratio and jet discharge angle, the optimal ratio of jet discharge momentum flux to jet cross-flow momentum flux (optimal momentum flux ratio) at which the modified separation efficiency of an AC reaches its maximum value is defined. At momentum flux ratios higher than the optimal value the jet hits the floor at higher velocities, resulting in excessive mixing between the jet and the ambient environment. In practice that would result in energy dissipation from the jet to the outdoor environment. Therefore, a too high momentum flux ratio causes an increased energy consumption by the air-curtain fan which should be prevented. Finally, the minimum deflection modulus to prevent breakthrough of the jet and its corresponding minimum momentum flux ratios are obtained based on a simplified equation and are compared to the values of optimal momentum flux ratios obtained from the CFD simulations. The following conclusions are provided:
The highest modified separation efficiency, i.e. $\eta^* = 0.963$, is obtained for a jet with $\beta = 18$ (at $\alpha_0 = 5^\circ$ and $10^\circ$).

The lowest modified separation efficiency $\eta^* = 0.915$ is obtained for a jet with $\beta = 45$ at $\alpha_0 = 20^\circ$.

The highest value of the optimal momentum flux ratio $\gamma = 1.46$ occurs for a jet with $\beta = 45$ at $\alpha_0 = 0^\circ$.

The lowest value of the optimal momentum flux ratio $\gamma = 0.66$ occurs for a jet with $\beta = 18$ and $\alpha_0 = 20^\circ$.

Jets with smaller height-to-width ratios provide a higher modified separation efficiency than jets with larger height-to-width ratios. Moreover, jets with the smallest height-to-width ratio ($\beta = 18$) reach their maximum modified separation efficiency at lower optimal momentum flux ratios.

Inclined jets with discharge angles $\alpha_0 = 5^\circ$ and $10^\circ$ provide a higher modified separation efficiency $\eta^*$ than straight ($\alpha_0 = 0^\circ$) jets and jets with discharge angle $\alpha_0 = 20^\circ$.

The values of the optimal momentum flux ratio calculated based on the CFD simulations range from $1.07 < \gamma < 1.46$ for jets at discharge angle $\alpha_0 = 0^\circ$, $0.93 < \gamma < 1.21$ for jets at discharge angle $\alpha_0 = 5^\circ$, $0.82 < \gamma < 1.03$ for jets at discharge angle $\alpha_0 = 10^\circ$, and $0.66 < \gamma < 0.85$ for jets at discharge angle $\alpha_0 = 20^\circ$. The highest values of $\gamma$ for each jet discharge angle correspond to the largest jet height-to-width ratio ($\beta = 45$).

The values of the optimal momentum flux ratio determined based on the CFD simulations deviate by up to 31.2% from the values defined by the simple analytical equation including a safety factor. This simplified equation can be used for the initial estimation of the required AC conditions to provide aerodynamic sealing in a given situation. However, in order to provide more accurate information, one should resort to the methods that better take into account the jet dynamics, e.g. CFD simulations with RANS modeling, or more advanced modeling with LES.

Acknowledgements

Twan van Hooff is currently a postdoctoral fellow of the Research Foundation – Flanders (FWO) and acknowledges its financial support (project FWO 12R9718N). The authors acknowledge the partnership with ANSYS CFD.
References


Ventilation and Air Conditioning associations.


Chapter 5

Impact of a wall downstream of an air curtain nozzle on air curtain separation efficiency

This chapter has been submitted for publication in a peer-reviewed journal:


Abstract

Air curtains (ACs) employ plane turbulent jets to separate two environments in terms of heat and mass transfer, while still allowing unrestricted access through the opening between these environments. Most previous studies focused on ACs discharged from nozzles located just above the opening. However, in some cases ACs have to be installed close to the ceiling at a substantial distance from the top of the opening. The AC blown downwards along the vertical wall then first resembles a wall jet and after reaching the top of the opening starts resembling a free jet. The present study analyzes the behavior and performance of an AC with upstream wall above the opening. 2D steady RANS CFD simulations are performed based on grid-sensitivity analyses and validation with experimental data for a wall jet and a free jet. The total opening height is 4 m and vertical walls of 0.5 m, 1 m and 2 m, partly closing this opening, are considered. AC performance is evaluated with the separation efficiency $\eta$ (based on infiltration) and the modified separation efficiency $\eta^*$ (based on infiltration and exfiltration). It is shown that the presence of the wall reduces jet decay. The higher the wall, the larger the jet momentum over the opening height. This reduces infiltration and increases $\eta$ but it increases exfiltration (decreases $\eta^*$). In practice, jet discharge velocity (jet momentum) will have to be adjusted to keep high $\eta^*$. 
5.1 Introduction

Air curtains (ACs) employ plane turbulent impinging jets (PTIJ) to separate two environments in terms of heat and mass transfer, while still allowing an unrestricted transfer of persons and products between these environments. Figure 1a is a schematic representation of a PTIJ discharged by a rectangular nozzle. The fluid mechanical behavior of a PTIJ involves the development of roll-up vortices, the related entrainment of ambient fluid, and the impingement and the emergence of counter-rotating vortices after impingement. ACs can be exposed to cross-jet temperature differences, pressure differences and/or concentration differences.

ACs are typically used at the entrance of stores, shops, hotels, restaurants and bars, to separate the indoor and outdoor environment while still allowing easy access, and to improve indoor thermal comfort and indoor air quality (e.g. [1-7]; see Fig. 1b). Applications of ACs can also be found inside buildings to separate smoking and non-smoking areas, in laboratories to reduce contamination hazard, in loading docks, in hospital operating theatres (e.g. [8]), in refrigerated trucks and in refrigerated display cabinets in supermarkets (e.g. [9-16]). They can also be used to maintain a stable indoor environment in incubators for neonates (at neonatology departments in hospitals) (e.g. [17,18]). Furthermore, they are employed in tunnels/channels to reduce smoke propagation in case of fire, etc. (e.g. [18-21]).

Depending on the application, various categories of ACs can be distinguished. The air flow provided by the AC can be pre-heated, pre-cooled or remain at the temperature of the supplied air. Costa et al. [4] provided an overview of studies that focused on the dimensioning of ACs and distinguished two categories of ACs: with and without air recirculation. In an AC with air recirculation, a duct is used to return the discharged air to the rear of the AC. For a vertically downwards blowing AC with recirculation this duct is connected to a grille installed in the floor where the opening is located. For an AC without air recirculation, the discharged air mixes with the ambient air (indoor and outdoor). Then the mixed air partly returns to the rear of the AC (Fig. 1b). For sufficient sealing, the jet has to be strong enough to cover the whole height of the opening. ACs can be installed above, below or next to the opening, thus representing a downward jet, an upward jet, or a horizontal jet, respectively.
The separation efficiency is a key performance parameter for ACs. It is determined by an equation based on the heat or mass transfer through the opening when the AC is “on” versus the heat or mass transfer when the curtain is “off” [4,22-24]. The separation efficiency depends on a wide range of jet parameters and environmental parameters, such as nozzle characteristics, the jet discharge parameters (i.e. mean jet velocity $U_0$, jet temperature $T_0$, and turbulence intensity $I_{u,0}$) and the air temperature, pressure and concentration in the two separated environments.

Generally, ACs are installed close to the opening to be sealed. The Eurovent code of good practice “Eurovent 16/1, Air curtain unit – Classification, test conditions and energy performance calculations” [25] states a maximum distance of 0.5 m between the discharge nozzle and the top of the opening. However, in practice, sometimes the AC discharge nozzle has to be installed close to the ceiling at a larger distance from the opening. Both in situations with a maximum distance of 0.5 m and in situations with a larger distance, the air jet flowing from the discharge nozzle of the AC towards the opening first develops as a wall jet, and after reaching the top of the opening this wall jet becomes a free jet, until a certain distance from the floor where adverse pressure build-up will mark the beginning of the jet impingement region.

Figures 2a and b are schematic representations of a wall jet and a free jet, respectively. A wall jet consists of an inner layer with a boundary layer flow and an outer layer with a free shear layer flow. Moving downstream from the discharge opening, a potential core region, a transitional region and finally a fully developed region (self-similar region) can be distinguished. A free jet on the other hand is not constrained by the presence of a wall. It consists of a potential core region, a transitional region, a self-similar region and
a termination region. In the potential core region, the centerline velocity is equal to 98-100% [1,28] of the jet discharge velocity. The free jet velocity profile shows a top-hat distribution and the shear layers start to evolve. In the transition region, the potential core is totally consumed by the shear layers and the centerline velocity starts to decay due to the entrainment of ambient air into the jet. In the self-similar region, the normalized cross-jet velocity profiles are identical at different distances from the discharge nozzle and show a Gaussian shape. Finally, in the termination region the jet becomes almost imperceptible due to diffusion with the ambient air. In this region the centerline velocity decreases sharply to zero [29].

Although many studies analyzed the behavior of wall jets and free jets and many studies analyzed the behavior of ACs, to the best of our knowledge, there is no previous study that analyzed the behavior of an AC consisting of both a clear and extensive wall jet and a subsequent free jet followed by impingement. Therefore, it is not clear to what extent the separation efficiency of an AC is influenced by installing the nozzle at different heights above the opening. This study presents numerical simulations with computational fluid dynamics (CFD) to investigate the behavior and separation efficiency of an AC when it first flows along a wall before reaching the opening. The study focuses on an isothermal AC without recirculation. The CFD simulations are supported by validation studies for the case of a wall jet and a free/impinging jet. The paper is structured as follows. Section 5.2 presents the validation studies. Section 5.3 provides the computational settings and parameters for the case studies. In section 5.4, the results of the case studies are outlined. Sections 5.5 (limitations and further work) and 5.6 (conclusions) conclude the paper.

5.2 CFD validation studies

The CFD simulations are performed based on the Reynolds-averaged Navier-Stokes (RANS) equations. While approaches such as large eddy simulation (LES) are intrinsically superior, they also entail a higher complexity and a much higher computational cost, and because of these reasons, in building simulation, RANS often remains the preferred choice [30]. This is in line with the fact that the vast majority of the scientific literature on CFD simulations of air curtains adopted the RANS approach. The validation studies
Figure 2: Schematic representation of (a) wall jet (modified from Kaffel et al. [26]) and (b) free jet (modified from Yue [27]) with indication of the different jet regions. b is nozzle width or jet discharge width, x is downstream direction coordinate, $U_m$ is the maximum fluid speed ($x$-velocity component) occurring at distance $y_m$ from the wall, $y_{1/2}$ is jet half-width

include a wall jet and a free jet, for which experimental data are available in the literature. In order to provide a quantitative assessment of the performance of different turbulence models the metric factor of 1.3 of the observations (FAC1.3) is applied:

$$FAC1.3 = \frac{1}{n} \sum_{i=1}^{n} N_i \quad \text{with} \quad N_i = \begin{cases} 1 & \text{for } 0.77 \leq \frac{P_i}{O_i} \leq 1.30 \\ 0 & \text{else} \end{cases}$$

(1)

with $P_i$ and $O_i$ the time-averaged values obtained from CFD simulations and experiments, respectively, and $n$ the number of data points.
5.2.1 Wall jet

The validation study for the wall jet employs the experimental data provided by Eriksson et al. [31], who performed laser Doppler velocimetry (LDV) experiments of an isothermal wall jet with $Re = 9,600$ discharged at the bottom of a water tank of 7 m long and 1.45 m wide, with the jet Reynolds number defined as $Re = (U_0b)/\nu$, with $b$ the nozzle height and $\nu$ the kinematic viscosity at 20°C. The jet nozzle has a height of $b = 0.0096$ m and it is present along the entire width of the tank. The 2D computational domain represents part of the tank and has a height of 0.9 m and a length of 5.0 m (Fig. 3a). The domain is discretized by a structured grid (Fig. 3b). A grid-sensitivity analysis with coarse, medium and fine grids is conducted. The number of cells over the height of the discharge nozzle are 20, 30 and 50 for the coarse, medium and fine grids, respectively (Fig. 3b), resulting in total cell counts of 87,300, 101,800 and 171,600, respectively. The maximum values of the dimensionless wall distance $y^+$ for the coarse, medium and fine grids are equal to 16, 1.7 and 1.0, respectively. The grid-sensitivity study indicates that the medium grid

![Figure 3: (a) Computational geometry of the wall jet study (not to scale). (b) View of coarse, medium and fine grid near nozzle exit. (c,d) Turbulence model evaluation on medium grid showing vertical profiles of $(U/U_m)$ by CFD and experiments at (c) $x/b = 20$ and (d) $x/b = 100$]
Impact of a wall downstream of an air curtain nozzle provides nearly grid-independent results and therefore this grid is retained for the validation study.

The mean jet discharge velocity at the nozzle exit \((U_0)\) is 1 m/s and the discharge turbulence intensity is 1%. The solid surfaces of the domain are specified as no-slip walls. Zero static gauge pressure is imposed at the outlet. The isothermal simulations are performed with ANSYS Fluent 15.0.7 [32]. The 2D steady RANS equations are solved with the standard \(k-\varepsilon\) model [33], the realizable \(k-\varepsilon\) turbulence model [34], the SST \(k-\omega\) model [35] and the re-normalization group (RNG) \(k-\varepsilon\) [36] model. Enhanced wall treatment (EWT) [32] is applied, in which a two-layer model is combined with enhanced wall functions. If the near-wall mesh is fine enough to be able to resolve the laminar sublayer (typically \(y^+ \approx 1\)), then the two-layer zonal model is used, which employs the Wolfshtein model [37] in the near-wall region. If the near-wall mesh is coarser, enhanced wall functions are employed which result from blending linear (laminar) and logarithmic (turbulent) laws-of-the-wall using a function suggested by Kader [38]. The SIMPLEC algorithm is utilized for pressure-velocity coupling, pressure interpolation is second order and second-order spatial discretization schemes are used for both the convection terms and the viscous terms of the governing equations. Convergence is assumed to be obtained when scaled residuals level off and reach \(10^{-6}\) for \(x\) and \(y\) momentum, \(10^{-5}\) for \(k\), \(\varepsilon\) and \(\omega\), and \(10^{-4}\) for continuity. Figures 3c and d present the results in terms of vertical profiles of the dimensionless mean \(x\)-velocity component \((U/U_m, \text{with } U_m \text{ the maximum jet velocity along the vertical line considered})\) at two distances from the discharge nozzle; \(x/b = 20\) and 100, as a function of the ratio \(y/y_{1/2}\) where \(y_{1/2}\) is the jet half-width. Based on the values of FAC1.3 (Table 1), the results from the realizable \(k-\varepsilon\) model with EWT show the best agreement with the experimental data at both \(x/b = 20\) (FAC1.3 = 0.81) and 100 (FAC1.3 = 0.91).

<table>
<thead>
<tr>
<th>Model</th>
<th>FAC1.3 (x/b = 20)</th>
<th>FAC1.3 (x/b = 100)</th>
</tr>
</thead>
<tbody>
<tr>
<td>SKE</td>
<td>0.65</td>
<td>0.88</td>
</tr>
<tr>
<td>RKE</td>
<td>0.81</td>
<td>0.91</td>
</tr>
<tr>
<td>RNG</td>
<td>0.78</td>
<td>0.79</td>
</tr>
<tr>
<td>SST</td>
<td>0.81</td>
<td>0.76</td>
</tr>
</tbody>
</table>
5.2.2 Free jet

The validation study for the free jet employs the experimental data provided by Maurel and Solliec [1], who performed LDV and particle image velocimetry (PIV) measurements for an isothermal free jet impinging on a flat plate at different distances from the discharge nozzle. The height of the impinging jet \( H \) in the set-up was variable from 0.1 m to 1.4 m, therefore, the opening ratio \( H/b \) could be varied from 20 to 50. The depth of the setup (in lateral direction) is 1.4 m. The nozzle has a width \( b = 0.02 \) m and is present along the entire depth of the setup. The jet discharge velocity for the validation study is \( U_0 = 20 \) m/s, resulting in a jet Reynolds number of \( Re = (U_0 b)/\nu = 25,640 \). For the validation study, the data for \( H/b = 25 \) are used, i.e. \( \nu = 0.5 \) m. The 2D computational geometry is depicted in Figure 4a. Note that only half the geometry is included in the simulations because of the symmetry of the flow. A grid-sensitivity study on three grids is conducted with the SST \( k-\omega \) turbulence model [35]. Figure 4b shows the grid resolution near the nozzle. The number of cells across half the width of the discharge nozzle is 4, 6 and 8 for the coarse, medium and fine grid, respectively, resulting in total cell counts of 5,777, 11,550 and 23,108. The grid-sensitivity study indicates that the medium grid provides nearly grid-independent results and therefore this grid is retained for the validation study.

A non-uniform velocity profile, based on the profile obtained at the nozzle exit from a CFD simulation including a smoothly-shaped contraction, with a centerline mean velocity of 20 m/s is applied at the velocity inlet and the discharge turbulence intensity is 2.8%. The solid surfaces, such as the top and the bottom of the geometric model are treated as no-slip walls. Zero static gauge pressure is imposed at the outlet. The isothermal simulations are performed with ANSYS Fluent 15.0.7 [32] with the 2D steady RANS equations in combination with the following turbulence models: the standard \( k-\varepsilon \) model [33], the RNG \( k-\varepsilon \) model [36], the realizable \( k-\varepsilon \) model [34] and the SST \( k-\omega \) model [35] in combination with EWT. The SIMPLEC algorithm is utilized for pressure-velocity coupling, pressure interpolation is second order and second-order discretization schemes are used for both the convection terms and the viscous terms of the governing equations. Convergence is assumed when scaled residuals level off and reach a minimum of \( 10^{-6} \) for \( x \) and \( y \) momentum, \( 10^{-5} \) for \( k, \varepsilon \) and \( \omega \), and \( 10^{-4} \) for continuity. Figure 4c compares the jet centerline dimensionless mean velocity profiles \((U/U_0)\). Close to the discharge nozzle \((x/H < 0.2)\), the CFD profiles obtained with different turbulence models are nearly identical, except for the profile provided by the SST \( k-\omega \).
model. The best agreement in the intermediate region is provided by the standard $k$-$\varepsilon$ model, especially at $0.2 \leq x/H \leq 0.6$. At $0.6 \leq x/H \leq 0.9$ all models slightly overestimate the experimental mean velocities. Based on the FAC1.3 values in Table 2 the RNG $k$-$\varepsilon$ model (FAC1.3 = 0.89) and the realizable $k$-$\varepsilon$ model (FAC1.3 = 0.89) provide the best agreement with the experimental data.
Table 2 Validation metrics for distributions of mean x-velocity component $U/U_m$

<table>
<thead>
<tr>
<th>Model</th>
<th>FAC1.3</th>
</tr>
</thead>
<tbody>
<tr>
<td>SKE</td>
<td>0.86</td>
</tr>
<tr>
<td>RKE</td>
<td>0.89</td>
</tr>
<tr>
<td>RNG</td>
<td>0.89</td>
</tr>
<tr>
<td>SST</td>
<td>0.83</td>
</tr>
</tbody>
</table>

Based on the results from the two validation studies, the realizable $k$-$\varepsilon$ model combined with EWT is used for the case studies\(^1\).

### 5.3 Case studies: computational settings and parameters

#### 5.3.1 Computational geometry and domain

The 2D geometry consists of two zones, one with polluted air (left) and one with clean air (right), separated by an AC installed above the opening connecting both zones (Fig. 5a). The AC is installed in the zone with clean air. It is an AC without recirculation so the air for the AC jet is extracted from the zone with clean air and after impingement on the floor the AC air can move either into the left zone or the right zone, or both, where it mixes with the air in that zone. The two zones have an identical geometry, each measuring $L \times H = 28.8 \text{ m} \times 4.8 \text{ m}$. The opening has a height of 4 m. The AC system has a height of 0.8 m and a nozzle width $b = 0.2 \text{ m}$. In the reference configuration, the AC nozzle is positioned at 4 m above the floor. In total, seven geometrical configurations are examined, which are schematically depicted in Figures 5b-h. The reference configuration is a free jet (no wall downstream the AC nozzle). Three configurations represent wall jets with different downstream wall heights: 0.5 m, 1.0 m and 2.0 m (Case A). These geometries represent variations to the case where it is assumed that the design requirements are such that the AC unit has to be placed against the ceiling at 4 m height.

---

\(^1\) Note that it is important that the turbulence models are able to accurately predict the length of the potential core due to its influence on the separation efficiency of an air curtain. The potential core is airtight and provides a perfect sealing against heat or mass transfer through an air curtain.
and where an optional (e.g. glass) wall can be installed below the AC. An example of such design requirement is a case in which no visual obstructions are allowed over the 4 m height to provide full visibility through a glass facade. For the sake of comparison, three additional configurations are included (Figs. 5f-h) where the AC is installed at the bottom end of the wall, so there is no wall downstream of the AC nozzle (Case B).

The computational domain replicates the geometry in Figure 5a, as shown in Figure 6a. The AC is represented in the domain by a solid volume and a velocity inlet represents the discharge nozzle opening with a width $b = 0.2$ m. The wall below the nozzle is modeled as a zero-thickness wall. The domain is divided into three zones (C1, C2, C3) as shown in Figures 5a and 6a in view of generating the computational grid and specifying
fixed pollutant mass fractions at a certain distance from the AC, as outlined in subsection 5.3.3.

5.3.2 Computational grid

A fully structured computational grid is applied. Figures 6b-h display the central part of the “medium” grid consisting of zone C2, which is discretized at a high grid resolution, especially near the AC and the discharge nozzle, where high gradients in mean velocity and turbulent kinetic energy are expected to occur. A grid-sensitivity analysis is conducted for the reference configuration. Therefore, two additional structured grids are created by refining and coarsening the medium grid with a linear refinement factor \( \sqrt{2} \). The grid-sensitivity analysis is performed with the realizable \( k-\varepsilon \) model with EWT based on the validation study in section 5.2. The number of cells over the width of the jet discharge nozzle are 14, 20 and 28 for the coarse, medium and fine grid, respectively. The total cell counts are 70,784 cells, 141,585 cells and 272,436 cells for coarse, medium and fine grid, respectively. Note that a high grid resolution is required given the small size of the jet discharge nozzle width and the anticipated high velocity and concentration gradients near the vertical zero-thickness wall.

5.3.3 Boundary conditions

At the discharge nozzle, a uniform mean velocity profile of 9 m/s is imposed and the discharge turbulence intensity is 5%. The discharge (nozzle) Reynolds number, based on \( b = 0.2 \) m and air at 20°C, is \( Re = 118,773 \). All solid surfaces including the zero-thickness wall below the nozzle are modeled as smooth no-slip walls. At the left and right vertical boundaries of the model, 0.5 Pa\(^2\) and zero static gauge pressure are imposed, respectively, both with a turbulence intensity of 5%.

All simulations are isothermal. In order to calculate the separation efficiency of the AC, a passive pollutant in the left zone is considered. The pollutant concentrations are applied by assigning fixed values of the pollutant mass fraction to the zones C1 (\( Y = 1 \)) and C3 (\( Y = 0 \)), while mixing is allowed to take place in C2.

---

2 Representative of cross-flow velocity equal to 0.9 m/s
Figure 6: (a) Computational domain with indication of dimensions and zones C1 to C3 for grid refinement and boundary conditions. (b-h) View of medium grid for (b) free jet; (c,f) 0.5 m wall; (d,g) 1 m wall; and (d,h) 2 m wall. Thickness of the indication of the wall is not to scale.
5.3.4 Solver settings

The simulations are performed with ANSYS Fluent 15.0.7 [32]. The 2D steady RANS equations are solved in combination with the realizable $k$-$\varepsilon$ model with EWT. An Eulerian advection-diffusion equation is used to model pollutant dispersion, with the turbulent Schmidt number taken equal to $Sc_t = 0.7$ [39]. The SIMPLEC algorithm is utilized for pressure-velocity coupling, pressure interpolation is standard and second-order spatial discretization schemes are used for both the convection terms and the viscous terms of the governing equations. Convergence is assumed to be obtained when the scaled residuals level off and reach a minimum of $10^{-6}$ for $x$ and $y$ momentum, $10^{-5}$ for $k$ and $\varepsilon$, $10^{-4}$ for continuity and $10^{-7}$ for species transport. In addition, velocity magnitudes near the bottom of the opening between the two zones are monitored to assess the solution convergence, which confirmed the presence of oscillatory convergence for Case B, which required averaging of the results over 80,000 iterations [40,41].

5.4 CFD simulations case studies: results

5.4.1 Grid-sensitivity analysis

Profiles of the dimensionless velocity magnitude ($|V|/U_0$), dimensionless turbulent kinetic energy ($k/U_0^2$) and pollutant mass fraction ($\gamma$) for the reference configuration with a free jet are compared along four lines (L1, L2, L3, L4) for the coarse, medium and fine grids. Figures 7b-d show the results along line L1 which intersects the nozzle at mid-width. While the results obtained on the coarse grid show significant differences from those obtained on the medium grid, the results of the medium grid and the fine grid are nearly identical. The same observations are made for lines L2 – L4 (not shown here for the sake of brevity).

Thus it is concluded that the medium grid provides fairly grid-independent results and, therefore, it is used for the CFD simulations of the case studies.
5.4.2 Case A

Figures 8a-d display contours of $|V|/U_0$ for the reference configuration and the three configurations in Case A. For each configuration, as the jet travels towards the floor, the jet centerline velocity decays due to the entrainment of ambient air and the growth of the shear layer at the jet edges. The jet slightly bends into zone C3 with clean air which is mainly caused by the static pressure difference between zones C1 and C3. As the vertical wall below the AC becomes longer, the bending of the jet decreases and it is more vertically oriented, which is attributed to the Coanda effect acting along the wall in combination with the lesser expansion of the jet due to the confinement by the wall. Figures 8c and d show that in these geometries, even beyond the bottom of the wall, the jet still moves a bit further towards the left, after which the bending
Figure 8: Reference case and Case A: (a-d) Contours of dimensionless mean velocity magnitude for (a) free jet; (b) wall of 0.5 m; (c) wall of 1 m; (d) wall of 2 m

towards the right and more into the zone with clean air occurs. As the vertical wall becomes longer, a jet with higher velocity impinges on the floor. This fact, in combination with the decrease in jet bending, causes a larger part of the impinging AC air to move into the room zone with polluted air (C1), compared to the reference configuration.

Figure 9 shows cross-jet profiles of the dimensionless mean $x$-velocity component $U/U_m$, where $U_m$ is the local maximum of the $x$-velocity component, i.e. along a given horizontal line, as a function of the ratio $y/b$. Each figure displays the typical top-hat profile just downstream of the nozzle exit after which entrainment and jet spreading occurs. Figures 9b-d clearly show the presence of the Coanda effect that increases in strength as the wall gets longer. Beyond $x/b = 12.5$, Figure 9a displays the typical, almost symmetrical, shape for the cross-jet profiles, however, in the situations in which a vertical wall is placed, a clear asymmetry emerges in these cross-jet velocity profiles and this asymmetry increases as the wall gets longer (Figs. 9b-d).
Figure 9: Reference configuration and configurations of Case A: Dimensionless mean x-velocity component $U/U_m$ as a function of $y/b$ for (a) free jet; (b) wall of 0.5 m; (c) wall of 1 m; (d) wall of 2 m
Figure 10 presents contours of pollutant mass fractions \( Y \) for (a,c,e,g) AC “on” and (b,d,f,h) AC “off”. Note that the simulations for AC “off” are needed to determine the separation efficiency in section 5.4.5. The behavior of the jet is clearly reflected in the figures with AC “on”. At the nozzle, the jet contains clean air only (Figs. 10a,c,e,g). As discussed in subsection 5.4.1, the bending of the AC towards the clean air zone is largest for the free jet and decreases as the height of the wall increases. This is also reflected in the pollutant mass fraction near the floor of the clean air zone, which is highest for the free jet, and decreases with increasing wall height for the cases with walls (Figs.
10a,c,e,g). Figures 10b,d,f,h with AC “off” depict a situation in which there is no flow from the AC, and in which the pollutant transport is mainly convective due to the pressure difference across the opening.

5.4.3 Case B

Figures 11a-c display contours of $|V|/U_0$ for the three configurations in Case B. For each configuration, no Coanda effect and no wall confinement are acting on the jet flow, nevertheless, also in these configurations bending of the jet towards the right zone is strongly reduced with increasing wall height and lower AC nozzle position. This is caused by the larger momentum of the jet along the opening height when the opening height decreases. As the vertical wall becomes longer, a jet with higher velocity impinges on the floor resulting in stronger horizontal wall jets above the floor after jet impingement.

![Figure 11: Case B: (a-c) Contours of dimensionless mean velocity magnitude for (a) wall of 0.5 m; (b) wall of 1 m; (c) wall of 2 m](image)

Figure 12 shows the cross-jet profiles of $U/U_m$ confirming the reduced bending to the left, moreover it shows that almost no jet bending occurs for a wall height of 2 m (Fig. 12c). A comparison of cross-jet profiles $U/U_m$ of Cases A and B in Figure 13 shows that for the geometries with a 0.5 m wall, 1 m wall and 2 m wall, the jet centerlines of Cases
A and B exhibit very similar bending along the opening height. The differences between the profiles of Cases A and B are largest near the end of the wall, where the results of Case B display the top-hat profile issued from the nozzle exit, while at this downstream position the jets in Case A have already suffered from entrainment in the shear layer (not at the wall-constrained side).

Figure 12: Case B: Dimensionless mean x-velocity component $U/U_m$ as a function of $y/b$ for (a) wall of 0.5 m; (b) wall of 1 m; (c) wall of 2 m

Figure 13: Comparison between Cases A and B: Dimensionless mean x-velocity component $U/U_m$ as a function of $y/b$ for (a) wall of 0.5 m; (b) wall of 1 m; (c) wall of 2 m
Figure 14 presents contours of the pollutant mass fractions (Y). For the situation with AC “on” (Fig. 14a,c,e), the mass fraction contours are almost identical as for case A (AC adjacent to ceiling). For the situation with AC “off” (Fig. 14b,d,f), a clear flow is again present from the left zone to the right zone, which is mainly driven by the imposed pressure difference.

Figure 14: Case B: Contours of pollutant mass fraction for (a,c,e,) air curtain ON; and (b,d,f,) air curtain OFF; for (a,b) 0.5 m wall; (c,d) 1 m wall; (e,f) 2 m wall

5.4.4 Maximum x-velocity component and jet half-width

Figure 15a shows the evolution of the dimensionless cross-jet maximum x-velocity component ($U_m/U_0$). For Case A, Figure 15a shows that $U_m/U_0$ near the bottom of the opening increases with increasing wall height, so that the highest value of $U_m/U_0$ is obtained for a jet with 2 m wall ($U_m/U_0 = 0.83$) and the lowest value of $U_m/U_0$ is obtained for a jet with 0.5 m wall ($U_m/U_0 = 0.67$). Similar findings are obtained for Case B. The highest value of $U_m/U_0$ is obtained for a jet with 2 m wall ($U_m/U_0 = 0.87$) and the lowest
value of $U_m/U_0$ is obtained for a jet with 0.5 m wall ($U_m/U_0 = 0.69$). For the configurations with 0.5 m wall, 1 m wall and 2 m wall, the values of $U_m/U_0$ near the bottom of the opening are higher for Case B than for Case A.

Figures 15b,c show the evolution of the dimensionless jet half-widths ($y_{1/2}/b$) for the reference configuration and for Cases A and B. The evolution of the jet half-width $y_{1/2}$ along the jet centerline characterizes the spreading of the jet. Note that the value of $y_{1/2}$ at a certain position $x/b$ is calculated as the horizontal distance between the jet centerline and the point where the mean $x$-velocity is equal to half the local jet maximum velocity $U_m$ (Fig. 2b). Due to jet bending, the cross-jet velocity profiles of all investigated configurations show asymmetrical distributions (see Figs. 9 and 12). This effect is more noticeable for the configurations of Case A than for Case B due to the presence of the Coanda effect. Therefore, the jet half-widths are defined separately for the left and for the right jet edges. Note that for Case A, due to the presence of the wall, the jet half-widths on the left jet edge are only given after the jet passes the wall. For Case A, the jet spreading at the left edge decreases with increasing wall height. At the right jet edge (Fig. 15c), in the region between the jet discharge nozzle ($x/b = 0.25$) and the distance of 0.5 m from the jet discharge nozzle ($x/b = 2.5$), the jet spreading is nearly identical for all studied wall heights. Starting from $x/b = 2.5$ to $x/b = 10.0$, the jet half-width growth increases with increasing wall height. The lowest values of the jet half-width and hence lowest spreading rates within this region occur for the free jet (reference case). For Case B and at both jet edges, the jet spreading consistently decreases with increasing wall height, i.e. the largest spreading along the whole jet height occurs for the jet with 0.5 m wall. For the geometries with a 0.5 m wall, 1 m wall and 2 m wall, the spreading at the left jet edge is lower for Case A than for Case B. In contrast, the spreading at the right jet edge is lower for Case B than for Case A.

5.4.5 Separation efficiency

The separation efficiency of the AC ($\eta$) is calculated based on the rate of pollutant mass transfer $Q_{ac}$ through the opening (from the left zone to the right zone) with an AC (infiltration) compared to the pollutant mass transfer rate $Q_0$ through the same opening without an AC (e.g. [24,42]):
The modified separation efficiency ($\eta^*$, see [43]) includes mass transfer due to both infiltration and exfiltration and due to the airflow from the AC to the left zone (if AC is “on”) and is calculated as:

$$\eta = 1 - \frac{Q_{ac}}{Q_0}$$ (2)
\[ \eta^* = 1 - \frac{Q_{ac} + Q_{ac}^*}{Q_0 + Q_{0}^*} \]  \hspace{1cm} (3) \]

with \( Q_{ac}^* \) the clean air mass transfer with AC, calculated as the sum of mass transfer by transport of clean air from the right zone through the opening with an AC to the left zone (exfiltration), and the mass transfer by the transport of clean air originating from the AC to the left zone; and \( Q_{0}^* \) the clean air mass transfer by transport of clean air originating from the right zone through the opening without an AC to the left zone (exfiltration). In the studied cases presented in this paper \( Q_{0}^* = 0 \), since there is no exfiltration through the opening without an AC due to the imposed pressure difference.

The transfer of the pollutant to the right zone (infiltration) is computed along line-2 at a distance \( y_2 = 4.0 \) m from the left edge of the discharge nozzle adjacent to the left zone (\( y = 0 \) m) (see Fig. 16), with:

\[ Q_i = \int_0^H Y \rho_a \left( \vec{V} \cdot \vec{n}_2 \right) dx \]  \hspace{1cm} (4) \]

with \( Q_i \) the pollutant mass transfer rate from the left side to the right side (with or without air curtain, i.e. \( Q_{ac} \) or \( Q_{0} \), respectively), \( Y \) the pollutant mass fraction, \( \rho_a \) the air density, \( \vec{V} \) the velocity vector, and \( \vec{n}_2 \) the outward vector normal to line-2, i.e. the vector corresponding to the direction of the pollutant mass transfer \( Q_i \) as indicated in Figure 16. The transfer of clean air from the nozzle exit and from the right zone to the left zone (exfiltration) is computed along line-1 at a distance \( y_1 = -4.0 \) m from the left discharge nozzle edge (\( y = 0 \)) (see Fig. 16) with:

\[ Q_{ac}^* = \int_0^H Y_{cl} \rho_a \left( \vec{V} \cdot \vec{n}_1 \right) dx \]  \hspace{1cm} (5) \]

with \( Y_{cl} \) the clean air mass fraction (\( = 1 - Y \)), and \( \vec{n}_1 \) the outward vector normal to the line-1, i.e. the vector corresponding to the direction of the clean air mass transfer \( Q_{ac}^* \) as
indicated in Figure 16. The distances of $y_1 = -4.0 \text{ m}$ and $y_2 = 4.0 \text{ m}$ are based on a sensitivity analysis with the aim to minimize the influence of the jet flow on the results for species mass transfer.

![Figure 16: Indication of the lines (line-1 and line-2) used for calculation of the separation efficiency $\eta$ and the modified separation efficiency $\eta^*$](image)

Also, the ratio of jet discharge momentum flux ($M_{ac}$) to cross-flow momentum flux ($M_{cf}$), called the momentum flux ratio $\gamma$, is calculated for each considered geometry:

$$\gamma = \frac{M_{ac}}{M_{cf}} = \frac{\rho_d b U_0^2}{\rho_d H_0^2 U_{cf}^2} = \frac{b U_0^2}{H_0 U_{cf}^2}$$  \hspace{1cm} (6)

with $U_{cf}$ the average mean $x$-velocity of the cross-flow through the opening between two zones created by the imposed pressure difference for the case without an AC, $H_0$ the height of the opening, which is defined as a distance from the end (the lowest edge) of the wall to the bottom of the opening ($= 4 \text{ m}, 3.5 \text{ m}, 3 \text{ m} \text{ and } 2 \text{ m}$ for free jet, $0.5 \text{ m}$ wall, $1 \text{ m}$ wall and $2 \text{ m}$ wall, respectively).

Table 3 provides the values of $\gamma$, $Q_{ac}$, $Q'_{ac}$ and $Q_0$ for Case A and B and the corresponding values of the separation efficiencies $\eta$ (Eq. (2)) and $\eta^*$ (Eq. (3)). The lowest value of $\eta$ ($= 0.943$) is obtained for the reference case (free jet). The results for Case A show that pollutant mass transfer $Q_{ac}$ to the right zone decreases with increasing wall height to a larger degree than the decrease of $Q_0$, and, therefore, $\eta$ increases with increasing wall height. The highest value ($\eta = 0.991$) is obtained for a jet with a $2 \text{ m}$ wall and the lowest
(\eta = 0.958) for jet with 0.5 m wall. In contrast, the clean air mass transfer \(Q'_{ac}\) to the left zone increases with increasing wall height. In line with this finding, the momentum flux ratio \(\gamma\), which characterizes the intensity of the jet relative to the cross-flow, increases with increasing wall jet height. The jet with 2 m wall shows the highest value of \(\gamma = 19.9\), while the lowest value is obtained for a free jet (\(\gamma = 5.1\)). As discussed in Section 4.4, the local x-velocity near the floor below the opening increases with increasing wall height. When the jet hits the bottom of the opening at a higher velocity, a large part of the clean air from the jet moves to the left zone. Therefore, \(\eta^*\) decreases with increasing wall height. The highest value is obtained for a free jet (\(\eta^* = 0.726\)) and the lowest value for a jet with 2 m wall (\(\eta^* = 0.332\)).

Table 3: Values of \(Q_{ac}\), \(Q_0\) and \(Q'_{ac}\) and corresponding separation efficiencies \(\eta\) and \(\eta^*\) for the considered cases

<table>
<thead>
<tr>
<th>Reference case</th>
<th>(\gamma = \frac{M_{ac}}{M_{cf}})</th>
<th>(Q_{ac}) (kg/s)</th>
<th>(Q_0) (kg/s)</th>
<th>(Q'_{ac}) (kg/s)</th>
<th>(\eta)</th>
<th>(\eta^*)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Free jet</td>
<td>5.1</td>
<td>0.249</td>
<td>4.340</td>
<td>0.939</td>
<td>0.943</td>
<td>0.726</td>
</tr>
<tr>
<td>Case A 0.5 m wall</td>
<td>6.8</td>
<td>0.152</td>
<td>3.587</td>
<td>0.971</td>
<td>0.958</td>
<td>0.687</td>
</tr>
<tr>
<td>Case A 1 m wall</td>
<td>9.4</td>
<td>0.085</td>
<td>2.846</td>
<td>1.022</td>
<td>0.970</td>
<td>0.611</td>
</tr>
<tr>
<td>Case A 2 m wall</td>
<td>19.9</td>
<td>0.015</td>
<td>1.641</td>
<td>1.081</td>
<td>0.991</td>
<td>0.332</td>
</tr>
<tr>
<td>Case B 0.5 m wall</td>
<td>6.9</td>
<td>0.154</td>
<td>3.616</td>
<td>0.978</td>
<td>0.957</td>
<td>0.687</td>
</tr>
<tr>
<td>Case B 1 m wall</td>
<td>9.2</td>
<td>0.076</td>
<td>2.876</td>
<td>1.036</td>
<td>0.974</td>
<td>0.613</td>
</tr>
<tr>
<td>Case B 2 m wall</td>
<td>19.2</td>
<td>0.011</td>
<td>1.667</td>
<td>1.051</td>
<td>0.993</td>
<td>0.363</td>
</tr>
</tbody>
</table>

As in Case A, \(\eta\) for Case B decreases with decreasing wall height so that the highest value (\(\eta = 0.993\) and corresponding \(\gamma = 19.2\)) is obtained for the jet with 2 m wall and the lowest value (\(\eta = 0.957\) and \(\gamma = 6.9\)) for a jet with 0.5 m wall. As observed for Case A, \(\eta^*\) decreases with increasing wall height: highest for a jet with 0.5 m wall (\(\eta^* = 0.687\)) and lowest for a jet with 2 m wall (\(\eta^* = 0.363\)).

For the geometries with a 1 m wall and 2 m wall, \(\eta\) and \(\eta^*\) show slightly higher values for Case B than for Case A. An exception is present for a wall height of 0.5 m; for this wall height both \(\eta\) and \(\eta^*\) are almost identical for Case A (\(\eta = 0.958, \eta^* = 0.687\)) and Case B (\(\eta = 0.957, \eta^* = 0.687\)).
The following observations hold for the separation efficiencies $\eta$ and $\eta^*$:

- The current jet discharge velocity $U_0$ (= 9 m/s) in combination with the opening height of 4 m (reference configuration, free jet) and the static pressure difference of 0.5 Pa between the two zones leads to a high $\eta$.
- Mounting the AC unit near the ceiling with a certain wall downstream of the nozzle (Case A) increases $\eta$ because of the higher jet momentum (stronger jet) present over the height of the opening. This is caused by the presence of the wall that constrains the jet expansion and therefore limits the decrease of the jet momentum along the opening height.
- Mounting the AC unit lower, with the discharge nozzle at the end of the wall (Case B), also increases $\eta$ because of the reduced opening height along which the jet acts.
- Therefore, the presence of a wall has a positive effect on $\eta$, irrespective of the position along the wall where the AC unit is installed. The differences in $\eta$ for Case A versus B are almost negligible for the configurations under study.
- The current jet discharge velocity $U_0$ (= 9 m/s) in combination with the opening height of 4 m (reference configuration) and the static pressure difference of 0.5 Pa between the two zones leads to a moderate modified separation efficiency $\eta^*$. This is attributed to the fact that a large amount of clean air from the jet is flowing to the left zone (exfiltration). The exfiltration is not taken into account in the definition of $\eta$, only in $\eta^*$.
- Mounting the AC unit near the ceiling with a certain wall downstream of the nozzle (Case A) or mounting the AC unit lower with the discharge nozzle at the end of the wall (Case B) decreases $\eta^*$ even further. This is because the higher jet momentum increases the exfiltration.

These observations give rise to the following practical conclusions:

- The separation efficiency $\eta$ can be preferred over the modified separation efficiency $\eta^*$ in situations where limiting infiltration is the main objective, irrespective of the air curtain energy consumption and of any exfiltration. Conversely, $\eta^*$ can be preferred over $\eta$ when both infiltration and exfiltration are of concern and/or when the energy consumption by the air curtain (heating, cooling, and fan power) are important.
- In situations with a vertical wall above the opening, this wall will act to increase $\eta$ and decrease $\eta^*$, therefore it might be necessary to reduce the jet discharge velocity, which will also allow to reduce the total AC energy consumption without reducing $\eta$ or $\eta^*$ compared to a situation without vertical wall.
5.5 Limitations and further work

The study is subjected to a number of limitations which can incite further work on this topic.

- The study only focused on a single opening height, a single jet discharge nozzle width, a single static pressure difference between the two separated zones and a single jet discharge velocity. A wider parametric study can include different opening heights and jet discharge nozzle widths, and especially – for a given opening height and width – varying discharge velocities, which should allow reaching higher modified separation efficiencies $\eta^*$ than in the present study.

- The study focused on isothermal conditions. Analyzing the influence of the presence of the wall on the (modified) separation efficiency of ACs with non-isothermal conditions between the separated zones and possibly a jet discharge temperature that is also different from that of the separated zones should be addressed in future studies.

- In the present study, a small and constant uniform static pressure difference of 0.5 Pa was imposed between the two zones. In reality, static pressure differences can be larger and can be influenced (in space and time) by wind and/or by building heating, cooling and ventilation systems. Including such external or internal perturbations is a topic for future research.

5.6 Conclusions

A numerical study with 2D steady RANS is carried out to predict the influence of an upstream wall on air curtain (AC) separation efficiencies. The simulations are based on grid-sensitivity analyses and on validation with measurements for a wall jet and a free jet. The situation under study is isothermal, without air recirculation and a small static pressure difference of 0.5 Pa exists between the two separated zones. The effect of different wall heights is analyzed: no wall (reference case), 0.5 m, 1.0 m and 2.0 m wall. For every wall height, a configuration with the AC unit installed against the ceiling (Case A) and a configuration with the AC unit installed near the bottom of the wall (Case B) are considered. The sealing efficiency is analyzed with a passive pollutant in one of the separated zones. Under these conditions, the following main conclusions are obtained:
For both Cases A and B, the presence of the wall leads to reduced jet bending and reduced jet decay. The higher the wall, the lower the jet decay. For Case A this is attributed to (1) the Coanda effect, (2) the fact that jet expansion is partly constrained by the wall, and (3) the fact that the static pressure difference acts over a lower (free) jet height. For Case B, this is attributed to the lower jet height.

The presence of a wall in both Cases A and B increases the separation efficiency $\eta$. The higher the wall, the higher $\eta$. This is due to the fact that a higher wall causes increased jet momentum over the opening height, which reduces infiltration.

The presence of a wall in both Cases A and B decreases the modified separation efficiency $\eta^\star$. The higher the wall, the lower $\eta^\star$. This is due to the fact that a higher wall causes increased jet momentum over the opening height, which increases exfiltration.

The differences in $\eta$ and $\eta^\star$ for Case A versus B are almost negligible for the configurations under study.

The current jet discharge velocity $U_0 (= 9 \text{ m/s})$ in combination with the opening height of 4 m and the static pressure difference of 0.5 Pa between the two zones leads to a moderate separation efficiency $\eta^\star$. This is attributed to the fact that a large amount of clean air from the jet is flowing to the left zone (exfiltration). The exfiltration is not taken into account in the definition of $\eta$, only in $\eta^\star$.

The separation efficiency $\eta$ can be preferred over the modified separation efficiency $\eta^\star$ in situations where limiting infiltration is the main objective, irrespective of the air curtain energy consumption and of any exfiltration. Conversely, $\eta^\star$ can be preferred over $\eta$ when both infiltration and exfiltration are of concern and/or when the energy consumption by the air curtain (heating, cooling, and fan power) is important.

In situations with a vertical wall above the opening, this wall will act to increase $\eta$ and decrease $\eta^\star$, therefore it might be necessary to reduce the jet discharge velocity, which will also allow to reduce the total AC energy consumption without reducing $\eta$ or $\eta^\star$ compared to a situation without vertical wall.

Acknowledgements

Twan van Hooff is currently a postdoctoral fellow of the Research Foundation – Flanders (FWO) and acknowledges its financial support (project FWO 12R9718N). The authors also gratefully acknowledge the partnership with ANSYS CFD.
References


This chapter states the main limitations of the thesis and, accordingly, provides recommendations for further research. A discussion on the main contributions of this thesis concludes this chapter.

6.1 Limitations and future work

The experimental work on plane turbulent impinging jets (PTIJ) presented in this thesis was limited to moderate Reynolds numbers (7,200 – 13,500). Although this range is found in practice and is thus realistic for many air curtain systems (ACs), future studies should include experiments for higher $Re$ values (higher jet discharge velocities) to address PTIJ (and ACs) at higher Reynolds numbers. The work can also be extended with experiments for inclined jets at different discharge angles. Moreover, 3D PIV measurements are suggested to address in detail the influence of sidewalls on the flow development within the potential, intermediate and impingement regions of the current jet configuration, as was done in studies on plane turbulent free jets by e.g. Deo et al. (2007) and Alnahhal and Panidis (2009). Furthermore, future research can focus on the analysis of instantaneous velocity and vorticity fields in order to identify and to analyze the Kelvin-Helmholtz instabilities that dominate the early stages of the jet transition process (from the potential core to the intermediate region).

The CFD study on generic PTIJ in this thesis showed that RANS was not able to accurately predict turbulence levels in the jet shear layer, especially near the inlet region,
and in the impingement region. The accuracy of numerical predictions in these regions would benefit from hybrid unsteady RANS-LES (URANS-LES, e.g. Kubacki et al. 2013) or LES (e.g. Zuckerman and Lior 2006). Moreover, temporal flow features, including jet flapping, development of vortical structures in the shear layers, etc. can be solved using URANS-LES or LES (e.g. Le Song and Prud’homme 2007, van Hooff et al. 2017). Future work can include an elaboration of hybrid URANS-LES models. Their advantage compared to LES is that high-resolution LES grids will be required only in the regions where LES is necessary, i.e. hybrid URANS-LES can couple a URANS model in the near-wall region with LES for the outer flow, reducing the need for a very fine near-wall grid to resolve the small-scale eddies near the wall when a full LES would be performed (e.g. Fröhlich and von Terzi 2008). This approach might therefore be a good compromise between computational demand and increased prediction accuracy.

The parametric study on PTIJs presented in Chapter 4 considered jets at four discharge angles: $\alpha_0 = 0^\circ$, $5^\circ$, $10^\circ$ and $20^\circ$. Future studies can consider more angles in the range of $5^\circ$ to $20^\circ$, and angles larger than $\alpha_0 = 20^\circ$ in order to extend the current investigation on the influence of jet discharge angles on jet separation efficiency and to define the limit of the jet discharge angle in obtaining the highest separation efficiency.

Both parametric studies, presented in Chapter 4 and Chapter 5, considered only one value of cross-jet pressure difference across the jet. Future studies can consider a range of pressure differences and its influence on the values of optimal momentum flux ratios. Moreover, these studies can be extended by analyzing the separation efficiency of ACs when it is subjected to external perturbations such as the wind, infiltration, and forces induced by the ventilation system. Furthermore, all studies in this thesis considered an isothermal situation. Future studies should investigate the influence of non-isothermal conditions between two spaces on jet separation efficiency. These studies can be also extended by assessing the influence of external surroundings (buildings) on jet separation efficiency, taking into account complex outdoor wind speed, turbulence intensity and temperature gradients. Finally, further investigations could be performed with these models to track the particles of the passive pollutant from the left (clean) to the right (polluted) side of the enclosure and to define the streamline patterns near the jet and within the whole domain to better understand the flow interactions between the jet and the cross flow. In addition, the validation of turbulence models could be extended to turbulence quantities and to pollutant concentrations.
6.2 Discussion

Although some limitations and possibilities for future work can be indicated, as discussed above, the studies performed and presented in this thesis are valuable and relevant with respect to both research on PTIJs and ACs.

Firstly, a literature study showed that both detailed experimental data and CFD validation studies on PTIJs at moderate Reynolds numbers and a jet height larger than 10 jet widths are scarce. The experimental study on PTIJs (Chapter 2) provided valuable experimental data for the whole vertical centerplane of the PTIJs at moderate Reynolds numbers. Alongside the information obtained on the jet dynamics, the data will be useful for the validation of numerical simulations (e.g. CFD) of PTIJs and ACs. Moreover, based on the validation study presented in Chapter 3, recommendations were provided with respect to the performance of the different RANS turbulence models for predicting PTIJ flows and assessing general flow characteristics. These recommendations can be relevant, for example, in application cases for prediction of heat and mass transfer through ACs.

Secondly, a literature study showed that most previous studies on ACs are application studies that are very case-specific. There is a clear lack of generic studies on ACs including the analysis of the complex impact of the jet parameters on the AC separation efficiency. The study presented in Chapter 4 provided a parametric analysis of several jet parameters (momentum flux ratio, jet height-to-width ratio, jet discharge angle) and their influence on jet separation efficiency. Moreover, a comparison was given between the momentum flux ratio required to provide the optimal aerodynamic sealing defined from CFD simulations and the analytically defined value based on the so-called deflection modulus (Hayes and Stoecker 1969). Based on the results it was recommended to use the analytical method only for the initial estimation for the AC boundary conditions, while, e.g., CFD simulations were recommended for more accurate estimations.

Finally, the majority of previous studies focused on ACs discharged from nozzles just above the door opening. However, in some cases ACs have to be installed close to the ceiling, i.e. at a substantial distance from the top of the door opening. Chapter 5 presented a generic analysis of the behavior and separation efficiency of ACs exposed to the influence of a vertical wall above the door opening. Moreover, the results
provided in Chapter 4 and Chapter 5 showed that separation efficiency of PTIJs can be improved by adjusting certain parameters of PTIJs that can be relevant in practical applications of ACs and can be used as starting point for both future research and practical AC design and implementation.

References


Chapter 7
Summary and conclusions

7.1 Summary and conclusions

This thesis contains an experimental and numerical study on plane turbulent impinging jets (PTIJs) at moderate Reynolds numbers. The results of this study provide both an analysis of the jet dynamics and an analysis of the influence of a range of jet parameters on jet separation efficiency focused on air curtain (AC) applications.

Chapter 2 of this thesis presented the results of the experimental analysis of PTIJs at moderate Reynolds numbers (7,200 – 13,500) with two height-to-width ratios $h_{jet}/w_{jet} = 22.5$ and 45. A reduced-scale water-filled model was developed to perform flow visualizations and PIV measurements. The flow visualizations showed the presence of cyclic flapping of the impinging jet and that the jet boundaries exhibited turbulent behavior with spanwise vortices developing in the outer region of the jet. The measurement data included time-averaged quantities of velocity, turbulence intensity and Reynolds shear stress measured in the vertical centerplane, as well as instantaneous fields of velocity and vorticity. The results provided information on the jet structure and showed that the PTIJs considered in the present study behaved as a plane turbulent free jet until 70 - 75% of the total jet height. The results showed that the jet at $h_{jet}/w_{jet} = 45$ and $AR = 37.50$ exhibited more intensive entrainment and a faster decay of the centerline velocity for the same value of the jet Reynolds number compared to jet with $h_{jet}/w_{jet} = 22.5$ and $AR = 18.75$. This is in line with the findings by van der Hegge Zijnen (1958) and Deo et al. (2007).

In addition to the aforementioned findings, the obtained results provided a data set suitable for validation of computational fluid dynamics (CFD) simulations in this thesis obtained with five commonly used Reynolds-averaged Navier-Stokes (RANS) turbulence models: (1) standard $k$-$\varepsilon$ model (SKE); (2) realizable $k$-$\varepsilon$ model (RKE); (3) renormalization group $k$-$\varepsilon$ model (RNG); (4) shear stress transport $k$-$\omega$ model (SST) and (5) linear pressure-strain Reynolds stress model (RSM). Chapter 3 presented the results of this
validation study for a PTIJ at $Re = 8,000$ and $Re = 13,000$ with $h_{jet}/w_{jet} = 22.5$. The numerically predicted distributions of mean velocity and turbulent kinetic energy were compared with the experimental data both along the jet centerline and along several cross-jet lines. The results of this study showed that for several of the lines of analysis considered, the differences in results provided by the considered turbulence models were negligibly small. The results showed that the RKE and RNG models accurately predicted the general jet flow patterns and characteristics. Based on these results it was recommended not to use SKE due to erroneous predictions of profiles of turbulent kinetic energy along the jet centerline, especially near the nozzle exit. Furthermore, the differences in turbulence model performance were analyzed and explained by different formulations for the eddy viscosity $\mu_t$ and the production term $G_k$. In addition, steady RANS models are not suitable for basic research focused on the effect of transient flow features (jet flapping, development and advection of vertical structures, etc.), which requires unsteady simulations and preferably hybrid URANS-LES or LES. Moreover, steady RANS models utilizing the Boussinesq eddy viscosity hypothesis for Reynolds stresses are in general less suitable for a detailed analysis of the impingement region due to an overestimation of turbulent kinetic energy in the impingement region. Nonetheless, RANS models are suitable to assess general flow characteristics, such as jet spreading rate, jet decay rate and estimation of the potential core region. These quantities are relevant, for example, in the prediction of heat and mass transfer through ACs in practical applications.

In Chapter 4, the generic situation of an isothermal PTIJ (AC) at $5,000 < Re < 30,000$ separating two environments subjected to a cross-jet pressure gradient was studied using 2D steady RANS CFD simulations. A parametric analysis was performed on the influence of several jet parameters, such as momentum flux ratio, jet height-to-width ratio and jet discharge angle on the jet separation efficiency. This separation efficiency took into account pollutant transfer through the opening as well as the mixing between the jet and the ambient air. For each jet configuration (depending on jet height-to-width ratio and jet discharge angle), the ratio of jet discharge momentum flux to jet cross-flow momentum flux (momentum flux ratio) at which the separation efficiency of an AC reached its optimum value was defined. The results showed that for this particular case and for identical momentum flux ratios, jets with smaller height-to-width ratios provided a higher separation efficiency than jets with larger height-to-width ratios. Moreover, jets with smaller height-to-width ratios reached their maximum separation efficiency at lower optimal momentum flux ratios. Furthermore, it was shown that inclined jets with discharge angles $\alpha_0 = 5^\circ$ and $10^\circ$ provided a higher separation efficiency than straight
Summary and conclusions

(α₀ = 0°) jets and jets with discharge angle α₀ = 20°. Finally, the minimum deflection modulus to prevent breakthrough of the jet and the corresponding minimum momentum flux ratios were obtained based on the conservation of momentum (simplified method) and were compared to the values of optimal momentum flux ratios obtained from CFD simulations. It was concluded that this simplified method can be used for an initial estimation of the AC boundary conditions required to provide aerodynamic sealing in a practical application. However, in order to provide more accurate estimations one should resort to more advanced methods that take into account the jet dynamics and transient environmental conditions for this jet, e.g. CFD simulations with for example RANS modeling.

Finally, Chapter 5 presented a numerical study to predict the influence of an upstream wall on AC separation efficiencies. The analysis was performed by steady isothermal 2D RANS simulations after validation with experimental data from literature for both a plane wall jet and a plane free jet. The effect of different wall heights was analyzed: no wall (reference case), 0.5 m, 1.0 m and 2.0 m wall. For every wall height, a configuration with the AC unit installed against the ceiling (Case A) and a configuration with the AC unit installed near the bottom of the wall (Case B) were considered. The numerical results indicated that for both Cases A and B the presence of the wall leads to reduced jet bending and reduced jet decay. The results showed that the presence of a wall in both Cases A and B increases the separation efficiency η (based on infiltration of pollutant) of the air curtain. This is due to the fact that a higher wall causes increased jet momentum over the opening height, which reduces infiltration. However, the presence of a wall in both Cases A and B decreases the modified separation efficiency η* (based on infiltration and exfiltration). This is due to the fact that a higher wall causes increased jet momentum over the opening height, which increases exfiltration.

From the research presented in this thesis it can be concluded that some of the analyzed steady RANS models (RKE and RNG) are suitable to predict isothermal PTIJ flows and therefore can be used in application studies on ACs. Regarding the separation efficiency of ACs in situations with cross-pressure gradients, the results suggested that the separation efficiency of ACs can be improved by limiting the jet height-to-width ratio, by an inclination of the AC (in this particular case to the outside counteracting the external force on the AC, e.g. wind) or by presence of a wall downstream the AC.
References


Adelya Khayrullina was born in 1987 in Belebey (Russian Federation).

She graduated with honors as Civil and Industrial Engineer from Ufa State Petroleum Technological University in 2009. Her wish to study energy-efficient, ecological and healthy buildings brought her to Eindhoven University of Technology where she enrolled to the master program Building Physics and Services in 2010, supported with a scholarship from the TU/e Talentship Program. She completed these studies with honors in 2012 with a project entitled *Wind energy potential in passages between buildings*, which was supervised by prof.dr.ir. B. Blocken, dr.ir. T. van Hooff and dr. H.A. Harwig.

Her PhD research work was conducted in a collaboration between the Department of the Built Environment (Unit of Building Physics and Services) and the Department of Applied Physics (Turbulence & Vortex Dynamics) at Eindhoven University of Technology (TU/e) under supervision of prof.dr.ir. B. Blocken, prof.dr.ir. G.J.F. van Heijst, and dr.ir. T. van Hooff.

In 2014 she received the Best Paper Award at the 6th International Symposium of Computational Wind Engineering (CWE2014) in Hamburg, Germany, for the CFD study of the aerodynamics of a train passing through an underground railroad platform.

During her PhD studies she was also an active board member of the PhD Network Bouwkunde and a member of the TU/e PhD Council.

Adelya is currently working as a consultant in building physics and sustainability at Cauberg Huygen bv (The Netherlands).
List of publications

1. ISI journal papers

1.1 Published


1.2 Submitted


2. Conference proceedings


Bouwstenen is een publicatiereeks van de Faculteit Bouwkunde, Technische Universiteit Eindhoven. Zij presenteert resultaten van onderzoek en andere activiteiten op het vakgebied der Bouwkunde, uitgevoerd in het kader van deze Faculteit.

Bouwstenen en andere proefschriften van de TU/e zijn online beschikbaar via: https://research.tue.nl/

Kernredactie
MTOZ
Reeds verschenen in de serie
*Bouwstenen*

nr 1
*Elan: A Computer Model for Building Energy Design: Theory and Validation*
Martin H. de Wit
H.H. Driessen
R.M.M. van der Velden

nr 2
*Kwaliteit, Keuzevrijheid en Kosten: Evaluatie van Experiment Klarendal, Arnhem*
J. Smeets
C. le Nobel
M. Broos
J. Frenken
A. v.d. Sanden

nr 3
*Crooswijk: Van ‘Bijzonder’ naar ‘Gewoon’*
Vincent Smit
Kees Noort

nr 4
*Staal in de Woningbouw*
Edwin J.F. Delsing

nr 5
*Mathematical Theory of Stressed Skin Action in Profiled Sheeting with Various Edge Conditions*
Andre W.A.M.J. van den Bogaard

nr 6
*Hoe Berekenbaar en Betrouwbaar is de Coëfficiënt k in x-ksioga en x-ks?*
K.B. Lub
A.J. Bosch

nr 7
*Het Typologisch Gereedschap: Een Verkennende Studie Omtrent Typologie en Omtrent de Aanpak van Typologisch Onderzoek*
J.H. Luiten

nr 8
*Informatievoorziening en Beheerprocessen*
A. Nauta
Jos Smeets (red.)
Helga Fassbinder (projectleider)
Adrie Proveniers
J. v.d. Moosdijk

nr 9
*Strukturering en Verwerking van Tijdgegevens voor de Uitvoering van Bouwwerken*
ir. W.F. Schaefer
P.A. Erkelens

nr 10
*Stedebouw en de Vorming van een Speciale Wetenschap*
K. Doevendans

nr 11
*Informatica en Ondersteuning van Ruimtelijke Besluitvorming*
G.G. van der Meulen

nr 12
*Staal in de Woningbouw, Korrosie-Bescherming van de Begane Grondvloer*
Edwin J.F. Delsing

nr 13
*Een Thermisch Model voor de Berekening van Staalplaatbetonvloeren onder Brandomstandigheden*
A.F. Hamerlinck

nr 14
*De Wijkgedachte in Nederland: Gemeenschapsstreven in een Stedebouwkundige Context*
K. Doevendans
R. Stolzenburg

nr 15
*Diaphragm Effect of Trapezoidally Profiled Steel Sheets: Experimental Research into the Influence of Force Application*
Andre W.A.M.J. van den Bogaard

nr 16
*Versterken met Spuit-Ferrocement: Het Mechanische Gedrag van met Spuit-Ferrocement Versterkte Gewapend Betonbalken*
K.B. Lubir
M.C.G. van Wanroy
nr 17  
De Tractaten van  
Jean Nicolas Louis Durand  
G. van Zeyl

nr 18  
Wonen onder een Plat Dak:  
Drie Opstellen over Enkele  
Vooronderstellingen van de  
Stedebouw  
K. Doevendans

nr 19  
Supporting Decision Making Processes:  
A Graphical and Interactive Analysis of  
Multivariate Data  
W. Adams

nr 20  
Self-Help Building Productivity:  
A Method for Improving House Building  
by Low-Income Groups Applied to Kenya  
1990-2000  
P. A. Erkelens

nr 21  
De Verdeling van Woningen:  
Een Kwestie van Onderhandelen  
Vincent Smit

nr 22  
Flexibiliteit en Kosten in het Ontwerpproces:  
Een Besluitvormingondersteunend Model  
M. Prins

nr 23  
Spontane Nederzettingen Begeleid:  
Voorwaarden en Criteria in Sri Lanka  
Po Hin Thung

nr 24  
Fundamentals of the Design of  
Bamboo Structures  
Oscar Arce-Villalobos

nr 25  
Concepten van de Bouwkunde  
M.F.Th. Bax (red.)  
H.M.G.J. Trum (red.)

nr 26  
Meaning of the Site  
Xiaodong Li

nr 27  
Het Woonmilieu op Begrip Gebracht:  
Een Speurtocht naar de Betekenis van het  
Begrip 'Woonmilieu'  
Jaap Ketelaar

nr 28  
Urban Environment in Developing Countries  
editors:  Peter A. Erkelens  
George G. van der Meulen (red.)

nr 29  
Strategische Plannen voor de Stad:  
Onderzoek en Planning in Drie Steden  
prof.dr. H. Fassbinder (red.)  
H. Rikhof (red.)

nr 30  
Stedebouwkunde en Stadsbestuur  
Piet Beekman

nr 31  
De Architectuur van Djenné:  
Een Onderzoek naar de Historische Stad  
P.C.M. Maas

nr 32  
Conjoint Experiments and Retail Planning  
Harmen Oppewal

nr 33  
Strukturformen Indonesischer Bautechnik:  
Entwicklung Methodischer Grundlagen  
für eine ‘Konstruktive Pattern Language’  
in Indonesien  
Heinz Frick arch. SIA

nr 34  
Styles of Architectural Designing:  
Empirical Research on Working Styles  
and Personality Dispositions  
Anton P.M. van Bakel

nr 35  
Conjoint Choice Models for Urban  
Tourism Planning and Marketing  
Benedict Dellaert

nr 36  
Stedelijke Planvorming als Co-Productie  
Helga Fassbinder (red.)
nr 37  
Design Research in the Netherlands  
editors: R.M. Oxman  
M.F.Th. Bax  
H.H. Achten

nr 38  
Communication in the Building Industry  
Bauke de Vries

nr 39  
Optimaal Dimensioneren van Gelaste Plaatliggers  
J.B.W. Stark  
F. van Pelt  
L.F.M. van Gorp  
B.W.E.M. van Hove

nr 40  
Huisvesting en Overwinning van Armoede  
P.H. Thung  
P. Beekman (red.)

nr 41  
Urban Habitat: The Environment of Tomorrow  
George G. van der Meulen  
Peter A. Erkelens

nr 42  
A Typology of Joints  
John C.M. Olie

nr 43  
Modeling Constraints-Based Choices for Leisure Mobility Planning  
Marcus P. Stemerding

nr 44  
Activity-Based Travel Demand Modeling  
Dick Ettema

nr 45  
Wind-Induced Pressure Fluctuations on Building Facades  
Chris Geurts

nr 46  
Generic Representations  
Henri Achten

nr 47  
Johann Santini Aichel: Architectuur en Ambiguité  
Dirk De Meyer

nr 48  
Concrete Behaviour in Multiaxial Compression  
Erik van Geel

nr 49  
Modelling Site Selection  
Frank Witlox

nr 50  
Ecolemma Model  
Ferdinand Beetstra

nr 51  
Conjoint Approaches to Developing Activity-Based Models  
Donggen Wang

nr 52  
On the Effectiveness of Ventilation  
Ad Roos

nr 53  
Conjoint Modeling Approaches for Residential Group preferences  
Eric Molin

nr 54  
Modelling Architectural Design Information by Features  
Jos van Leeuwen

nr 55  
A Spatial Decision Support System for the Planning of Retail and Service Facilities  
Theo Arentze

nr 56  
Integrated Lighting System Assistant  
Ellie de Groot

nr 57  
Ontwerp Leren, Leren Ontwerpen  
J.T. Boekholt

nr 58  
Temporal Aspects of Theme Park Choice Behavior  
Astrid Kemperman

nr 59  
Ontwerp van een Geïndustrialiseerde Funderingswijze  
Faas Moonen
nr 60
Merlin: A Decision Support System for Outdoor Leisure Planning
Manon van Middelkoop

nr 61
The Aura of Modernity
Jos Bosman

nr 62
Urban Form and Activity-Travel Patterns
Daniëlle Snellen

nr 63
Design Research in the Netherlands 2000
Henri Achten

nr 64
Computer Aided Dimensional Control in Building Construction
Rui Wu

nr 65
Beyond Sustainable Building
editors: Peter A. Erkelens
Sander de Jonge
August A.M. van Vliet
co-editor: Ruth J.G. Verhagen

nr 66
Das Globalrecyclingfähige Haus
Hans Löfflad

nr 67
Cool Schools for Hot Suburbs
René J. Dierkx

nr 68
A Bamboo Building Design Decision Support Tool
Fitri Mardjono

nr 69
Driving Rain on Building Envelopes
Fabien van Mook

nr 70
Heating Monumental Churches
Henk Schellen

nr 71
Van Woningverhuurder naar Aanbieder van Woongenot
Patrick Dogge

nr 72
Moisture Transfer Properties of Coated Gypsum
Emile Goossens

nr 73
Plybamboo Wall-panels for Housing
Guillermo E. González-Beltrán

nr 74
The Future Site-Proceedings
Ger Maas
Frans van Gassel

nr 75
Radon transport in Autoclaved Aerated Concrete
Michel van der Pal

nr 76
The Reliability and Validity of Interactive Virtual Reality Computer Experiments
Amy Tan

nr 77
Measuring Housing Preferences Using Virtual Reality and Belief Networks
Maciej A. Orzechowski

nr 78
Computational Representations of Words and Associations in Architectural Design
Nicole Segers

nr 79
Measuring and Predicting Adaptation in Multidimensional Activity-Travel Patterns
Chang-Hyeon Joh

nr 80
Strategic Briefing
Fayez Al Hassan

nr 81
Well Being in Hospitals
Simona Di Cicco

nr 82
Solares Bauen: Implementierungs- und Umsetzungs- Aspekte in der Hochschulausbildung in Österreich
Gerhard Schuster
nr 83  
Supporting Strategic Design of Workplace Environments with Case-Based Reasoning  
Shauna Mallory-Hill

nr 84  
ACCEL: A Tool for Supporting Concept Generation in the Early Design Phase  
Maxim Ivashkov

nr 85  
Brick-Mortar Interaction in Masonry under Compression  
Ad Vermeltfoort

nr 86  
Zelfredzaam Wonen  
Guus van Vliet

nr 87  
Een Ensemble met Grootstedelijke Allure  
Jos Bosman  
Hans Schippers

nr 88  
On the Computation of Well-Structured Graphic Representations in Architectural Design  
Henri Achten

nr 89  
De Evolutie van een West-Afrikaanse Vernaculaire Architectuur  
Wolf Schijns

nr 90  
ROMBO Tactiek  
Christoph Maria Ravesloot

nr 91  
External Coupling between Building Energy Simulation and Computational Fluid Dynamics  
Ery Djunaedy

nr 92  
Design Research in the Netherlands 2005  
editors: Henri Achten  
Kees Dorst  
Pieter Jan Stappers  
Bauke de Vries

nr 93  
Ein Modell zur Baulichen Transformation  
Jalil H. Saber Zaimian

nr 94  
Human Lighting Demands: Healthy Lighting in an Office Environment  
Myriam Aries

nr 95  
A Spatial Decision Support System for the Provision and Monitoring of Urban Greenspace  
Claudia Pelizaro

nr 96  
Leren Creëren  
Adri Proveniers

nr 97  
Simlandscape  
Rob de Waard

nr 98  
Design Team Communication  
Ad den Otter

nr 99  
Humaan-Ecologisch Georiënteerde Woningbouw  
Juri Czabanowski

nr 100  
Hambase  
Martin de Wit

nr 101  
Sound Transmission through Pipe Systems and into Building Structures  
Susanne Bron-van der Jagt

nr 102  
Het Bouwkundig Contrapunt  
Jan Francis Boelen

nr 103  
A Framework for a Multi-Agent Planning Support System  
Dick Saarloos

nr 104  
Bracing Steel Frames with Calcium Silicate Element Walls  
Bright Mweene Ng’andu

nr 105  
Naar een Nieuwe Houtskeletbouw  
F.N.G. De Medts
nr 106 and 107

Niet gepubliceerd

nr 108

Geborgenheid
T.E.L. van Pinxteren

nr 109

Modelling Strategic Behaviour in Anticipation of Congestion
Qi Han

nr 110

Reflecties op het Woondomein
Fred Sanders

nr 111

On Assessment of Wind Comfort by Sand Erosion
Gábor Dezsö

nr 112

Bench Heating in Monumental Churches
Dionne Limpens-Neilen

nr 113

RE. Architecture
Ana Pereira Roders

nr 114

Toward Applicable Green Architecture
Usama El Fiky

nr 115

Knowledge Representation under Inherent Uncertainty in a Multi-Agent System for Land Use Planning
Liying Ma

nr 116

Integrated Heat Air and Moisture Modeling and Simulation
Jos van Schijndel

nr 117

Concrete Behaviour in Multiaxial Compression
J.P.W. Bongers

nr 118

The Image of the Urban Landscape
Ana Moya Pellitero

nr 119

The Self-Organizing City in Vietnam
Stephanie Geertman

nr 120

A Multi-Agent Planning Support System for Assessing Externalities of Urban Form Scenarios
Rachel Katoshevski-Cavari

nr 121

Den Schulbau Neu Denken, Fühlen und Wollen
Urs Christian Maurer-Dietrich

nr 122

Peter Eisenman Theories and Practices
Bernhard Kormoss

nr 123

User Simulation of Space Utilisation
Vincent Tabak

nr 124

In Search of a Complex System Model
Oswald Devisch

nr 125

Lighting at Work: Environmental Study of Direct Effects of Lighting Level and Spectrum on Psycho-Physiological Variables
Grazyna Górnicka

nr 126

Flanking Sound Transmission through Lightweight Framed Double Leaf Walls
Stefan Schoenwald

nr 127

Bounded Rationality and Spatio-Temporal Pedestrian Shopping Behavior
Wei Zhu

nr 128

Travel Information: Impact on Activity Travel Pattern
Zhongwei Sun

nr 130

Co-Simulation for Performance Prediction of Innovative Integrated Mechanical Energy Systems in Buildings
Marija Trćka

nr 131

Niet gepubliceerd
nr 132
Architectural Cue Model in Evacuation Simulation for Underground Space Design
Chengyu Sun

nr 133
Uncertainty and Sensitivity Analysis in Building Performance Simulation for Decision Support and Design Optimization
Christina Hopfe

nr 134
Facilitating Distributed Collaboration in the AEC/FM Sector Using Semantic Web Technologies
Jacob Beetz

nr 135
Circumferentially Adhesive Bonded Glass Panes for Bracing Steel Frame in Façades
Edwin Huveners

nr 136
Influence of Temperature on Concrete Beams Strengthened in Flexure with CFRP
Ernst-Lucas Klamer

nr 137
Sturen op Klantwaarde
Jos Smeets

nr 138
Lateral Behavior of Steel Frames with Discretely Connected Precast Concrete Infill Panels
Paul Teewen

nr 139
Integral Design Method in the Context of Sustainable Building Design
Perica Savanović

nr 140
Household Activity-Travel Behavior: Implementation of Within-Household Interactions
Renni Anggraini

nr 141
Design Research in the Netherlands 2010
Henri Achten

nr 142
Modelling Life Trajectories and Transport Mode Choice Using Bayesian Belief Networks
Marloes Verhoeven

nr 143
Assessing Construction Project Performance in Ghana
William Gyadu-Asiedu

nr 144
Empowering Seniors through Domotic Homes
Masi Mohammadi

nr 145
An Integral Design Concept for Ecological Self-Compacting Concrete
Martin Hunger

nr 146
Governing Multi-Actor Decision Processes in Dutch Industrial Area Redevelopment
Erik Blokhuis

nr 147
A Multifunctional Design Approach for Sustainable Concrete
Götz Hüsken

nr 148
Quality Monitoring in Infrastructural Design-Build Projects
Ruben Favié

nr 149
Assessment Matrix for Conservation of Valuable Timber Structures
Michael Abels

nr 150
Co-simulation of Building Energy Simulation and Computational Fluid Dynamics for Whole-Building Heat, Air and Moisture Engineering
Mohammad Mirsadeghi

nr 151
External Coupling of Building Energy Simulation and Building Element Heat, Air and Moisture Simulation
Daniel Cóstola
Adaptive Decision Making In Multi-Stakeholder Retail Planning
Ingrid Janssen

Landscape Generator
Kymo Slager

Constraint Specification in Architecture
Remco Niemeijer

A Need-Based Approach to Dynamic Activity Generation
Linda Nijland

Modeling Office Firm Dynamics in an Agent-Based Micro Simulation Framework
Gustavo Garcia Manzato

Lightweight Floor System for Vibration Comfort
Sander Zegers

Aanpasbaarheid van de Draagstructuur
Roel Gijsbers

'Village in the City' in Guangzhou, China
Yanliu Lin

Climate Risk Assessment in Museums
Marco Martens

Social Activity-Travel Patterns
Pauline van den Berg

Sound Concentration Caused by Curved Surfaces
Martijn Vercammen

Design of Environmentally Friendly Calcium Sulfate-Based Building Materials: Towards an Improved Indoor Air Quality
Qingliang Yu

Beyond Uniform Thermal Comfort on the Effects of Non-Uniformity and Individual Physiology
Lisje Schellen

Sustainable Residential Districts
Gaby Abdalla

Towards a Performance Assessment Methodology using Computational Simulation for Air Distribution System Designs in Operating Rooms
Mônica do Amaral Melhado

Strategic Decision Modeling in Brownfield Redevelopment
Brano Glumac

Pamela: A Parking Analysis Model for Predicting Effects in Local Areas
Peter van der Waerden

A Vision Driven Wayfinding Simulation-System Based on the Architectural Features Perceived in the Office Environment
Qunli Chen

Measuring Mental Representations Underlying Activity-Travel Choices
Oliver Horeni

Modelling the Effects of Social Networks on Activity and Travel Behaviour
Nicole Ronald

Uncertainty Propagation and Sensitivity Analysis Techniques in Building Performance Simulation to Support Conceptual Building and System Design
Christian Struck

Numerical Modeling of Micro-Scale Wind-Induced Pollutant Dispersion in the Built Environment
Pierre Gousseau
<table>
<thead>
<tr>
<th>nr 175</th>
<th>Modeling Recreation Choices over the Family Lifecycle</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Anna Beatriz Grigolon</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 176</th>
<th>Experimental and Numerical Analysis of Mixing Ventilation at Laminar, Transitional and Turbulent Slot Reynolds Numbers</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Twan van Hooff</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 177</th>
<th>Collaborative Design Support: Workshops to Stimulate Interaction and Knowledge Exchange Between Practitioners</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Emile M.C.J. Quanjel</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 178</th>
<th>Future-Proof Platforms for Aging-in-Place</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Michiel Brink</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 179</th>
<th>Motivate: A Context-Aware Mobile Application for Physical Activity Promotion</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Yuzhong Lin</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 180</th>
<th>Experience the City: Analysis of Space-Time Behaviour and Spatial Learning</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Anastasia Moiseeva</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 181</th>
<th>Unbonded Post-Tensioned Shear Walls of Calcium Silicate Element Masonry</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Lex van der Meer</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 182</th>
<th>Construction and Demolition Waste Recycling into Innovative Building Materials for Sustainable Construction in Tanzania</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Mwita M. Sabai</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 183</th>
<th>Durability of Concrete with Emphasis on Chloride Migration</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Przemysław Spiesz</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 184</th>
<th>Computational Modeling of Urban Wind Flow and Natural Ventilation Potential of Buildings</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Rubina Ramponi</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 185</th>
<th>A Distributed Dynamic Simulation Mechanism for Buildings Automation and Control Systems</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Azzedine Yahiaoui</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 186</th>
<th>Modeling Cognitive Learning of Urban Networks in Daily Activity-Travel Behavior</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Şehnaz Cenani Durmazoğlu</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 187</th>
<th>Functionality and Adaptability of Design Solutions for Public Apartment Buildings in Ghana</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Stephen Agyefi-Mensah</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 188</th>
<th>A Construction Waste Generation Model for Developing Countries</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Lilliana Abarca-Guerrero</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 189</th>
<th>Synchronizing Networks: The Modeling of Supernetworks for Activity-Travel Behavior</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Feixiong Liao</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 190</th>
<th>Time and Money Allocation Decisions in Out-of-Home Leisure Activity Choices</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Gamze Zeynep Dane</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 191</th>
<th>How to Measure Added Value of CRE and Building Design</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Rianne Appel-Meulenbroek</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 192</th>
<th>Secondary Materials in Cement-Based Products: Treatment, Modeling and Environmental Interaction</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Miruna Florea</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nr 193</th>
<th>Concepts for the Robustness Improvement of Self-Compacting Concrete: Effects of Admixtures and Mixture Components on the Rheology and Early Hydration at Varying Temperatures</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Wolfram Schmidt</td>
</tr>
</tbody>
</table>
nr 194 Modelling and Simulation of Virtual Natural Lighting Solutions in Buildings
Rizki A. Mangkuto

nr 195 Nano-Silica Production at Low Temperatures from the Dissolution of Olivine - Synthesis, Tailoring and Modelling
Alberto Lazaro Garcia

nr 196 Building Energy Simulation Based Assessment of Industrial Halls for Design Support
Bruno Lee

nr 197 Computational Performance Prediction of the Potential of Hybrid Adaptable Thermal Storage Concepts for Lightweight Low-Energy Houses
Pieter-Jan Hoes

nr 198 Application of Nano-Silica in Concrete
George Quercia Bianchi

nr 199 Dynamics of Social Networks and Activity Travel Behaviour
Fariya Sharmeen

nr 200 Building Structural Design Generation and Optimisation including Spatial Modification
Juan Manuel Davila Delgado

nr 201 Hydration and Thermal Decomposition of Cement/Calcium-Sulphate Based Materials
Ariën de Korte

nr 202 Republiek van Beelden: De Politieke Werkingen van het Ontwerp in Regionale Planvorming
Bart de Zwart

nr 203 Effects of Energy Price Increases on Individual Activity-Travel Repertoires and Energy Consumption
Dujuan Yang

nr 204 Geometry and Ventilation: Evaluation of the Leeward Sawtooth Roof Potential in the Natural Ventilation of Buildings
Jorge Isaac Perén Montero

nr 205 Computational Modelling of Evaporative Cooling as a Climate Change Adaptation Measure at the Spatial Scale of Buildings and Streets
Hamid Montazeri

nr 206 Local Buckling of Aluminium Beams in Fire Conditions
Ronald van der Meulen

nr 207 Historic Urban Landscapes: Framing the Integration of Urban and Heritage Planning in Multilevel Governance
Loes Veldpaus

nr 208 Sustainable Transformation of the Cities: Urban Design Pragmatics to Achieve a Sustainable City
Ernesto Antonio Zumelzu Scheel

nr 209 Development of Sustainable Protective Ultra-High Performance Fibre Reinforced Concrete (UHPFRC): Design, Assessment and Modeling
Rui Yu

nr 210 Uncertainty in Modeling Activity-Travel Demand in Complex Urban Systems
Soora Rasouli

nr 211 Simulation-based Performance Assessment of Climate Adaptive Greenhouse Shells
Chul-sung Lee

nr 212 Green Cities: Modelling the Spatial Transformation of the Urban Environment using Renewable Energy Technologies
Saleh Mohammadi
A Bounded Rationality Model of Short and Long-Term Dynamics of Activity-Travel Behavior
Ifigeneia Psarra

Effects of Pricing Strategies on Dynamic Repertoires of Activity-Travel Behaviour
Elaheh Khademi

Handstorm Principles for Creative and Collaborative Working
Frans van Gassel

Light Conditions in Nursing Homes: Visual Comfort and Visual Functioning of Residents
Marianne M. Sinoo

Woonsporen: De Sociale en Ruimtelijke Biografie van een Stedelijk Bouwblok in de Amsterdamse Transvaalbuurt
Hüseyin Hüsnu Yegenoglu

Studies on User Control in Ambient Intelligent Systems
Berent Willem Meerbeek

Daily Livings in a Smart Home: Users’ Living Preference Modeling of Smart Homes
Erfaneh Allameh

Smart Home Design: Spatial Preference Modeling of Smart Homes
Mohammadali Heidari Jozam

Wonen: Discoursen, Praktijken, Perspectieven
Jos Smeets

Personal Control over Indoor Climate in Offices: Impact on Comfort, Health and Productivity
Atze Christiaan Boerstra

Personalized Route Finding in Multimodal Transportation Networks
Jianwe Zhang

The Design of an Adaptive Healing Room for Stroke Patients
Elke Daemen

Experimental and Numerical Analysis of Climate Change Induced Risks to Historic Buildings and Collections
Zara Huijbregts

Wind Flow Modeling in Urban Areas Through Experimental and Numerical Techniques
Alessio Ricci

Clever Climate Control for Culture: Energy Efficient Indoor Climate Control Strategies for Museums Respecting Collection Preservation and Thermal Comfort of Visitors
Rick Kramer

Fatigue Life Estimation of Metal Structures Based on Damage Modeling
Sarmediran Silitonga

A multi-agents and occupancy based strategy for energy management and process control on the room-level
Timilehin Moses Labeodan

Environmental assessment of Building Integrated Photovoltaics: Numerical and Experimental Carrying Capacity Based Approach
Michiel Ritzen

Performance of Admixture and Secondary Minerals in Alkali Activated Concrete: Sustaining a Concrete Future
Arno Keulen
| nr 232 | World Heritage Cities and Sustainable Urban Development: Bridging Global and Local Levels in Monitoring the Sustainable Urban Development of World Heritage Cities |
| nr 233 | Stage Acoustics and Sound Exposure in Performance and Rehearsal Spaces for Orchestras: Methods for Physical Measurements |
| nr 234 | Municipal Solid Waste Incineration (MSWI) Bottom Ash: From Waste to Value Characterization, Treatments and Application |
| nr 235 | Large Eddy Simulations Applied to Wind Loading and Pollutant Dispersion |
| nr 236 | Alkali Activated Slag-Fly Ash Binders: Design, Modeling and Application |
| nr 237 | Sodium Carbonate Activated Slag: Reaction Analysis, Microstructural Modification & Engineering Application |
| nr 238 | Shopping Behavior in Malls |
| nr 239 | Smart Grid-Building Energy Interactions: Demand Side Power Flexibility in Office Buildings |
| nr 240 | Modeling Taxis Dynamic Behavior in Uncertain Urban Environments |
| nr 241 | Gap-Theoretical Analyses of Residential Satisfaction and Intention to Move |
| nr 242 | Travel Satisfaction and Subjective Well-Being: A Behavioral Modeling Perspective |
| nr 243 | Building Energy Modelling to Support the Commissioning of Holistic Data Centre Operation |
| nr 244 | Regret-Based Travel Behavior Modeling: An Extended Framework |
| nr 245 | Towards Robust Low-Energy Houses: A Computational Approach for Performance Robustness Assessment using Scenario Analysis |
| nr 246 | Development of sustainable and functionalized inorganic binder-biofiber composites |
| nr 247 | A Multiscale Analysis of the Urban Heat Island Effect: From City Averaged Temperatures to the Energy Demand of Individual Buildings |
| nr 248 | Design Method for Adaptive Daylight Systems for buildings covered by large (span) roofs |
| nr 249 | Hardening, high-temperature resistance and acid resistance of one-part geopolymers |
nr 250
Effects of the built environment on dynamic repertoires of activity-travel behaviour
Aida Pontes de Aquino

nr 251
Modeling for auralization of urban environments: Incorporation of directivity in sound propagation and analysis of a framework for auralizing a car pass-by
Fotis Georgiou

nr 252
Wind Loads on Heliostats and Photovoltaic Trackers
Andreas Pfahl

nr 253
Approaches for computational performance optimization of innovative adaptive façade concepts
Roel Loonen

nr 254
Multi-scale FEM-DEM Model for Granular Materials: Micro-scale boundary conditions, Statics, and Dynamics
Jiadun Liu

nr 255
Bending Moment - Shear Force Interaction of Rolled I-Shaped Steel Sections
Rianne Willie Adriana Dekker

nr 256
Paralympic tandem cycling and hand-cycling: Computational and wind tunnel analysis of aerodynamic performance
Paul Fionn Mannion

nr 257
Experimental characterization and numerical modelling of 3D printed concrete: Controlling structural behaviour in the fresh and hardened state
Robert Johannes Maria Wolfs

nr 258
Requirement checking in the building industry: Enabling modularized and extensible requirement checking systems based on semantic web technologies
Chi Zhang

nr 259
A Sustainable Industrial Site Redevelopment Planning Support System
Tong Wang

nr 260
Efficient storage and retrieval of detailed building models: Multi-disciplinary and long-term use of geometric and semantic construction information
Thomas Ferdinand Krijnen

nr 261
The users' value of business center concepts for knowledge sharing and networking behavior within and between organizations
Minou Weijjs-Perrée

nr 262
Characterization and improvement of aerodynamic performance of vertical axis wind turbines using computational fluid dynamics (CFD)
Abdolrahim Rezaeiha

nr 263
In-situ characterization of the acoustic impedance of vegetated roofs
Chang Liu

nr 264
Occupancy-based lighting control: Developing an energy saving strategy that ensures office workers' comfort
Christel de Bakker

nr 265
Stakeholders-Oriented Spatial Decision Support System
Cahyono Susetyo

nr 266
Climate-induced damage in oak museum objects
Rianne Aleida Luimes

nr 267
Towards individual thermal comfort: Model predictive personalized control of heating systems
Katarina Katic
nr 268  
Modelling and Measuring Quality of Urban Life: Housing, Neighborhood, Transport and Job  
Lida Aminian

nr 269  
Optimization of an aquifer thermal energy storage system through integrated modelling of aquifer, HVAC systems and building  
Basar Bozkaya

nr 270  
Numerical modeling for urban sound propagation: developments in wave-based and energy-based methods  
Raúl Pagán Muñoz

nr 271  
Lighting in multi-user office environments: improving employee wellbeing through personal control  
Sanae van der Vleuten-Chraibi

nr 272  
A strategy for fit-for-purpose occupant behavior modelling in building energy and comfort performance simulation  
Isabella I. Gaetani dell’Aquila d’Aragona

nr 273  
Een architectuurhistorische waardestelling van naoorlogse woonwijken in Nederland: Het voorbeeld van de Westelijke Tuinsteden in Amsterdam  
Eleonore Henriette Marie Mens

nr 274  
Job-Housing Co-Dependent Mobility Decisions in Life Trajectories  
Jia Guo

nr 275  
A user-oriented focus to create healthcare facilities: decision making on strategic values  
Emilia Rosalia Catharina Maria Huisman
Plane turbulent impinging jets (PTIJs) at moderate Reynolds numbers are used in a wide range of applications. A typical example in building engineering is an air curtain (AC), which can be used to separate two environments in terms of heat and mass transfer, for example, at entrances of buildings to reduce heat losses, in clean rooms and hospitals to prevent contamination and inside buildings and tunnels to prevent smoke propagation between two or more zones.

This thesis provides high-quality experimental data for a detailed analysis of the fluid dynamics of PTIJs. The experimental data are also used to validate steady Reynolds-averaged Navier-Stokes (RANS) computational fluid dynamics (CFD) simulations of PTIJ flows. The validated model is applied in a generic study in which an isothermal jet separates two environments subjected to a cross-jet pressure gradient. The results of this study provide insights into the separation efficiency of PTIJs and the sensitivity of the separation efficiency to jet height-to-width ratio, jet angle and the ratio of jet discharge momentum flux to jet cross-flow momentum flux (momentum flux ratio). Finally, the separation efficiency of ACs is analyzed for the situation where an AC is subjected to the influence of a vertical wall above the door opening.

Recommendations are provided with respect to the performance of the different RANS turbulence models for predicting PTIJ flows. It is shown that the separation efficiency of PTIJs can be improved by adjusting certain parameters of PTIJs. These recommendations and findings can be relevant in practical applications of ACs and can be used for both future research and AC design and implementation.