Linköping Studies in Science and Technology
Dissertation No. 1639

A NEAR-FIELD STUDY OF MULTIPLE INTERACTING JETS: CONFLUENT JETS

SHAHRIAR GHAHREMANIAN

Linköping University
INSTITUTE OF TECHNOLOGY

Division of Energy Systems
Department of Management and Engineering
Linköping University, SE-581 83 Linköping, Sweden

2014

Copyright © Shahriar Ghahremanian, “Unless otherwise noted”.

“A Near-Field Study of Multiple Interacting Jets: Confluent Jets”

Linköping Studies in Science and Technology, Dissertation No. 1639

ISBN: 978-91-7519-161-4

ISSN: 0345-7524

Printed in Sweden by LiU-Tryck, Linköping, 2014.

Distributed by:

Linköping University

Department of Management and Engineering

SE-581 83 Linköping, Sweden

Tel: +46 13 281000

Abstract

This thesis deals with the near-field behavior of confluent jets, which can be of interest in many engineering applications such as design of a ventilation supply device. The physical effect of interaction between multiple closely spaced jets is studied using experimental and numerical methods. The primary aim of this study is to obtain a better understanding of flow and turbulence behavior of multiple interacting jets and is to gain an insight into the confluence of jets occurring in the near-field of multiple interacting jets.

The array of multiple interacting jets is studied when they are placed on a flat and a curved surface. To obtain the boundary conditions at the nozzle exits of the confluent jets on a curved surface, the results of numerical prediction of a cylindrical air supply device using two turbulence models (realizable $k - \epsilon$ and Reynolds stress model) are validated with hot-wire anemometry (HWA) near different nozzles discharge in the array. A single round jet is then studied to find the appropriate turbulence models for the prediction of the three-dimensional flow field and to gain an understanding of the effect of the boundary conditions predicted at the nozzle inlet. In comparison with HWA measurements, the turbulence models with low Reynolds correction ($k - \epsilon$ and shear stress transport [SST] $k - \omega$) give reasonable flow predictions for the single round jet with the prescribed inlet boundary conditions, while the transition models ($k - kl - \omega$ and transition SST $k - \omega$) are unable to predict the flow in the turbulent region. The results of numerical prediction (low Reynolds SST $k - \omega$ model) using the prescribed inlet boundary conditions agree well with the HWA measurement in the near-field of confluent jets on a curved surface, except in the merging region.

Instantaneous velocity measurements are performed by laser Doppler anemometry (LDA) and particle image velocimetry (PIV) in two different configurations, i.e., a single row of parallel coplanar jets and an inline array of jets on a flat surface. The results of LDA and PIV are compared, which exhibit good agreement except near the nozzle exits.

The streamwise velocity profile of the jets in the initial region shows a saddle back shape with attenuated turbulence in the core region and two off-centered narrow peaks. When confluent jets issue from an array of closely

spaced nozzles, they may converge, merge, and combine after a certain distance downstream of the nozzle edge. The deflection plays a salient role for the multiple interacting jets (except in the single row configuration), where all the jets are converged towards the center of the array. The jet position, such as central, side and corner jets, significantly influences the development features of the jets, such as velocity decay and lateral displacement. The flow field of confluent jets exhibits asymmetrical distributions of Reynolds stresses around the axis of the jets and highly anisotropic turbulence. The velocity decays slower in the combined region of confluent jets than a single jet. Using the response surface methodology, the correlations between characteristic points (merging and combined points) and the statistically significant terms of the three design factors (inlet velocity, spacing between the nozzles and diameter of the nozzles) are determined for the single row of coplanar parallel jets. The computational parametric study of the single row configuration shows that spacing has the greatest impact on the near-field characteristics.

Sammanfattning

Denna avhandling behandlar närområdet av sammanverkande jetstrålar (så kallade confluent jets), som kan vara av intresse för en rad tekniska tillämpningar, såsom utformning av ett ventilationsdon. De fysikaliska effekterna av samverkan mellan flera tätt placerade jetstrålar studeras med hjälp av olika experimentella och numeriska metoder. Det primära syftet med denna studie är att belysa en bättre förståelse för flödet och turbulens beteende av flera samverkande jetstrålar. Det huvudsakliga målet är att få en inblick i hur jetstrålarna flyter samman i närområdet hos flera samverkande strålar.

Matrisen med flera samverkande strålar studeras när de placeras på en plan eller en krökt yta. För att erhålla randvillkor vid dysans utlopp för de sammanflytande jetstrålarna på en krökt yta, valideras resultaten av numeriska beräkningar för ett cylindriskt ventilationsdon med hjälp av två turbulensmodeller (realizable $k - \epsilon$ och Reynolds stressmodell) med mätningar med varmtrådsanemometer (HWA) från olika dysor nära utloppet i uppställningen. En ensam rund stråle studeras sedan för att finna lämpliga turbulensmodeller som kan förutsäga det tredimensionella flödesfältet och förstå effekten av de randvillkor som predikteras vid dysans utlopp. I jämförelse med HWA mätningar, ger turbulensmodeller med låg Reynoldskorrigerig ($k - \epsilon$ och shear stress transport [SST] $k - \omega$) rimliga flödesprognoser för en enstaka runda strålar med de givna ingående randvillkoren, medan övergångsmodeller ($k - kl - \omega$ och SST $k - \omega$) inte kan förutsäga flödet i den turbulenta regionen. Resultaten av numeriska simuleringar (låg Reynolds SST $k - \omega$ modell), med användning av de givna inloppsvillkoren överensstämmer väl med HWA-mätningen i närområdet av sammanfallande jetstrålar på en krökt yta, utom i det sammanfallande området.

Momentana hastighetsmätningar görs med laser Doppler anemometri (LDA) och particle image velocimetry (PIV) i två olika konfigurationer; en enda rad med parallella och en matris av jetstrålar; båda på en plan yta. Resultaten av LDA och PIV jämförs, och visar en god överensstämmelse förutom mycket nära munstyckena.

Strålarnas hastighetsprofil i det initiala området är sadelformad med svag turbulens i strålens kärnområde och två ocentrerade smala toppar i utkanten

av strålen. När samverkande jetstrålar utgår från en uppsättning av tätt placerade dysor kan de konvergera, gå ihop, och förenas på ett visst avstånd nedströms från dysorna. Avböjningen spelar en framträdande roll för flera samverkande strålar utom i enradskonfigurationen, där alla strålarna inte konvergerar mot mitten. Strålens läge i matrisen, t.ex. centrala, sido och hörnstrålar, påverkar kraftigt utvecklingsfunktionerna i strålarna, till exempel maxhastigheten och strålarnas bredd. Jetstrålarna uppvisar en asymmetrisk fördelning av Reynolds spänningar runt strålens mittlinje och kraftigt anisotropisk turbulens. Hastigheten avtar långsammare i det kombinerade området av samverkande jetstrålar i jämfört med en enda separat stråle. Med hjälp av svarsyta metodik, bestäms sambanden mellan karakteristiska punkter (de interaktion och kombinerade punkterna) och statistisk signifikanta termer för tre designfaktorer (inloppshastighet, avstånd mellan dysorna och dysornas diameter) för enda rad av parallella strålar på samma plan. Den parametriska beräkningen av enradiga konfigurationen visar att avståndet mellan dysorna har störst inverkan på närområdets egenskaper.

List of publications

The present doctoral dissertation is based on the following papers:

Paper I: Ghahremanian, S., Moshfegh, B., 2014. A study on proximal region of low Reynolds confluent jets Part I: Evaluation of turbulence models in prediction of inlet boundary conditions. *ASHRAE Transactions* 120, Part 1, NY-14-021, 256-270.

Paper II: Ghahremanian, S., Moshfegh, B., 2013. Evaluation of RANS Models in Predicting Low Reynolds, Free, Turbulent Round Jet. *Journal of Fluids Engineering* 136, 011201.

Paper III: Ghahremanian, S., Svensson, K., Tummers, M.J., Moshfegh, B., 2014. Near-field development of a row of round jets at low Reynolds numbers. *Experiments in Fluids* 55, 1789.

Paper IV: Ghahremanian, S., Moshfegh, B., 2014. A study on proximal region of low Reynolds confluent jets Part II: Numerical verification of the flow field. *ASHRAE Transactions* 120, Part 1, NY-14-022, 271-285.

Paper V: Ghahremanian, S., Svensson, K., Tummers, M.J., Moshfegh, B., 2014. Near-field mixing of jets issuing from an array of round nozzles. *International Journal of Heat and Fluid Flow* 47, 84-100.

Paper VI: Ghahremanian, S., Moshfegh, B., 2014. Investigation in the near-field development of a row of round jets. Submitted for publication.

The following papers were published during my PhD study, but they are not included in his thesis:

Janbakhsh, S., Moshfegh, B., Ghahremanian, S., 2010. A Newly Designed Supply Diffuser for Industrial Premises. *International Journal of Ventilation* 9, 59-68.

Ghahremanian, S., Moshfegh, B., 2011. Numerical and experimental verification of initial, transitional and turbulent regions of free turbulent round jet, *20th AIAA Computational Fluid Dynamics Conference* Honolulu, Hawaii, USA.

Svensson, K., Ghahremanian, S., Moshfegh, B., Tummers, M., 2012. Numerical and experimental investigation of flow behavior in a confluent jet ventilation system for industrial premises, *Proceedings of the 10th International Conference on Industrial Ventilation*, Paris, France.

Acknowledgement

Foremost, I would like to thank my supervisor, Professor Bahram Moshfegh, for his support and advice throughout my PhD. Thanks to Bahram who introduced me to the scientific research and gave me the opportunity to carry out this study.

I would like to thank Dr. Ir. Mark J. Tummers for all his practical support and invaluable advice through the experimental studies and writing the articles. I would like to thank PhD student Klas Svensson for his fruitful contributions to the measurements and to coauthoring the articles. I am so grateful to associate professor Taghi Karimipناه for all his kind support and advice. I would like to thank Professor Hazim Awbi for his valuable comments on my papers.

I would like to thank Mr. Ulf Larsson, assistant professor Mathias Cehlin, Mr. Hans Lundström, Dr. Claes Blomqvist, Mr. Staffan Nygren, Mrs. Eva Wännström, PhD students (Mr. Amir Sattari, Mr. Mohammad Jahedi, Mr. Abolfazl Hayati, Mr. Alan Kabanshi), Mr. Taha Arghand (research student), and many others from the University of Gävle. The author is thankful for the assistance received by personnel at the laboratory of Delft University of Technology especially Mr. Bart Hoek and Mr. Erwin de Beus. I would like to thank Mr. Sohil Shahabi.

The author gratefully acknowledges the financial support received from the University of Gävle (Sweden), Stravent Oy (Finland) especially Mr. Timo Karkulahti and Sparbankensstiftelse Nya.

I am eternally grateful to my father Hossein (who was a compassionate and decent doctor, may he rest in peace), my mother Homa, my sister Shahrzad and her husband Mohammad, my brother Shayan, my in law family (Mostafa, Mahin, Payman, Reza, Sanaz and Sadaf) and other family members for their continuous support and encouragement.

I owe so much to my beloved wife, Setareh. With her unwavering love, deep understanding, great faith, endless support, positive encouragement, and patience I could accomplish my PhD studies. Words cannot express my deep gratitude for everything you, Setareh, have done for me. I will really miss when we were classmates, colleagues, experimental collaborators, and officemates. Arwin, my lovely son, you have brought great joy and happiness to our life, and you have given me more energy to complete this study. Your kind visits at my office and asking for “fika” gave me such pleasure.

Table of contents

1	Introduction	1
1.1	Application of confluent jets in ventilation	1
1.2	Motivation for this study	2
1.3	Aim	3
1.4	Research process	4
1.5	Methods	5
1.6	Limitations	5
1.7	Appended papers	6
1.8	Co-author statement	7
1.9	Other publications	7
1.10	Outline of the appended papers	8
2	Literature Review	11
2.1	Single jet	11
2.1.1	Influence of jet exit conditions	11
2.1.2	Turbulence modeling	17
2.2	Twin jets	18
2.3	Multiple jets	23
3	Methods	25
3.1	Experimental fluid dynamics	25
3.1.1	Hot-wire anemometry (HWA)	25
3.1.2	Laser Doppler anemometry (LDA)	28
3.1.3	Particle image velocimetry (PIV)	34
3.2	Computational fluid dynamics (CFD)	41
3.2.1	Governing equations	42
3.2.2	Boussinesq approach versus Reynolds stress model	44
3.2.3	Realizable $k - \epsilon$ model (R $k - \epsilon$)	44

3.2.4	The shear stress transport (SST) $k - \omega$ model with low Reynolds correction	46
3.2.5	The Reynolds stress model (RSM)	49
3.2.6	Low Reynolds (low Re) $k - \epsilon$	52
3.2.7	Transition models ($k - kl - \omega$ and Transition SST $k - \omega$)	54
3.2.8	Near wall treatment in the $k - \epsilon$ and RSM	57
3.3	Response surface methodology	58
4	Case Study	63
4.1	Inlet boundary conditions of confluent jets on a curved surface	63
4.2	Single round jet	70
4.3	Confluent jets on a curved surface	74
4.4	Parallel, coplanar, single row of jets	76
4.5	Confluent jets on a flat surface	82
4.6	Parametric study of a coplanar parallel row of jets	86
5	Results	91
5.1	Experimental study	91
5.1.1	Single row of round jets	91
5.1.2	Array of multiple round jets	92
5.1.3	Comparison of the single row and array setups	95
5.2	Numerical study	98
5.2.1	Inlet boundary conditions of confluent jets on a curved surface	98
5.2.2	Single round jet	99
5.2.3	Confluent jets on a curved surface	101
5.3	Parametric study	102
6	Conclusions	107

7	Looking to the Future Studies	110
	References	112

1 Introduction

Multiple interacting jets that issue from an array of closely spaced round nozzles are known as confluent jets. Multiple interacting jets merge (coalesce), combine with one another and move as a single jet at a certain distance downstream. Multiple interacting jets are utilized in several applications, such as air supply devices of ventilation systems, air curtains, injection systems for thrust augmentation and noise control in S/VTOL aircrafts, cooling of devices (e.g., film cooling of turbine blades and electronic devices), effluents in sewage water, die injection, combustion nozzles or fuel injectors, fuel-air mixing and dilution holes in gas turbines, burners and chemical reactors, and pollutant dispersion and plume dilution from chimney (exhaust) stacks with optimal spacing.

1.1 Application of confluent jets in ventilation

During the past three decades, the global demand for primary energy has nearly doubled and the demand for electrical energy has approximately tripled. In most IEA countries, the building sector accounts for more than 40% of society's total primary energy use and contributes to more than 15% of the total CO₂ emissions. Energy demand in the built environment is a growing issue; the building sector accounts for nearly 35% of total energy use and 15% of total CO₂ emissions (Sayigh, 2013). According to the roadmap for moving to a competitive low-carbon economy, the EU intends to reduce energy use (Sayigh, 2013) and greenhouse gas (GHG) emissions by 20% by 2020 and by 80% by 2050. In Sweden, the Environmental Advisory Council has stated that the demand for purchased energy in the building sector should decrease by at least 30% by 2025 compared to 2000. One of the priority areas for achieving these goals is energy efficiency of buildings. One of Sweden's 16 national environmental qualities is a good built environment; "Cities, towns and building communities must provide a good living environment and contribute to improving the regional and global environment". Thus, there is a significant need to develop sustainable buildings and innovative system solutions to reduce primary energy use and power demand and to increase the quality of indoor climate.

1 Introduction

Nowadays, heating and cooling of buildings accounts for more than half of all residential energy usage in developed countries. The main objective for heating, ventilation and air-conditioning (HVAC) systems in buildings is to ensure a healthy and comfortable indoor climate, preferably at a low cost with minimal energy usage and environmental impact. Poor indoor environmental conditions, such as in offices and classrooms, impose large costs in health-care, administration, and lost productivity. This stresses the importance of well-functioning HVAC systems in the built environment. Due to the low ventilation performance and the energy efficiency of conventional ventilation systems such as mixing ventilation (Cao et al., 2014), it is important to develop air supply systems (e.g., confluent jets systems) that may improve indoor air quality, energy and cost efficiency, and thermal comfort (Janbakhsh and Moshfegh, 2014a; Karimipanah et al., 2007). The ventilated air from a supply device that uses confluent jets spreads over greater distances (Cho et al., 2008) and can work for both heating and cooling purposes due to the higher momentum than other ventilation systems. The fresh air from a confluent jet supply device mixes better with the surrounding air because of the interactions between the multiple jets (Janbakhsh and Moshfegh, 2014a; Janbakhsh and Moshfegh, 2014b). Thus, a confluent jet supply device requires less energy than in the mixing or displacement ventilation systems due to lower pressure drop (Karimipanah et al., 2008). Ventilation systems that are based on multiple interacting jets have lower installation costs because the system can use a low capacity mechanical fan, and there is no need to use noise absorption devices (silencers or mufflers)¹ that cause pressure drops. Understanding the confluence of multiple interacting jets is important for airflow control, air quality control, and energy efficiency, all of which are major concerns in any modern, efficient and expedient air supply design.

1.2 Motivation for this study

As described in the above section, the massive concerns on the emissions of GHG and energy efficiency measures in the built environment have resulted in the building industry to seek for a new ventilation supply device. The new supply device has to provide the proper indoor climate in an

¹ From a product brochure available at http://www.stravent.fi/uploads/file/pdf_eng/eng_S11.pdf

1.3 Aim

energy efficient and economical way with lower environmental impact. One of the factors that initially promoted this study is the application of multiple interacting jets in the design of air supply devices with applications in residential and industrial ventilation systems. Understanding the flow behavior and the confluence of multiple interacting jets is key to controlling the near-field fluid flow and thus being able to improve the efficiency of the supply devices that are based on multiple jets.

Extensive and substantial research has been carried out on the flow characteristics of a single jet and twin jets. In contrast to these fundamental configurations, the flow configuration of the multiple interacting jets and prediction of the jets confluence are complex and has not been thoroughly investigated. Flow features such as jet deflection, initial and transition regions, converging, merging, and combined regions needs to be explored. The dependencies of the merging and combined points on the other variables (such as nozzle diameter, spacing between nozzles or inlet velocity) need to be studied. Extensive investigations and a thorough understanding of these complex flow features are necessary before this supply device concept can be optimally developed.

Previous studies provided the motivation to perform investigations on multiple interacting jets with similar experimental and numerical techniques. Hot wire anemometry (HWA), laser Doppler anemometry (LDA) and particle image velocimetry (PIV) can be used as experimental tools for detailed measurement. The advancement in computational fluid dynamics (CFD) makes it a suitable research tool for detailed exploration of flow field analysis and for parametric study in the case where it is difficult and costly to accomplish it by measurement. Experimental parts of the study and validation of CFD simulations by detailed results from measurements can be of interest for other researchers in turbulence modelling.

1.3 Aim

The main objective of this study is to gain insight into the complex phenomena that occur in the near-fields of two multiple jet configurations (a single row of parallel coplanar jets and an in-line array of confluent jets) in which the airflow from the round jets issues into a stagnant environment. The primary aim of the investigation of multiple jets is to gain a better understanding of the flow behavior (such as confluence of the jets) and the

1 Introduction

turbulent properties by experimental and numerical methods. The objective of this study is to validate the performance of various turbulence models and to test their capabilities in predicting the near field of confluent jets. The study of near field of a single jet is also aimed at serving as a reference for comparison to the features of confluent jets.

The study of multiple interacting jets is intended to provide information about the mean flow (e.g., development of the mean velocity and the RMS, velocity decay and growth of the RMS, Reynolds normal and shear stresses) and turbulence properties (e.g., turbulent kinetic energy production and anisotropy of the turbulence). The goal is to obtain data that can serve as benchmark cases for future studies on multiple interacting jets. The experimental and numerical investigation aims to provide information about features of the jet, such as jet-to-jet interaction, jet deflection (convergence), jet position dependency and characteristic regions (initial and transition, converging, merging, combined and turbulent) as well as characteristic points (merging and combined points). The dependencies of the merging and combined points on other variables (such as nozzle diameter, spacing between nozzles or inlet velocity) are also aimed for investigation in this study.

1.4 Research process

In this study, the scientific research process can be described by a series of inter-related steps. Observations may generate a series of questions and multiple ideas can then be constructed. Controlled experiments and simulations are then designed and performed to answer questions and to explain the educated guesses. The investigation of confluent jets is based on four research parts: experimental study, numerical study, validation study and parametric study. Previous research on two fundamental configurations (a single jet and twin jets) and multiple jets are also compiled and studied. Controlled experiments are then conducted, the results are collected, and the details of the measurements are recorded. The measurements provide a picture of the complex flow behavior and it is difficult to measure all variables, thus numerical simulations of the flow field are performed. Before using the numerical results, the simulation cases are compared and validated with the experimental data. Numerical simulations allow the examination of more details and other aspects that are not measured. The

1.5 Methods

important variables are then identified, and the influences of different parameters on the confluent jets are examined and are studied using numerical techniques.

1.5 Methods

This study attempted to investigate the characteristics of the flow field and turbulence behavior of multiple interacting jets. A series of experimental measurement have been performed using Particle Image Velocimetry (PIV), Laser Doppler Anemometry (LDA) and Hot-Wire Anemometry (HWA). Computational Fluid Dynamics (CFD) using the Reynolds Averaged Navier-Stokes (RANS) equations is employed for turbulence modelling. Different turbulence models are evaluated during the numerical simulations of multiple interacting jets. The results of each numerical study are validated with the results of the experimental studies. The results of the PIV measurements are compared with several LDA profiles in single row and array setups. The velocity and the RMS of the velocity are evaluated in the overlap regions of the PIV recording sessions. In addition, repeated measurements are used to evaluate the results of the experimental studies. The high frequency HWA, the LDA, the two-dimensional PIV measurements and the three-dimensional numerical simulations provide high quality detail information for a further study on the confluent jets. To determine and quantify the relationship between the characteristic points and a group of significant factors (inlet bulk velocity, spacing between nozzles and diameter of nozzles), the response surface method is used for the parametric study. The direct effect of the inlet Reynolds number is assessed in all of the experimental and numerical cases as a parametric study.

1.6 Limitations

The research process and methods applied in this study has several limitations. The experimental methods for the observations and validation are limited to HWA, LDA and PIV. The numerical simulations, which are limited to the finite volume method and steady state time-averaged method (RANS equations), are used to extract additional information. The numerical predictions are validated by the experimental results in a limited

1 Introduction

number of locations. The parametric study is limited to the response surface method and the numerical predictions. This study also has configuration, experimental and numerical limitations. The configuration parameters, such as the shape (short-height contraction), diameter ($d_0 = 0.0058$ m), spacing ($2.82d_0$), a range of inlet Reynolds number and arrangement of the array (in-line) of nozzles are kept constant during the experimental, numerical and validation studies except in the parametric study. In the experimental study, only the axial component of the velocity and its RMS are measured with either the single HWA probe or the LDA instrument. Because measurements are impossible to make exactly at the nozzle exits, all of the measurements that are used for validation are collected at a short distance downstream of the nozzle edge. The two components of the instantaneous velocities are simultaneously measured by a standard PIV system that uses one camera in specific two-dimensional planes. The measurements of the mean velocities and some of the Reynolds stresses with the standard PIV system are made in the mixing layer of the jets near the nozzle exits. The numerical predictions are limited to the commercial CFD code ANSYS Fluent and to physical models, such as the steady state RANS turbulence models in the software.

1.7 Appended papers

This thesis is based on the following papers:

Paper I: Ghahremanian, S., Moshfegh, B., 2014. A study on proximal region of low Reynolds confluent jets Part I: Evaluation of turbulence models in prediction of inlet boundary conditions. *ASHRAE Transactions* 120, Part 1, NY-14-021, 256-270.

Paper II: Ghahremanian, S., Moshfegh, B., 2013. Evaluation of RANS Models in Predicting Low Reynolds, Free, Turbulent Round Jet. *Journal of Fluids Engineering* 136, 011201.

Paper III: Ghahremanian, S., Svensson, K., Tummers, M.J., Moshfegh, B., 2014. Near-field development of a row of round jets at low Reynolds numbers. *Experiments in Fluids* 55, 1789.

Paper IV: Ghahremanian, S., Moshfegh, B., 2014. A study on proximal region of low Reynolds confluent jets Part II: Numerical verification of the flow field. *ASHRAE Transactions* 120, Part 1, NY-14-022, 271-285.

1.8 Co-author statement

Paper V: Ghahremanian, S., Svensson, K., Tummers, M.J., Moshfegh, B., 2014. Near-field mixing of jets issuing from an array of round nozzles. *International Journal of Heat and Fluid Flow* 47, 84-100.

Paper VI: Ghahremanian, S., Moshfegh, B., 2014. Investigation in the near-field development of a row of round jets. Submitted for publication.

1.8 Co-author statement

The main supervisor of this study was Prof. Bahram Moshfegh.

Paper I, II, IV and VI:

The studies were planned by the author (Shahriar Ghahremanian) and by Prof. Bahram Moshfegh based on the initial idea of Prof. Moshfegh. The measurements, numerical simulations and parametric study were performed by the author. The results were analyzed, interpreted and put together by the author under the supervision of Professor Moshfegh. Papers I, II, IV and VI were written entirely by the author. However, comments and advices were received from Prof. Moshfegh.

Paper III and V:

The studies were planned by the author (Shahriar Ghahremanian) and by Prof. Bahram Moshfegh. The measurements were carried out in close collaboration between the author and Klas Svensson (PhD student). The author carried out the majority of PIV measurements, while the majority of LDA measurements were done by Klas. Both PIV and LDA measurements were performed under supervision of Dr. Ir. Tummers. The measurement results were analyzed, interpreted and put together by the author and Klas under the supervision of Professor Moshfegh and Dr. Tummers. As a result, Paper III and V were written by the author and Klas. However, valuable comments and advices were received from Prof. Moshfegh and Dr. Tummers.

1.9 Other publications

Janbakhsh, S., Moshfegh, B., Ghahremanian, S., 2010. A Newly Designed Supply Diffuser for Industrial Premises. *International Journal of Ventilation* 9, 59-68.

1 Introduction

Ghahremanian, S., Moshfegh, B., 2011. Numerical and experimental verification of initial, transitional and turbulent regions of free turbulent round jet, *20th AIAA Computational Fluid Dynamics Conference* Honolulu, Hawaii, USA.

Svensson, K., Ghahremanian, S., Moshfegh, B., Tummers, M., 2012. Numerical and experimental investigation of flow behavior in a confluent jet ventilation system for industrial premises, *Proceedings of the 10th International Conference on Industrial Ventilation*, Paris, France.

1.10 Outline of the appended papers

Due to the importance of providing accurate inlet boundary conditions, the air supply device of confluent jets is numerically simulated with a two-equation turbulence model (Realizable $k - \epsilon$ model [R $k - \epsilon$]) and a six-equation turbulence model (Reynolds Stress Model [RSM]) in the first paper (Ghahremanian and Moshfegh, 2014b). The axial velocity and turbulence intensity along two perpendicular diameters of several nozzles in the array of multiple interacting jets are validated with HWA measurements taken near the nozzle exits. The results of the numerical predictions of short contraction nozzles on a cylindrical supply device are compared to flow issuing from a long pipe and are presented in the first paper (Ghahremanian and Moshfegh, 2014b).

To evaluate the predicted inlet boundary conditions at the nozzle exit, the three-dimensional flow field of a free round jet is numerically simulated in the second paper (Ghahremanian and Moshfegh, 2014a) with the prescribed inlet profile from the first paper (Ghahremanian and Moshfegh, 2014b). Due to the anomaly reported in the simulation of a single jet, four turbulence models are employed for the numerical simulation of the entire field in the second paper (Ghahremanian and Moshfegh, 2014a). The development of a single jet contains three regions (initial, mixing transition and turbulent regions), thus two two-equation turbulence models ($k - \epsilon$ and Shear Stress Transport [SST] $k - \omega$) with a low Reynolds number correction, a three-equation ($k - kl - \omega$) transition turbulence model and a four-equation (SST eddy-viscosity) transition turbulence model are used to numerically simulate the entire field of the single jet from a short contraction nozzle. The axial velocity and its RMS are compared to the HWA measurement data along cross-sectional profiles at several distances from the nozzle exit. The second

1.10 Outline of the appended papers

paper (Ghahremanian and Moshfegh, 2014a) presents the well-known characteristics (i.e., self-similarity) of the turbulent region of a single free jet and compares the spreading rate of a single jet with the results of previous studies. The second paper also investigates the predicted dissipation rates of turbulent kinetic energy at several profiles.

To obtain additional insight about multiple round jets and fill the knowledge gap between the fundamental configurations (single jet and twin jets) and an array of confluent jets, the instantaneous velocity is measured in the center-plane of a single row of nozzles using the PIV technique. The axial velocity and its RMS are also measured along one cross-sectional profile in the center-plane and two geometrical centerlines of the jets using the LDA technique. The third paper (Ghahremanian et al., 2014a) presents measurements from the PIV and LDA techniques and investigates the mechanisms that are involved in the merging and combined regions of the single row of jets. The development of the velocity and Reynolds stresses are also presented in the third paper (Ghahremanian et al., 2014a). In the sixth paper, the flow field in the proximal region of a single row of round jets is then numerically predicted by using the low Reynolds $k - \epsilon$ model. The results of the simulation are validated with measurement data acquired by PIV. The streamwise velocity of PIV and simulation along the geometrical centerline of central jets and along the cross sectional profiles at different distances from the nozzles are compared and presented in the sixth paper. The characteristic points of numerical simulation are determined and are compared to the PIV for different regions in-between confluent jets.

The flow from an array of multiple interacting jets placed on a curved surface (cylinder) is numerically simulated with the prescribed inlet boundary conditions from the first paper (Ghahremanian and Moshfegh, 2014b). The axial velocity and its RMS are measured in the center-plane of one column of nozzles using HWA. The fourth paper (Ghahremanian and Moshfegh, 2014c) verifies the results of these numerical simulations using the measurement data and investigates the jet-to-jet interactions, the jet position dependency and the jet deflection in the array of confluent jets.

The fifth paper (Ghahremanian et al., 2014b) focuses on the near-field of an array of multiple interacting jets. The two-dimensional velocity and its RMS are measured on four center-planes and three in-between planes using PIV. The results of the PIV measurements are compared with LDA data

1 Introduction

along three cross-sectional profiles and one geometrical centerline. The LDA measurements also provide information about the velocity and the RMS of the velocity in regions with high gradients, such as near the nozzle exits. The conformity of the axial velocity and its RMS in the symmetry planes of the array and in the overlap regions between the PIV recording sessions are studied, and the development of the Reynolds normal and shear stresses on the studied planes are presented in the fifth paper (Ghahremanian et al., 2014b). The production of turbulent kinetic energy due to the Reynolds normal and shear stresses and the anisotropy tensor on the measured planes are investigated.

To compare the findings from all of the appended papers, the maximum axial velocities in the single row of nozzles from the third paper (Ghahremanian et al., 2014a) and in the array setup from the fifth paper (Ghahremanian et al., 2014b) are compared with the centerline velocity of the single jet that was presented in the second paper (Ghahremanian and Moshfegh, 2014a). The behaviors of the jets in different positions of the single row and array setups are investigated by comparing the locations of the maximum velocity and the central streamlines.

The parametric study is presented in all of the papers (Ghahremanian and Moshfegh, 2014a, b, c; Ghahremanian et al., 2014a, b) by investigating the influence of the Reynolds number on the characteristics of the flows from the single jet and multiple interacting jets. A computational parametric study is conducted on the flow field of confluent jets in the single row configuration. The effects of inlet velocity, spacing between nozzles, and nozzle diameter on the flow development are analyzed by using the response surface method and are presented in the sixth paper. The simulation cases are determined based on Box-Behnken design. Correlations between three design factors and the characteristic points are estimated in the form of full quadratic regression equations at different regions between confluent jets. The contour and surface plots for the response of merging and combined points to two design factors are shown for different regions between the jets.

2 Literature Review

This chapter presents a global overview of a single jet, twin jets and multiple jets on the basis of currently available literature.

2.1 Single jet

2.1.1 Influence of jet exit conditions

When a flow leaves the leading edge of the nozzle, it becomes unstable due to infinitesimal disturbances in the velocity profiles at almost all Reynolds numbers. Two length scales can characterize a round jet: the inlet boundary shear-layer thickness (e.g., momentum thickness) and the nozzle diameter. In the near-field, the large-scale coherent structures depend on the inlet momentum thickness and shear layer instabilities (Crow and Champagne, 1971), while the jet diameter controls the length scale beyond the near-field. While the inlet boundary conditions of the round jet affect the initial large-scale vortices by forming axisymmetric vortex rings near the nozzle exit and helical instabilities downstream (sometimes at a transition between two states; Dimotakis et al. (1983)), the length it requires to become self-similar can be different for the mean flow properties and higher order moments. Therefore, providing accurate inlet boundary conditions is important for the correct prediction of the transitional region. Reviews of previous investigations of turbulent round free jets reveal the importance of inlet boundary conditions for the downstream development of the jets. Factors of the inlet boundary conditions that can influence the free round jet development include the nozzle type or geometry, free stream (or co-flow) to jet density ratio and free stream to jet speed ratio, nozzle Reynolds number, and the inlet velocity profile, inlet turbulence intensity profile (or inlet Reynolds stress profiles at the nozzle exit). For example, a top hat and a saddle-back initial velocity profile can be produced by a smooth contraction nozzle and a sharp-edged orifice, respectively. Although the flow features from these two nozzle geometries are different, both are widely used in industrial applications. The former initial velocity profile results in attenuated turbulence at the nozzle exits, which makes it suitable for industrial applications due to the minimization of the pressure drop in

2 Literature Review

the supply pipes. The latter profile results in a rapid increase in the jet spreading rate due to the presence of an inward lateral velocity component.

Bradshaw (1966) was one of the first studies to deal directly with the effect of inlet conditions on free shear layers. He used pitot tube measurements to show that the spreading and evolution of a round jet (e.g., virtual origin) depends on the inlet conditions and Reynolds number. He concluded that the fully developed turbulent mixing layer depends on the momentum thickness of the initial boundary layer, irrespective of whether the initial boundary is laminar or turbulent. He also reported large shifts in the virtual origin position caused by varying the Reynolds number of the round jet.

Ricou and Spalding (1961), Sami et al. (1967), Oosthuizen (1983), Obot et al. (1984) and Abdel-Rahman et al. (1997) found that, when the exit Reynolds number increases, the mean centerline velocity decays and the radial spreading slows (centerline velocity decay constant increases), the virtual origin moves continuously in the upstream direction, the turbulence intensity decreases (which causes a longer decay of the mean centerline velocity and less mixing of the jet flow with the surrounding region), the skewness and flatness deviate less from Gaussian values (0 and 3, respectively), or a Gaussian probability density distribution forms near the jet axis (that causes less intermittency), the energy content increases at high frequencies (so small eddies appear and vortex stretching increases), and the entrainment coefficient decreases.

Crow and Champagne (1971) found a decrease in the displacement thickness of a round jet with an increase in the Reynolds number. They (Crow and Champagne, 1971) also found that varying the Reynolds number has no significant influence on the Strouhal number, which indicates that the interaction of large-scale vortices is insensitive to changes in the Reynolds number.

Round jets with laminar and turbulent inlets were studied by Hussain and Zedan (1978), who showed that the momentum thicknesses at different Reynolds numbers do not affect the parameters of the self-preserving regions of a round jet except for the virtual origin. In addition, when the inlet boundary condition is turbulent, the mean velocity and turbulence intensity profiles do not reach self-similarity simultaneously. Zaman and Hussain (1984) studied laminar and turbulent inlet boundary layers of a

2.1 Single jet

round jet and illustrated that the former flow has larger dispersions in characteristic variables and pairing jitters in the structure than the latter. Zaman and Hussain (1984) examined large scale round jets and reported that contoured structures at a certain distance from the leading edge of the nozzle are not noticeably dependent on the range of Reynolds numbers for the inlet.

Studies of the effect of the Reynolds number on the scalar field by Ricou and Spalding (1961), Yule (1978), Dimotakis et al. (1983), Koochesfahani and Dimotakis (1986) and Miller and Dimotakis (1991) indicated a decrease in entrainment, a reduction of scalar fluctuations and a rapid mixing transition with an increase in Reynolds number. Thus, a more turbulent (homogeneous and chaotic) and more qualitative well-mixed state of the scalar field is evident when the Reynolds number is large. In comparison, low Re free shear flows contain orderly vortices in a growing three-dimensional transition region that depends on the nozzle inlet profile.

Visualizations by Mungal and Hollingsworth (1989) and experimental investigations by Panchapakesan and Lumley (1993) and Hussein et al. (1994) indicated that the normalized mean velocity profiles and spreading rates of a round jet at sufficiently high Reynolds numbers cannot be attributed to a Reynolds number effect.

LDA measurements of ventilated and unventilated (with and without front plates) round jets by Abdel-Rahman et al. (1997) indicated that unventilated round jets have lower decay rates of the mean centerline velocity, spreading rate, kinematic momentum flux and kinematic mass flux and behave more intermittently than ventilated round jets due to reduced mixing with the surroundings. Larger lateral skewness and flatness moments across the jet show that unventilated round jets behave more intermittently.

Papadopoulos and Pitts (1999) studied influence of the Reynolds number and turbulence intensity on the near-field of a round jet and reported that the turbulence intensity per unit area of the inlet boundary condition is an important source of shear layer growth, potential core break-up and its transition into the self-preserving region.

Gouldin et al. (1986) completely summarized the round jet data from previous experimental results and attributed the cause of the variations in the jet spreading rate, virtual origin and centerline mean velocity to inlet condition sensitivity. Two strategies were suggested to overcome the

2 Literature Review

problem of choosing inlet conditions: design a well-defined laminar inlet condition or a turbulent inlet boundary layer. The difficulty with the former idea is the large-scale transition modeling of laminar to turbulent flow, while the difficulty with the latter is the need for extensive measurements (e.g., mean properties, higher moments and their correlations) to specify the turbulent boundary. Despite the problems of modeling the transitional and downstream regions discussed previously, Gouldin et al. (1986) presented that the two-dimensional simulations and experiments of a free shear turbulent flow from a pipe are in good agreement.

A study of the mean flow behavior of short and long (i.e., nozzle height - diameter ratio) square-edged round jets by Obot et al. (1984) indicated that a short nozzle spreads and decays more rapidly and entrains three times more fluid than a tall nozzle. Their measurements illustrate that the entrainment for each nozzle in the near-field is essentially independent of the Reynolds number (1.3 and 2.2×10^4), but clear evidence of a Reynolds number effect exists at large axial locations. Their results indicate that the entrainment rate is a strong function of the nozzle design and that a small overshoot can be observed in jet profiles at the nozzle exit at low Reynolds numbers but not in high Reynolds number flows.

Preliminary results presented by Ashforth-Frost and Jambunathan (1997) indicated that the length of the potential core of a fully developed velocity exit profile is up to 7% longer than a flat profile, which was corroborated by the findings of Lepicovsky (1989). Despite the higher initial turbulence intensity of a fully developed jet, the results reflect the fact that in the initial region, the turbulence intensity for the flat-profile jet increases much more rapidly than for the fully developed jet. These phenomena can be attributed to the higher velocity gradients of the flat-profile jet, which in turn lead to faster spreading of the mixing layer to the jet axis than in the fully developed exit profile.

The mixing performances of three axisymmetric nozzles, including a smooth contraction (contoured) nozzle, a sharp-edged orifice and a long pipe, were investigated by Mi et al. (2001a), and the first two nozzles were studied by Quinn and Militzer (1989) and Quinn (2006). They concluded that the mixing, velocity decay and jet spreading rate in both the near-field and developing regions of the orifice are significantly greater than for a pipe outlet. They also showed that breaking the axisymmetry and enhancing the

2.1 Single jet

three-dimensionality in the mixing layer of round jets (from orifice and contoured nozzles) results in increased entrainment and mixing rates. For the axial velocity spectra, a higher velocity gradient in the shear layer and greater mean velocity along the potential core of an orifice (compared to a contoured nozzle) implies that large-scale coherent vortices roll up from the shear layer and are then convected downstream at a higher speed. In contrast, a fully developed pipe flow has a significantly higher turbulence intensity than the flow from any other nozzle the vorticity is distributed throughout the flow rather than being concentrated at the edge of the nozzle. Both of these effects lead to a reduction of the number of coherent large-scale structures in the near-field of the pipe flow.

Mi et al. (2001a) and Xu and Antonia (2002) showed that large-scale turbulent structures form from a jet exiting from a long pipe much farther downstream (approximately six jet diameters) than from a smooth contraction nozzle (approximately one jet diameter), which results in a clear difference in the potential cores of the two jets. The roll-up of the thin laminar boundary in the near-field of a smooth contraction causes well-defined toroidal vortices, which engulf both the ambient air and the jet and propagate downstream with pairing and break-up processes. These streamwise vortices enhance the entrainment and turbulent mixing of the contraction nozzle, which is absent in the shear layer of a pipe jet. The frequency spectra of the velocity fluctuations in the shear layer of the two jets indicate that the pipe outlet has no broad hump and no preferred frequencies or dominant coherent structures. Measurements (Mi and Nathan, 2004) of the dominant peak in the velocity power spectrum showed a higher Strouhal number (vortex formation frequency) for a sharp-edged orifice nozzle than for a smooth contoured nozzle. These results (Mi and Nathan, 2004) are consistent with the findings of Wong et al. (2005), who showed that shedding of fewer and moderate-size axisymmetric and well-structured vortices from a smooth contoured jet leads to low frequencies. In contrast, the greater shedding from a variety of sizes of asymmetric vortices for the sharp-edged orifice could be attributed to higher frequencies. Additional support for the difference in the Strouhal number of the orifice nozzle and smooth contoured nozzle was demonstrated by Mi et al. (2007) using the Particle Image Velocimetry (PIV) technique. Their linear constant spreading rate of the fully developed region is significantly greater than those reported by Wygnanski and Fiedler (1969) for short convergent

2 Literature Review

nozzles. These empirical correlations make it clear that the constants are dependent on the nozzle geometry. Mi et al. (2001b) performed an additional study on the influence of two different exit conditions of a heated air jet, including a smooth contraction and a long pipe, on the passive scalar field. The findings support the analytical results and conclusions of George (1989). The exit velocity profiles of unheated air from these two nozzles were also considered by Mi et al. (2001b), who showed that the boundary layer displacement thickness and momentum thickness for a smooth contraction nozzle is thinner than that for a long pipe jet due to primary vortices that originate from the instability within the mixing layer. In the exit plane of the smooth contraction, uniform scalar (concentration/temperature) and velocity fields have a strong gradients in the shear layer that yield spatial coincidences of the maximum gradients of the mean values. In contrast, the mean scalar and velocity profiles do not have similar forms.

Results from X-wire measurements of flows from a contraction nozzle and a long pipe by Antonia and Zhao (2001) indicate that the peak values of the Reynolds stresses in the near region are greater for the jet from a contraction nozzle than that from a pipe, which shows stronger shear in the mixing layer. Streamwise development of axial and radial velocity fluctuations (RMS) of both flows monotonically and smoothly increases to a constant value in the self-similar region. It is interesting to note that despite the different initial conditions, the mean velocities and Reynolds stresses are nearly identical for the two axisymmetric flows in the self-similar region.

Velocity measurements of jets exiting from a smooth contraction nozzle and a long pipe by Xu and Antonia (2002) indicate that the flow from the former nozzle develops much more rapidly (i.e., the mean velocity decays and becomes self-similar more rapidly) than the flow from the latter. The pipe jet produces a larger initial mixing layer thickness that results in a lower frequency instability (i.e., longer wavelength structures), which develops and pairs at greater downstream distances. They also inferred higher growth rate of the velocity field for the contraction jet than the pipe jet. Following the suggestions by George (1989) and Hussein et al. (1994), the normalization of turbulence intensities and Reynolds shear stresses in this study (Xu and Antonia, 2002) by the axial mean velocity instead of the half-radius illustrates the influence of the inlet boundary conditions. Using this normalization, the turbulence intensities and Reynolds stresses are

2.1 Single jet

wider for the contraction jet than the pipe jet. The greater axial turbulence intensity compared to the radial intensity reflects the anisotropy in both flows. The distributions of the mean axial velocity, turbulence intensities and Reynolds stresses produced smaller differences as the axial distance increased (Xu and Antonia, 2002).

A critical review of round turbulent jets was provided by Ball et al. (2012) and includes the effect of the Reynolds number and inlet conditions. Their review (Ball et al., 2012) provides physical insight into the evolution of theoretical, experimental and computational methods for round jets.

2.1.2 Turbulence modeling

The 20th-century notion that “turbulence forgets its origins”, or universal (or quasi-universal) similarity (small-scale), was analytically reviewed and argued for by George (1989) and George and Davidson (2004). The universal far field properties are a corollary of the energy cascade principle of sufficiently large Reynolds numbers, which states that the flow is much less sensitive to the inlet conditions than to the large-scale properties. In this classical belief, the effect of the inlet conditions can be eliminated only by a shift in the source point (e.g., virtual origin) where the jet originates from (Richards and Pitts, 1993; Townsend, 1976). However, George and Davidson (2004) related the production of asymptotic dependence of the inlet boundary condition to necessary physics (vorticity production, convection and diffusion) and provided several examples. By proposing a better rescaling based on the half width of a jet, George and Davidson (2004) also showed that the mean velocity profile and the Reynolds stress profile in a special non-dimensional term are independent of all of the inlet conditions in the self-preserving region of wakes and all similar canonical flows, but the growth rate and other moment profiles are not. Earlier experiments on circular jets by Antonia and Zhao (2001) and Mi et al. (2001b) found evidence for this independence. A Direct Numerical Simulation (DNS) of self-similar regions of round jets was performed by Boersma et al. (1998) for different orifice inflow conditions, a top-hat profile and an artificial profile. Their results support the suggestion by George (1989) that the details of self-similarity depend on the inlet conditions. The evidence presented by Boersma et al. (1998) explains that scatter in experimental data may not be solely due to experimental uncertainties but may be attributed to the nozzle’s geometry or velocity

2 Literature Review

profile (incorrect scaling). It appears that the anisotropy of large-scale motion should be considered when turbulence is associated with dissipative and inertial range scales. This suggestion leads to another aspect of the study by George and Davidson (2004), in which the inlet conditions appear in the modeling coefficient (i.e., turbulent viscosity) of the Reynolds Averaged Navier- Stokes (RANS) equations. RANS turbulence models do not represent the necessary physics of the inlet, but Large Eddy Simulations (LES) retain the necessary physics of the inlet conditions in all regions of canonical flows. These results reveal the importance of supplying appropriate inlet boundary conditions for the numerical simulation of round jets and confluent round jets, where small changes at the inlet can significantly affect the subsequent evolution of the jets.

2.2 Twin jets

Twin jets usually consist of two co-planar identical plane (or round) nozzles of the same width (or diameter) that are separated from each other by a distance S . The nozzle spacing (S) ratio or nozzle width (or diameter) can be used as a characteristic length for the twin jet configuration. A symmetry plane (or line) bisects the two jets and is the location where they meet each other. The twin jets pass an initial region immediately after they leave the nozzles, and a low pressure (sub-atmospheric) is then created due to mutual entrainment, which is known as the re-circulation region. The twin jets deflect towards each other due to a decrease of the sub-atmospheric pressure in the converging region. If we assume that the streamwise direction is positive, then the mean streamwise velocity becomes negative in the symmetry plane (line) until the inner shear layers of the two jets merge. At the merging point, which is located on the symmetry plane (line), the mean streamwise velocity becomes zero; the region downstream of this point is called the merging region. The interactions between two jets increase and the mean streamwise velocity at the symmetry plane (line) increases to a maximum at the combined point. The combined region begins immediately after the combined point, at which the twin jets have completely combined and resemble a single free jet. These three regions are typical for twin jets and, depending on the design, some may appear together.

2.2 Twin jets

An analysis of dual jet (from parallel slot nozzles) flow was presented by Miller and Comings (1960), in which the flow region was divided into jet convergence and combined jet regions. They reported that the low static pressure between the jets causes jet convergence, which creates a free stagnation point and considerable reversal of the flow at the plane of symmetry. They concluded that a pressure greater than atmospheric redirects the merging jet streams into a single jet that is symmetrical about the center-plane. They also concluded that if a dual jet is replaced by a single jet nozzle of the same size but with a higher outlet velocity, the mean velocity after a certain distance downstream is indistinguishable from the original field in all respects.

Tanaka (1970) studied the interference of two-dimensional parallel jets by measuring the pressure and velocity. They showed that within the area of attraction of the two jets, the jet axis follows an arc and deflects because of unsymmetrical mixing on each side of the jet. Conservation of momentum flux in the flow direction, which differs from a single jet, was also illustrated. The entrainment of the surrounding fluid by a turbulent jet creates a low pressure region with a negative streamwise velocity between the two jets (concave near the nozzle) due to symmetrical recirculation eddies and high pressure with a positive velocity due to merging downstream of the nozzles, while the static pressure of a single jet is constant in the cross section. The outer edge of a curved jet has a larger turbulent energy distribution than the inner edge due to the centrifugal force and jet deflection. Tanaka (1970) also found that the growth in width of one of the dual jets is greater than that of a single jet and that the maximum velocity decay along the mid-nozzle streamline is faster than in a single jet. His study considered the effect of spacing on the velocity decay, the radius of the maximum velocity arc, the lateral distribution of velocity, the turbulence and static pressure.

Tanaka (1974) reported experimental results of two-dimensional parallel jets in the combined region and showed that the velocity profiles of the combined flow agree with the single jet studies but have differences in the turbulence intensity distribution. The static pressure increases more than the momentum diffusion, which leads to a greater spread of the combined jet flow than for a single jet. He concluded that the positions of the maximum velocity and maximum pressure of the combined jet is independent of the Reynolds number and depends only on the geometrical pattern. He

2 Literature Review

described the dual jet as a complex jet flow that is more difficult to measure and that has different distributions of the velocity, turbulence and pressure along the flow of the combined jet compared to a single jet. He concluded that the maximum velocity of twin jets in the combined region decays faster than a single plane jet, the width of the twin jet in the combined region remains linear, and the width increases with spacing. He also found that the momentum flux at each cross section in the combined region is conserved only for some spacings between jets.

Tanaka and Nakata (1975) evaluated the interference of triple jets with different discharge velocities in the near zone using hot-wire anemometry and oil film visualization. The entrainment area and the area surrounding the central jet are different from the dual jets in Tanaka's previous studies. This study focused on the velocity variation and did not report on the identical discharge velocities from the nozzles. The classification of the flow pattern reveals that for a velocity ratio of approximately 0.865 (between the central and side jets), the central jets divide into two and merge with the side jets.

Becker and Booth (1975) mapped the concentration in the interaction zone of two inclined free jets (only converging angles) using the smoke scattered light technique. They concluded that before meeting, the two jets are strongly deflected towards each other. They suggested using one-half of the distance between the virtual origins, or more generally the distance between the nozzle centers, as a length scale. Their results indicate that the mixing volume with the entrained fluid decreases with increasing intersection angle.

Yuu et al. (1979) measured turbulent properties of two interacting parallel plane jets using hot-wire anemometry. Yuu et al. (1979) used two ventilated jets between which the secondary flow could be entrained and found that the interaction is weaker than in an unventilated configuration. The merging point was determined by the coincidence of the trajectory of the jet centerline with the symmetry plane. Similarity of the longitudinal velocity, root mean square of the turbulent velocity and autocorrelation coefficients occurred downstream of the merging point (i.e., the combined region). However, the gradient of the half width of the jet is smaller than that for a single plane jet due to the inward deflection of the jets. The energy spectrum and probability density function for both the converging and combined regions were similar to the slope of the Kolmogorov inertial sub

2.2 Twin jets

range $(-5/3)$ and Gaussian curve, respectively. Finally, they concluded that the turbulence is homogeneous and isotropic at small scales except for in the merging region.

Leschziner and Rodi (1981) numerically studied several discretization schemes and turbulence models of twin parallel jets. They compared the numerical solutions with measurements at the center plane of twin parallel jets and concluded that the standard $k - \varepsilon$ model by Launder and Spalding fails to accurately represent the effects of turbulence in the re-circulating flows because it does not account for important secondary strain effects (curvature). As a result, they proposed several modifications to the some turbulence model, mainly for the normal stress, that improved the agreement with the experimental data. A negative velocity in the mixing zone of center plane before the merging point was observed in their investigation.

Okamoto et al. (1985) investigated the interaction of twin round turbulent jets using velocity and static pressure measurements. This article describes how twin jets interact and join in an ellipse at a downstream distance and become similar to a circular jet further downstream. The deflection of the jets causes a shift of the maximum velocity position from the nozzle axis to a midpoint (symmetry) between the jets. In addition, the decay of the maximum velocity of the twin jets is consistent with that for a single jet, when a high Reynolds number flow issues from two nozzles that are separated by a large spacing.

Hong and Chaung (1987) reported the solution for twin plane jets using a kinetic theory approach (constructing a probability density function via integration of a Green's function over the source distributions according to the boundary conditions) without turbulent transport coefficients or eddy viscosity. The mixing mechanism, Reynolds stress and turbulent energies (second-order correlations of turbulent transport phenomena) were predicted using this approach.

Dual jets in tubular confinements with various nozzle configurations were studied by Koller-Milojevie and Schneider (1993). Flow visualizations and LDA measurements were used to investigate low Reynolds number jets with small diameters. Jets with intersecting axes (divergent angles) of dual-hole nozzles showed considerably greater entrainment rates and turbulence level than non-intersecting dual nozzles.

2 Literature Review

Moustafa (1994) conducted an experimental study of high-speed twin turbulent circular jets with pitot tubes and showed that the main features of a high-speed twin jets are similar to those of a low-speed incompressible twin jets.

Lai and Nasr (1998) compared the performance of three turbulence models (standard $k - \epsilon$, RNG and Reynolds stress model [RSM]) for two-dimensional parallel jets. The flow is divided into four regions, including the recirculation flow, converging, merging and combined regions, and all of the regions were analyzed by LDA measurements and CFD simulations. RSM had the closest static pressure distribution to the experimental results but was very sensitive to different discretization schemes. The slightly over prediction of the non-dimensional merging length is in the same range as in the standard $k - \epsilon$ and RSM models. The latter model does not produce better predictions than the former for this complex flow.

Figure 1: Two parallel plane jets (Nasr and Lai, 2000)

Anderson and Spall (2001) experimentally and numerically investigated the flow field of twin plane jets. They carried out hot-wire anemometry measurements inside a bounded space with end walls. They used the standard $k - \epsilon$ model and the Reynolds stress model (RSM) with a uniform

2.3 Multiple jets

inlet velocity profile for the two-dimensional numerical simulations. The predicted characteristic points by the numerical models have a good accuracy. They found that the merging point is present when the velocity on the symmetry plane is equal to zero or the streamwise normal stress rapidly increases toward an asymptote, and the combined point has a maximum velocity along the symmetry plane.

Yin et al. (2007) experimentally determined the flow field characteristics in the mixing region of twin jets from velocity measurements. They indicated that by increasing the Reynolds number or decreasing the spacing between the jets, the turbulence energy and interference increases, i.e., the twin jets attract strongly. They observed a clear separation of the twin jets near the nozzle, and interaction of the two jets, and mixing and merging to a single jet.

2.3 Multiple jets

Corrsin (1944) published systematic measurements downstream of a row of nozzles but did not demonstrate quantitatively, that the flow downstream from a grid is unstable and that the instability results in a rapid amalgamation of adjacent jets that issue from open parts of the grid. Corrsin (1944) measured total-head and temperature using a hypodermic needle total-head tube and a copper constantan thermocouple, respectively. He intended to stabilize the flow downstream of the grid, but two important phenomena were mentioned and illustrated: the amalgamation of jets after a certain distance to create a single jet and the deflection of side jets toward the center of the grid.

Systematic measurements downstream of a row of nozzles were performed by Knystautas (1962), Marsters (1979); Tanaka and Nakata (1975), Pani and Dash (1983) and Cho et al. (2008). Two important phenomena were identified in these studies: the confluence of the jets into a single jet after a certain distance and the deflection of peripheral jets towards the center of the linear array.

Villiermaux and Hopfinger (1994) presented a detailed analysis of multiple jets emerging through a perforated plate using Hot-wire Anemometry, Pitot tube and smoke visualization. In this setup, the perforated plate extended all the way to the walls of the wind tunnel, and

2 Literature Review

there was no entrainment of ambient air from the sides. While merging of the multiple jets still occurs in this configuration, the merging is due to jet spreading rather than the deflection of the jets towards the center and, is related to the confined arrangement of the multiple jets. The results show that the merging length of the jets oscillated and was dependent on the Reynolds number below a Reynolds number of $3 - 4 \times 10^3$.

Further analysis of multiple jets has been conducted in research related to combustion or chemical mixing (Tatsumi et al., 2010; Yimer et al., 1996). Yimer et al. (1996) studied multiple jet burners with a central (fuel) jet surrounded by a ring of either four or six (air) jets and found that the jets from the outer nozzles curve towards the centerline of the array while merging with each other and the center jet. This overall contraction is the result of the entrainment action of each jet, which draws in fluid from the surroundings and thus produces suction on its neighbors.

The experimental data and numerical simulations in Böhm et al. (2010); Rieth et al. (2014); Stein et al. (2011) show three regions. An initial region with a constant jet velocity and low velocities between the jets is located near the nozzle plate. The next region is dominated by strong jet-to-jet interactions and a decay in the jet velocity. In the final stages, individual jets can no longer be observed, and the turbulence decays (Böhm et al., 2010; Stein et al., 2011). While the general behavior is similar to the studies described above, the unconfined jets also showed a strong convergence along the side of the array. This lateral displacement had a large impact on the flow field development. The regions on multiple jets described in these studies illustrate the similarities between multiple round jets and simpler geometrical configurations such as twin plane jets.

Multiple jets impinging on a flat plate have been studied experimentally by Geers et al. (2006); Geers et al. (2005) and numerically by Thielen et al. (2005); Thielen et al. (2003). The results show both strong jet-to-jet interactions and symmetry breakdown phenomena within multiple impinging jet configurations.

3 Methods

The experimental, numerical, and statistical methods described in this chapter are used in this thesis and the appended papers are presented. First, hot-wire anemometry, laser Doppler anemometry and particle image velocimetry are presented as the techniques for instantaneous velocity measurement. The experimental fluid dynamics is followed by computational fluid dynamics as the numerical prediction technique. Next, response surface method is discussed as the statistical technique for parametric study.

3.1 Experimental fluid dynamics

3.1.1 Hot-wire anemometry (HWA)

Constant Temperature Anemometry (CTA) is an experimental technique that was used to perform the velocity measurements in this study. In HWA, the convective heat transfer from a heated wire sensor to the surrounding fluid is attributed to the fluid velocity. This occurs by keeping a very fine wire at a constant temperature using a feedback electronic circuit, which makes it possible to measure the velocity and its fluctuations at fine scales and high frequencies.

The HWA technique is simple to use, widely available, well established, accurate, reliable, and has low initial and maintenance costs. Using HWA, the flow parameters can be measured near the walls. The output signal from the HWA sensor in the form of a continuous analogue voltage provides the instantaneous velocity. This provides no loss of the output information and makes it possible to extract all statistical parameters, such as the mean, root mean square (RMS), higher order of moments (skewness, kurtosis), probability density functions and higher order spectra.

The HWA probes influence and disturb the flow due to its intrusion. The HWA probes cannot sense the occurrence of reversals in the flow direction; and therefore do not detect the correct velocity. Eddy shedding from the cylindrical sensors may cause prong vibrations that can disturb the measurements, and contamination of the HWA probes by dust or other materials can affect the quality of the velocity measurements. The HWA

3 Methods

sensors require careful handling due to their fragility, which is more important in measurements close to the walls (e.g., the nozzle exit). Measurements by HWA in non-isothermal environments can cause problems because temperature variations produce substantial changes in the calibration function.

3.1.1.1 Principles of CTA operation

The main components of a CTA circuit are a Wheatstone bridge, a feedback (servo) amplifier, an electric-testing sub-circuit, a signal conditioner and a probe. The hot-wire sensor and a variable resistor to control the sensor temperature are placed on opposite arms of the Wheatstone bridge. When the flow has no velocity, the Wheatstone bridge is balanced with no voltage drop. When the flow velocity increases, the sensor resistance decreases and the current in the sensor increases by the feedback amplifier due to a voltage drop across the bridge diagonal. The servo amplifier keeps the sensor resistor in the Wheatstone bridge constant; therefore, the heat loss from the sensor wire is directly represented by the output voltage of the bridge, and the flow velocity can then be computed from this heat loss. To set up an anemometer, a signal conditioner is also required to adjust the high/low pass filter, the voltage offset and the gain. The high gain of the current-regulating amplifier leads to a balance in the bridge, which is nearly independent of the flow velocity. Although the time constant of the sensor is on order of a few microseconds, some optimization of the output frequency must be performed to adjust the CTA. The voltage offset brings the measurement signal into the range that the acquisition system (analog/digital board) expects. The high pass filter eliminates the direct current (DC) component of the CTA signal when high frequency fluctuations contaminate the spectral analysis. The contributions of the unwanted high frequencies in the spectrum must be eliminated when the timescale of the CTA signal is longer than the total length of the data record. Although a stationary signal can be generated by applying a high pass filter, the low-frequency oscillations may also be eliminated. A low pass filter must be set up to attenuate the low frequency noise and to avoid aliasing (i.e., violating the Nyquist criterion) by estimating the highest frequency in the flow.

The CTA hardware setup consists mainly of static and dynamic bridge balancing. Static balancing refers to the adjustment of the overheat ratio,

3.1 Experimental fluid dynamics

and dynamic balancing refers to the square wave test. The working temperature of the sensor is determined by the overheat adjustment. By adjusting the decade resistor in the right bridge arm, the desired working temperature of the sensor is established when the bridge is operating. The overheat ratio relates the sensor resistance at the operating and ambient temperatures. The square wave test has two purposes: to optimize the bandwidth of the combined anemometer/sensor circuit and to check the operational stability of the servo-loop. The time for the bridge to reach a balance after applying a square wave signal is related to the bandwidth of the probe/anemometer. The optimized response can be achieved by adjusting the amplifier filter and gain.

3.1.1.2 Theory of operation

The general equation of the HWA is derived to examine the response under steady state and fluctuating conditions. The energy balance for a small circular element of the HWA depends on the current, heat transfer coefficient, flow temperature and wire temperature. By assuming negligible radial variations in temperature over the cross-section of the wire and negligible radiation heat transfer, the general equation is (Bruun, 1995; Højstrup et al., 1976):

$$\frac{\rho_{wire} c_{wire}}{k_{wire}} \frac{\partial T_{wire}}{\partial t} = \frac{\partial^2 T_{wire}}{\partial x^2} - \overbrace{\left(\frac{h_{ambient} \pi d_{wire}}{A_{wire} k_{wire}} - \frac{\alpha_{wire} I^2 \rho_{ref}}{A_{wire}^2 k_{wire}} \right)}^{Biot\ number/l_{wire}} T_{wire} + \frac{h_{ambient} \pi d_{wire}}{A_{wire} k_{wire}} T_{ambient} - \frac{I^2 \rho_{ref}}{A_{wire}^2 k_{wire}} (\alpha_{wire} T_{ref} - 1) \quad (1)$$

The steady state solution of the general equation can be found by assuming that:

$$h_{ambient} \pi d_{wire} > \frac{\alpha_{wire} I^2 \rho_{ref}}{A_{wire} k_{wire}} \quad (2)$$

$$\frac{\bar{T}_{wire} - \bar{T}_{ambient}}{T_{mean,wire} - \bar{T}_{ambient}} = \left(1 - \frac{\cosh[\sqrt{Bi/l_{wire}x}]}{\cosh[Bi]} \right) / \left(1 - \frac{\tanh[\sqrt{Bi/l_{wire}x}]}{Bi} \right) \quad (3)$$

which is only a function of the Biot number. By assuming uniform flow conditions over the wire, a heat balance can be written between the heat

3 Methods

dissipation by the wire and the heat losses due to conduction and convection. Bradshaw (1971) showed that if $Bi > 3$, the heat balance equation becomes:

$$E^2 = A + BU^n \quad (4)$$

Using $n = 0.5$ results in the famous analytical solution of the flow around a cylinder that was obtained by King (1914).

3.1.1.3 Velocity sensitivity (HWA calibration)

An HWA must be calibrated by establishing a relationship between a set of known velocities and the bridge voltages. The calibration is performed by exposing the sensor to a free jet or a low-turbulence flow in a wind tunnel. After measuring the velocity and the corresponding output voltage, two types of relationships can be made using curve fitting: an exponential function (King equation) or a quartic polynomial function. The accuracy of the former curve fit is lower than the latter, especially in a turbulent flow, due to the dependency of the power n on the velocity (Bruun, 1979). The calibration using the quartic polynomial must be performed for the entire range of desired velocities.

3.1.1.4 Accuracy of the measurements

According to ISO/IEC Guide 98-3:2008, each individually measured quantity and the output results contain uncertainties at a certain confidence level. Therefore, the overall accuracy of each individually acquired voltage and obtained velocity must be estimated in HWA measurements. The different sources of uncertainties (such as calibration, instruments and measurement conditions) must be considered to estimate the accuracy of each individual sample in the velocity series. The largest contribution of uncertainties originates from the calibration and the temperature variation of HWA. Experimental conditions (e.g., probe position, ambient pressure and humidity variation), curve fitting (linearization) to the calibration results and the resolution of the analog/digital board during data acquisition make up as the other sources of uncertainty.

3.1.2 Laser Doppler anemometry (LDA)

The LDA technique was introduced by Yeh and Cummins (1964), and then widely employed in many research and industrial applications for

3.1 Experimental fluid dynamics

measurements in turbulent flows. The instantaneous velocity vector can be accurately measured by the LDA method. In this section, the basic principles of dual-beam LDA configuration are briefly described. A more comprehensive and complete description of Laser Doppler Anemometry can be found in Absil (1995); Albrecht et al. (2010); Durrani and Greated (1977); Durst et al. (1981); Somerscales (1981); Tummers (1999).

The LDA is a non-intrusive technique, which is the main advantage over conventional measuring methods (e.g., HWA or pitot tubes). The Doppler frequency is linearly related to the velocity component, which the single calibration factor depends on the geometry of optical arrangement and frequency of laser light. The spatial and temporal resolution of LDA technique is high, especially the former in comparison to HWA (Tummers, 1999). Another superiority of the LDA technique over HWA is the ability of measuring reverse flow.

The single point measurement by LDA technique is complex and relatively expensive. The sampling time of the velocity of randomly distributed tracer particles in the measuring volume becomes random, which makes the data processing complicated. The spectral analysis of velocity fluctuations measured by LDA is not possible by using the standard methods such as fast Fourier transform due to the random sampling time. The randomly sampled data of the moving tracer particles depends on the flow velocity, which means that a velocity bias may occur. Since LDA requires scattering light from particles, the flow must be seeded and the tracer particles must follow the flow faithfully.

The Doppler shift is the basic physical principal of LDA. LDA is an indirect measuring technique, since this system measures the velocity of tracer particles that are following the flow assuming without any slip. In this technique, a monochromatic coherent laser with frequency f_{beam} is split to two equal intensity beams.

$$f_{beam} = \frac{\text{speed of light } (c)}{\lambda_{beam}} \quad (5)$$

Since the laser beams are intense and directional, a measurable amount of radiation is scattered from tracer particles. The crossing point of the two beams creates the measurement volume, in which the interference of two scattering waves on a stationary heterodyne detector creates a fringe pattern.

3 Methods

Due to the Doppler shift, both scattered beams consist of light waves with slightly different frequencies, f_{wave1} and f_{wave2} . By optical mixing of the scattered waves on the surface of a square-law photo detector, the difference frequency (i.e., Doppler frequency $f_{Doppler}$), which shows the oscillation of the output signal, is obtained.

$$f_{wave1} = f_{beam} + \frac{\mathbb{V}_{particle}(e_{particle-detector} - e_{wave1})}{\lambda_{beam}} \quad (6)$$

$$f_{wave2} = f_{beam} + \frac{\mathbb{V}_{particle}(e_{particle-detector} - e_{wave2})}{\lambda_{beam}} \quad (7)$$

$$f_{Doppler} = f_{wave1} - f_{wave2} = \frac{\mathbb{V}_{particle}(e_{wave1} - e_{wave2})}{\lambda_{beam}} \quad (8)$$

$$f_{Doppler} = \frac{2 \sin \frac{\theta}{2}}{\lambda_{beam}} |\mathbb{V}_{particle}| \sin \alpha \quad (9)$$

where $\mathbb{V}_{particle}$ is the velocity vector of the seeding particle, $e_{particle-detector}$ is the unit vector showing the direction of scattered light from moving particle on the detector, e_{wave1} and e_{wave2} are the unit vectors denoting the direction of split incident beams, λ_{beam} is the wavelength of the laser light and the angles θ and α denote the angle between the two split light beams and the angle of flow direction with respect to the perpendicular of the beam bisector. It is noteworthy to mention that the difference frequency (8) does not depend on the position of the receiver for the dual-beam configuration. In order to collect enough scattered beams from the moving particles and to provide strong output signals, the optical aperture of the photo detector is required to be large. According to equation (9), the frequency difference is linearly proportional to the velocity component normal to the interference fringes.

3.1.2.1 Tracer particles

The seed or tracer particles need to faithfully track the fluid motion; therefore, they need to be sufficiently small. Rapid changes in the fluid velocity must be followed by the tracer particles; particles that are too large or too heavy would exhibit a delayed response. The lag time of spherical particles, τ_l , in a viscous fluid at a low Reynolds number is defined as an associated characteristic time scale by the Stokes drag law:

3.1 Experimental fluid dynamics

$$\tau_l = \frac{(\rho_p - \rho)d_p^2}{18\mu} \quad (10)$$

where ρ_p is the particle density, ρ is the fluid density, and μ is the dynamic viscosity of the fluid. The typical diameter of the particles, d_p , is from 0.1 to 10 microns and 10 to 100 microns for gas flows and liquid flows, respectively. Dispersion of prefabricated solid particles or the generation of atomized liquid droplets are typical methods for seeding particles into the gas flows. The most common technique for generating aerosol tracers for PIV or LDA measurements in airflows is to use several Laskin nozzles together with various nonpoisonous liquids, such as glycerol/water or olive oil. The pressurized air will atomize the liquid inside a container and produce bubbles that contain an aerosol of particles. After the bubbles rise to the surface, the smallest particles are transported out of the airtight container, while larger particles remain in the liquid after collision with a horizontal circular impactor plate. A pressure difference (e.g., 0.5 to 1.5 bar) between the inlet and outlet of the liquid container causes seeding with the desired mean particle diameters ($d_p \cong 1$ micron).

Generally, the light scattered by small particles is a function of the ratio of the refractive index of the particles to the surrounding medium and the size, shape and orientation of the particles. In addition, the scattering of light depends on the polarization and observation angle (Albrecht et al., 2010; Raffel et al., 2007). On the other hand, the particle diameter should not be too small to ensure enough light scattering; therefore, a trade-off between particle size and laser light intensity must be found. By performing this optimization, the particles will scatter a sufficient light intensity and also follow the fluid motion accurately with a smaller lag time than the typical time scale of the flow.

3.1.2.2 Fringe model

Since the concept of fringe model is simple, it has been used for clarifying some principle aspects of dual beam LDA configurations (Absil, 1995). The fringe model is an alternative way to provide a physical meaning to the conversion factor of Doppler frequency to the velocity (9). However, the fringe model should be used with some doubt, since it is not correct (Durst, 1982) in the way that the interference occurs on the surface of square-law photo detector and not in the measuring volume.

3 Methods

The intensity of light in the overlap region of the both incident laser beams varies periodically and the spacing between two consecutive maxima in the interference pattern is defined by $\frac{\lambda_{beam}}{2 \sin \frac{\theta}{2}}$. This quantity is the fringe distance, which is the proportionality constant between the velocity component of a tracer particle perpendicular to the fringe plane and the measured Doppler frequency (9).

3.1.2.3 Directional ambiguity

The Doppler frequency recorded by the square-law photomultiplier depends on the magnitude of velocity of the scattering tracer particle through the measurement volume not on the direction. In order to remove directional ambiguity in the velocity measurements with LDA, the frequency of one of the incident beams is shifted by a constant value f_{shift} . The frequency shifting can be achieved with an acousto-optic Bragg cell. Due to the frequency shifting, the relationship between Doppler frequency and particle velocity is given by (assuming $f_{shift} \ll f_{beam}$)

$$f_{Doppler} = f_{shift} + \frac{2 \sin \frac{\theta}{2}}{\lambda_{beam}} |\mathbf{v}_{particle}| \sin \alpha \quad (11)$$

The frequency shifting provides a fixed offset to the relationship between the Doppler frequency and particle velocity. If the f_{shift} is chosen larger than the $f_{Doppler}$, which corresponds to the smallest velocity in the flow, each absolute value of the Doppler frequency is uniquely attributed to one velocity value and the directional ambiguity is removed. According to the fringe model, the frequency shifting of one of the incident beams results in a fringe pattern moving with a velocity equals to $\frac{f_{shift} \times \lambda_{beam}}{2 \sin \frac{\theta}{2}}$.

3.1.2.4 Velocity bias

The velocity of small particles, which are randomly distributed in space, are measured with a random sampling time by LDA when the tracer seeding move through the measuring volume. The time between consecutive samples, i.e. inter-arrival time, is assumed to obey Poisson statistics. George (1978) has theoretically showed that if there is no correlation between instantaneous data rate and the instantaneous velocity of the fluid, the mean velocity and variance of velocity fluctuations can be computed from simple

3.1 Experimental fluid dynamics

arithmetic averages of velocity samples, similar to calculations from HWA data. However, McLaughlin and Tiederman (1973) recognized earlier that there is a correlation, commonly referred to as velocity bias, between instantaneous data rate and the instantaneous velocity of the fluid when LDA measurements are made in a time-varying flows. The effect of velocity bias on arithmetic averaging of one-dimensional flow velocity is exemplified by Tummers (1999). Tummers (1999) illustrated that the arithmetic average of velocity measured by LDA is higher than the true mean velocity and even the effect of velocity bias increases with increasing turbulence intensity. Hence, the probability of measured velocity is biased towards high velocities and the velocity statistics processed as arithmetic averages of LDA samples are erroneous in the presence of velocity bias (Geers, 2004; Tummers, 1999).

There are two categories of data-processing methods to correct the velocity bias, namely sampling procedures or weighting factors. The sampling procedures can be used for acquiring the unbiased velocity data by reconstructing the original signal or by approximately equidistant sampling of velocity. The data density (mean data rate $\times$ characteristic time scale of the velocity) is an important factor and requires being high in sampling procedures. The weighting factors can also be used to compute the bias-free moments of velocity from discrete randomly-sampled data. A correct estimation for the first moment and the second central moment of velocity can take the following form

$$\bar{u} = \frac{\sum_{i=1}^N u_i w_i}{\sum_{i=1}^N w_i} \quad (\text{mean velocity}) \quad (12)$$

$$\overline{u'^2} = \frac{\sum_{i=1}^N u_i'^2 w_i}{\sum_{i=1}^N w_i} \quad (\text{variance of velocity}) \quad (13)$$

where the index i denotes the i -th velocity sample ($u_i = \bar{u} + u'$), the total number of samples is N and w is the weighting factor.

There are several frequently weight factors, such as inverse velocity, transit time, inter-arrival time and free-running processor. Among the commonly used weight factors, transit time weight factor is reliable at all data densities and exhibits a very low bias, while the inter-arrival time weight factor shows acceptable results for data densities exceeding a minimum limit and the free-running processor yields a maximum error (Albrecht et al., 2010). The inverse velocity weight factor uses the

3 Methods

magnitude of the velocity and assumes a spherical measurement volume, which means all three components of velocity must be measured otherwise a systematic over-correction occurs (Albrecht et al., 2010; Tummers, 1999). Since almost all the commercial signal processors provide reliable estimates of the transit time for each tracing particle move through the measurement volume together with the reliability of transit-time weighting to compensate for the velocity bias, transit time weight factor is recommended to be used by Albrecht et al. (2010); Tummers (1999).

3.1.3 Particle image velocimetry (PIV)

PIV is a measurement technique in which small tracer particles that have almost the same density as fluid are seeded into the flow. A digital camera records light scattering of the tracer particles along a plane of illumination. Illumination by laser light at least twice within a short time interval, enables digital recording on a sequence of frames. The displacement pattern of the tracer particles is determined using a correlation theorem. By knowing the displacement and time interval, instantaneous velocities in the fluid flow can be measured or evaluated.

This section discusses the most important aspects of single-exposure double-frame particle image velocimetry. More comprehensive treatments of the PIV technique can be found in Westerweel (1993), Raffel et al. (2007) and Adrian and Westerweel (2010).

3.1.3.1 Image acquisition

Double short-pulsed light sheet illuminates the particles. Illuminating the micrometer-sized particles requires an intense light source. The pulse duration of Nd:YAG lasers ($\lambda = 532$ nm) is on the order of several nanoseconds with a constant pulse energy, which is an advantage of this type of solid-state laser to prevent from visible blurring of the particle images. The mono-chromatic and collimated light rays make Nd:YAG lasers ideal for use in PIV systems. Low timing jitter and lighting homogeneity are other reasons for using Nd:YAG lasers for PIV measurements.

The Nd:YAG lasers emit light repetitively at rates between 10 and 30 Hz (Adrian and Westerweel, 2010) and two lasers are combined into a dual-cavity system with precision optics to produce double pulses with the time interval of a few microseconds. A quality switch (Q-switch) can be operated

3.1 Experimental fluid dynamics

inside the lasers by allowing the cavity to resonate at the most energetic pulse to adjust the time interval between two independent illuminations of the tracer particles. Q-switches are usually composed of a polarizer and a Pockels cell.

The laser beam has to be formed into a light sheet using optical lenses. A cylindrical lens (e.g., plano-concave) is essential for generating a thin light sheet from a sufficiently small and divergent beam. The height and thickness of the light sheet can be simultaneously controlled using a second spherical lens (e.g., plano-convex). To correct the diverging light sheet into a parallel constant-height illumination plane, a third cylindrical lens can be used.

The scattered light of particles crossing the light sheet is subsequently recorded into a solid-state image detector. A pinhole camera and a compound lens are required to record the locations of the tracer particles in the illuminated volume of fluid. The intensity distribution of the scattered light should accurately map the location of each tracer particle onto the image plane (e.g., a solid-state sensor or photographic film). Presently, PIV is usually performed with digital cameras with solid-state image detectors, Charge Coupled Devices (CCDs) or Complementary Metal Oxide Semiconductors (CMOSs). Digital cameras with CCD chips have better spatial resolution and can record two temporally spaced images. CCD sensors convert the light energy during an exposure into electric charge. A CCD chip contains an array (typically 0.3 to 16 million) of individual semiconductor elements (typically 3 -10 μm) called pixels.

Unavoidable noise sources for the image detectors include Johnson thermal noise, shot or readout noise, and switching noise. CCD sensors must be cooled during recording to avoid image degradation by thermal effects on the semiconductor. Because every 6-7 $^{\circ}\text{C}$ temperature increase doubles the rate of thermal noise generation, cooled CCD chips are required in scientific imaging. The conversion or quantization of the electron charge to voltage is called shot noise. The shot noise increases with an increase of the readout frequency, which limits the number of image acquisitions per unit of time (typically 3 – 200 frames per second).

The directional ambiguity can be solved by exposing different frames to the separated light of the sequential recording. For example, a single exposure double frame PIV requires a multiple oscillator Nd:YAG laser and

3 Methods

a high-speed camera. The camera must acquire two images on a full frame CCD chip within a time interval of a few microseconds. In the frame transfer sensor, a temporary memory array on the CCD chip receives all of the pixel charges immediately after the first exposure. After the second exposure of the pixels of the image detector, the pair of images is transferred to the memory of a PC using a sequential read-out (progressive scan) on a row-by-row basis to calculate the velocity vectors (Adrian and Westerweel, 2010).

The image created by the scattered light from a small-size seeding particle has a finite diameter due to the effect of diffraction and aberration by the lens. A Fraunhofer diffraction pattern or an Airy disk is caused by the finite wavelength of the light and the finite diameter of the lens aperture, and does not include image broadening by geometric aberrations of the lens. The minimum diffraction-limited spot diameter d_{diff} can be obtained for a particular imaging configuration (Adrian and Westerweel, 2010; Raffel et al., 2007):

$$d_{diff} = 2.44(1 + M)\lambda f^\# \quad (14)$$

$$f^\# = \frac{f}{D_a} \quad (15)$$

where $f^\#$ denotes the f -number of the imaging lens, f is the focal length and D_a is the aperture diameter, M denotes the ratio of the image distance from the effective center of the lens to the object distance from the effective center of the lens (i.e., image magnification factor), and λ is the laser light wavelength. An estimate of the particle image diameter d_τ with a Gaussian shape on the image sensor can be obtained from the following formula:

$$d_\tau \cong (M^2 d_p^2 + d_{diff}^2)^{0.5} \quad (16)$$

A $1 \mu\text{m}$ particle in a typical PIV system with a Nd:YAG laser ($\lambda = 532 \text{ nm}$), a magnification factor of 0.1 and a f -number of 5.6 generates an image with a diameter of $8 \mu\text{m}$. Optimizing the particle image diameter after focusing the camera in the PIV technique minimizes the uncertainty in the velocity measurements and increases the particle image intensity. To perform this optimization, the terms of the diffraction-limited imaging should be adjusted. However, the size of the field of view and the imaging sensor determines the image magnification factor, and choosing the laser

3.1 Experimental fluid dynamics

fixes the wavelength of the light, so the f -number can mostly control the particle image diameter. A trade-off between the particle image diameter and the particle image intensity must be considered; i.e., an increase in the f -number results in a larger d_τ , while the decrease of D_a decreases the intensity of the collected light on the imaging sensor. Suitable selection of the seeding particles to increase the particle image intensity (the contrast of the PIV recordings) is more economically feasible than increasing the laser power.

3.1.3.2 Digital analysis

The entire field of view of the flow in the single-exposure double-frame PIV recording is divided into a grid of comparatively small interrogation spots. It is assumed that the velocity field is smooth and the motion is uniform in the interrogation spots; i.e., there is no distortion in the particle image during the time interval between the two exposures. The pair of two singly exposed PIV recordings are evaluated by locally cross-correlating each interrogation spot in the first image with the corresponding spot in the second image.

The interrogation window size and weighting function can be changed during the cross-correlation procedure and optimized locally based on the local seeding density and flow gradients. A higher spatial resolution and accuracy can be reached using an adaptive interrogation window. Non-uniform weighting functions other than square interrogation windows, such as round Gaussian, elongated elliptical Gaussian and cosine functions, can be used to calculate the cross-correlation between image pairs.

The same PIV image pair is repeatedly interrogated while the dimension of the interrogation grid is reduced in each subsequent pass until a desired refined spot is achieved. In the first pass, a large interrogation area (e.g., 32×32 pixels) is cross-correlated, and the spurious vectors are identified and replaced by interpolated vectors. The resulting vector field is interpolated to match a finer grid (e.g., 16×16 pixels), which is used for the subsequent interrogation pass. After substituting the spurious vectors, the procedure is repeated until the desired resolution is obtained.

The flows contain a wide range of length scales, thus finer grids of velocity vectors are an advantage. The use of a larger interrogation area in the multigrid procedure results in a smaller in-plane loss of correlation

3 Methods

and confident detection of spurious vectors. An analysis of the performance of the multigrid procedure (Adrian and Westerweel, 2010) showed this approach is not necessarily time consuming.

Fast Fourier transformation is used for the efficient numerical computation of the cross-correlation. The Fourier transform of the interrogation spot in the first image is multiplied by the complex conjugate of the Fourier transform of the corresponding interrogation spot in the second image. The result of this multiplication is then inverse Fourier transformed (FT^{-1}) to yield the displacement cross-correlation peak.

The displacement correlation peak must be detected at sub-pixel accuracy after cross-correlation of the interrogation spots from the image pairs. Due to discretization of the input data, the highest correlation peak can be estimated at an integer number of pixels with an uncertainty of $\pm 1/2$ pixel. A sub-pixel displacement estimator can improve the detection of the location of the correlation peak. Because correlation peaks have short width, three adjoining pixels of the correlation peak can be taken into account for the estimation. This method is called the three-point estimator.

Centroiding and function fitting have been used to estimate the location of the correlation peak in the past. Separating correlation peaks from background noise with centroiding is unambiguous, but this method works best with wide correlation peaks. A more robust and widespread three-point estimator is to fit a Gaussian function to the narrow correlation peaks, such as an airy-shaped intensity distribution from a particle image of 2-3 pixels. The correlation between two Gaussian functions results in a Gaussian function. The formula for the Gaussian peak fit is defined as:

$$f(x) = C \exp \left[\frac{-(x_P - x)^2}{k} \right] \quad (17)$$

$$(x_P - i) = \frac{\ln(P_{(i-1,j)}/P_{(i+1,j)})}{2 \ln(P_{(i-1,j)}P_{(i+1,j)}/P_{(i,j)}^2)} \quad (18)$$

$$(y_P - j) = \frac{\ln(P_{(i,j-1)}/P_{(i,j+1)})}{2 \ln(P_{(i,j-1)}P_{(i,j+1)}/P_{(i,j)}^2)} \quad (19)$$

where x_P and y_P are the exact coordinates of the peak, i and j are the integer coordinates of the peak, $P_{(i,j)}$ is the value of the correlation function at (i, j) , and $P_{(i-1,j)}$, $P_{(i+1,j)}$, $P_{(i,j-1)}$ and $P_{(i,j+1)}$ are the four adjoining

3.1 Experimental fluid dynamics

correlation values. The two-dimensional Gaussian peak estimator provides even higher accuracy using more immediate values around the correlation peak. The 2-D Gaussian fit can recover the aspect ratio and skew of the correlation maximum (Raffel et al., 2007; Ronneberger et al., 1998). To increase the accuracy, a three-point estimator using a Gaussian peak fit can be used to determine the four highest displacement correlation peaks. A bias error contributes to the three-point estimator if the diameter of the particle image is too small ($d_\tau < 1.5$ pixel). Displacement bias errors tend to have integral values, which is called peak locking. To decrease this effect, the particle image diameter must be adjusted around two pixels ($d_\tau > 2$ pixels).

The locus of the correlation peak with respect to the center of the interrogation spot represents the average in-plane particle-image displacement vector ΔX . The local mean velocity vector u can then be determined by dividing this by the time between the two exposures Δt . In this PIV technique, different sizes and/or slightly displaced interrogation spots can be chosen to compensate for the in-plane loss of cross-correlation.

The result of a successful interrogation is the identification (maximization) of the displacement correlation peak, which has been investigated extensively by Keane and Adrian (1992); Keane and Adrian (1990, 1991). They used Monte-Carlo simulations to determine the requirements for the experimental parameters to yield optimal performance of the PIV analysis (Westerweel, 1993). The height and width of the correlation peak due to the displacement can be increased by the particle diameter, the particle density (N_I) in the interrogation spots of both images and the number of exposures (N_P). The particle image density (N_I) can be computed from the particle concentration (c), light sheet thickness (Δz_0), interrogation spot area (A_I) and magnification factor (M) as follows:

$$N_I = \frac{c\Delta z_0 A_I}{M^2} \quad (20)$$

N_I must be more than 10 (average) to unambiguously detect the peak from the noise. At the same time, the loss of pairs effect can be minimized by following the golden quarter rules. The height and width of the correlation peak can be reduced by the in-plane loss of correlation (F_I), the out-of-plane loss of correlation (F_0) and the velocity gradients (F_Δ) over the interrogation spot. The maximum F_I of the particle image pattern should not

3 Methods

exceed one quarter of the width (d_I) of an interrogation spot on the image plane:

$$M|u_{max}|\Delta t < \frac{1}{4}d_I \quad (21)$$

where u_{max} is the maximum in-plane velocity. The F_0 must also be limited to one quarter of Δz_0 :

$$|w_{max}|\Delta t < \frac{1}{4}\Delta z_0 \quad (22)$$

where w_{max} is the maximum velocity normal to the laser sheet.

Strong spatial gradients over the interrogation spot can broaden the displacement correlation peak and decrease its amplitude, which can prevent the detection of the peak. This effect due to velocity gradients is called detection bias. When

$$M|\Delta u|\Delta t \leq d_\tau \quad (23)$$

the detection probability is acceptable, and Δu is the velocity difference over the interrogation spot.

Gradient bias is an additional effect of velocity variations at small interrogation spots with a limited light sheet thickness. When the correlation peak is statistically biased towards velocities that are lower than the average velocity in the measured volume, the gradient bias occurs. Therefore, the gradient bias should not exceed the maximum velocity gradient within the interrogation spot. The following criterion must be met to minimize the gradient bias:

$$M|\Delta u|\Delta t \leq 0.05\sqrt{A_I} \quad (24)$$

Therefore, to obtain a sufficiently accurate velocity vector at an interrogation spot, several criteria with the parameters described above should be met.

3.1.3.3 Detection of spurious vectors

Erroneous correlation peaks due to noise or artifacts may be detected during the evaluation procedure. The magnitude or direction of spurious velocity vectors, or outliers, markedly deviate from their immediate

3.2 Computational fluid dynamics (CFD)

neighbors. Several automatic statistical algorithms have been used to detect and eliminate outliers before performing further processing of the flow field. The most commonly used technique is a local approach for PIV data validation called median filtering, which was proposed by Westerweel (1994); Westerweel and Scarano (2005). The median center vector is calculated from the eight immediate neighbors that are located in a three by three matrix. The median vector from all of the neighboring vectors can be calculated with respect to either the magnitude of the velocity vector or the individual components. The absolute deviation (residual) of a measured velocity vector ($u_{(i,j)}$) at position (i,j) and the median of the surrounding vector ($u_{median-surrounding(i,j)}$) at (i,j) should remain below a threshold:

$$|u_{(i,j)} - u_{median-surrounding(i,j)}| < \alpha \sigma_{surrounding(i,j)} \quad (25)$$

where $\sigma_{surrounding(i,j)}$ is the standard deviation of the magnitudes of the surrounding vectors, and α is a confidence parameter. The probability of having at least two spurious vectors in the immediate neighborhood is given by a binomial distribution in which the probability is less than 14.9% (Westerweel, 1994). The spurious vectors are detected and discarded, and the empty spaces are filled with vectors that are generated by bilinear interpolation of the neighboring vectors. The bilinear interpolation fulfills the continuity (Raffel et al., 2007).

3.2 Computational fluid dynamics (CFD)

Computational fluid dynamics (CFD) is a numerical method for the prediction (simulation) of fluid flow systems using computers. CFD is a powerful additional problem-solving method and is a vital component in many applications. The CFD technique has increasingly been integrated in a wide range of industrial and scientific communities due to the rapid development of affordable high-performance computers. CFD can be used in both the early and final design stages of products and processes. Applying CFD can substantially reduce the cost and time of new designs. Because the CFD technique can produce extremely large and detailed results, parametric studies and optimization can be performed at a lower cost than corresponding experimental measurements. Development of the CFD could not reach its ultimate capabilities due to the complexity of fluid flows, therefore, there is no substitute for measurement. However, CFD

3 Methods

simulations can be carried out in locations where taking measurements is difficult or impossible due to hazardous conditions and the size of the systems.

Pre-processing, solving and post-processing are the three building blocks of a successful CFD simulation. Pre-processing consists of specifying the domain geometry, discretizing (grid or mesh generation) the computational domain into nodes or cells, and specifying the physical models (e.g., turbulence models, incompressibility, steady or unsteady state), fluid properties, boundary conditions, initial conditions, convergence criteria and numerical schemes to approximate the derivatives and pressure-velocity coupling. The definition of the computational domain and grid generation of the region of interest are the most important and time-consuming (almost 50%) stages of a successful simulation. To perform a grid-independent simulation, mesh generation should be repeated by refining an initial coarse grid until the CFD results do not change. The optimized grid-independent mesh is usually fine in areas that contain rapid changes and coarse in regions with small variations.

The solutions to the governing differential equations can be approximated using distinct numerical techniques, including finite difference, finite element, finite volume, and spectral methods. For example, in the finite volume method, the governing equations of the fluid flow are integrated over all of the finite volumes in the computational domain. The resulting integrals are then converted/discretized into a system of algebraic equations that are solved in an iterative algorithm until obtaining a converged solution with small residuals.

CFD simulations can produce an extremely large data volume of results with an almost unlimited level of detail. To obtain insight and meaningful outputs, post-processing of the CFD simulations is important. Post-processing includes displaying the geometry and grid of the computational domain and accessing the results to display x - y graphs, 2-D or 3-D vectors and/or contour plots, and animations of the dynamic data.

3.2.1 Governing equations

The characteristics of turbulence, which is a flow property rather than a fluid, include irregularity, diffusivity, three-dimensionality, time-dependency, and dissipativity. A turbulent flow contains unsteady eddies at a

3.2 Computational fluid dynamics (CFD)

wide range of scales larger than molecular scales (continuum phenomena). Kolmogorov (1941) found that micro-scales of length, time and velocity can be uniquely determined by the dissipation rate ϵ and viscosity ν . Small eddies, which are statistically isotropic at sufficiently high Reynolds numbers and have a universal statistical form, dissipate turbulent kinetic energy into heat. The turbulent kinetic energy is extracted from the largest eddies by the mean flow and is transferred to successively smaller scales by the energy cascade process until being dissipated by viscous processes.

Isothermal fluid flow is governed by the conservation laws of mass and momentum. The Navier-Stokes equations cannot be solved analytically except for very simple cases. The equation system (i.e., Navier-Stokes) can be solved numerically without turbulence, which is called Direct Numerical Simulation (DNS). DNS requires the resolution of the entire range of spatial and temporal scales of the turbulence. Due to the high computational cost even for flows at low Reynolds numbers, DNS is a powerful tool in fundamental research of turbulence and is useful in the development of turbulence models for practical applications.

Resolving the large scales that contain energy and modelling the smaller scales is called Large Eddy Simulation (LES). The anisotropic large eddies, which extract turbulent kinetic energy from the three-dimensional unsteady mean flow, are dependent on the geometry and boundary conditions. LES has a lower computational cost than DNS due to the elimination of the small scales.

By separating the instantaneous velocity and pressure into mean and fluctuating parts (Reynolds decomposition), the time-averaged continuity and Reynolds equations for steady state, three-dimensional, incompressible and turbulent conditions are given by:

$$\frac{\partial U_i}{\partial x_i} = 0 \quad (26)$$

$$\underbrace{\frac{\partial(U_j U_i)}{\partial x_j}}_{\text{Convective acceleration}} = \underbrace{-\frac{1}{\rho} \frac{\partial p}{\partial x_i}}_{\text{Pressure gradients}} + \underbrace{\frac{\partial}{\partial x_j} \left(\underbrace{\nu \frac{\partial U_i}{\partial x_j}}_{\text{Viscosity}} - \overline{u'_i u'_j} \right)}_{\text{Divergence of forcing terms (stress)}} \quad (27)$$

3 Methods

where $\overline{u'_i u'_j}$ is the second moment of the statistical correlation and is known as the Reynolds stress. To close this system of equations, the Reynolds stress term requires modeling, which is called turbulence modeling. The turbulence model are selected based on some considerations, e.g., the physics of the flow, the insight into the capabilities and limitations of turbulence models, the attempt for the specific problem by other researchers, the accuracy needed, the available computational resources and time.

3.2.2 Boussinesq approach versus Reynolds stress model

A popular and common approach to modelling the Reynolds stresses in equation (27) is to apply the Boussinesq hypothesis (Hinze, 1975), which relates them to the mean velocity gradients as follows:

$$-\overline{u'_i u'_j} = \nu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \quad (28)$$

The Boussinesq hypothesis is employed in the two equations turbulence models (e.g., Realizable $k - \epsilon$). Only two additional transport equations for the turbulence kinetic energy, k (29), and the turbulence dissipation rate, ϵ (30), need to be solved. Therefore, ν_t is assumed to be an isotropic scalar quantity and is computed as a function of k and ϵ . This approach requires a relatively low computational cost that is associated with the modeling the turbulent viscosity (ν_t).

The transport equations for the Reynolds stresses can be solved with an additional equation for the turbulence dissipation rate ϵ . This requires solving five more transport equations, so the Reynolds stress model requires greater computational expense but is superior to the $k - \epsilon$ model in cases of turbulence anisotropy.

3.2.3 Realizable $k - \epsilon$ model (R $k - \epsilon$)

The realizable $k - \epsilon$ model (R $k - \epsilon$) was proposed by Shih et al. (1995). The transport equations for k and ϵ in R $k - \epsilon$ are given by:

$$\frac{\partial}{\partial x_j} (k U_j) = \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + 2 \nu_t S_{ij} S_{ij} - \epsilon \quad (29)$$

3.2 Computational fluid dynamics (CFD)

where $\sigma_k = 1.0$, and $S_{ij} = \frac{1}{2} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right)$

The transport equation for the dissipation rate, ϵ , is based on an exact and dynamic equation for the transport of the mean-square vorticity fluctuation. The new form of this equation has a desirable feature that provides better spectral energy transfer, because the production term (second portion of the right-hand side of equation (30)) does not contribute to the production of k . Equation (30) can have a non-singular feature by eliminating k in the denominator of the third term on the right-hand side.

$$\frac{\partial}{\partial x_j} (U_j \epsilon) = \frac{\partial}{\partial x_j} \left[\left(v + \frac{v_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + 2C_1 S_{ij} S_{ij} \epsilon - C_2 \frac{\epsilon^2}{k + \sqrt{v\epsilon}} \quad (30)$$

where $C_1 = \max \left[0.43, \frac{\eta}{\eta+5} \right]$, $\eta = S_{ij} \frac{k}{\epsilon}$, $C_2 = 1.9$, $\sigma_\epsilon = 1.2$.

The turbulent viscosity, v_t , is modeled as:

$$v_t = C_\mu \frac{k^2}{\epsilon} \quad (31)$$

where k and ϵ are the turbulent kinetic energy and the dissipation rate of k , respectively, but C_μ is no longer constant. In the Realizable $k - \epsilon$ model, C_μ is a function of the mean strain S_{ij} , rotation rates $\tilde{\Omega}_{ij}$, angular velocity ω_k and turbulence fields (k and ϵ) and C_μ is computed from:

$$C_\mu = \frac{1}{A_0 + A_s \frac{kU^*}{\epsilon}} \quad (32)$$

where

$$U^* = (S_{ij} S_{ij} + \tilde{\Omega}_{ij} \tilde{\Omega}_{ij}) \quad (33)$$

and

$$\tilde{\Omega}_{ij} = \overline{\Omega_{ij}} - \epsilon_{ijk} \omega_k \quad (34)$$

where $\overline{\Omega_{ij}}$ is the mean rate-of-rotation tensor in a rotating reference frame with an angular velocity ω_k . The constants of the turbulent viscosity modeling are:

3 Methods

$$A_0 = 4.04, \quad A_s = \sqrt{6} \cos \left(\frac{1}{3} \cos^{-1} \left(\sqrt{6} \frac{S_{ij}S_{jk}S_{ki}}{(S_{ij})^{1.5}} \right) \right) \quad (35)$$

The standard value of $C_\mu = 0.09$ for an inertial sub-layer in an equilibrium boundary layer can be computed from equation (32). By applying the new formulation of C_μ , the normal Reynolds stresses $(\overline{u'_i u'_i})$ can be realizable (positive) even when the mean strain rate is large; therefore, inequality (36), which is derived from the Boussinesq relationship, is satisfied. In addition, the Schwarz inequality (37) for shear stresses cannot be violated.

$$\frac{1}{3C_\mu} > \frac{k}{\epsilon} \frac{\partial U_i}{\partial x_j} \quad (36)$$

$$\overline{u_\alpha u_\beta}^2 \leq \overline{u_\alpha^2 u_\beta^2} \quad (37)$$

3.2.4 The shear stress transport (SST) $k - \omega$ model with low Reynolds correction

The Boussinesq hypothesis is employed in the low Reynolds number shear stress transport (SST) $k - \omega$ model. Only two additional transport equations for the turbulence kinetic energy, k , and the specific dissipation rate, ω , must be solved. Therefore, v_t is assumed to be an isotropic scalar quantity and is computed as a function of k and ω . This approach has a lower computational cost that is associated with modeling the turbulent viscosity (v_t) than the alternative approach of solving six transport equations for each of the terms of the Reynolds stresses plus the additional scale-determining equation.

Menter (1994) developed the shear-stress transport (SST) $k - \omega$ model by blending a robust formulation of the standard $k - \omega$ model (Wilcox, 2010) for free shear flows (e.g., round jets) in the near-wall region with free stream independence of the transformed $k - \epsilon$ in the far field. The transport equations for k and ω in the SST $k - \omega$ model are given by:

$$\frac{\partial}{\partial x_j} (k U_j) = \frac{\partial}{\partial x_j} \left[\left(v + \frac{v_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + \tilde{G}_k - \beta^* k \omega \quad (38)$$

3.2 Computational fluid dynamics (CFD)

$$\frac{\partial}{\partial x_j}(U_j \omega) = \frac{\partial}{\partial x_j} \left[\left(v + \frac{v_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\alpha}{\mu_t} \tilde{G}_k - \beta \omega^2 + D_\omega \quad (39)$$

The turbulent Prandtl numbers σ_k and σ_ω for k and ω , respectively, are used in the effective diffusivities.

$$\sigma_k = \frac{1}{\tanh(\Phi_1^4)/1.176 + (1 - \tanh(\Phi_1^4))} \quad (40)$$

$$\sigma_\omega = \frac{1}{\tanh(\Phi_1^4)/2 + (1 - \tanh(\Phi_1^4))/1.168} \quad (41)$$

$$\Phi_1 = \min \left[\max \left(\frac{\sqrt{k}}{0.09\omega y}, \frac{500\mu}{\rho y^2 \omega} \right), \frac{4\rho k}{1.168 D_\omega^+ y^2} \right] \quad (42)$$

$$D_\omega^+ = \max \left[2\rho \frac{1}{1.168\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-10} \right] \quad (43)$$

y is the distance to the next surface, and D_ω^+ is the positive portion of the cross-diffusion term. The turbulent viscosity, v_t , is defined as follows, where $S = \sqrt{2S_{ij}S_{ij}}$ is the strain rate magnitude:

$$v_t = \frac{k/\omega}{\max \left[\frac{1}{\alpha^*}, S \frac{\tanh(\Phi_2^2)}{0.31\omega} \right]} \quad (44)$$

$$\Phi_2 = \max \left[2 \frac{\sqrt{k}}{0.09\omega y}, \frac{500\mu}{\rho y^2 \omega} \right] \quad (45)$$

The coefficient α^* damps the turbulent viscosity and causes a low Reynolds number correction, which is written as below:

$$\alpha^* = \left(\frac{0.024 + k/6\nu\omega}{1 + k/6\nu\omega} \right) \quad (46)$$

where $\tanh(\Phi_1)$ and $\tanh(\Phi_2)$ are blending functions that approach one in the near-wall region, which activates the standard $k - \omega$ model, and become zero away from surface, which activates the transformed $k - \epsilon$ model. The addition of the production terms (G_k and G_ω) to the k -equation and the ω -equation improves the accuracy of the standard $k - \omega$ model for predicting free shear flows. The term $\tilde{G}_k$ describes the production of

3 Methods

turbulence kinetic energy and is computed as follows for the incompressible form:

$$\tilde{G}_k = \min(v_t S^2, 10\beta^* k\omega) \quad (47)$$

$$\beta^* = 0.09 \left(\frac{4/15 + (k/8\nu\omega)^4}{1 + (k/8\nu\omega)^4} \right) \quad (48)$$

The second term of the right-hand side of the ω transport equation represents the production of specific dissipation, and the coefficient α is given by:

$$\alpha = \frac{\left(\frac{0.075}{0.09} - \frac{0.41^2}{2\sqrt{0.09}} \right) \tanh(\Phi_1^4) + (1 - \tanh(\Phi_1^4)) \left(\frac{0.0828}{0.09} - \frac{0.41^2}{1.168\sqrt{0.09}} \right)}{\alpha^*} \left(\frac{1/9 + k/2.95\nu\omega}{1 + k/2.95\nu\omega} \right) \quad (49)$$

For the incompressible form, the third term of the right-hand side of the ω transport equation represents the dissipation of ω , and the term β is evaluated as follows:

$$\beta = 0.075 \tanh(\Phi_1^4) + 0.0828(1 - \tanh(\Phi_1^4)) \quad (50)$$

The transformation of the standard $k - \epsilon$ model into equations that are based on k and ω leads to the introduction of a cross-diffusion term, D_ω , which can be computed as:

$$D_\omega = 2.336(1 - \tanh(\Phi_1^4)) \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (51)$$

D_ω is generated from the $k - \epsilon$ model and vanishes in the $k - \omega$ model when the blending function equals one.

In the mixing layer of free shear flows, the introduction of blending functions in the cross-diffusion term improves the production of specific dissipation, ω , which in turn enhances the dissipation of turbulent kinetic energy, k ; i.e., the net production of k is reduced. Although vortex stretching is very small in axisymmetric boundary layers, it causes the energy transfer from large to small eddies and is nonzero for an

3.2 Computational fluid dynamics (CFD)

axisymmetric mean flow, which corresponds to a radial stretch of the vortex rings.

3.2.5 The Reynolds stress model (RSM)

The Reynolds-averaged Navier-Stokes equations are closed by solving the transport equations for the Reynolds stresses and the transport equation for the dissipation rate. Seven additional transport equations are solved using this approach and the isotropic eddy-viscosity hypothesis is not needed. The exact form of the transport equations for the Reynolds stresses $\overline{u'_i u'_j}$ is derived as follows:

$$\begin{aligned}
 \underbrace{\frac{\partial}{\partial x_k} (\overline{U_k u'_i u'_j})}_{C_{ij} \equiv \text{Convection}} &= \underbrace{\frac{\partial}{\partial x_k} \left[-\overline{u'_i u'_j u'_k} - \frac{p'}{\rho} (\delta_{jk} u'_i + \delta_{ik} u'_j) \right]}_{D_{T,ij} \equiv \text{Turbulent Diffusion}} \\
 &+ \underbrace{\frac{\partial}{\partial x_k} \left[v \frac{\partial \overline{u'_i u'_j}}{\partial x_k} \right]}_{D_{L,ij} \equiv \text{Molecular Diffusion}} - \underbrace{\left(\overline{u'_i u'_k} \frac{\partial U_j}{\partial x_k} + \overline{u'_j u'_k} \frac{\partial U_i}{\partial x_k} \right)}_{P_{ij} \equiv \text{Stress production}} \\
 &- \underbrace{2v \frac{\partial \overline{u'_i}}{\partial x_k} \frac{\partial \overline{u'_j}}{\partial x_k}}_{\epsilon_{ij} \equiv \text{Dissipation}} + \underbrace{\frac{p'}{\rho} \left(\frac{\partial \overline{u'_i}}{\partial x_k} + \frac{\partial \overline{u'_j}}{\partial x_k} \right)}_{\Phi_{ij} \equiv \text{Pressure strain}}
 \end{aligned} \tag{52}$$

The terms in these equations, including C_{ij} (convection), $D_{L,ij}$ (molecular diffusion) and P_{ij} (production), do not require modeling. However, $D_{T,ij}$ (turbulent diffusion), ϵ_{ij} (dissipation) and Φ_{ij} (pressure strain) must be modeled to close the equations. The following section describes the modeling assumptions that are required to close the equation set. $D_{T,ij}$ can be modeled by the generalized gradient-diffusion model of Daly and Harlow (1970):

$$D_{T,ij} = C_s \frac{\partial}{\partial x_k} \left(\rho \frac{\overline{k u'_k u'_\ell}}{\epsilon} \frac{\partial \overline{u'_i u'_j}}{\partial x_\ell} \right) \tag{53}$$

This equation can result in numerical instabilities, so it was simplified by Lien and Leschziner (1994) to use a scalar turbulent diffusivity as follows:

$$D_{T,ij} = \frac{\partial}{\partial x_k} \left(\frac{v_t}{\sigma_k} \frac{\partial \overline{u'_i u'_j}}{\partial x_k} \right) \tag{54}$$

3 Methods

The turbulence viscosity, ν_t , is modeled as (7), where k and ϵ are the turbulent kinetic energy and dissipation rate of k , respectively, and the constant $C_\mu = 0.09$. The kinetic energy is given by:

$$k = \frac{1}{2} \overline{u'_i u'_i} \quad (55)$$

The transport equation for the turbulence kinetic energy is solved to obtain the wall boundary conditions for the Reynolds stresses with $\sigma_k = 0.82$. Equation (29) can also be obtained by contracting the modeled equation for the Reynolds stresses (equation (52)).

The scalar dissipation rate of k , ϵ , is computed with a model transport equation and is given by:

$$\frac{\partial}{\partial x_j} (U_j \epsilon) = \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_{\epsilon 1}} \right) \frac{\partial \epsilon}{\partial x_j} \right] - C_{\epsilon 1} \overline{u'_i u'_i} \frac{\partial U_i}{\partial x_j} \frac{\epsilon}{k} - C_{\epsilon 2} \frac{\epsilon^2}{k} \quad (56)$$

where $\sigma_{\epsilon 1} = 1.0$, $C_{\epsilon 1} = 1.44$, and $C_{\epsilon 2} = 1.92$. The dissipation tensor, ϵ_{ij} , is modeled with the isotropic dissipation assumption as:

$$\epsilon_{ij} = \frac{2}{3} \delta_{ij} \epsilon \quad (57)$$

The pressure-strain term, Φ_{ij} , is modeled by a classical linear approach, see Fluent (2012). The modeling of Φ_{ij} uses the following decomposition:

$$\Phi_{ij} = \Phi_{ij,1} + \Phi_{ij,2} + \Phi_{ij,w} \quad (58)$$

where $\Phi_{ij,1}$ is the slow pressure-strain term (or the return-to-isotropy term) and is modeled as:

$$\Phi_{ij,1} = -C_1 \frac{\epsilon}{k} \left(\overline{u'_i u'_j} - \frac{2}{3} \delta_{ij} k \right) \quad (59)$$

$\Phi_{ij,2}$ is the rapid pressure-strain term and is modeled as:

$$\Phi_{ij,2} = -C_2 \left((P_{ij} - C_{ij}) - \frac{1}{3} \delta_{ij} (P_{kk} - C_{kk}) \right) \quad (60)$$

$\Phi_{ij,w}$ is the wall-reflection term that is responsible for the redistribution of the normal stresses near the wall. It is likely to damp the stresses normal

3.2 Computational fluid dynamics (CFD)

to the wall while enhancing the stresses parallel to the wall and is modeled as:

$$\begin{aligned}\Phi_{ij,w} = C_1^w \frac{\epsilon}{k} & \left(\overline{u'_k u'_m n_k n_m} \delta_{ij} - \frac{2}{3} \overline{u'_i u'_k n_j n_k} - \frac{2}{3} \overline{u'_j u'_k n_i n_k} \right) \frac{C_\ell k^{3/2}}{\epsilon b} \\ & + C_2^w \left(\Phi_{km,2} n_k n_m \delta_{ij} - \frac{2}{3} \Phi_{ik,2} n_j n_k \right. \\ & \left. - \frac{2}{3} \Phi_{jk,2} n_i n_k \right) \frac{C_\ell k^{3/2}}{\epsilon b}\end{aligned}\quad (61)$$

where n_k is the n_k component of the unit normal to the wall, b is the normal distance to the wall, and $C_\ell = C_\mu^{3/4}/\kappa$, where $\kappa = 0.4187$ is the von Karman constant. Using the enhanced wall treatment for near wall flows, the RSM model needs to be modified for the constants C_1 , C_2 , C_1^w and C_2^w as a function of the Reynolds stress invariants near the wall. According to Launder and Shima (1989):

$$C_1 = 1 + 2.58A\sqrt{A_2}(1 - \exp[-(0.0067Re_t)^2]) \quad (62)$$

$$C_2 = 0.75\sqrt{A} \quad (63)$$

$$C_1^w = -\frac{2}{3}C_1 + 1.67 \quad (64)$$

$$C_2^w = \max\left(\frac{2}{3} + \frac{1}{6C_2}, 0\right) \quad (65)$$

where the turbulent Reynolds number, flatness parameter A and tensor invariants A_2 and A_3 are defined as:

$$Re_t = \left(\frac{k^2}{\nu\epsilon}\right) \quad (66)$$

$$A = 1 - \frac{9}{8}(A_2 - A_3) \quad (67)$$

$$A_2 = a_{ik}a_{ki} \quad (68)$$

$$A_3 = a_{ik}a_{kj}a_{ji} \quad (69)$$

where a_{ij} is the Reynolds stress anisotropy tensor, which is defined as:

$$a_{ij} = \frac{\overline{u'_i u'_j}}{k} - \frac{2}{3}\delta_{ij} \quad (70)$$

3.2.6 Low Reynolds (low Re) $k - \epsilon$

A general form for the turbulence model equations described above can be written as follows:

$$\overline{u_j} \frac{\partial \overline{\phi}}{\partial x_j} - \frac{\partial}{\partial x_j} \left[\Gamma_{\phi,eff} \frac{\partial \overline{\phi}}{\partial x_j} \right] = S_\phi \quad (71)$$

where ϕ represents the variables, $\Gamma_{\phi,eff}$ is the effective diffusion coefficient, and S_ϕ represents the source term of an equation, and all of the coefficients and source terms can be found in Table 1. Equations (28) and (71) briefly express the mathematical formula of the selected turbulence models. The low Re $k - \epsilon$ model consists of the three modified damping functions, f_μ , $C_{\epsilon 1}^*$ and $C_{\epsilon 2}^*$, and two additional terms (D and E) compared to the standard $k - \epsilon$ model. The inclusion of the damping functions accounts for the viscosity effects, and the additional terms account for anisotropic dissipation. All of the model values become constants at high Reynolds numbers, while C_μ and $C_{\epsilon 2}^*$ depend on the turbulence Reynolds number (Re_t) at low Reynolds numbers. The term D is included only for computational reasons because the zero value of ϵ has several disadvantages in the k -equation. The value of D becomes negligible in regions with high Reynolds numbers. E has no physical importance and is used only to increase the computed dissipation rate, which allows the realistic prediction of turbulent kinetic energy. These major modifications to the low Re $k - \epsilon$ model improve the performance in regions with low Re numbers.

3.2 Computational fluid dynamics (CFD)

Table 1: Coefficients and source terms for the low Reynolds models

ϕ	$\Gamma_{\phi,\text{eff}}$	S_ϕ	Modeling and their constants
Low Re $k - \epsilon$	k	$\nu + \frac{\nu_t}{\sigma_{k,t}}$	$v_t = \frac{f_\mu C_\mu k^2}{\epsilon}; f_\mu = \exp\left[-\frac{3.4}{\left(1 + \frac{Re_t}{50}\right)^2}\right];$
		$G_k - \epsilon + D$	$D = 2v_t \left(\frac{\partial k^{0.5}}{\partial x_\perp}\right)^2; E = \frac{2\mu v_t}{\rho} \left(\frac{\partial^2 u_\parallel}{\partial^2 x_\perp}\right)$
	ϵ	$\nu + \frac{\nu_t}{\sigma_{\epsilon,t}} - \frac{C_{\epsilon 1}^* G_k \epsilon}{\rho k} - \frac{C_{\epsilon 2}^* \epsilon^2}{k} + E$	$C_{\epsilon 1}^* = 1.44; C_{\epsilon 2}^* = 1.92[1 - 0.3\exp(-Re_t^2)]; G_k = 2v_t S_{ij} S_{ij}$

3.2.7 Transition models ($k - kl - \omega$ and Transition SST $k - \omega$)

The two-equation Reynolds-Averaged Navier-Stokes (RANS) turbulence models with the Boussinesq approximation simulate the flow as fully turbulent. Additional equations, which are not necessarily required with the Boussinesq approximation to solve closure problems, may provide more accurate predictions of the laminar, transition from laminar to turbulent, and turbulent regimes (especially the onset of the transition). Two different transition turbulence models, a three-equation ($k - kl - \omega$) transition model and a four-equation (transition SST $k - \omega$) transition model, are included in the round jet study. These turbulence models were mostly tuned to predict the transition in the boundary layers. The general form (71) of the transition turbulence models can be used, and the coefficients and source terms can be found in Table 2. In the transition $k - kl - \omega$ model, the transport equations of the turbulent kinetic energy, k_T , and the laminar kinetic energy, k_L , are solved separately. These two transport equations and the transport equation of the specific dissipation rate (turbulence frequency), ω , compose a three-equation closed system and determine the turbulent eddy viscosity, $\nu_t = \frac{k_L + k_T}{\omega}$. The terms k_L and k_T represent the pre-transitional (laminar) velocity fluctuations and the magnitude of the fluctuations of the fully turbulent flow, respectively. The production (P_{k_L}) of k_L is assumed from the interactions between the mean shear and pre-transitional velocity fluctuations; i.e., from large-scale turbulent fluctuations. The energy transfer from laminar kinetic energy (k_L) to turbulent kinetic energy (k_T) expresses the transition process. P_{k_T} represents the turbulent production that is generated by turbulent fluctuations and contains a damping function due to intermittency. The term k_T consists of small-scale energy ($k_{T,s}$) and large-scale energy ($k_{T,l}$), and D_T and D_L represent the dissipation of turbulent and laminar kinetic energies, respectively. The averaged breakdown of the streamwise fluctuations into turbulence during the bypass transition is included as R . The natural transition production term (R_{NAT}) represents the breakdown to turbulence due to instabilities. The onset of the transition is assumed to arise at sufficiently short time scales of turbulence production dynamics compared to the time scale of molecular diffusion.

3.2 Computational fluid dynamics (CFD)

The TrSST $k - \omega$ model (Menter et al., 2006) uses the damping functions of the underlying turbulence model to predict the transition. A local correlation-based transition model has been proposed by coupling two new equations to the original SST $k - \omega$, thus forming a four-equation model. One transport equation for the intermittency, γ , determines the onset of the transition, while one transport equation for the transition momentum thickness Reynolds number, Re_{θ_t} , is included as a scalar quantity to prevent additional nonlocal operations. Re_{θ_t} depends on the local and free stream turbulent intensities, which are attributed to the diffusion of the experimentally corrected free stream value of Re_{θ_t} into the boundary layer. The original production (P_k) and destruction (D_k) terms of the turbulent kinetic energy from the SST turbulence model are used in the TrSST $k - \omega$ model. The effective intermittency (γ_{eff}) is only used in $\widetilde{P}_k$ and $\widetilde{D}_k$ to control the source terms in the k -equation; it is not employed to multiply the eddy viscosity. The inclusion of γ_{eff} has two advantages. The first is enhancing the robustness of the model due to an indirect use of intermittency in the momentum equation because of the difficulty in linearizing the empirical correlations in the source terms. The second is improving the prediction of the effects of high levels of turbulence in the free stream; i.e., the small values of intermittency cannot remove the local eddy viscosity when the eddy viscosity of the free stream is large. The final change to the SST $k - \omega$ is redefining the blending functions to always become one when a laminar layer is present.

3 Methods

Table 2: Coefficients and source terms for the transition models

ϕ	$\Gamma_{\phi, \text{eff}}$	S_ϕ	Modeling and their constants
k_T	$\nu_t + \frac{\nu_t}{\sigma_k}$	$P_{k_T} + R - \omega k_T - D_T + \bar{u}_j \frac{\partial k_T}{\partial x_j}$	$k_T = k_{T,l} + k_{T,s}; k_{T,s} = \frac{\lambda_{eff} k_T}{\lambda_T} e^{-\left(\frac{C_{SSV\Omega}}{k_T}\right)^2}; D_T = \nu \frac{\partial \sqrt{k_T}}{\partial x_j} \frac{\partial \sqrt{k_L}}{\partial x_j}; D_L = \nu \frac{\partial \sqrt{k_L}}{\partial x_j} \frac{\partial \sqrt{k_L}}{\partial x_j}$
k_L	ν	$P_{k_L} - R - D_L + \bar{u}_j \frac{\partial k_L}{\partial x_j}$	$P_{k_T} = S^2 \lambda_{eff} \left(1 - e^{-\frac{\lambda_{eff} \sqrt{k_T}}{\lambda_T A_v \sqrt{\nu \omega}}}\right) \left(\min\left(\frac{k_L}{C_{int} k_{TOTAL}}, 1.0\right)\right) \left(\frac{1}{A_0 + A_s(S/\omega)}\right) \sqrt{k_{T,s}}$
Transition $k - kl - \omega$	$\nu + \frac{\nu_t}{\sigma_\omega}$	$\frac{\omega}{k_T} \left(C_{\omega 1} P_{k_T} + \left(\frac{C_{\omega R}}{f_W} - 1\right) R\right) - C_{\omega 2} \omega^2 + C_{\omega 3} f_\omega \alpha_T f_W^2 \sqrt{k_T} d^3 + \bar{u}_j \frac{\partial \omega}{\partial x_j}$	$R = R_{BP} + R_{NAT}; R_{BP} = \frac{C_{Rk_L \omega \lambda_T}}{\lambda_{eff}} \left(1 - e^{-\frac{\psi_{BP}}{A_{BP}}}\right); R_{NAT} = C_{R,NAT} k_L \Omega \left(1 - e^{-\frac{\psi_{NAT}}{A_{NAT}}}\right); P_{k_L} = S^2 \min\left\{f_{\tau,l} C_{11} \lambda_{eff} \left(\frac{\Omega \lambda_{eff}^2}{\nu}\right) \sqrt{k_T} + \beta_{TS} C_{12} Re_\Omega d^2 \Omega, \frac{(k_{T,l} + k_L)}{2S}\right\};$
k	$\nu + \sigma_k \nu_t$	$\bar{P}_k - \bar{D}_k$	$\bar{P}_k = \gamma_{eff} P_k; \bar{D}_k = \min(\max(\gamma_{eff}, 0.1), 1.0) D_k; F_1 = \max(F_{1orig}, e^{\left(-\frac{\gamma \sqrt{k}}{120\nu}\right)});$
Transition SST $k - \omega$	$\nu + \sigma_k \nu_t$	$\alpha \frac{P_k}{\nu_t} - D_\omega + C d_\omega$	$F_{\theta t} = \min\left\{\max\left[e^{-[Re_\omega/(E+5)]^2} e^{-(\nu U^2/375\Omega \widetilde{Re_{\theta t}})^4}, 1.0 - \left(\frac{\gamma - 1/c_{e2}}{1 - 1/c_{e2}}\right)^2\right], 1.0\right\}$
γ	$\nu + \frac{\nu_t}{\sigma_\gamma}$	$P_{\gamma 1}(1 - \gamma) + P_{\gamma 2}(1 - \gamma c_{\gamma 2})$	$P_{\gamma 1} = F_{length} S [\gamma F_{onset}]^{c_{\gamma 3}}; P_{\gamma 2} = (2c_{\gamma 1}) \Omega \gamma F_{turb}; F_{turb} = e^{-\left(\frac{Re_t}{4}\right)^4};$
$Re_{\theta l}(\nu + \nu_t)$	$\sigma_{\theta t}(\nu + \nu_t)$	$P_{\theta t}$	$P_{\theta t} = \frac{c_{\theta t} U^2}{500\nu} (Re_{\theta t} - \widetilde{Re_{\theta t}})(1.0 - F_{\theta t}); Re_v = \frac{\nu^2}{\nu} \Omega; Re_\theta = \frac{Re_{\theta max}}{2.193};$

3.2 Computational fluid dynamics (CFD)

3.2.8 Near wall treatment in the $k - \epsilon$ and RSM

The viscosity-affected near wall layer up to the viscous sub-layer is completely resolved using an enhanced wall treatment. The first near wall node is placed at $y^+ = 1$, which provides sufficiently fine near-wall mesh to resolve the viscous sub-layer, so the integral part of the enhanced wall treatment is identical to the two-layer approach. The traditional two-layer approach specifies both ϵ and ν_t in the near-wall cells and subdivides the entire domain into a viscosity-affected region and a fully turbulent region. The wall distance-based turbulent Reynolds number, Re_b , determines the demarcation of the two regions as:

$$Re_b = \frac{bk^{0.5}}{\nu} \quad (72)$$

where b is the normal distance to the nearest wall cell centers, which is independent of the mesh topology.

The $k - \epsilon$ models (R $k - \epsilon$ and low Re $k - \epsilon$) and the Reynolds stress model (RSM) are employed in the fully turbulent region; i.e., for $Re_b > Re_b^*$; $Re_b^* = 200$. When Re_b is less than Re_b^* , the near wall treatment (the one-equation model of Wolfshtein (1969)) is used to capture the viscosity-affected near wall region. The momentum equations and the turbulent kinetic energy equation (5) are solved in the one-equation model. Here, the dissipation rate of k is computed inside the viscosity-affected region with an algebraic relation for ϵ (not by solving the transport equation (18)), which is given by:

$$\epsilon = \frac{k^{3/2}}{\ell_\epsilon} \quad (73)$$

where the turbulent length scale, ℓ_ϵ , proposed by Chen and Patel (1988) is given by:

$$\ell_\epsilon = bC_\epsilon^*(1 - \exp[-(Re_b/A_\epsilon)]) \quad (74)$$

A blending of ϵ is used for a smooth transition between the explicitly specified dissipation rate in the inner region and the value of ϵ obtained by solving the transport equation (18) in the outer region. The constants in the equations are taken from Chen and Patel (1988) and are as follows:

3 Methods

$$C_\ell^* = C_\ell^{-1}, A_\epsilon = 2C_\ell^* \quad (75)$$

The turbulence viscosity $\nu_{t,ewt}$ consists of an enhanced wall treatment part (two-layer approach) that is smoothly blended with the high-Re outer region turbulence viscosity ν_t . The proposed turbulence viscosity of Jongen (1998) is given by:

$$\nu_{t,ewt} = \lambda_\epsilon \nu_t + (1 - \lambda_\epsilon) \nu_{t,2layer} \quad (76)$$

where ν_t is identical to the high-Reynolds definition (7). The two-layer turbulent viscosity, $\nu_{t,2layer}$, is described as follows:

$$\nu_{t,2layer} = C_\mu \ell_\mu k^{0.5} \quad (77)$$

in which the length scale that appears in equation (38) is computed from:

$$\ell_\mu = bC_\ell^* (1 - \exp[-(Re_b/A_\mu)]) \quad (78)$$

where $A_\mu = 70$. A blending function, λ_ϵ , prevents mismatches of ν_t in the outer layer with the one that obtained by the Wolfshtein (1969) model at the edge of the viscosity-affected region. The blending function is defined as:

$$\lambda_\epsilon = \begin{cases} 0 & \text{in the vicinity of wall} \\ 1 & \text{far away from wall} \end{cases} \quad (79)$$

$$\lambda_\epsilon = \frac{1}{2} \left[1 + \tanh \left(\frac{Re_b - Re_b^*}{A} \right) \right] \quad (80)$$

The width of blending function, A , is constant and is defined as:

$$A = \frac{|\Delta Re_b|}{\tanh^{-1}(0.98)} \quad (81)$$

where $0.05Re_b^* < \Delta Re_b < 0.2Re_b^*$, which gives $0.01\lambda_\epsilon$ of its far field value.

3.3 Response surface methodology

A parametric study is the analysis of the trend between dependent and independent variables to improve the design of a system, process or product. In a traditional parametric study, the effect of one independent variable on a response (dependent) of interest is investigated at a time. Only one variable is varied, while the other influencing factors remain constant. This

3.3 Response surface methodology

technique cannot provide the relationship between the response and interactive effects between the associated variables. A large number of experiments as well as increased time and cost are required to conduct parametric studies one variable at a time. A multivariate mathematical and statistical approach can be used to determine the relationship between a response and several associated factors (variables). The response surface methodology is one type parametric study; it is a collection of mathematical and statistical techniques that are used to develop an approximate model between a response ($y_{response}$) and real design factors ($x_1, x_2, \dots, x_n$) (Khuri and Mukhopadhyay, 2010). A parametric study that uses the response surface methodology consists of five stages. The first is to determine which independent variables are significant in terms of affecting a dependent response. A wide range of factors may influence a particular response, but it is only effective to select the significant variables with the greatest influence. Superior ($x_{i|max}$) and inferior ($x_{i|min}$) levels of each independent factor must be selected. The narrower the range between the upper and lower levels of variables that is determined, the more accurately the model is approximated.

The real design factors (x_i) differ in terms of the order of magnitude and the dimensional unit. The investigation of the response surface by regression models is influenced by the high or low values of the real variables or their units. The real design factors need to be codified to dimensionless values that are proportional to their levels. The real variables (x_i) can commonly be transformed to coded factors (X_i) according to a determinate design by the following equation:

$$X_i = 2 \frac{x_i - x_{i|mean}}{x_{i|max} - x_{i|min}} \quad (82)$$

where $x_{i|mean}$ is the arithmetic mean of the upper ($x_{i|max}$) and lower ($x_{i|min}$) levels of the real design factors. Equation (82) produces coded variables in the design space, which is bounded by the upper ($X_{i|max} = +1$) and lower ($X_{i|min} = -1$) levels with the central point at $X_{i|mean} = 0$.

The design of the experiment influences the accuracy and effectiveness of the model that is approximated by the response surface methodology, and determines the experimental points, the levels of the variables, and the numbers of blocks and runs. The experiment is designed as an n -

3 Methods

dimensional space that is limited to the inferior and superior levels of the independent variables. Two important models are commonly used in the response surface methodology: the first-order model and the second-order model. In the former model, such as a 2^n factorial design, the response does not have any curvature and is related to the independent variable by a linear relationship. The latter polynomial function contains additional terms that are necessary to describe a second-order interaction. The approximated response surface (Y) is represented by the following full quadratic equation, which describes the two-way interaction of n design factors ($x_1, x_2, \dots, x_n$):

$$y_{response} = \beta_0 + \sum_{i=1}^n \beta_i x_i + \sum_{i=1}^n \beta_{ii} x_i^2 + \sum_{1 \leq i < j}^n \sum \beta_{ij} x_i x_j + \varepsilon \quad (83)$$

where β_0 , β_i , β_{ii} and β_{ij} represent the intercept, linear, quadratic and interaction coefficients of the approximated regression model, respectively, and ε is the random error, which is assumed to have a normal distribution with the mean at zero.

The 3^n factorial, central composite and Box-Behnken design are the most frequently used second-order symmetrical designs (Khuri and Mukhopadhyay, 2010). The full three-level (i.e., low, middle and high) factorial design includes all of the combinations of the n design variables. The number of runs for $n > 2$ becomes very large compared to other second-order designs, so the 3^n factorial design loses its practical efficiency due to the increased cost. The central composite design (Box and Wilson, 1951) is augmented by the 2^n factorial design and $2n$ additional runs together with the central point of the design space. A response surface investigation by the central composite design first fits a first-order model, and additional runs determine the importance of the quadratic and interaction terms. The Box-Behnken design requires only three levels ($-1, 0, +1$) for each independent variable and includes inter-blocking combinations of the 3^n factorial model. The total number of runs for the Box-Behnken design is defined as $No. = 2n(n - 1) + C_0$, where n is the number of independent real factors, and C_0 is the number of central points. In comparison, the full three-level factorial and central composite designs use $No. = 3^n$ and $No. = 2^n + 2n + C_0$ runs, respectively. The total number of runs in the Box-Behnken model is less than in the other two designs for $n > 2$. Comparisons between the designs described above can be found in Ferreira et al. (2007), who showed that the

3.3 Response surface methodology

Box-Behnken design is slightly more efficient than the other two models. The efficiency of the designs is defined as the number of coefficients in the approximated full-quadratic regression equation divided by the number of runs. Another advantage of the Box-Behnken design is that it avoids cases with simultaneous extreme conditions (i.e., at least one of the variables is kept at the center point level $X_{i|mean} = 0$).

After acquiring the results of different cases based on the chosen (Box-Behnken) design, a second-order mathematical equation is statistically approximated by least-squares multiple regression. The fitted model describes the behavior of the response according to the design factors. The least squares method minimizes the sum of the squared residuals of the overall approximation. The difference between the real and approximated response values is called the residual.

The mathematical model of the relationship between the response and design factors must be evaluated in the domain of study. The first reliable term to determine the quality of the approximated equation is the residuals. The residuals are expected to have a normal Gaussian shape, and the residual probability is expected to follow a straight line. Several patterns of the normal probability of the residuals, such as following a curved line, having curved tails or changes in the slope of the line, are indications of non-normality, skewness or an unidentified variable, respectively. The appearance of a point far from the straight line indicates the existence of an outlier in the approximated model. The residuals of the approximated values should be scattered randomly about zero; any pattern, such as uneven spreading or a curvilinear pattern, indicates a non-constant variance or a missing higher-order term. The statistical significance of the design factors in the approximated regression model can be determined using the probability value (P -value) in the analysis of variance (ANOVA). A P -value < 0.05 indicates that the design factor or its square or interaction is statistically significant at a 95% confidence level. A backward elimination approach based on a stepwise regression model can be used to select significant variables. Each step consists of removing the term with the largest P -value from the full quadratic model. In every step, the deletion of the term should not cause an appreciable increase in the sum of the squares of the residual, which can be assessed by a statistical test. The elimination process terminates when further improvement in the regression equation is below a critical value. The surface plots can be depicted from the

3 Methods

approximated full-quadratic relationship between the response and design factors. The most common graphical representation is a plot of the relationship between the response and two real variables, while the other factors remain constant.

4 Case Study

Experiments were performed in four different configurations: a single round jet, a single row of coplanar parallel jets, an array of round jets on a flat surface and on a curved surface. In this chapter, the test room, which was used for flow measurement of the single jet and the confluent jets on a curved surface, is discussed. The flow measurements and its numerical prediction conducted for the single round jet and multiple interacting jets on the curved surface are presented in this chapter. The wind tunnel that provided the airflow for the single row of jets and confluent jets on a flat surface is discussed. The flow measurements performed in the single row of jets and multiple interacting jets on a flat surface are presented.

4.1 Inlet boundary conditions of confluent jets on a curved surface

Previous research (see the section 2.1.1) has shown that the shape, height, contraction ratio and Reynolds number of the nozzle influence the inlet profile and upstream and downstream flow regions (initial, transition and turbulent) of a single round jet. Therefore, the inlet boundary conditions of confluent jets must be predicted numerically due to the difficulties in taking measurements at nozzle exits. A prototype for a supply device of confluent jets is modeled with two RANS turbulence models (Realizable $k - \epsilon$ ($R k - \epsilon$) and the Reynolds Stress Model (RSM)), and the results of the numerical simulation were then evaluated with HWA near the nozzle outlets.

All of the HWA experiments were performed at the University of Gävle (Department of Building, Energy and Environmental Engineering) laboratory. The size of the well-insulated test room is 4.2 m (length) $\times$ 3.6 m (width) $\times$ 2.5 m (height), and the temperature can be controlled to within 0.1 °C. The test room is located inside an operating room that is insulated from the ambient air of the main laboratory. The wooden walls of the test room include a water pipe system for temperature control. The doors of the test and operating rooms remained closed during all of the measurements. All internal heat generators (e.g., lights in the test chamber) were turned off

4 Case Study

during the measurements, to keep the model and test room as isothermal as possible.

Two labeling methods were used for the confluent jets on the curved surface. The first method was standard labeling of the individual jets based on their position in the array. The array of nozzles is labeled by the row and column numbers (Figure 3); i.e., 1R-4C stands for the nozzle from the first row from the top and the fourth column from the left as viewed from the front. In this configuration, three different types of confluent jets are present based on the number of direct neighboring jets: jets in the side rows or columns with five neighboring jets (Side Jets [SJ]; e.g., 1R-4C, 4R-6C and 6R-4C), jets at the corners with three neighboring jets (Corner Confluent Jets [CoJ]; e.g., 1R-6C and 6R-6C) and jets in the middle of the array with eight neighboring jets (Confluent Jets [CJ]; e.g., 3R-4C and 4R-4C).

The airflow is supplied by a centrifugal mechanical fan with straight radial blades. The power of the blower is controlled by a frequency regulator to obtain the desired airflow rate. The supply-air pipe is connected to the fan and an orifice flowmeter, and a flexible rubber duct is connected to a straight 2-m-long aluminum pipe to absorb mechanical vibrations of the fan. The model is placed at the end of the 2 m long pipe. No filter, such as honeycomb, was used for this measurement. The air exits the test room freely through a rectangular exhaust on one of the side walls.

Figure 2: Top-view sketch of the test room.

4.1 Inlet boundary conditions of confluent jets on a curved surface

The air supply device (Figure 3) is mounted at the center of the longest wall of the test room for the HWA measurements. A square array of thirty-six nozzle outlets with identical exit diameters (d_0) of 0.0058 m is located on a cylindrical tube that is blocked from one side (Figure 2 and Figure 3). Low velocity airflow enters the supply device axially from a long pipe and then radially exits from the nozzles. The instantaneous axial velocity across two local coordinates (x and z) at different nozzles in the array was measured by HWA at the closest possible distance ($0.26d_0$) to the nozzle exits for $Re = 3290$. The predicted axial velocities and their root mean square (RMS) were verified with measurements.

Figure 3: Test room (side view), supply device for the study of confluent jets on the curved surface (front view) and nozzle design.

A 55P11 constant-temperature hot-wire anemometric bridge was used to measure the velocity. The straight single-sensor miniature wire with especially long prongs (5 mm) is made by DANTEC, and the one-dimensional platinum-plated tungsten probe is 5 μ m in diameter and 1.25 mm long. The general-purpose probe was operated at a temperature of 200 $^{\circ}$ C with an overheat ratio of approximately 0.64. The hot-wire probe was

4 Case Study

calibrated by low-speed and high-speed open loop wind tunnels that are located in a temperature-controlled test room. Fourth-order polynomial least squares regression was used to evaluate the velocity calibration data. The data were acquired with a National Instrument NI USB-6215.

The turbulence levels measured in this study are less than 9 %. Bruun (1995) showed that rectification problems are very limited below this level because of homogeneous and isotropic turbulence. The relative expanded uncertainty of the velocity measurements for the single-sensor hot-wire probe in the air is 4.2 %. The sources of uncertainty and their relative standard uncertainties can be summarized as follows: calibration (0.02), linearization (curve fitting) (0.005), analogue/digital board resolution (0.0013), probe position (≈ 0), air and sensor temperature variations (0.01), ambient pressure variations (0.006) and humidity (≈ 0).

A temperature sensor was placed near the anemometer probe to ensure isothermal conditions. The position temperature sensor did not disturb the flow and recorded the heating of the air by the hot-wire probe. Several thermocouples were installed on the walls, ceiling, floor, and exhaust and at different heights in the center of the room to control the temperature of the test rig.

A LabVIEW computer program was used for highly automated data gathering because of the large volume of data and the long acquisition times. A traversing system with an uncertainty of 0.0125 mm was used for the automated movement of the hot-wire probe and the temperature sensor. The three-dimensional traversing system was also controlled by the LabVIEW program.

The axial velocity component in the nozzle cross section was measured using a single-wire sensor. Each set of measurements includes a series of data along the cross section of the nozzles at a distance of $0.26 d_0$ downstream from the exit section with a separation of $0.17 d_0$. Each measurement covered $1.55 d_0$ of the nozzle exit across the nozzle center, which started and ended in the stagnant surrounding fluid.

The general measurement procedure was as follows:

- The blower power was adjusted to provide the desired outlet velocity.
- The acquisition parameters were set in the LabVIEW program.

4.1 Inlet boundary conditions of confluent jets on a curved surface

- The hot-wire sensor was positioned at its starting point (e.g., $y = 0.26 d_0$ downstream of one nozzle exit and 1.5 mm below or to the side of the nozzle).
- The LabVIEW program was launched.
- The direction of probe movement was set to guarantee the statistical independence between different situations and the hot-wire swept diameter of the target nozzles.
- Each data point was logged at 5 kHz for 180 seconds.

The air supply device was modelled to obtain all of the inlet boundary conditions for all of the nozzles. Numerical predictions were also performed for a different nozzle shape (i.e., long pipe outlet) and three Reynolds numbers to compare the nozzle profiles. The computational domain is a cylindrical tube that is blocked on one side (Figure 3). The tube has one round inlet with a diameter $D = 0.125$ m into which the airflow enters axially at a low velocity and is then radially issued from thirty-six nozzle outlets that are located on the body of the model. The columns of nozzle axes are coplanar and parallel on a curved surface, while the rows of nozzle axes are coplanar and do not intersect in the downstream direction. The rows of nozzle axes are aimed obliquely away from each other, and a diverging configuration is formed by the axes. Despite the 15° angle between the axes of the jets in the rows, the length (or arc length) of the spacing ($S = 2.82 d_0$) between the rows and columns of nozzle axes was designed to be identical for the confluent jets on the curved surface. The pressure outlet boundaries are placed at $100 d_0$ from the openings by axial extrusion of the nozzle outlets. Overpressure is present inside the model and no back flow occurs in the nozzles, thus the extruded small pipes over the nozzles do not affect the nozzle profiles. The geometrical details and fluid properties are summarized in Table 3.

A pipe is also simulated with the standard $k - \epsilon$ model ($S k - \epsilon$) and the Reynolds stress model (RSM) to provide the inlet profile for the supply device. The diameter (D) and mesh resolution of the simulated pipe are identical to the model of supply device. A constant velocity ($U_{inlet\ pipe}$) is used as the inlet for the pipe calculation, but the long length of the pipe ($16 D$) ensures fully-developed pipe flow at the exit. The axial velocity profile at the supply device inlet (or pipe outlet) and the associated turbulence intensity are verified with hot-wire anemometry. The radial profile of pipe

4 Case Study

flow $\frac{U}{U_{centerline}}$ is fairly consistent with the empirical power-law profile $\left(1 - \frac{r}{D}\right)^{\frac{1}{n}}$ with $n = 6.9$, which confirms that the flow at the pipe exit is fully developed turbulent flow (Munson et al., 2009; Schlichting and Gersten, 2000). The velocity components, associated Reynolds stresses ($\overline{u'_i u'_j}$), turbulent kinetic energy (k) and dissipation rate (ϵ) (or specific dissipation rate (ω)) at the pipe outlet are used as inlet profiles in the supply device model (Ghahremanian and Moshfegh, 2014b).

Table 3: Geometrical and flow details.

D (main pipe diameter)	0.125 m	L (main pipe length)	$59d_0$
d_0 (nozzle diameter)	$2r_0 = 0.0058$ m	ρ (air density)	$1.225 \frac{\text{kg}}{\text{m}^3}$
S (spacing)	$\frac{\pi \times D}{24} = 2.82 d_0$	μ (air viscosity)	$1.7894 \times 10^{-5} \frac{\text{kg}}{\text{m s}}$
Height of nozzles	0.0023 m	$\text{Re}_D = \frac{U_{\text{inlet pipe}} \times D}{\nu}$	3550, 5495, 7608
Contraction ratio of nozzles	4	$\text{Re} = \frac{U_{\text{bulk}} \times d_0}{\nu}$	2125, 3290, 4555

The three-dimensional computational grid is generated with hexahedral cells so the total number of cells, excluding the boundary extrusions, is less than 5×10^6 (Figure 4). The mesh is refined near the solid walls, and the wall nearest y^+ is less than one to make the all-boundary layer be compatible with the two-layer model. Special attention was also paid to controlling the dimensionless streamwise, normal and spanwise spacings, x^+ , y^+ and z^+ , to avoid generating cells with high aspect ratio that would degrade the numerical accuracy. The model decomposition and structured mesh are generated with Gambit 2.4.6.

4.1 Inlet boundary conditions of confluent jets on a curved surface

Figure 4: Grid of planes at $z = 0.111$ m (a) and the nozzles (b).

The research finite volume solver Fluent 12.1.4 was used to simulate numerically the airflow profile of confluent round jet nozzles. The governing and transport equations are solved with a memory-efficient algorithm (Table 4). Both $R k - \epsilon$ and RSM are employed for this prediction. The simulations were conducted on a computer with two processors that each consist of four cores and 16 GB system memory. All of the normalized residuals of the transport equations in the converged solutions of both turbulent models are less than 10^{-6} .

Table 4: Numerical scheme.

Solver	3-Dimensional, Double precision, Steady state, Pressure-based
Pressure-Velocity coupling scheme	SIMPLE
Spatial discretization	
Gradient	Least squares cell based
Pressure	Second order
Momentum	
Turbulent kinetic energy	Second order upwind
Turbulent dissipation rate	
Reynolds stresses	

4 Case Study

To examine the mesh independence, the model was solved at three scales of grid densities, including 208, 576 and 1280 hexahedral cells for each nozzle exit plane. The computational grid of the nozzle exit plane is illustrated in Figure 4. The predicted velocity profiles of the nozzles were compared with the measurement results. The results revealed that a scaled grid density of 208 is unsatisfactory (especially with RSM), and the case with 1280 cells produced almost the same information as a scaled grid density of 576. Therefore, the results from calculations that used 576 cells at the nozzle outlet plane are presented in this paper. The computational staggered grid is generated with hexahedral cells and consists of 12,800,000 cells after a grid independence analysis.

4.2 Single round jet

A free round jet is considered to obtain the physical characteristics of a low Reynolds number jet, to find an appropriate RANS turbulence model to predict the entire field (initial, transition and turbulent regions) of a round jet and to verify the predicted inlet boundary conditions. Anomalies in numerical simulations of the turbulent region of free round jets have been investigated using RANS turbulence models by many researchers (Cho and Chung, 1992; Ghirelli, 2007; Launder et al., 1973; McGuirk and Rodi, 1977; Morse, 1977; Pope, 1978; Spalding, 1971; Wilcox, 2010). All of those investigations were performed on two-dimensional axisymmetric jets with high Reynolds numbers, and most issued from a long pipe. RANS two-equation turbulence models have also been developed for the turbulent regions of the flows; i.e., they are not adjusted for the initial and transition regions. RANS two-equation models provide incorrect prediction of the fully developed region of two-dimensional round jets (Wilcox, 2010). Due to the nature of confluent jets, three-dimensional and low Reynolds number characteristics must be added to these issues, which makes the numerical simulation of the entire domain of a round jet more challenging.

The measurements on a single free round jet were taken in the same test room that was described in section 4.1 using the same HWA equipment and test facilities. A nozzle (Figure 3) with an identical diameter ($d_0 = 0.0058$ m) and shape was placed on a cylindrical air supply, which was located in the same position in the test room as described in section 4.1. The axial

4.2 Single round jet

instantaneous velocities along several cross-sectional profiles of the single jet ranging from $0.06 d_0$ to $50 d_0$ for $Re = 3461$ were measured by HWA.

The computational domain is an incomplete conical frustum (Figure 5) in which the nozzle exit is located at the center of the smallest face (Figure 6). The round jet is assumed to be unventilated (with a front plate) due to the short height of contraction. The Reynolds numbers (Re) of the nozzle exit based on the bulk velocities are 2125, 3461 and 4554.

Figure 5: Computational domain for single round jet.

The mean longitudinal velocity profiles of the nozzle have two overshoots, which are attributed to the small convergence in the nozzle and are similar to the flow from orifice nozzles in Mi et al. (2001a); Mi et al. (2001b) and Deo et al. (2007). The centrifugal acceleration of the fluid occurs due to the streamline curvature that is initiated within the contraction and does not end at the exit plane. The resulting centrifugal forces must then be balanced by a transverse pressure gradient that increases the centerline velocity (vena contracta phenomena). This streamline curvature is absent in flows from parallel exits (Mi et al., 2000; Mi et al., 2001a).

The computational staggered grid is generated with hexahedral cells and consists of 960000 cells after a grid independence analysis. The grid independence study was performed for $Re = 3461$ with three grid densities,

4 Case Study

including 420000, 960000 and 1280000. The latter two grid arrangements yield similar results; therefore, the grid with 960000 structured hexahedral cells is chosen considering the computer resources. The selected grid consists of 576 cells in the nozzle inlet plane (Figure 6). Pressure outlet conditions were specified on the large face and the right side boundary (Figure 6). The effect of the height of the frustum on the flow was examined, and a domain extension larger than $253 d_0$ was found to have no noticeable effect on the flow field.

Figure 6: Computational grid (nozzle grid and entire domain) and measured lines.

A brief review of the physics of a single round jet and numerical predictions of round jets by other researchers is provided in the introduction. As was mentioned, the selected turbulence model for the round jet in this study should be able to numerically predict the flow field from the edge of the nozzle outlet to the mixing transition regime and the turbulent region.

4.2 Single round jet

Two key points play the most important roles in choosing the RANS turbulence model for simulations of a free round jet: the anomaly in the turbulent region and the mixing transition. The former refers to the higher (almost 25%) predicted spreading rate than was measured for the round jet by the $k - \epsilon$ model, while the latter requires further discussion. No general statement can be made about the failure reasons of the transition prediction by a specific turbulence model, which means that the models require additional investigation. The turbulence model equations can be integrated through the transition from laminar to turbulent flow, although the predicted transition occurs at Reynolds numbers that are too low. It appears to be difficult to implement the $k - \epsilon$ model for flow at the transition, because if k goes to zero, the ϵ equation becomes singular unless ϵ becomes zero. The parameter k is present in the denominator of the dissipation term of the ϵ -equation. The $k - \epsilon$ model can resolve laminar flow for some ratios of ϵ/k , but the $k - \epsilon$ model is much less sensitive to the level of turbulence in the free stream due to the odd differential equation for ϵ (Wilcox, 2010). Introducing damping functions and additional closure coefficients, such as in the low Reynolds number $k - \epsilon$ model of Launder and Sharma (1974), the $k - \epsilon$ model may also provide realistic prediction of flows with a mixing transition.

The prediction of the turbulent region of a round jet by the $k - \omega$ model includes smoother velocity profiles near the free stream (Wilcox, 2010). The addition of the cross-diffusion term increases ω and dissipation in the k -equation; the $k - \omega$ model provides a satisfactory computed spreading rate for the round jet. The $k - \omega$ model also predicts the mixing transition at low Reynolds numbers, and the addition of one coefficient damps the turbulent viscosity and causes the low Reynolds correction. These two examples show that the mixing transition can be simulated numerically using the low Reynolds correction.

The intermittent behavior of the near field of a round jet has been investigated in numerical and experimental studies (Gilliland et al., 2012; Ranga Dinesh et al., 2010). Among the turbulence models that consider intermittency, two transition models, the $(k - kl - \omega)$ model (Walters and Cokljat, 2008; Walters and Leylek, 2004) and the TrSST $k - \omega$ (Menter et al., 2006) model, were selected for this study. The transition $k - kl - \omega$ model can numerically predict the transition, while there is not much

4 Case Study

investigation on free round jets with this model. Intermittency produces turbulence, which is defined in the transition $k - kl - \omega$ model as a damping function. The transition from laminar flow to turbulence due to instabilities in the flow is defined as a production term in the transition $k - kl - \omega$ model. The outer region of the flow from a turbulent round jet has less intermittency, which the presence of ω can help to predict. The coupling of the original transport equations of the SST $k - \omega$ model with the transport of intermittency and the momentum-thickness Reynolds number results in the transition SST $k - \omega$ model. The TrSST $k - \omega$ model contains an empirical correlation to address flows with low turbulence in the free stream region. The rapid decay of the turbulence intensity at an inlet depends on the inlet viscosity ratio, and the TrSST $k - \omega$ model can accurately predict the flow behavior with a large inlet viscosity ratio.

The research finite-volume solver Fluent 12.1.4 was employed to numerically simulate the airflow of a three-dimensional round jet. The governing and transport equations are solved with segregated memory-efficient algorithms. The double precision, steady state and pressure-based solver is used for the low Re $k - \epsilon$ and SST $k - \omega$ models. For the discretization, the non-linear terms are calculated with a second-order upwind scheme, and the viscous terms are calculated with a second-order central scheme. The under-relaxation factors for the pressure, momentum, turbulent kinetic energy and turbulent dissipation rate (specific dissipation rate) are set to 0.3, 0.7, 0.5 and 0.5, respectively. The SIMPLE algorithm solves the pressure-velocity coupling. If the sum of the absolute normalized residuals of the transport equations in the turbulent models for all of the cells in the flow domain is less than 10^{-6} , the solutions were considered to have converged.

4.3 Confluent jets on a curved surface

Measurements of the instantaneous axial velocity were performed using the same HWA equipment, test room and procedure that was described in section 4.1. Measurements were taken along the cross section of the nozzles in column 4 at several downstream distances ranging from $0.34 d_0$ to $20 d_0$ at steps of $0.17 d_0$. Each cross-sectional traverse covered the complete profile of three and a half jets in column 4. Measurements were also performed along the geometrical centerline of four jets, CJ (3R-4C), SJ (6R-

4.3 Confluent jets on a curved surface

4C and 3R-6C) and CoJ (6R-6C), with the same steps. Both labeling methods depicted in Figure 3 are used to present the results.

The research finite-volume solver Fluent 13 was employed to numerically simulate the airflow of half of an in-line array of thirty-six round jets, or three columns of six jets. Half of the six columns, which are divided by a symmetry plane (Figure 7), were modeled to reduce the computational complexity, time and cost of the simulations.

The computational domain (Figure 7) begins from a curved solid wall (at a radius of 0.0625 m or 2.46 in), $D / d_0 = 21.55$, and contains eighteen nozzles. It then extends radially to the pressure outlet faces at $100d_0$. The longest arc length between the pressure outlet and the symmetry plane is $261d_0$, which corresponds to an angle of 135° . The center-to-center angle between two columns of jets is 15° , and the total angle between the symmetry plane of the array and column six is 37.5° ; the 135° angle is greater than a right angle and is therefore wide enough to contain the pressure outlet. The geometrical details and fluid properties are summarized in Table 3.

Figure 7: Computational domain.

4 Case Study

To examine the mesh independence, the model was solved with three scaled grid densities: 208, 576, and 1280 hexahedral cells per nozzle exit plane. The computational grid of the nozzle exit planes is illustrated in Figure 6. Identical cell distributions are used in the three test cases; the greater the number of cells, the finer the resolution near the walls. The predicted velocity profiles at the outlet of the nozzles were compared with the measurements. The results revealed that the grid density of 208 is unsatisfactory and that the case with 1280 cells produces nearly the same information as the grid density of 576. Therefore, the results for the numerical prediction that used 576 cells in the nozzle's outlet plane are presented in this paper.

The computational staggered grid is generated with hexahedral cells, so twelve million cells are used for the selected case from grid independence. The supply device is simulated with the Realizable $k - \epsilon$ model ($R k - \epsilon$) and the Reynolds stress model (RSM) to provide the inlet profile for the confluent jets (Ghahremanian and Moshfegh, 2014b). The simulated supply device (Ghahremanian and Moshfegh, 2014b) has an identical nozzle diameter (d_0) and mesh resolution to the current model. The axial velocity profile of the nozzle outlets and its associated turbulence intensity are verified with hot-wire anemometry (Ghahremanian and Moshfegh, 2014b). The inlet boundary conditions (velocity components, turbulent kinetic energy (k) and specific dissipation rate (ω)) at the nozzle outlets from the verified numerical prediction of the supply device are employed; they were presented in section 4.1.

The numerical predictions of confluent jets on the cylindrical supply device were performed with the turbulence model that had the best performance in the single round jet study. The shear stress transport $k - \omega$ model with the low Reynolds correction was used to simulate the flow from the confluent jets. The governing and transport equations are solved with a memory-efficient algorithm (Table 4). All of the normalized residuals of the transport equations in both turbulent models are less than 10^{-6} in the converged solutions.

4.4 Parallel, coplanar, single row of jets

A single row of jets can represent the midpoint between the fundamental configurations, such as the single jet or twin jets, and an array

4.4 Parallel, coplanar, single row of jets

setup. The investigation of a single row of jets can link the fundamental jet configurations to the array setup. Single round (or plane) jets and twin plane jets have been extensively studied, but research about a single row of jets is scarce. Therefore, Particle Image Velocimetry (PIV) and Laser Doppler Anemometry (LDA) were used to investigate the flow in the near-field of a parallel coplanar row (1×6) of round jets. The main aim of this study is to gain insight into the confluence of jets in the near-field of one row of round jets using detailed two-dimensional measurements. The investigation aims to provide information on jet features, such as the development of the mean velocity, maximum velocity decay and characteristic points (merging and combined points), as well as the influence of the position of the jet in the row. To assess the dependence on the Reynolds number, all of the measurements were performed for three Reynolds numbers ($Re = 2125, 3290$ and 4555).

There are two main purposes of the LDA measurements. Firstly, the usage of LDA enabled the measurements of the jet exit profile close to the nozzles. This was very difficult using PIV due to the sharp velocity gradients near the nozzle edges and the laser light reflections. Secondly, one LDA profile and two geometrical centerline within the flow regime was used for comparison of the main findings reported by the PIV measurements.

Study of multiple interacting jets requires well-defined boundary conditions. The velocity and turbulence profiles should also be similar for all confluent jets at the nozzles edge. The flow at the upstream of nozzle plate should be without any swirling. A wind tunnel was designed and constructed to create a uniform low-turbulence flow upstream of the nozzle plate (Figure 8). The design of wind tunnel complied with the above-mentioned requirements and was based on the rules of thumb devised by Mehta and Bradshaw (1979). A centrifugal fan (ITHO, MD-63) ventilated air into a round pipe upstream of a rotary flow meter (Instromet, Silvolde NL, Q-75-E). The round pipes with length of 1 m were placed upstream and downstream of flow meter in order to measure the flow accurately. A variable power supply was used to control the airflow rate. The airflow was transferred to the test chamber (box) using square supply channels, bends, a diffuser, a settling chamber and a contraction (Geers et al., 2006). The 90° corner bends with square cross-sections guided the airflow with circular-corner vanes to avoid the creation of streamwise swirl or Dean vortices. A

4 Case Study

square wide angle diffuser with an area ratio of 6.25, a length of 0.56 m and an angle of 30° was installed with two wire-gauze screens (Groth and Johansson, 1988) at the inlet and outlet to prevent airflow separation. The diffuser results in low-speed airflow that decreases pressure losses and reduces the variations of the mean and fluctuating velocities. The settling chamber consisted of four equal-sized square sections, in which a honeycomb (Lumley, 1964) were placed in the first section to remove swirl and variation of lateral mean velocity. Each section was 0.125 m long and had a cross-sectional area of $0.5 \text{ m} \times 0.5 \text{ m}$. Since the relatively long honeycomb cells ($23.5 \times$ cell diameter) was expected to increase the turbulence level of the flow, wire-mesh screens were inserted consecutively between sections. A fine mesh screen was installed at the end of settling chamber to avoid production of flow instabilities. A square Plexiglas contraction with an area of $0.3 \text{ m} \times 0.3 \text{ m}$ was used to reduce the cross-section of the settling chamber. A contraction allowed that the flow in settling chamber remained at low speed. The contraction reduced pressure losses and the variations of lateral velocity with respect to average velocity. The contraction with the inlet area over outlet area ratio of 2.78 was designed based on a fifth-order polynomial (Bell and Mehta, 1988, 1989) to produce a straight inlet and outlet. The nozzle plate was attached to the outlet of the contraction.

The jets issue into a large enclosure with dimensions of $1 \text{ m} \times 1 \text{ m} \times 2 \text{ m}$ (height $\times$ width $\times$ length). The enclosure prevents disturbances from air movement in the room and ensures equal seeding density in the jet fluid and in the surrounding fluid. The nozzle plate was mounted in the center of one of the sidewalls of the enclosure. Two other sidewalls were equipped with high transmittance windows ($0.2 \text{ m} \times 0.2 \text{ m}$) to provide optical access. Therefore, no measurement equipment had to be placed inside the enclosure, which prevented flow disturbances. The accuracy of the flow meter and the air tightness of the wind tunnel were tested by integrating the inlet velocity profiles (acquired with LDA) near the inlet. The average difference in airflow between the flow meter reading and the integrated velocity profile was less than 2%.

4.4 Parallel, coplanar, single row of jets

Figure 8: Top-view sketch of the wind tunnel.

Two types of confluent jets are in the single row configuration (Figure 9) based on the number of neighboring jets: jets at the ends of the row, which have one neighboring jet (side jets [SJ]), and jets in the middle of the row, which have two neighboring jets (central jets [CJ]). The nozzle axes of both SJs and CJs are coplanar and parallel. The mean velocity components and Reynolds stresses were determined along one center-plane (extended up to $29d_0$) perpendicular to the nozzles using PIV. LDA measurements were performed to determine the mean velocity and its RMS along one profile close to the nozzle ($y/d_0 = 0.086$), one cross-sectional profile ($y/d_0 = 4.2$) and two geometrical centerlines (CJ I and CJ II). The flows in the overlap regions of two consecutive PIV recording sessions could be verified due to the care taken in making the measurements. The locations of the PIV and LDA measurements and the single row setup are shown in Figure 9.

The green line at 514.5 nm of a 4 W Ar-Ion laser (Spectra Physics, model 2016) was used to measure the axial velocity component. The laser beam was divided and one of the beams was frequency pre-shifted over 40 MHz by a Bragg cell to detect flow reversals. The length and diameter of the measurement volume are 1.0 mm and 0.1 mm, respectively. The scattered light was collected in a side scatter to reduce the effective length of the measurement volume to approximately 0.15 mm. The output signal of the photo multiplier was electronically down-mixed to an effective pre-shift

4 Case Study

frequency of 5 MHz and subsequently fed into a Dantec BSA signal processor.

Figure 9: Locations and size of the PIV measurement planes, locations of the LDA measurements and nozzle shape geometry.

The fringe spacing was determined by a graphical method (described in detail by Absil (1995)), which had an uncertainty of approximately 1%. Approximately 6000 velocity samples were taken at each measurement point. The mean data rate varied between points in the flow due to variations in the local velocities. As a result, the measurement times varied between 35 to 600 s. Velocity statistics, such as the mean velocity and the Reynolds normal stress were computed using transit time weighting to avoid errors due to the velocity bias. The overall uncertainties of the LDA measurements are approximately 2 % of the bulk velocity for the mean velocity and 8 % of the local maximum for the Reynolds stress.

The trace particles were illuminated by a double-pulsed Nd:Yag laser (Continuum Minilite) with a pulse energy of 25 mJ and a pulse duration of approximately 5 ns. A 1-mm-thick laser sheet was produced by mounting light sheet optics directly after the laser. The scattered light from the seeding particles was recorded on a CCD sensor of a PCO SensiCam camera with a resolution of 1280×1024 pixels. A Nikon micro-NIKKOR lens with a 105

4.4 Parallel, coplanar, single row of jets

mm focal length and a numerical aperture of 11 was used with the camera. The size of the field of view was typically 48 mm $\times$ 38 mm. CamWare v3.08 and an ILA timing unit were used to control the camera, to synchronize the triggering of the laser shots and to acquire images. The time interval between the two laser pulses were 39, 25 and 18 μ s for nozzle exit Reynolds numbers of 2125, 3290 and 4555, respectively. A time lag of 0.5 s between two consecutive samples of the velocity field was used to capture statistically independent image pairs. Approximately 4000 image pairs were acquired in each ensemble to accurately determine the flow statistics to the second moments. The LaVision software DaVis v8.0.8 was used to analyze the image pairs and calculate the vectors.

Table 5: PIV recording parameters for all measurements.

Focal length of lens (mm)	105		
Numerical aperture	11		
Field of view (mm ²)	48 $\times$ 38		
Final interrogation area size (pixels)	16 $\times$ 16		
Final interrogation area size (mm ²)	0.25 $\times$ 0.25		
Interrogation overlap (%)	25		
Reynolds number	2125	3290	4555
Pulse delay (μ s)	39	25	18
Bulk velocity at nozzle exit, U_b (m/s)	5.6	8.6	12.0
Number of image pairs	4000		

The image pairs were processed in two consecutive multi-pass steps with decreasing interrogation areas after image preprocessing by particle

4 Case Study

intensity normalization (min/max = 2 pixels). In the first step, 25% overlapping square interrogation areas with sizes of 32×32 pixels were cross-correlated. After filtering out spurious vectors, the resulting particle displacements were used as window displacements for a cross correlation with an interrogation area of 16×16 pixels ($0.25 \text{ mm} \times 0.25 \text{ mm}$) with 25% overlapping square interrogation areas and Whittaker reconstruction. A local median filter was used to remove spurious vectors during the iterative processing, and the resulting empty spaces were filled by interpolated vectors. The recording parameters of the PIV system for all of the measurements are summarized in Table 5. According to the practical guidelines described by Adrian and Westerweel (2010), the uncertainty of the instantaneous velocity field was estimated at approximately 5% of the bulk velocity (U_b). The maximum statistical error due to the finite number of samples (with 95% confidence) was approximately $\pm 0.008 U_b$ for the mean velocities and approximately $\pm 0.003 U_b^2$ and $\pm 0.002 U_b^2$ for the Reynolds normal and shear stresses, respectively. The latter represents approximately 6% of the maximum values of the stresses in the shear layers. The bulk velocity (U_b) is determined by dividing the average airflow rate by the total area of the nozzles ($6\pi d_0^2/4$). The airflow rate is based on the air properties (temperature, static pressure and humidity) in the test room, which are recorded during each measurement.

4.5 Confluent jets on a flat surface

As described in the literature review section, detailed experimental investigations on the confluence of an array of free round jets are scarce. Therefore, the two-dimensional turbulent velocity field (scalar mean properties) near confluent jets issuing into a stagnant environment was studied using Particle Image Velocimetry (PIV) and Laser Doppler Anemometer (LDA) techniques. The main objective was to gain insight into the complex phenomena that occur in the initial, converging and transitional, merging and combined regions by studying the mean flow and turbulence statistics. The study of jet-to-jet interactions, jet position dependency within the array, Reynolds number dependency and turbulence is intended to provide information about the jet characteristics, such as the mean velocity profiles (and their decay), jet deflection (convergence),

4.5 Confluent jets on a flat surface

Reynolds stresses, normal and shear turbulence production terms and the anisotropy of turbulence at different stages of development.

The planar PIV measurements were performed at seven planes perpendicular to the nozzle plate (Figure 10), and each plane with $z = \text{constant}$ included of four or five PIV recording sessions. As shown in Figure 10, the planes at $z = S$, $z = 0$, $z = -S$ and $z = -2S$ are nozzle center planes, while those at $z = 0.5S$, $z = -0.5S$ and $z = -1.5S$ are between the nozzle centers. The short-height contraction nozzles and diameters were identical to those in the studies described previously. The PIV results are consistent with the values measured using LDA over the complete dynamic velocity range. Some slight differences were observed and are related to the PIV resolution in velocity fields with very sharp gradients. Particular care was taken to verify the symmetry in the studied array and the conformity in the overlap region between subsequent PIV recording sessions.

Table 6: Summary of the PIV measurements.

Plane	z -coordinate	Plane position	Number of subsequent recording sessions	Maximum, covered streamwise distance (y/d_0)
1	$-S$	Nozzle center	4	20.6
2	$-0.5S$	Symmetry	4	20.6
3	0	Nozzle center	5	25.75
4	$0.5S$	In-between	4	20.6
5	S	Nozzle center	5	25.75
6	$1.5S$	In-between	4	20.6
7	$2S$	Nozzle center	5	25.75

Measurements were made at $z = S$ to verify the flow symmetry of the jet array. The range in the x -direction was limited to $1.6d_0 \leq x \leq 6.1d_0$ for each plane ($z = \text{constant}$). This increases the spatial resolution compared to measuring across the full width of the nozzle plate. The planes ($z = \text{constant}$) extend from the nozzle at $y = 0$ to a maximum distance of $25.7d_0$ ($\approx 150 \text{ mm}$). Up to five recording sessions were needed to cover this range

4 Case Study

in the x -direction due to the relatively small field of view (typically $48 \text{ mm} \times 38 \text{ mm}$).

The fields of view in the different recording sessions overlapped slightly. This has the advantage that regions with high in-plane loss of pairs near the edge of the field of view can be removed without creating gaps in the velocity fields. LDA measurements were taken along a limited number of lines in the plane $z = 0$, including $y/d_0 = 0.086$, $y/d_0 = 0.86$, $y/d_0 = 4.9$, $y/d_0 = 7.8$ and a centerline traverse at $x = 0$. A summary of the locations of the PIV measurements is given in Table 6.

The image pairs were processed in two consecutive multi-pass steps of decreasing interrogation areas after image preprocessing by particle intensity normalization ($\text{min/max} = 2$ pixels) due to reflections from the nozzle plate. In the first step, 50% overlapping square interrogation areas with window sizes 32×32 pixels were cross-correlated. After filtering out spurious vectors, the resulting particle displacements were used as window displacements for a cross correlation with an interrogation area of 16×16 pixels ($0.3 \text{ mm} \times 0.3 \text{ mm}$) with a 50% overlapping adaptive weighting function and Whittaker reconstruction. A local median filter was used to remove spurious vectors during the iterative processing, and the resulting empty spaces were filled by interpolation from neighboring interrogation areas. The recording parameters of the PIV system for all of the measurements are summarized in

Table 7. The relative expanded uncertainty of the PIV velocity measurements in air is 5.1%. The sources of uncertainty and their relative standard uncertainties are as follows: slip error in the tracer dynamics (1%), magnification in the image mapping (0.3%), displacement-correlation peak in interrogation (1.5%), sampling error of the statistical properties (0.5%) and test facilities (e.g. leakage of wind tunnel) (1.8%).

The measurements were mainly taken in one quarter of the array, which is bounded by the horizontal ($z = -0.5S$) and vertical ($x = -0.5S$) symmetry planes (see Figure 10). A third symmetry plane spans along the diagonal ($x = z$) of the studied part of the array. Considering all three symmetry planes, six unique jet positions are found among the nine jets in the region of measurements (three jets along the diagonal ($x = z$) and three jet pairs located symmetrically around the diagonal). The six jets can be grouped based on the number of neighboring jets. This results in three types of jets:

4.5 Confluent jets on a flat surface

jets near the center of the nozzle array (Central Jets [CJ]), jets along the sides of the array (Side Jets [SJ]) and jets at the corners of the array (Corner Jets [CoJ]), which have eight, five and three neighboring jets, respectively.

Figure 10: Locations and sizes of the PIV measurement planes, locations of the LDA measurements and nozzle shape geometry.

4 Case Study

Table 7: PIV recording parameters for all of the measurements.

Focal length of lens (mm)	105		
Numerical aperture	11		
Field of view (mm ²)	47.8×38.2		
Final interrogation size (pixels)	16×16		
Final interrogation size (mm ²)	0.3×0.3		
Interrogation overlap (%)	50		
Reynolds number	2125	3290	4555
Pulse delay (μs)	39	25	18
Average velocity at nozzle exit (m/s)	5.6	8.6	12
Number of snapshots	4000		

4.6 Parametric study of a coplanar parallel row of jets

The study of low Reynolds number round jets by Todde et al. (2009) shows that the inlet velocity (V) affects the length of the potential core and the mixing layer thickness. Therefore, the inlet velocity may affect the merging point of multiple interacting jets. The dimensional analysis by Knystautas (1964) shows that the spacing (S) between the nozzles and the diameter (d) of the nozzles are the most significant parameters that influence the combined point of a series of co-planar parallel jets. In this study, the responses of the merging point and combined points are assumed to be mainly influenced by three design factors: the inlet velocity (V), spacing (S) and nozzle diameter (d). The independent parameters (V , S and d) vary in ranges that may increase the interaction between the jets compared to the factors used in the verification case ($V = 8.355$ m/s, $S = 2.82 d_0$ and $d_0 = 0.0058$ m). Three levels, low (-1), middle (0) and high (+1), are assigned to each natural variable (i.e., inlet velocity, spacing and

4.6 Parametric study of a coplanar parallel row of jets

diameter) as required by the Box-Behnken design and are shown in Table 8. A three-level, three-factor Box-Behnken design is applied to generate the design matrix for the numerical simulation by the low $Re\ k - \epsilon$ model as is shown in Table 9. A total of 13 unique cases are modeled numerically, and two replications at the central point (V_{00} , S_{00} and d_{00}) are ignored due to the small variations in the results of identical CFD simulations.

Table 8: Box-Behnken design matrix for three uncoded variables.

V [m/s]		S [m]		d [m]	
V_{00}	10	S_{00}	0.012	d_{00}	0.008
$V_{\pm}$	12	$S_{\pm}$	0.014	d_{00}	0.008
	8		0.010		
$V_{\pm}$	12	S_{00}	0.012	$d_{\pm}$	0.009
	8				0.007
V_{00}	10	$S_{\pm}$	0.014	$d_{\pm}$	0.009
			0.010		0.007

Each case uses a specified combination of inlet velocity, spacing between the nozzles and diameter of nozzles (Table 9). The dimensionless spacing (S/d) varies between 1.11 and 2, the Reynolds number varies between 3835 and 7395, the total airflow rate fluctuates between 1.8 and 4.6 lit/s, and the equivalent diameter ($d_{eq} = \sqrt{6}d$) varies between 0.0171 and 0.022 m.

First, a numerical model based on the PIV measurements on a single row of parallel co-planar round jets as described above is predicted. The results of the numerical model are then validated by comparing the velocity profiles in various regions to the PIV data. The turbulence model (low Reynolds number $k - \epsilon$) for all of the numerical predictions is selected from those that were verified and shown to provide the best overall performance in another study of a single round jet by the authors (Ghahremanian and Moshfegh, 2014a).

4 Case Study

Table 9: Details of the numerical prediction cases.

Case	V [m/s]	S [$\times d_{00}$]	A [$\times \frac{\pi d_{00}^2}{4}$)]	$\frac{S}{d}$	Re_d	Q , [m ³ /s]	d_{eq} [m]
PIV							
Low Re	8.355	0.725	0.725^2	2.82	3290	0.0013	0.0142
$k - \epsilon$							
1	10	1.5	1	1.50	5478	0.0030	0.0196
2	12	1.75	1	1.75	6574	0.0036	0.0196
3	12	1.25	1	1.25	6574	0.0036	0.0196
4	8	1.75	1	1.75	4382	0.0024	0.0196
5	8	1.25	1	1.25	4382	0.0024	0.0196
6	12	1.5	1.125^2	1.33	7395	0.0046	0.0220
7	12	1.5	0.875^2	1.71	5752	0.0028	0.0171
8	8	1.5	1.125^2	1.33	4930	0.0031	0.0220
9	8	1.5	0.875^2	1.71	3835	0.0018	0.0171
10	10	1.75	1.125^2	1.56	6163	0.0038	0.0220
11	10	1.75	0.875^2	2.00	4793	0.0023	0.0171
12	10	1.25	1.125^2	1.11	6163	0.0038	0.0220
13	10	1.25	0.875^2	1.43	4793	0.0023	0.0171

A : Area of one nozzle, Q : Total Airflow rate, d_{eq} : Equivalent diameter

Second, nine numerical models are built based on the design factors of the Box-Behnken design. The other four numerical simulations have different inlet boundary conditions. Details of the numerical setup and procedure are described below.

The computational domain for the validation case consists of a single row of parallel, co-planar round jets with equidistant spacing ($S_0 = 2.82 d_0$). The short converging nozzles have a diameter of $d_0 = 0.0058$ m and a height of $0.4 d_0$. A similar bulk velocity ($V_b = 8.355$ m/s) and Reynolds

4.6 Parametric study of a coplanar parallel row of jets

number ($Re = 3290$) to the PIV measurements are used. The computational domain is depicted in Figure 11. One quarter of the real domain (i.e., half of three jets) is numerically predicted due to symmetry. Using the horizontal and vertical symmetry planes ($z = 0$ and $x = -0.5 S_0$, respectively) reduces the computational cost. The computational boundaries are extended enough to ensure that the free conditions at the pressure outlets do not affect the predicted results. The inlet boundary conditions at the nozzle exits are taken from the PIV results. The PIV measurements could not acquire the instantaneous velocity near the nozzle exits due to the laser light reflections, thus the nozzles in the numerical simulation are modified to a cylindrical shape and extended to $y/d_0 = 1.03$ (Figure 11). Profiles of the velocity, turbulent kinetic energy and its dissipation rate are generated from piecewise spline curve-fits of the PIV results along the x -direction of each nozzle. Due to the geometrical symmetry, the third unmeasured component of the velocity (W) and its RMS ($\sqrt{w'^2}$) are assumed to be equal to the other lateral component (U and $\sqrt{u'^2}$). A no-slip condition is applied to the walls.

Gambit 2.4.6 is used to construct the three-dimensional geometry and generate the structured grid for the validation case and the other nine cases in the parametric study. The same mesh strategy as was used in the validation study is used, in which the expansion strategy of the grid is based on the nozzle diameter and the spacing between nozzles. The mesh is refined near the solid walls, and the wall nearest y^+ is kept below one to make the boundary layers compatible with the low Reynolds correction. Special attention was also paid to controlling the dimensionless streamwise, normal and span-wise spacings to avoid generating cells with high aspect ratios, which would degrade the numerical accuracy. The hexahedral cells near the nozzle exits are refined due to the high velocity gradients. To examine mesh independence, the validation model was solved with three different mesh densities and grid configurations.

4 Case Study

Figure 11: Computational domain and nozzle shape for the validation case.

The research finite-volume solver ANSYS Fluent 14.5.7 was employed to numerically simulate the airflow of the single row of parallel co-planar round jets. The governing and transport equations are solved with segregated memory-efficient algorithms. The double precision, steady state and pressure-based solver is used for the Low $Re\ k - \epsilon$ model. For the discretization, the non-linear terms are calculated with a second-order upwind scheme, and the viscous terms are calculated with a second-order central scheme. The under-relaxation factors for the pressure, momentum, turbulent kinetic energy and turbulent dissipation rate (specific dissipation rate) are set to 0.3, 0.7, 0.5 and 0.5, respectively. The SIMPLE algorithm solves the pressure-velocity coupling. If the sum of the absolute normalized residuals of the transport equations in the turbulent model for all of the cells in the flow domain is less than 10^{-6} , the solutions were considered to have converged.

5 Results

This section reports the results of both the experimental and the numerical studies of the multiple interacting jets. Section 5.1 presents the LDA and PIV measurement results of confluent jets. Section 5.2 reports the numerical study of multiple interacting jets. Section 5.3 presents the result of parametric study on a single row of interacting jets.

5.1 Experimental study

5.1.1 Single row of round jets

The three jets in the single row study have potential cores with indistinguishable velocity profiles in the initial region ($0 \leq y/d_0 \leq 2$). Within this region, there is no significant dependency on the jet position for the one row of jets. Based on the LDA measurements, the flow very close to the nozzle ($y/d_0 = 0.086$) has a velocity profile with a saddle-back shape that contains low turbulence in the central region and a sharp narrow peak in the turbulent mixing layers (Figure 14). The velocities in the x -direction along the geometrical centerlines of the jets increase in the potential core regions due to the vena contracta effect, while the RMS remains almost constant.

Figure 12 shows a comparison between the LDA measurements and the PIV results for the cross-sectional profile at $y/d_0 = 4.2$. The streamwise position (y), mean velocity (V) and RMS ($\sqrt{v'^2}$) are made dimensionless with the nozzle diameter, d_0 , and the bulk velocity, U_b , respectively. At $y/d_0 = 4.2$, the agreement between PIV and LDA is satisfactory in both high velocity regions and in the low velocity regions between the jets, although small differences can be observed between CJ II and SJ. The values of RMS, $\sqrt{u'^2}$, from both measurement techniques have similar patterns, although there is a mismatch of the peaks within the inward facing shear layer of both CJs.

5 Results

Figure 12: Profile of the dimensionless velocity in the y -direction V (left) and the RMS of the velocity $\sqrt{v'^2}$ (right) measured by PIV (solid lines) and LDA (symbols) at $y/d_0 = 4.2$ for $Re = 3290$.

The velocity profile of the jets in the single row setup becomes blunt and conical in the merging region ($4.5 \lesssim y/d_0 \lesssim 14.3$), and the velocity along the geometrical centerline begins to decay (Figure 15). The thickness of the peaks of the normal Reynolds stresses become wider, and the turbulence from the mixing layer approaches the geometrical centerline of the jets. The central jets (CJs) merge closer to the edge of the nozzles than the side jet. All of the jets form a flat velocity profile with relatively low Reynolds stresses in the combined region ($y/d_0 \gtrsim 14.3$).

5.1.2 Array of multiple round jets

Identical saddle-back velocity profiles with low turbulence at the core of the jets were revealed at the front of the initial region of multiple jets on the flat surface ($0 \leq y/d_0 \leq 2$) by the LDA measurements (Figure 14). Within the initial region, the jet takes the shape of a top hat with a clear potential core zone of nearly constant velocity and low turbulence. Immediately after the nozzle exit, turbulence starts being produced by the thin mixing layers. The normal Reynolds stresses within this region have approximately equal magnitudes, although asymmetrical maximum levels are observed throughout the shear layers. Within the initial region, the jet behavior does not depend on the positions of the jets; i.e., there is no significant difference between the CJs, SJs and CoJs.

At approximately $y/d_0 = 2$, the jets enter the converging region. All of the jets converge toward the center of the nozzle plate, and the SJs show the greatest deflections. The majority of the turbulent kinetic energy (k) is

5.1 Experimental study

produced in the converging region. For the CJs, the main production of k is related to the shear stress. For the SJs, additional production of k due to the normal Reynolds stresses is responsible for the higher turbulence levels than in the CJs. In addition, the instabilities around the shear layer of the SJs form earlier than for the CJs. The development of normal Reynolds stresses is also dependent on the Reynolds number.

Figure 13: Two sequential (0.5 s) calculated, instantaneous velocity vector fields (left) from the most upstream PIV recording session at $z = 0$ and close-up views of vortices of the side jet (right); $Re = 3290$.

Figure 13 presents two typical instantaneous velocity fields from the first recording session of the plane at $z = 0$. Entrainment and interaction with neighboring jets and the surrounding fluid are clearly demonstrated for the studied confluent jets. Velocity vectors without turbulence structures form in the central region of each jet up to $y/d_0 = 4$ and $y/d_0 = 2.5$ for the CJs and the SJ, respectively. The instabilities that form at certain distances from the nozzle edges are the result of induction of the surrounding fluid and primary coherent structures that evidently formed in an earlier stage for the SJ than for the CJs. In Figure 13, the resulting primary structures from the roll-up of the initial shear layer are evident in the CJs and the SJ at these downstream

5 Results

distances. These large-scale vertical structures engulf and entrain ambient air between the jets and attempt to dominate the initial growth by pairing and pinching the potential core. The pinch-off of the tip of the potential core occurs closer to the nozzle's edge for the SJ than for the CJs because more of the surrounding ambient fluid is entrained than the fluid between the jets. The primary coherent structure will propagate downstream of the potential core of the jets and is shown in close-up views in Figure 13.

For the SJ, the downstream development of large-scale coherent structures reveals non-axisymmetric behavior after the initial region, which contrasts with the quasi-axisymmetric trend in the near field of the CJs. The lack of ring-like coherent structures in the very near-field is consistent with the presence of a thin initial mixing layer. These thin mixing layers are formed by azimuthally coherent structures that prevent upstream separation, which ensures an axisymmetric trend and a more stable mixing layer at the nozzle exit. The stable mixing layer in the proximal region of the nozzles causes slow fluctuations and almost no vortex roll-up. The flow from the confluent jets becomes three-dimensional after a certain axial distance from the nozzles due to the entrainment of ambient fluid and convergence of the jets. The mixing of confluent jets involves self-induction and vortex stretching, similar to single jets, as well as convergence, merging and combining, which makes the fluid flow three-dimensional.

The velocity profiles in the converging region ($2 \leq y/d_0 \leq 6$) become conical as the jets experience a mixing transition and begin to form a three-dimensional flow field. Considerable asymmetry in the distribution of the Reynolds stresses occurs in the mixing layers in the converging region. As the jets begin to converge, the anisotropy of the Reynolds stresses increases, and the instabilities in the shear layer gradually penetrate deeply enough to reach the center of the confluent jets. Due to the greater lateral displacement and turbulence production, the pinch-off of the potential core occurs earlier for the SJs than for the CJs. The rate of convergence decreases when the Reynolds number is lower, while no signs of Reynolds number dependency were observed in the decay of the maximum velocity.

As the convergence progresses, the confluent jets eventually start to merge. The SJs and CoJs merge more quickly than the CJs due to the larger deflection, while adjacent central jets remain separated for a greater distance. Interestingly, the CoJs converge less than the SJs, which delays the

5.1 Experimental study

interaction with the neighboring jets. In the merging region ($6 \leq y/d_0 \leq 15$), the Reynolds stresses begin to decrease, which also reduces the anisotropy. Initially, the decrease is faster in the outer part of the flow field, which causes the maximum Reynolds stresses to be found in the region of the CJs.

A flat profile for the streamwise velocity does not contain distinct traces from the individual jets forms in the combined region ($y/d_0 \geq 15$). As the velocity profile becomes flat in the core of the combined jet, the Reynolds stresses progressively decay, which leads to very low turbulence in the central parts of the combined jet. The combined jet is thinner than the array due to convergence of the SJs and CoJs. The maximum velocity of the combined jet does not decay because the streamwise velocity is almost constant; this behavior is not observed in either the twin jets or the single jet (Figure 15).

5.1.3 Comparison of the single row and array setups

The maximum streamwise velocity of each of the measured planes in the array setup can be compared with the centerline velocity of a single round jet with the same nozzle profile and Reynolds number (Figure 15). In the initial region, the centerline velocity of the single jet and the maximum velocities at planes 1, 3 and 5 decrease slightly and are followed by slight recoveries before strong decays. The maximum velocities in planes 1 ($z = -S$), 3 ($z = 0$) and 5 ($z = S$) have similar behavior to the developed region of a single jet except for the difference in the decay rate. The velocity of a single jet continues to decay, while the maximum velocities in planes 1 to 5 reach plateaus. The maximum velocities in the planes between the jet rows (i.e., planes 2, 4 and 6) begin to increase until they reach the combined maximum velocity. The maximum velocity in plane 7 begins to decay strongly earlier than in the other center planes (planes 1, 3 and 5), and the decay rate is sharper than those planes. The maximum velocity in plane 6 increases more rapidly than in planes 2 and 4 and does not reach the combined maximum velocity. The different behavior of the maximum velocities in planes 6 and 7 is due to the convergence and merging of the SJs and the CoJ; similar behavior also occurs in twin jets.

Tanaka (1970, 1974) defined the merging (stagnation) point as the location where the velocity changes direction and the pressure shows a

5 Results

maximum positive value. Nasr and Lai (1997a, b, 2000) defined the merging point as the location along the symmetry between twin plane jets where the streamwise velocity becomes zero and the pressure reaches a maximum. Anderson and Spall (2001) defined the merging point as the distance to the stagnation point or the location where the streamwise normal Reynolds stress increases toward an asymptote, and Vouros and Panidis (2008) suggested that the merging point is located where the minimum streamwise mean velocity between two jets exceeds 10% of the maximum streamwise mean velocity of the flow field. Kumar and Kumar Das (2011) used the intersection of the contour lines where $U = 0$ and $V = 0$ to determine the merging point.

The combined point is defined (Tanaka, 1970, 1974) as the location along the symmetry plane between twin plane jets where the streamwise mean velocity increases to a maximum value. Anderson and Spall (2001) characterized the combined point as the distance at which the trough between two peaks in the velocity profile disappears and only a single maximum exists. Therefore, the gradient of the streamwise velocity in the transverse direction becomes zero ($\partial U / \partial x = 0$) and the curvature between the local maxima and minima is zero (Karimpour et al., 2011; Kumar and Kumar Das, 2011). A short summary of the merging points and combined points reported in previous studies is given in Table 10.

The setup of a single row of jets can be compared to an array of 6×6 round jets with identical nozzle shapes, Reynolds numbers and center-to-center spacing to investigate the influence of additional surrounding jets on the development of the flow field. The velocity profile near to the nozzle exit, the potential core region, the growth in thickness of the mixing layer and the initial, merging and combined regions are present in both the one row setup and the array of 6×6 jets. In contrast, all of the jets (especially the SJ) in the 6×6 array setup deflect towards the center of the array and form a converging region between the initial and merging regions. The deflection of jets in the 6×6 array setup makes the merging region shorter with higher turbulence. The flow of maximum velocity of SJs in the 6×6 array setup follows a concave curvature with higher Reynolds stresses than in the one row setup.

5.1 Experimental study

Table 10: Comparison of experimental and numerical studies of twin jets with this study.

		Dimensionless Spacing	Merging point (y_{mp}/d_0)	Combined point (y_{cp}/d_0)	y_{mp}/y_{cp}
Present study (single row, Re = 3290)	CJ I - CJ I		4.5		0.31
	CJ I - CJ II	2.82	4.7	14.4	0.33
	CJ II - SJ		5.2		0.36
Present study (array, Re = 3290)	CJ		≈ 6.6	≈ 15	0.44
	SJ	2.82	≈ 5.9	≈ 10	0.59
Durve et al. (2012)		3.5	2.7	11	0.245
		9	11.5	17.95	0.641
		13	14.9	22.89	0.651
		18.25	18.7	27	0.693
			$0.721 S/d_0 + 2.06$ Tu -2.453	$1.231 S/d_0 +$ 2.06 Tu -2.453	
Fujisawa et al. (2004)		3	3	6	0.5
Harima et al. (2001)		2		25	
		4		52	
		8		90	
Anderson and Spall (2001)		9	11.7	19.30	0.606
		13	15	22.6	0.664
		18.25	17.5	25.3	0.692
Nasr and Lai (1997a, b, 2000)	2.5, 4.25, 7.5, 11.25, 15		$1.049 (S/d_0)^{0.948}$	3.514 $(S/d_0)^{0.673}$	0.411 $(S/d_0)^{0.155}$
Lin and Sheu (1990)	30		21.6	30	0.72
	40		27.6	40	0.69
			$0.48 S/d_0 + 8.98$		
Ko and Lau (1989)	2.5		1.5	10.5	0.142
Murai et al. (1976)		4.1		9.8	
		17.5		21	
		20.6		23.7	
		23.7		26	
		31		31	
Tanaka (1970, 1974)			For $8.5 < y < 16$: $5.06 (S/d_0)^{0.27}$		0.715
	8.5-25		For $y > 16$: 0.667 (S/d_0)	$7.08 (S/d_0)^{0.27}$	$0.094(S/d_0)$ _{0.73}
Miller and Comings (1960)	6		6.8	12.4	0.55

5.2 Numerical study

5.2.1 Inlet boundary conditions of confluent jets on a curved surface

Figure 14: Axial velocity (V) and its RMS along the cross-section of nozzle 4R-4C of an array on a cylinder (measured by HWA and predicted by R $k-\epsilon$ and RSM at $x/d_0=0.26$) and of a CoJ of an array on a flat plate (measured by LDA at $x/d_0=0.086$) and of nozzle CJ I of a single row of round jets (measured by LDA at $x/d_0=0.086$); $Re=3290$.

Using a short-height contraction nozzle causes the flow to follow the nozzle curvature and results in narrow convergence of the fluid streamlines. The axial velocity profile along the diameter of each nozzle of confluent jets

5.2 Numerical study

had a saddle-back shape with two distinct off-center overshoots (Figure 14), which is different from the fully developed profile of flow from a long pipe (Figure 9 in Ghahremanian and Moshfegh (2014b)). While the spanwise velocities of each nozzle have similar profiles, the vertical velocity profile of a contraction nozzle is different from flow from a long pipe. Contraction nozzles attenuate the turbulence at the nozzle exits and generate low turbulence throughout the core region of the jets. Numerical simulations of the confluent-jet supply device with both RANS models ($k - \epsilon$ and RSM) produced greater turbulence intensities at the nozzle exits than the measurement results (Figure 14).

5.2.2 Single round jet

Two two-equation, low Reynolds number RANS turbulence models (Low Re $k - \epsilon$ and Low Re SST $k - \omega$), a three-equation transition turbulence model ($k - kl - \omega$) and a four-equation transition turbulence model (SST eddy-viscosity) were evaluated based on how they numerically simulated a round jet. The results from the numerical simulations of the three regions (initial, transition and turbulence) of a round jet were compared with hot-wire anemometry measurements (Figure 5 in Ghahremanian and Moshfegh (2014a)) and showed that the numerical predictions with the low Reynolds turbulence models (low Re SST $k - \omega$ and low Re $k - \epsilon$) agreed well with the measurements.

The centerline decay of the axial velocity increases to a maximum immediately after the nozzle edge due to the vena contracta effect (Figure 15). The turbulent region, which begins after the transition region, follows the well-known self-similarity characteristic. The axial velocity profile near the nozzle edge has a saddle-back shape, while after a small downstream distance from the nozzle, the velocity profile becomes shaped as a top hat. As the jet develops downstream, the axial velocity profile becomes conical and spreads. The axial velocity profile in the turbulent region reaches a self-similar shape that matches a Gaussian function and spreads linearly. Two sharp and narrow peaks are present in the RMS profile of the axial velocity very near the nozzle edge, while the RMS of the axial velocity has a low value close to the central region of the jet. The RMS profile forms two wider peaks, while its value at the center of the jet increases.

5 Results

Figure 15: Axial velocity decay and its RMS along the geometrical centerline of a single jet, nozzle CJ I in a single row of round jets, nozzle 3R-4C in an array of jets on a cylinder and a CJ in an array of jets on flat plate.

5.2 Numerical study

5.2.3 Confluent jets on a curved surface

The confluent jets on the curved surface show an initial region in which the maximum velocity increases after the jet issues from the nozzle exit. The jets in this region are indistinguishable (e.g., at a radial distance of $1 d_0$ in Figure 16) and have similar saddle-back profiles to the single jet described above. After the initial region, the SJs and CoJs bend toward the center of the array and form the converging region. The velocity profiles of all three types of confluent jets become conical in this region. While the SJs and CoJs show faster decreases of the maximum velocity than the CJs, the position of the maximum velocity becomes closer to the neighboring jets. The deformation of the SJs and CoJs is much stronger than the CJs in the converging region (see Figure 16). The convergence process continues until the SJs and CoJs merge to their neighboring jets. The numerical simulation of the confluent jets shows that the merging process that occurs between the jets in the columns is much stronger than that between the jets in the rows (e.g., at a radial distance of greater than $7.5 d_0$ in Figure 16). At a radial distance of $7.5 d_0$, the jets in column six merge earlier than the other jets in the middle columns due to the entrainment of ambient air and the greater turbulent kinetic energy in the outward-facing SJs and CoJs. The numerically predicted profiles of the axial velocity and its RMS in column 4 agree well with the measurements, except in the merging region. The numerical simulation of confluent jets reveals that the sub-atmospheric static pressure between the SJs (or CoJs) and their neighboring jets leads to convergence and merging of the jets. The greater turbulent kinetic energy in the outward-facing SJs and CoJs than in the CJs increases the interaction between the jets.

The comparison of the axial velocities along the geometrical centerline of the confluent jets shows good consistency between the measurements and the numerical predictions. The core region, which is an initial region between the jets to the uniform velocity along the geometrical centerline, is present for the CJs, and the length is nearly identical to that of the single round jet except for in the SJs and CoJs. The mixing transition region and interaction region of the confluent jets after the core region begin earlier than with the single jet, which is more obvious for the SJs and CoJs. In the transition, region large-scale vortices interact and facilitate momentum transport, especially near the SJs and CoJs. In sharp contrast to the CJ, the deflection of the maximum axial velocity of a SJ in column 4 ends up on the

5 Results

geometrical centerline of the adjacent jet after 4.2S. Downstream of the combined point, the fluid motion of the combined jet behaves as a self-similar round jet; i.e., the spreading rate, decay rate and higher turbulence moments approach a single dimensionless value. In the combined region, individual jets cannot be identified because the axial velocity profile becomes nearly flat. This similarity between the combined jet and a single round jet requires further investigation.

Figure 16: Contour plots of the velocity at different radial distances from the supply device.

5.3 Parametric study

First, the numerical simulation of a single row of co-planar parallel jets is validated with PIV data along the geometrical centerlines of the central jets and along the cross-sectional profiles. The characteristic points (merging point and combined point) of the numerical case are also compared with those extracted from the PIV measurements and are found to be in good agreement.

The responses of the characteristic points, including the merging point (MP) and combined points (CP1 and CP2), to the design factors, such as the inlet bulk velocity (V), spacing between the nozzles (S) and the nozzle diameter (d), are then determined in three different regions between the jets. The results of 13 unique numerical predictions (CFD) based on the Box-Behnken design are summarized in Table 11. The shortest distance of the merging point (MP) and combined point (CP2) to the nozzle edge occurs in

5.3 Parametric study

region I – I of case 12, which has the shortest spacing between the nozzle centers of all the simulated cases. The greatest distances to the MP, CP1 and CP2 occur in case 4, which has the slowest bulk velocity (V) and greatest spacing (S).

The relationship between the responses (MP, CP1 and CP2) and the set of quantitative design factors (V , S and d) can be obtained using the response surface method (RSM), and a regression model is determined based on the data collected using the Box-Behnken design. The full quadratic model fitted to each response consists of the linear and squared independent design factors as well as the interaction between the two variables. The statistical significance of the design factors in the approximated regression model can be determined using the probability value (P -value) in the analysis of variance (ANOVA). A P -value < 0.05 indicates that the design factor or its square or two-way interaction is statistically significant at a 95% confidence level. The backward elimination approach based on a stepwise regression model can be used to select the significant variables. Each step consists of removing the term with the largest P -value from the full quadratic model. The elimination process terminates when further improvement of the regression equation is less than a critical value.

Table 12 shows estimates of the uncoded coefficients of the regression equation for each response term (MP, CP1 and CP2) in the regions between the jets (I – I, I – II and II – SJ). The estimated regression equation represents the relationship between each response and the design factors, and a value for the response can be calculated by inserting an actual (uncoded) factor. A general form of the regression equation for all of the response terms in the different regions between the jets is as follows:

MP or CP1 or CP2

$$= C_0 + C_1V + C_2S + C_3d + C_4V^2 + C_5S^2 + C_6d^2 + C_7VS + C_8Vd + C_9Sd \quad (84)$$

where C_0 to C_9 are represented in Table 12.

The numerically predicted and statistically approximated characteristic points (MP, CP1 and CP2) are compared, and the agreement between the approximation and the predicted values is satisfactory. The regression equations in the different regions show that the spacing between the nozzles

5 Results

has the most significant influence on the merging point, while the diameter of the nozzles has a moderate impact. The $V \times S$ term is the only two-way interaction term that remained in the regression equations of the MP in the different regions, although all of the square interaction terms are removed. The inlet velocity V has a marginal effect on the approximation of the merging point in regression model in all three in-between regions. The square of the nozzle diameter d^2 has the greatest impact on the approximation of the combined points in the regression equation in the region between two CJ I nozzles. The negative contribution of d^2 implies that the increase of this term reduces CP1 and CP2 in region I – I. In comparison, the two-way interaction term $S \times d$ has the second largest influence on CP2 in region I – I. Two linear terms, S and d , have the greatest impacts on approximation model of CP1 in the regions on the two sides of CJ II. The contributions of the coefficients of the regression model for estimating CP2 in the region II – SJ are similar to those CP2 in region I – I except for the inclusion of S^2 , which has a moderate contribution. The two-way interaction of S and d also plays an important role in the approximation equation of CP2 in all of the in-between regions.

In the initial region, there is no significant dependency on the Reynolds number for the one row of jets. The velocity and Reynolds stress profiles have no observable Reynolds number dependencies after the combined point in the single row setup.

5.3 Parametric study

Table 11: Experimentally measured (PIV) and numerically predicted (CFD) characteristic points.

Region		I - I			I - II			II - SJ		
Characteristic point	MP	CP1	CP2	MP	CP1	CP2	MP	CP1	CP2	
PIV ($\times d_0$)	4.5	28.8	14.40	4.7	26.7	14.40	5.2	20.10	14.40	
Low Re $k - \epsilon$ ($\times d_0$)	5.0	29.34	14.46	4.68	29.34	13.84	4.68	15.96	14.05	
1 ($\times d_{00}$)	1.75	26.92	10.25	1.73	19.96	10.29	1.74	12.14	10.89	
2 ($\times d_{00}$)	2.44	29.77	11.55	2.43	22.12	11.55	2.43	13.16	11.90	
3 ($\times d_{00}$)	0.83	23.39	8.40	0.83	17.75	8.52	0.87	10.81	9.08	
4 ($\times d_{00}$)	2.96	31.94	12.25	3.06	24.12	12.43	3.07	15.12	12.80	
5 ($\times d_{00}$)	0.99	23.44	8.53	0.99	17.76	8.53	0.99	10.93	9.22	
6 ($\times d_+$)	1.16	17.84	8.99	1.14	18.44	9.00	1.14	11.22	9.72	
7 ($\times d_-$)	2.34	29.58	11.32	2.34	21.77	11.32	2.33	13.08	11.83	
8 ($\times d_+$)	1.29	24.86	9.14	1.29	18.70	9.28	1.30	11.42	10.02	
9 ($\times d_-$)	2.49	29.65	11.49	2.48	21.79	11.49	2.49	13.20	12.00	
10 ($\times d_+$)	1.89	27.69	10.64	1.91	20.48	10.64	1.92	12.39	11.14	
11 ($\times d_-$)	3.35	33.42	12.89	3.35	24.10	12.89	3.34	14.43	13.09	
12 ($\times d_+$)	0.34	17.77	7.30	0.40	17.34	7.67	0.38	10.84	7.80	
13 ($\times d_-$)	1.51	25.87	9.76	1.51	19.27	9.76	1.52	11.79	10.52	

Table 12: Analysis of the response surface regression with the backward elimination method

Characteristic point	Region	I - I			I - II			II - SJ		
		Model	Coefficients	P-value	Coefficients	P-value	Coefficients	P-value		
MP	Constant	-0.0259	-	-0.0261	-	-0.0335	-	-		
		Linear								
	V (m/s)	0.00244	0.002	0.00244	0.001	0.00313	0.004	0.004		
	S (m)	5.875	0.0	5.813	0.0	6.56	0.0	0.0		
	d (m)	-3.125	0.0	-3.0	0.0	-3.125	0.0	0.0		
		Two-way interaction								
	V × S (m²/s)	-0.25	0.029	-0.25	0.016	-0.313	0.025	0.025		
	Constant	-1.362	-	-0.0149	-	-0.1122	-	-		
		Linear								
	V (m/s)	0.0568	0.015	-	-	0.1006	0.073	0.073		
CP1	S (m)	16.31	0.0	9.25	0.0	14.69	0.0	0.0		
	d (m)	285.6	0.227	8.13	0.0	5.75	0.001	0.001		
		Square interaction								
	d² (m²)	-13286	0.028	-	-	-	-	-		
		Two-way interaction								
	V × S (m²/s)	-	-	-	-	-0.937	0.05	0.05		
	V × d (m²/s)	-7.75	0.012	-	-	-	-	-		
	Constant	-0.0271	-	-0.047	-	-0.0475	-	-		
		Linear								
	V (m/s)	0.00319	0.006	0.00456	0.006	0.00375	0.007	0.007		
CP2	S (m)	1.81	0.0	5.88	0.0	7.11	0.0	0.0		
	d (m)	10.14	0.01	-5.87	0.002	8.37	0.003	0.003		
		Square interaction								
	S² (m²)	-	-	-	-	-385	0.034	0.034		
	d² (m²)	-1321	0.017	-	-	-1538	0.034	0.034		
		Two-way interaction								
	V × S (m²/s)	-0.312	0.018	-0.438	0.01	-0.375	0.033	0.033		
	S × d (m²)	1000	0.001	625	0.047	1500	0.001	0.001		

6 Conclusions

The results of this study show the flow characteristics in the near field of a single jet and multiple interacting jets using several experimental methods and turbulence models. The study confirmed the confluence and interaction between multiple jets in different configurations.

The overall conclusion of this investigation is that the flow characteristics of multiple interacting jets can be measured and predicted with an adequate degree of accuracy with the employed techniques. Measurements of the mean and fluctuating velocity in the near-field of confluent jets were collected using the hot-wire anemometry (HWA), laser Doppler anemometry (LDA) and particle image velocimetry (PIV) techniques. The PIV technique was used to provide instantaneous and mean velocity fields and turbulence properties in the proximal region of different arrays of multiple interacting jets. The PIV measurements proved to be essential for detecting several salient features of the dynamics of confluent jets. The LDA measurements were performed for comparison and provided more detailed insight into the near-field of multiple interacting jets.

This study showed that the flow field of multiple interacting jets could be numerically predicted by Computational Fluid Dynamics (CFD). The results of the computational simulations were validated by comparing the predictions with experimentally measured flow characteristics. The degree of agreement between the numerically predicted values and the experimentally obtained results was found to depend on the inlet boundary conditions, the location within the flow field and the employed turbulence model. The degree of agreement between the CFD results and the measurements for various variables was also found to be different. Numerical predictions by two turbulence models (realizable $k - \epsilon$ and Reynolds stress model) of the supply device producing the confluent jets is validated by HWA measurements near the nozzles exit in both lateral and vertical direction. The low Reynolds number $k - \epsilon$ and the SST $k - \omega$ with low Reynolds correction show the best overall performance in order to predict the whole field (laminar, transition and turbulent regions) of a round jet. The velocity and its RMS measured with HWA in the near-field of confluent jets on a curved surface are consistent with the results of numerical simulations using the SST $k - \omega$ with low-Reynolds correction

6 Conclusions

and using the prescribed inlet boundary conditions. This study shows that the near flow-field of a single row of parallel coplanar jets predicted by numerical methods is validated by PIV measurement. The results of numerical prediction by the low Reynolds $k - \epsilon$ model agree with the experimental data at the center-plane of the single row of jets.

The results showed that the axial velocity profile near the exit of short-height contraction nozzles had a saddle-back shape. The axial velocity along the geometrical centerline of the jets increased sharply in the initial region, while the axial velocity profiles formed a top hat shape. The RMS of the axial velocity was attenuated to low values in the core region of the jets in the initial region, while the RMS formed two off-centered narrow peaks in the mixing layer. The two peaks of the RMS of the axial velocity in the mixing layers of the jets became thicker, while the velocity profile become conical, and its geometrical centerline value began to decrease after the mixing transition region. The flow field of the multiple interacting jets contained converging, merging, and combined regions, where the starting point of each region depended mostly on the configuration of the nozzles. In the turbulent region, the decay of the maximum axial velocity of the confluent jets reached a plateau, while the centerline velocity of the single jet continued to decay.

The results also showed that the deflection of the confluent jets enhanced the turbulent fluctuations of velocity, which resulted in higher level and more rapid development of turbulent kinetic energy in the outward-facing side jets. Higher values of the lateral Reynolds normal stress within the inner face of the mixing layer of the side jets caused anisotropic turbulence. The anisotropy caused by the axial Reynolds normal stress was dominant in the flow field of multiple interacting jets except on the outward-facing side jets with higher lateral Reynolds normal stresses. The core region of the combined jet became isotropic when the Reynolds shear stress was dominant. In regions with mean flow curvature, the distribution of Reynolds stresses was an asymmetric. This study revealed that the maximum Reynolds normal stress of multiple interacting jets had a lower value than that observed for a single round jet.

In the current work, response surface methodology together with numerical prediction in fluid dynamics is also used to study the influence of a number of design factors (such as inlet velocity, spacing between nozzles

6 Conclusions

and nozzle diameters) on the near-field of a single row of parallel coplanar jets. After applying the backward elimination approach based on stepwise regression for removing the statistically insignificant terms, the correlations between characteristic points (merging and combined points) and three design factors are determined. The spacing variable shows the lowest P -value and the highest coded coefficient in all of the approximated regression models, which exhibits the greatest impact on the merging and combined points in various regions between the jets. All of the square terms are removed from the response equations of MP, and only one two-way interaction term ($V \times S$) remains in the regression model, with a marginal effect. The diameter of the nozzles has the second influence on the approximation model of CP1 in the regions on both sides of CJ II. The square of the nozzle diameter (d^2) contributes in the regression equations of CP1 and CP2 in some regions between the jets. The squared factor S^2 appears only in the response equation of CP2 in the region II-SJ.

7 Looking to the Future Studies

Several important questions were raised by this study, and they need to be investigated. What is the influence of the boundary conditions at the nozzle exit on the interaction of confluent jets? What is the effect of temperature on the confluence of multiple interacting jets? How do jets interact with each other, and what is the nature of the interactions? What causes the merging points and combined points? How do coherent structures behave during the evolution of the flow in different regions between the confluent jets?

Understanding the effects of the boundary conditions at the origin on the development of the flow from multiple interacting jets is necessary to establish, e.g., how the characteristic points and regions depend on the boundary conditions at the nozzle exit. The same attention that different nozzle types (e.g., long pipe, orifice plate or chevron nozzle) for the single round jet received could be considered in future studies of confluent jets. The role of boundary conditions at the nozzle exit on the route to the mixing transition and to the combined jet would be useful to study.

The studies described above are the first to provide detailed experimental data for a coplanar row of jets and an array of round jets at low Reynolds numbers using a two-dimensional measurement technique. The measurement results may be valuable for the validation and development of computational techniques and turbulence models that are used to predict numerically the flow field of multiple round jets.

While the obtained empirical data-set for this study is useful for investigating the near-field of confluent jets, experiments with time-resolved PIV are expected to increase the detailed understanding of the dynamic processes in the interaction region of the multiple jet configurations. Because coherent structures play a significant role in turbulence dynamics, proper identification of these well-organized repeating eddies is necessary to gain insight into the momentum transport phenomena in the flow from multiple interacting jets. Several methods, such as the vorticity magnitude, second invariant of the velocity gradient tensor, kinematic vorticity number, proper orthogonal decomposition, or dynamic mode decomposition (DMD) may be used to extract the dominant flow structures. Accurate data for identifying structures must also be obtained

7 Looking to the Future Studies

from numerical (e.g., LES) or experimental (e.g., time resolved PIV) approaches.

References

- Abdel-Rahman, A.A., Chakroun, W., Al-Fahed, S.F., 1997. LDA measurements in the turbulent round jet. *Mechanics Research Communications* 24 (3), 277-288.
- Absil, L.H.J., 1995. Analysis of the Laser Doppler Measurement Technique for Application in Turbulent Flows. PhD thesis, Delft University of Technology, Delft, The Netherlands.
- Adrian, R.J., Westerweel, J., 2010. Particle Image Velocimetry. Cambridge University Press.
- Albrecht, H.E., Damaschke, N., Borys, M., Tropea, C., 2010. Laser Doppler and Phase Doppler Measurement Techniques. Springer.
- Anderson, E.A., Spall, R.E., 2001. Experimental and numerical investigation of two-dimensional parallel jets. *Journal of Fluids Engineering - Transactions of the ASME* 123 (2), 401-406.
- Antonia, R.A., Zhao, Q., 2001. Effect of initial conditions on a circular jet. *Experiments in Fluids* 31 (3), 319-323.
- Ashforth-Frost, S., Jambunathan, K., 1997. Effect of nozzle geometry and semi-confinement on the potential core of a turbulent axisymmetric free jet. *International Communications in Heat and Mass Transfer* 23 (2), 155-162.
- Ball, C.G., Fellouah, H., Pollard, A., 2012. The flow field in turbulent round free jets. *Prog. Aeosp. Sci.* 50, 1-26.
- Becker, H.A., Booth, B.D., 1975. Mixing in the interaction zone of two free jets. *AIChE Journal* 21 (5), 949-958.
- Bell, J.H., Mehta, R.D., 1988. Contraction design for small low-speed wind tunnels, National Aeronautics and Space Administration, Ames Research Center; Stanford University, Department of Aeronautics and Astronautics, Joint institute for aeronautics and acoustics p. 37.
- Bell, J.H., Mehta, R.D., 1989. Boundary-layer predictions for small low-speed contractions. *AIAA Journal* 27 (3), 372-374.

References

- Boersma, B.J., Brethouwer, G., Nieuwstadt, F.T.M., 1998. A numerical investigation on the effect of the inflow conditions on the self-similar region of a round jet. *Physics of Fluids* 10 (4), 899-909.
- Böhm, B., Stein, O., Kempf, A., Dreizler, A., 2010. In-nozzle measurements of a turbulent opposed jet using PIV. *Flow, Turbulence and Combustion* 85 (1), 73-93.
- Box, G.E.P., Wilson, K.B., 1951. On the experimental attainment of optimum conditions. *Journal of the Royal Statistical Society. Series B (Methodological)* 13 (1), 1-45.
- Bradshaw, P., 1966. The effect of initial conditions on the development of a free shear layer. *Journal of Fluid Mechanics* 26 (02), 225-236.
- Bradshaw, P., 1971. *An introduction to turbulence and its measurement*. Pergamon Press.
- Bruun, H.H., 1979. Interpretation of hot-wire probe signals in subsonic airflows. *Journal of Physics E: Scientific Instruments* 12 (12), 1116.
- Bruun, H.H., 1995. *Hot-Wire Anemometry - Principles and Signal Analysis* Oxford University Press, New York, NY.
- Cao, G., Awbi, H., Yao, R., Fan, Y., Sirén, K., Kosonen, R., Zhang, J., 2014. A review of the performance of different ventilation and airflow distribution systems in buildings. *Building and Environment* 73 (0), 171-186.
- Chen, H.C., Patel, V.C., 1988. Near-wall turbulence models for complex flows including separation. *The American Institute of Aeronautics and Astronautics (AIAA)* 26 (6), 641-648.
- Cho, J.R., Chung, M.K., 1992. A $k-\epsilon-\gamma$ equation turbulence model. *Journal of Fluid Mechanics* 237, 301-322.
- Cho, Y., Awbi, H.B., Karimipناه, T., 2008. Theoretical and experimental investigation of wall confluent jets ventilation and comparison with wall displacement ventilation. *Building and Environment* 43 (6), 1091-1100.
- Corrsin, S., 1944. Investigation of the behavior of parallel two-dimensional air jets. NASA No. 4H24.

References

- Crow, S.C., Champagne, F.H., 1971. Orderly structure in jet turbulence. *Journal of Fluid Mechanics* 48 (3), 547-591.
- Daly, B.J., Harlow, F.H., 1970. Transport equations in turbulence. *Physics of Fluids* 13 (11), 2634-2649.
- Deo, R.C., Mi, J., Nathan, G.J., 2007. The influence of nozzle-exit geometric profile on statistical properties of a turbulent plane jet. *Experimental Thermal and Fluid Science* 32 (2), 545-559.
- Dimotakis, P.E., Miake-Lye, R.C., Papantoniou, D.A., 1983. Structure and dynamics of round turbulent jets. *Physics of Fluids* 26 (11), 3185-3192.
- Durrani, T.S., Greated, C.A., 1977. *Laser Systems In Flow Measurement*. Plenum Press, New York.
- Durst, F., 1982. Review—combined measurements of particle velocities, size distributions, and concentrations. *Journal of Fluids Engineering* 104 (3), 284-296.
- Durst, F., Melling, A., Whitelaw, J.H., 1981. *Principles And Practice Of Laser-Doppler Anemometry*. Academic Press, New York.
- Durve, A., Patwardhan, A.W., Banarjee, I., Padmakumar, G., Vaidyanathan, G., 2012. Numerical investigation of mixing in parallel jets. *Nuclear Engineering and Design* 242 (0), 78-90.
- Ferreira, S.L.C., Bruns, R.E., Ferreira, H.S., Matos, G.D., David, J.M., Brandão, G.C., da Silva, E.G.P., Portugal, L.A., dos Reis, P.S., Souza, A.S., dos Santos, W.N.L., 2007. Box-Behnken design: An alternative for the optimization of analytical methods. *Analytica Chimica Acta* 597 (2), 179-186.
- Fluent, A., 2012. *Fluent 14.5, ANSYS Fluent Documentation, Theory guide*.
- Fujisawa, N., Nakamura, K., Srinivas, K., 2004. Interaction of two parallel plane jets of different velocities. *J Visual-Japan* 7 (2), 135-142.
- Geers, L.F.G., 2004. Multiple impinging jet arrays: an experimental study on flow and heat transfer. PhD Thesis thesis, Delft University of Technology, Delft.

References

- Geers, L.F.G., Hanjalic, K., Tummers, M.J., 2006. Wall imprint of turbulent structures and heat transfer in multiple impinging jet arrays. *Journal of Fluid Mechanics* 546, 255-284.
- Geers, L.F.G., Tummers, M.J., Hanjalic, K., 2005. Particle imaging velocimetry-based identification of coherent structures in normally impinging multiple jets. *Physics of Fluids* 17 (5), 055105-055113.
- George, W.K., 1989. The self-preservation of turbulent flows and its relation to initial conditions and coherent structures. Hemisphere, New York.
- George, W.K., Davidson, L., 2004. Role of initial conditions in establishing flow behavior. *AIAA Journal* 42 (3), 438-446.
- George, W.K.J., 1978. Processing of random signals, The Dynamic Flow Conference, pp. 757-793.
- Ghahremanian, S., Moshfegh, B., 2014a. Evaluation of RANS models in predicting low reynolds, free, turbulent round jet. *Journal of Fluids Engineering* 136 (1), 1 - 13.
- Ghahremanian, S., Moshfegh, B., 2014b. A study on proximal region of low Reynolds confluent jets Part 1: Evaluation of turbulence models in prediction of inlet boundary conditions. *ASHRAE Transactions* 120 (1), 256-270.
- Ghahremanian, S., Moshfegh, B., 2014c. A study on proximal region of low Reynolds confluent jets Part 2: Numerical verification of the flow field. *ASHRAE Transactions* 120 (1), 271-285.
- Ghahremanian, S., Svensson, K., Tummers, M.J., Moshfegh, B., 2014a. Near-field development of a row of round jets at low Reynolds numbers. *Experiments in Fluids* 55 (8), 1-18.
- Ghahremanian, S., Svensson, K., Tummers, M.J., Moshfegh, B., 2014b. Near-field mixing of jets issuing from an array of round nozzles. *International Journal of Heat and Fluid Flow* 47, 84-100.
- Ghirelli, F., 2007. $k\epsilon\alpha$: A three-equation eddy-viscosity model of turbulence. *International Journal of Numerical Methods for Heat & Fluid Flow* 17 (2), 140-164.

References

- Gilliland, T., Ranga-Dinesh, K.K.J., Fairweather, M., Falle, S.A.E.G., Jenkins, K.W., Savill, A.M., 2012. External intermittency simulation in turbulent round jets. *Flow, Turbulence and Combustion* 89 (3), 385-406.
- Gouldin, F.C., Schefer, R.W., Johnson, S.C., Kollmann, W., 1986. Nonreacting turbulent mixing flows. *Progress in Energy and Combustion Science* 12 (4), 257-303.
- Groth, J., Johansson, A.V., 1988. Turbulence reduction by screens. *Journal of Fluid Mechanics* 197, 139-155.
- Harima, T., Fujita, S., Osaka, H., 2001. Mixing and diffusion processes of twin circular free jets with various nozzle spacing, 5th World Conference on Experimental Heat Transfer, Fluid Mechanics, and Thermodynamics, pp. 1017-1022.
- Hinze, J.O., 1975. *Turbulence*, 2nd ed. McGraw-Hill.
- Højstrup, J., Rasmussen, K., Larsen, S.E., Højstrup, J., Rasmussen, K., Larsen, S.E., 1976. Dynamic calibration of temperature wires in still air. *Disa Inf.* 20, 22-30.
- Hong, Z.C., Chaung, S.H., 1987. Kinetic theory approach to twin plane jets turbulent mixing analysis. *AIAA Journal* 26 (3), 303-310.
- Hussain, A.K.M.F., Zedan, M.F., 1978. Effects of the initial condition on the axisymmetric free shear layer: Effects of the initial momentum thickness. *Physics of Fluids* 21 (7), 1100-1112.
- Hussein, J.H., Capp, S.P., George, W.K., 1994. Velocity measurements in a high-Reynolds-number, momentum-conserving, axisymmetric, turbulent jet. *Journal of Fluid Mechanics* 258, 31-75.
- Janbakhsh, S., Moshfegh, B., 2014a. Experimental investigation of a ventilation system based on wall confluent jets. *Building and Environment* 80 (0), 18-31.
- Janbakhsh, S., Moshfegh, B., 2014b. Numerical study of a ventilation system based on wall confluent jets. *HVAC&R Research* 20 (8), 846-861.

References

Jongen, T., 1998. Simulation and modeling of turbulent incompressible fluid flows. Ph.D. Thesis thesis, Ecole polytechnique fédérale de Lausanne EPFL, Lausanne.

Karimipناه, T., Awbi, H.B., Moshfegh, B., 2008. The air distribution index as an indicator for energy consumption and performance of ventilation systems. *Journal of the Human-Environment System* 11 (2), 77-84.

Karimipناه, T., Awbi, H.B., Sandberg, M., Blomqvist, C., 2007. Investigation of air quality, comfort parameters and effectiveness for two floor-level air supply systems in classrooms. *Building and Environment* 42 (2), 647-655.

Karimpour, A., Kaye, N., Khan, A., 2011. CFD study of merging turbulent plane jets. *Journal of Hydraulic Engineering* 137 (3), 381-385.

Keane, R., Adrian, R., 1992. Theory of cross-correlation analysis of PIV images. *Applied Scientific Research* 49 (3), 191-215.

Keane, R., Adrian, R.J., 1990. Optimization of particle image velocimeters. I. Double pulsed systems. *Measurement Science and Technology* 1 (11), 1202.

Keane, R., Adrian, R.J., 1991. Optimization of particle image velocimeters: II. Multiple pulsed systems. *Measurement Science and Technology* 2 (10), 963.

Khuri, A.I., Mukhopadhyay, S., 2010. *Response Surface Methodology*. Wiley Interdisciplinary Reviews: Computational Statistics 2 (2), 128-149.

King, L.V., 1914. On the convection of heat from small cylinders in a stream of fluid: Determination of the convection constants of small platinum wires with applications to hot-wire anemometry. *Philosophical Transactions of the Royal Society of London. Series A, Containing Papers of a Mathematical or Physical Character* 214 (509-522), 373-432.

Knystautas, R., 1962. The turbulent jet from a series of holes in line. MERL Report 62-1, McGill University, Canada.

Knystautas, R., 1964. The turbulent jet from a series of holes in line. *Aeronautical Quarterly* 15, 1-28.

References

- Ko, N.W.M., Lau, K.K., 1989. Flow structures in initial region of two interacting parallel plane jets. *Experimental Thermal and Fluid Science* 2 (4), 431-449.
- Koller-Milojevie, D., Schneider, W., 1993. Free and confined jets at low Reynolds numbers. *Fluid Dynamics Research* 12 (6), 307-322.
- Kolmogorov, A.N., 1941. The local structure of turbulence in incompressible viscous fluid for very large reynolds numbers. *Dokl. Akad. Nauk SSSR* 30 (4).
- Koochesfahani, M.M., Dimotakis, P.E., 1986. Mixing and chemical reactions in a turbulent liquid mixing layer. *Journal of Fluid Mechanics* 170, 83-112.
- Kumar, A., Kumar Das, M., 2011. Study of a turbulent dual jet consisting of a wall jet and an offset jet. *Journal of Fluids Engineering- Transactions of the ASME* 133 (10), 101201-101211.
- Lai, J., Nasr, A., 1998. Two parallel plane jets: comparison of the performance of three turbulence models. *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering* 212 (6), 379-391.
- Launder, B.E., Morse, A.P., Rodi, W., Spalding, D.B., 1973. Prediction of free shear flows: A comparison of the performance of six turbulence models, *Proceedings of the NASA Langley Free Shear Flows Conference*, NASA, Washington, DC, pp. 361-422.
- Launder, B.E., Sharma, B.I., 1974. Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc. *Letters in Heat and Mass Transfer* 1 (2), 131-137.
- Launder, B.E., Shima, N., 1989. Second-moment closure for the near-wall sublayer - Development and application. *The American Institute of Aeronautics and Astronautics (AIAA)* 27 (10), 1319-1325.
- Lepicovsky, J., 1989. The role of nozzle exit boundary layer velocity gradient in mixing enhancement of free jets, *Forum on Turbulent Flows: The Third Joint ASCE/ASME Mechanics Conference*, La Jolla, San Diego, CA, pp. 41 - 47.

References

- Leschziner, M.A., Rodi, W., 1981. Calculation of annular and twin parallel jets using various discretization schemes and turbulence-model variations. ASME, Transactions, Journal of Fluids Engineering 103, 352-360.
- Lien, F.S., Leschziner, M.A., 1994. Assessment of turbulence-transport models including non-linear rng eddy-viscosity formulation and second-moment closure for flow over a backward-facing step. Computers & Fluids 23 (8), 983-1004.
- Lin, Y.F., Sheu, M.J., 1990. Investigation of two plane parallel unventilated jets. Experiments in Fluids 10 (1), 17-22.
- Lumley, J.L., 1964. Passage of a turbulent stream through honeycomb of large length-to-diameter ratio. Journal of Fluids Engineering 86 (2), 218-220.
- Marsters, G.F., 1979. Measurements in the flow field of a linear array of rectangular nozzles. Journal of Aircraft 17 (11), 774-780.
- McGuirk, J.J., Rodi, W., 1977. The calculation of three-dimensional free jets, Symposium on turbulent shear flows, Pennsylvania State University.
- McLaughlin, D.K., Tiederman, W.G., 1973. Biasing correction for individual realization of laser anemometer measurements in turbulent flows. Physics of Fluids (1958-1988) 16 (12), 2082-2088.
- Mehta, R.D., Bradshaw, P., 1979. Design rules for small low speed wind tunnels. Aeronautical Journal.
- Menter, F.R., 1994. Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal 32 (8), 1598-1605.
- Menter, F.R., Langtry, R.B., Likki, S.R., Suzen, Y.B., Huang, P.G., Volker, S., 2006. A Correlation-Based Transition Model Using Local Variables---Part I: Model Formulation. Journal of Turbomachinery 128 (3), 413-422.
- Mi, J., Kalt, P., Nathan, G., Wong, C., 2007. PIV measurements of a turbulent jet issuing from round sharp-edged plate. Experiments in Fluids 42 (4), 625-637.

References

- Mi, J., Nathan, G.J., 2004. Mean velocity decay of axisymmetric turbulent jets with different initial velocity profiles, 4th International Conference on Fluid Mechanics, Dalian, China, pp. 618-621.
- Mi, J., Nathan, G.J., Luxton, R.E., 2000. Centreline mixing characteristics of jets from nine differently shaped nozzles. *Experiments in Fluids* 28 (1), 93-94.
- Mi, J., Nathan, G.J., Nobes, D.S., 2001a. Mixing characteristics of axisymmetric free jets from a contoured nozzle, an orifice plate and a pipe. *Journal of Fluids Engineering- Transactions of the ASME* 123 (4), 878-883.
- Mi, J., Nobes, D.S., Nathan, G.J., 2001b. Influence of jet exit conditions on the passive scalar field of an axisymmetric free jet. *Journal of Fluid Mechanics* 432, 91-125.
- Miller, D.R., Comings, E.W., 1960. Force-momentum fields in a dual-jet flow. *Journal of Fluid Mechanics* 7 (2), 237-256.
- Miller, P.L., Dimotakis, P.E., 1991. Reynolds number dependence of scalar fluctuations in a high Schmidt number turbulent jet. *Physics of Fluids A: Fluid Dynamics* 3 (5), 1156-1163.
- Morse, A.P., 1977. Axisymmetric turbulent shear flows with and without swirl PhD thesis, London University, England,
- Moustafa, G.H., 1994. Experimental investigation of high-speed twin jets. *AIAA Journal* 32 (11), 2320-2322.
- Mungal, M.G., Hollingsworth, D.K., 1989. Organized motion in a very high Reynolds number jet. *Physics of Fluids A: Fluid Dynamics* 1 (10), 1615-1623.
- Munson, B.R., Young, D.F., Okiishi, T.H., Huebsch, W.W., 2009. *Fundamentals of Fluid Mechanics*, 6 ed. Wiley.
- Murai, K., Taga, M., Akagawa, K., 1976. An experimental study on confluence of two two-dimensional jets. *Bulletin of JSME* 19 (134), 958-964.

References

- Nasr, A., Lai, J.C.S., 1997a. Comparison of flow characteristics in the near field of two parallel plane jets and an offset plane jet. *Physics of Fluids* 9 (10), 2919-2931.
- Nasr, A., Lai, J.C.S., 1997b. Two parallel plane jets: mean flow and effects of acoustic excitation. *Experiments in Fluids* 22 (3), 251-260.
- Nasr, A., Lai, J.C.S., 2000. The effects of nozzle spacing on the development of two parallel plane jets. *International Journal of Transport Phenomena* 2 (1), 57-70.
- Obot, N.T., Graska, M.L., Trabold, T.A., 1984. The near field behavior of round jets at moderate reynolds numbers. *The Canadian Journal of Chemical Engineering* 62 (5), 587-593.
- Okamoto, T., Yagita, M., Watanabe, A., Kawamura, K., 1985. Interaction of twin turbulent circular jet. *Bulletin of JSME* 28 (238), 617-622
- Oosthuizen, P.H., 1983. An experimental study of low reynolds number turbulent circular jet flow, American Society of Mechanical Engineers, Applied Mechanics, Bioengineering, and Fluids Engineering Conference. ASME Paper 83-FE-36, Houston, TX.
- Panchapakesan, N.R., Lumley, J.L., 1993. Turbulence measurements in axisymmetric jets of air and helium. Part 1. Air jet. *Journal of Fluid Mechanics* 246 (-1), 197-223.
- Pani, B., Dash, R., 1983. Three dimensional single and multiple free jets. *Journal of Hydraulic Engineering* 109 (2), 254-269.
- Papadopoulos, G., Pitts, W.M., 1999. A generic centerline velocity decay curve for initially turbulent axisymmetric jets. *Journal of Fluids Engineering- Transactions of the ASME* 121 (1), 80-85.
- Pope, S.B., 1978. An explanation of the turbulent round-jet/plane-jet anomaly. *AIAA Journal* 16, 279-281.
- Quinn, W.R., 2006. Upstream nozzle shaping effects on near field flow in round turbulent free jets. *European Journal of Mechanics - B/Fluids* 25 (3), 279-301.

References

- Quinn, W.R., Militzer, J., 1989. Effects of nonparallel exit flow on round turbulent free jets. *International Journal of Heat and Fluid Flow* 10 (2), 139-145.
- Raffel, M., Willer, C., Wereley, S., Kompenhans, J., 2007. Particle Image Velocimetry: A practical guide, Second Edition ed. Springer.
- Ranga Dinesh, K.K.J., Savill, A.M., Jenkins, K.W., Kirkpatrick, M.P., 2010. LES of intermittency in a turbulent round jet with different inlet conditions. *Computers & Fluids* 39 (9), 1685-1695.
- Richards, C.D., Pitts, W.M., 1993. Global density effects on the self-preservation behaviour of turbulent free jets. *Journal of Fluid Mechanics* 254 (-1), 417-435.
- Ricou, F.P., Spalding, D.B., 1961. Measurements of entrainment by axisymmetrical turbulent jets. *Journal of Fluid Mechanics* 11 (1), 21-32.
- Rieth, M., Proch, F., Stein, O.T., Pettit, M.W.A., Kempf, A.M., 2014. Comparison of the sigma and smagorinsky LES models for grid generated turbulence and a channel flow. *Computers & Fluids* 99, 172-181.
- Ronneberger, O., Raffel, M., Kompenhans, J., 1998. Advanced evaluation algorithms for standard and dual plane particle image velocimetry, 9th International Symposium on Applications of Laser Techniques to Fluid Mechanics, Lisbon. Portugal.
- Sami, S., Carmody, T., Rouse, H., 1967. Jet diffusion in the region of flow establishment. *Journal of Fluid Mechanics* 27 (02), 231-252.
- Sayigh, A., 2013. Sustainability, Energy and Architecture: Case Studies in Realizing Green Buildings. Elsevier Science.
- Schlichting, H., Gersten, K., 2000. Boundary-Layer Theory, 8th ed. Springer, New York, NY.
- Shih, T.-H., Liou, W.W., Shabbir, A., Yang, Z., Zhu, J., 1995. A new $k-\epsilon$ eddy viscosity model for high reynolds number turbulent flows. *Computers & Fluids* 24 (3), 227-238.

References

- Somerscales, E.F.C., 1981. Tracer methods: Laser Doppler Velocimeter in: Emrich, R.J. (Ed.), *Methods of experimental physics*. Academic Press, London, pp. 93-239.
- Spalding, D.B., 1971. Concentration fluctuations in a round turbulent free jet. *Chemical Engineering Science* 26 (1), 95-107.
- Stein, O., Böhm, B., Dreizler, A., Kempf, A., 2011. Highly-resolved LES and PIV Analysis of Isothermal Turbulent Opposed Jets for Combustion Applications. *Flow, Turbulence and Combustion* 87 (2-3), 425-447.
- Tanaka, E., 1970. The interference of two-dimensional parallel jets : 1st report, experiments on dual jet. *Bulletin of JSME* 13 (56), 272-280.
- Tanaka, E., 1974. The interference of two-dimensional parallel jets : 2nd report, experiments on the combined flow of dual jet. *Bulletin of JSME* 17 (109), 920-927.
- Tanaka, E., Nakata, S., 1975. The interference of two-dimensional parallel jets : 3rd report, the flow region near the nozzle in triple jets. *Transactions of the Japan Society of Mechanical Engineers* 41 (342), 537-545.
- Tatsumi, K., Tanaka, M., Woodfield, P.L., Nakabe, K., 2010. Swirl and buoyancy effects on mixing performance of baffle-plate-type miniature confined multijet. *Int J Heat Fluid Fl* 31 (1), 45-56.
- Thielen, L., Hanjalić, K., Jonker, H., Manceau, R., 2005. Predictions of flow and heat transfer in multiple impinging jets with an elliptic-blending second-moment closure. *International Journal of Heat and Mass Transfer* 48 (8), 1583-1598.
- Thielen, L., Jonker, H.J.J., Hanjalić, K., 2003. Symmetry breaking of flow and heat transfer in multiple impinging jets. *International Journal of Heat and Fluid Flow* 24 (4), 444-453.
- Todde, V., Spazzini, P.G., Sandberg, M., 2009. Experimental analysis of low-Reynolds number free jets: Evolution along the jet centerline and Reynolds number effects. *Experiments in Fluids* 47 (2), 279-294.
- Townsend, A.A., 1976. *The structure of turbulent shear flow*, Second edition ed. Cambridge University Press, Cambridge.

References

- Tummers, M.J., 1999. Investigation of a turbulent wake in an adverse pressure gradient using laser Doppler anemometry. PhD thesis, Delft University of Technology, Delft, The Netherlands.
- Villiermaux, E., Hopfinger, E.J., 1994. Periodically arranged co-flowing jets. *Journal of Fluid Mechanics* 263, 63-92.
- Vouros, A., Panidis, T., 2008. Influence of a secondary, parallel, low Reynolds number, round jet on a turbulent axisymmetric jet. *Experimental Thermal and Fluid Science* 32 (8), 1455-1467.
- Walters, D.K., Cokljat, D., 2008. A three-equation eddy-viscosity model for Reynolds-averaged Navier-Stokes simulations of transitional flow. *Journal of Fluids Engineering- Transactions of the ASME* 130 (12), 121401-121414.
- Walters, D.K., Leylek, J.H., 2004. A new model for boundary layer transition using a single-point RANS approach. *Journal of Turbomachinery* 126 (1), 193-202.
- Westerweel, J., 1993. Digital Particle Image Velocimetry - Theory and Application. PhD thesis, Delft University of Technology,
- Westerweel, J., 1994. Efficient detection of spurious vectors in particle image velocimetry data. *Experiments in Fluids* 16 (3-4), 236-247.
- Westerweel, J., Scarano, F., 2005. Universal outlier detection for PIV data. *Experiments in Fluids* 39 (6), 1096-1100.
- Wilcox, D.C., 2010. *Turbulence Modeling for CFD*, 3rd ed. DCW Industries, Inc.
- Wolfshtein, M., 1969. The velocity and temperature distribution in one-dimensional flow with turbulence augmentation and pressure gradient. *International Journal of Heat and Mass Transfer* 12 (3), 301-318.
- Wong, C.Y., Nathan, G.J., Mi, J.C., 2005. The near-field structure of free jets from a contoured nozzle, a long pipe and an orifice plate, Fourth Australian Conference on Laser Diagnostics in Fluid Mechanics and Combustion, The University of Adelaide, South Australia, Australia, pp. 145-148.

References

- Wynnanski, I., Fiedler, H., 1969. Some measurements in the self-preserving jet. *Journal of Fluid Mechanics* 38 (3), 577-612.
- Xu, G., Antonia, R.A., 2002. Effect of different initial conditions on a turbulent round free jet. *Experiments in Fluids* 33 (5), 677-683.
- Yeh, Y., Cummins, H.Z., 1964. Localized fluid flow measurements with an he-ne laser spectrometer. *Applied Physics Letters* 4 (10), 176-178.
- Yimer, I., Becker, H.A., Grandmaison, E.W., 1996. Development of flow from multiple-jet burners. *The Canadian Journal of Chemical Engineering* 74 (6), 840-851.
- Yin, Z., Zhang, H., Lin, J., 2007. Experimental study on the flow field characteristics in the mixing region of twin jets. *Journal of Hydrodynamics, Ser. B* 19 (3), 309-313.
- Yule, A.J., 1978. Large-scale structure in the mixing layer of a round jet. *Journal of Fluid Mechanics* 89 (3), 413-432.
- Yuu, S., Shimoda, F., Jotaki, T., 1979. Hot wire measurement in the interacting two-plane parallel jets. *AIChE Journal* 25 (4), 676-685.
- Zaman, K.B.M.Q., Hussain, A.K.M.F., 1984. Natural large-scale structures in the axisymmetric mixing layer. *Journal of Fluid Mechanics* 138, 325-351.

Appended papers

Papers

The articles associated with this thesis have been removed for copyright reasons. For more details about these see:

<http://urn.kb.se/resolve?urn=urn:nbn:se:liu:diva-113259>