



Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin

School of Engineering

Department of Mechanical, Manufacturing and Biomedical Engineering

Characterisation of the Flow and Heat Transfer Behaviour Associated with a Wall- Bounded Dual Jet Flow

Paula Murphy

March 3, 2024

Supervisor: Dr Séamus O'Shaughnessy

A dissertation submitted to the University of Dublin in partial fulfilment of
the requirements for the degree of Doctor in Philosophy

Declaration

I declare that this thesis has not been submitted as an exercise for a degree at this or any other university and it is entirely my own work.

I agree to deposit this thesis in the University's open access institutional repository or allow the Library to do so on my behalf, subject to Irish Copyright Legislation and Trinity College Library conditions of use and acknowledgement.

I consent to the examiner retaining a copy of the thesis beyond the examining period, should they so wish (EU GDPR May 2018).

Signed: _____

Paula Murphy, August 2024

Abstract

The purpose of this investigation is to characterise the fluid flow and heat transfer behaviour of a dual jet comprising a wall jet, discharged immediately parallel to a solid boundary, flowing alongside a parallel jet whose exit is offset from the solid boundary by some distance, *i.e.*, an offset jet. Despite the widespread industrial applications of dual jet flows, which include turbine blade shielding, heat exchangers, fuel injection systems, wastewater evacuation process, air-conditioning, and noise suppression technology, the flow and heat transfer characteristics of a wall-bounded dual jet remains severely underrepresented in the published literature. As a result, the heat transfer capabilities and driving flow mechanisms associated with dual jet flows are not yet fully understood, meaning they cannot be optimised towards specific applications.

Most notably, the lack of verified, repeatable experimental flow and heat transfer measurements in the published literature has led to significant discrepancies across the published numerical findings, which furthers the requirement for the provision of a suitable means of validation with simulating dual jet flows. The present study is thus concerned with characterising the transfer of heat to a dual jet flow for varying Reynolds number, offset ratio and velocity ratio through experimental means, as well as acquiring an understanding of the underlying flow phenomena which drive this heat transfer behaviour. The findings of the experimental investigation are hence used to develop a validated numerical dual jet model and improve on the dual jet numerical modelling techniques available in the literature.

For the experimental investigation, air is adopted as the working fluid and each jet Reynolds number is varied in the range $5500 \leq Re \leq 12000$, while the offset ratio and velocity ratio are varied in the ranges $1 \leq OR \leq 7$ and $0.5 \leq V_r \leq 2$, respectively. The initial heat transfer study subjects the airflow to a constant heat flux generation of 1670 W/m^2 in the bounding wall and the surface Nusselt number distribution is measured using infrared thermography techniques. The velocity field is subsequently captured using particle image velocimetry for a more limited offset ratio range of $1 \leq OR \leq 3$, while the range of Re and V_r remain consistent.

The results reveal a distinct time-averaged local Nusselt number profile in the wall boundary, where the magnitude and positions of a local Nu_x minimum and maximum can readily be controlled through adjusting Re , OR and V_r . This is attributed to changes in the size of the recirculation zone, which ultimately dictates the extent of the wall jet deflection and, subsequently, the thickness, length and streamwise position of a small separation region induced adjacent to the solid wall boundary. Analysis of the surface-averaged Nusselt number indicates a linear relationship between $\overline{Nu}_x$ and Re , where $\overline{Nu}_x$ rises linearly with increasing jet Reynolds number, while higher values of $\overline{Nu}_x$ are noted for lower OR . The time-resolved flow data indicates the occurrence of periodic vortex shedding, the frequency of which is dependent on Re , OR and V_r , however, a limited range of unsteady V_r values are identified.

The numerical investigation uses Ansys Fluent 2023 R2 to model an identical dual jet flow configuration by means of the $\gamma - Re_\theta$ transition model. The numerical data is acquired using a steady state solver for a limited subset of the experimental range of jet parameters, *i.e.*, $5500 \leq Re \leq 12000$, $V_r = 1$ and $OR = 1$ and 3 . A velocity inlet is applied at the entrance to each jet, where the associated velocity profile, turbulence intensity and turbulent viscosity ratio are tuned appropriately with respect to the experimental data. The results reveal a successful emulation of the experimental heat transfer findings, thus enabling a full validation of the numerical model setup within the accepted uncertainty limits. The numerical model hence introduces significant improvements to dual jet numerical modelling compared to that available in the published literature, despite a slightly slower rate of jet development, where the resulting heat transfer behaviour can be considered more reliable and better aligned with physical reality.

Acknowledgments

First and foremost, I would like to thank my supervisor, Dr Séamus O'Shaughnessy, for his continued support and guidance throughout the course of my PhD. I would also like to extend a special thanks to Professor Darina Murray for helping me to get started on my journey and initially providing me with the opportunity to pursue a PhD. The completion of this project would not have been possible without the expertise and advice offered to me by Dr Tim Persoons regarding multiple aspects of my experimental setup, it was very much appreciated, and I am extremely grateful. Similarly, I would like to thank Dr Michael Gibbons for advising me on IR thermography and the necessary post-processing.

I would also like to thank everyone in the workshop for aiding me in the manufacture of my experimental setup. I would like to especially thank Gerard Byrne for the many months of hands-on assistance offered to me in the lab and motivational conversations. Your practical experience was invaluable towards the completion of this project and I am grateful for all the advice offered. I would also like to thank everyone in the office for all the intellectual conversations over the past few years, as well as providing me with much needed support and entertainment.

I would like to thank the Irish Research Council (IRC) for the financial support offered to me during the course of this PhD (grant number GOIPG/ 2019/3719). I would also like to thank the Irish Centre for High-end Computing (ICHEC) for the use of their computational facilities (project number tceng044c).

Finally, I would like to thank my friends and family for all the support offered to me over the past 4 years. I would like to especially thank my parents for all the love and support over the years, your advice and guidance has been much appreciated. Last, but certainly not least, I would like to thank my partner, Dean, for supporting me through all these years, the completion of this thesis would not have been possible without you.

Contents

Declaration.....	ii
Abstract.....	iv
Acknowledgments	vi
Contents	viii
List of Figures.....	xii
List of Tables	xx
Nomenclature.....	xxii
Chapter 1 Introduction.....	1
1.1. Overview of Dual Jets	3
1.2. Research Aims and Objectives	5
Chapter 2 Dual Jet Flow	9
2.1. Turbulent Free Jets	9
2.1.1. Single Plane Jet.....	10
2.1.2. Plane Parallel Jets	11
2.2. Wall-Bounded Turbulent Jets.....	12
2.2.1. Wall Jets	13
2.2.2. Offset Jets	15
2.3. Dual Jet Flow	17
2.4. Summary	19
Chapter 3 Literature Review.....	21
3.1. Hydrodynamic Studies of Dual Jet Flows	21
3.2. Heat Transfer Characteristics of Dual Jet Flows	29
3.3. Summary	34
3.3.1. Research Objectives	36
Chapter 4 Experimental Investigation	41
4.1. Experimental Rig Setup.....	41
4.1.1. Airflow Measurement and Conditioning	41

4.1.2.	Heat Transfer Study: Thermography System	49
4.1.3.	Flow Visualisation Study: Particle Image Velocimetry Setup.....	56
4.2.	Experimental Procedure	63
4.2.1.	Heat Transfer Investigation	63
4.2.2.	Flow Visualisation Investigation	63
4.3.	Data Reduction.....	65
4.3.1.	Heat Transfer Study.....	65
4.3.2.	PIV Study	71
4.4.	Calibration of Instrumentation	77
4.4.1.	Thermocouple.....	77
4.4.2.	Volumetric Flowrate.....	77
4.4.3.	IR Camera.....	81
4.4.4.	PIV Spatial Calibration.....	85
4.5.	Experimental Uncertainty Analysis.....	87
4.5.1.	Thermocouple Uncertainty	89
4.5.2.	Airflow Uncertainty.....	89
4.5.3.	IR Camera Uncertainty	90
4.5.4.	Uncertainty in Heat Transfer Measurements	91
4.5.5.	Uncertainty in Particle Image Velocimetry	92
Chapter 5	Experimental Setup Validation	95
5.1.	Effectiveness of Airflow Conditioning	95
5.1.1.	Flow Quality at Jet Nozzle Exit.....	95
5.1.2.	Two-Dimensionality	97
5.2.	Repeatability of Measured Data	99
5.3.	Comparison with Published Literature.....	100
5.3.1.	Single Wall Jet.....	100
5.3.2.	Single Offset Jet	103
Chapter 6	Dual Jet Experimental Results	109
6.1.	Flow Behaviour	109
6.1.1.	Time-Averaged Analysis.....	109
6.1.2.	Time-Resolved Analysis	115
6.2.	Heat Transfer Characteristics	119
6.2.1.	Time-Averaged Analysis.....	119

6.3.	Effect of Confinement	122
6.4.	Effect of Reynolds Number	125
6.4.1.	Flow Field Characteristics	125
6.4.2.	Heat Transfer Behaviour.....	132
6.5.	Effect of Offset Ratio	136
6.5.1.	Flow Field Characteristics	136
6.5.2.	Heat Transfer Behaviour.....	142
6.6.	Effect of Velocity Ratio	146
6.6.1.	Flow Structure	146
6.6.2.	Heat Transfer Characteristics	156
6.7.	Summary	162
Chapter 7	Numerical Investigation	165
7.1.	Review of Computational Fluid Dynamics	165
7.1.1.	Fundamental Equations	165
7.1.2.	Meshing Considerations in CFD	167
7.1.3.	Turbulence Modelling	168
7.1.4.	Verification and Validation in Numerical Modelling	171
7.2.	Numerical Modelling in Dual Jet Studies.....	173
7.3.	Numerical Approach	175
7.3.1.	Model assumptions	175
7.3.2.	Computational Domain.....	176
7.3.3.	Turbulence Model.....	177
7.3.4.	Governing Equations	178
7.3.5.	Boundary Conditions	179
7.3.6.	Mesh Design.....	180
7.3.7.	Discretisation Scheme	180
7.3.8.	Pressure-Velocity Coupling.....	181
7.3.9.	Turbulent Prandtl Number.....	182
7.3.10.	Solution Approach.....	182
Chapter 8	Numerical Model Results.....	183
8.1.	Tuning of Model Parameters	183
8.1.1.	Turbulence Model.....	183
8.1.2.	Discretisation Scheme	185

8.1.3.	Inlet Boundary Conditions.....	185
8.1.4.	Turbulent Prandtl Number.....	190
8.1.5.	Summary of Tuning Parameters	191
8.2.	Assessment of Model Implementation	192
8.2.1.	Assessment of Mesh Quality	192
8.2.2.	Model Verification and Validation	193
8.3.	Comparison with Experimental Results	198
8.3.1.	Flow Characteristics	198
8.3.2.	Heat Transfer Behaviour	202
8.4.	Summary	204
Chapter 9	Conclusions.....	205
9.1.	Future Work	208
References		209
Appendix A	Thermocouple Calibration Curves.....	221
Appendix B	Tabulated Data for mp , cp , UVC and LVC	225
Appendix C	Calibration Certification	227

List of Figures

Figure 1.1: Schematic of turbine blade shielding in gas turbine engine [7].....	2
Figure 1.2: Schematic of single (left) vs dual (right) lip ring design used for cooling during blown film extrusion process [8].....	2
Figure 1.3: Noise suppression at exit of turbofan engine through the use of a co-flowing bypass duct, as provided by (a) Sadeghian [17] and (b) Ji et al. [13]	3
Figure 1.4: Dual jet flow schematic.	4
Figure 2.1 General flow structure of a free plane jet [40].....	10
Figure 2.2: Fluid flow schematic for plane parallel jets.....	12
Figure 2.3: Fluid flow schematic of a plane wall jet.....	13
Figure 2.4: Typical streamwise velocity profiles obtained within the self-similar region of a planar wall jet, as reported by (a) Eriksson et al. [72] and (b) AbdulNour et al. [68].....	14
Figure 2.5: Variation in the local heat transfer coefficient in the solid boundary when subject to a plane wall jet flow, as provided by (a) AbdulNour et al. [68] and (b) Akfirat [73]	15
Figure 2.6: Fluid flow schematic for an offset jet.	15
Figure 2.7: Variation in the local Nusselt number in the solid boundary when subject to a plane offset jet flow, as provided by (a) Kim et al. [79] ($Re = 39000$) and Song et al. [81].	16
Figure 2.8: Flow structure of a dual jet, adapted from Wang and Tan [19].	17
Figure 3.1: (a) Mean velocity vector plot and (b) von Kármán vortex interaction as acquired by Wang and Tan [19] using particle image velocimetry.	22
Figure 3.2: Change in (a) streamwise velocity and (b) turbulence intensity with downstream distance, as reported by Li et al. [2] by means of LES, where L is analogous to the jet width w , i.e., $L = w$	23
Figure 3.3: Contours and vector plot of velocity magnitude for $OR = 5$ as per Li et al. [92] by means of the $k-\epsilon$ turbulence model, where $V_r =$ (a) 0.25, (b) 0.5, (c) 0.75, (d) 1, (e) 1.33, (f) 2, and (g) 4... ..	23
Figure 3.4: Streamline plot of a dual jet indicating the presence of two counter-rotating vortices [1]	24
Figure 3.5: Identification of (a) UVC, LVC and mp and (b) cp as per Kumar and Das [1] by means of the $k-\epsilon$ turbulence model.	24
Figure 3.6: Streamline plot acquired by Mondal et al. [20] using the $k-\epsilon$ turbulence model for a dual jet with $OR =$ (a) 0.7 and (b) 0.6, demonstrating the stable and unstable flow regimes, respectively.....	25
Figure 3.7: Streamline plots representing the effect of varying OR on the dual jet flow field, as acquired by Kumar [83] using the $k-\epsilon$ turbulence model.	26
Figure 3.8: Decay of maximum velocity with downstream distance for a dual jet with $Re = 15,000$ for $OR =$ (a) 2 and (b) 6, as acquired by Kumar [83] using the $k-\epsilon$ turbulence model.	26
Figure 3.9: Contours of (a) streamwise velocity and (b) static pressure for $Re = 15,000$ and $OR = 8$ obtained by Hnaïen et al. [25] using the $k-\omega$ turbulence model.	27

Figure 3.10: Streamline plots for a dual jet with varying OR and $Re = 15000$ as per Assoudi et al. [26], acquired using the Reynolds stress model (RSM).	28
Figure 3.11: Definition of (a) wall inclination angle β and (b) offset jet ejection angle α in the context of the dual jet flow configuration, as per Hnaïen et al. [10] and Hnaïen et al. [86], respectively.	29
Figure 3.12: Nux distribution along the solid wall, as per Vishnuvardhanarao and Das [28], for a dual jet with varying Re and $OR = 1$, acquired using the $k-\epsilon$ turbulence model, where (a) $Vr = 1$ (b) $Vr = 0.25$ (c) $Vr = 0.5$ and (d) $Vr = 4$	30
Figure 3.13: Variation in Nux profile with respect to changing OR for $Re = 15000$ as per (a) Farooq and Das [29] and (b) Kumar [83] by means of the $k-\epsilon$ turbulence model.	31
Figure 3.14: Variation in the Nux profile with respect to Re for a dual jet as per (a) Mondal et al. [22] ($OR = 1$) and (b) Hnaïen et al. [84] ($4 \leq OR \leq 19$) by means of the $k-\epsilon$ and $k-\omega$ turbulence models, respectively.	32
Figure 3.15: Variation in the Nux profile for a dual jet with respect to (a) Re , as per Singh and Dewan [24] by means of the $k-\epsilon$ turbulence model for $OR = 1$ and (b) β , as per Hnaïen et al. [10] by means of the $k-\omega$ turbulence model for $Re = 15,000$ and $OR = 8$	33
Figure 3.16: Variation in the Nux profile for a dual jet with $Re = 15000$ with respect to (a) varying amplitude, as per Singh et al. [87] by means of the $k-\epsilon$ turbulence model for $OR = 6$ and (b) varying co-flow velocity, as per Mondal et al. [102] by means of the $k-\omega$ turbulence model for $OR = 8$	34
Figure 4.1 Experimental Rig Setup. (a) Isometric view, (b) plan view, and (c) front elevation view.	42
Figure 4.2: Dual jet offset ratio changed from (a) $OR = 2$ to (b) $OR = 7$ by means of interchangeable offset plate inserts.	43
Figure 4.3: Schematic of airflow chamber design	44
Figure 4.4: Airflow chamber design for each jet, (a) complete view (b) internal view of airflow conditioning components.	45
Figure 4.5: Final honeycomb flow straightener design.	46
Figure 4.6: Arrangement of flow conditioning components inside the airflow chamber, where $x = 0$ coincides with the contraction entry. (1) indicates the first mesh screen location, (2) marks the position of the honeycomb, its associated depth represented by the blue lines, (3) indicates the second mesh screen location and (4) is the nozzle contraction.	48
Figure 4.7: Flowchart of flow control function implemented in the LabVIEW VI.	49
Figure 4.8: a) UWF wall test section design, b) Placement of wall test section in experimental setup with full downstream insert.	51
Figure 4.9: Section view of foil tensioning design for UWF test section.	52
Figure 4.10: Foil tensioning mechanism.	53
Figure 4.11: Position of thermal camera with respect to heated foil.	54
Figure 4.12: Paint layer thickness image acquired from Labscope software. (a) raw image, and (b) annotated image	55
Figure 4.13: Flowchart of testing function implemented in the LabVIEW VI for the heat transfer study.	56
Figure 4.14: Typical arrangement for 2C-2D PIV experiment, as per Raffel et al. [132].	57
Figure 4.15: Particle image velocimetry setup.	57
Figure 4.16: Alignment of the laser sheet in the PIV experimental setup.	60
Figure 4.17: Surface temperature contour plot of test foil (a) before and (b) after the foil edge detection, where the jet exit is located along the left side of the image and the flow direction is from left to right	67
Figure 4.18: Downstream temperature gradient along centreline of test foil before foil edge detection.	67

Figure 4.19: Flow of thermal energy into the foil and paint layers of a $dx \times dz$ foil and paint element.	68
Figure 4.20: Foil energy balance as per equation 4.18 in the context of the experimental setup.	69
Figure 4.21: Comparison of sample instantaneous flow field (a) before and (b) after POD post-processing..	75
Figure 4.22: Thermocouple calibration curves for thermocouples representing (a) T_w and (b) T_o	77
Figure 4.23: Experimental setup for volumetric flowrate calibration.	78
Figure 4.24: Sample velocity profile acquired inside the PVC inlet pipe leading to the wall jet at the maximum blower output, where the measured and predicted data points are highlighted.	79
Figure 4.25: Differential pressure to volumetric flowrate calibration curves for (a) the wall jet and (b) the offset jet.	81
Figure 4.26: (a) Position of heater block for in-situ counts-to-temperature calibration. (b) Schematic of calibration heater block instrumentation.	83
Figure 4.27: (a) Schematic of thermocouple placement on underside of foil. (b) Thermal image captured of foil during counts-to-temperature calibration $^{\circ}\text{C}$	83
Figure 4.28: Comparison of counts to temperature calibration curves from in-situ calibration and blackbody calibration.	84
Figure 4.29: (a) IR camera spatial calibrator (b) placement of spatial calibrator in experimental setup.	84
Figure 4.30: (a) Sample surface temperature contour plot acquired during the IR camera spatial calibration and (b) location of temperature peaks as per the MATLAB script.	85
Figure 4.31: (a) PIV spatial calibration target (b) Final spatial calibration and distortion correction of PIV field-of-view.	86
Figure 4.32: Relationship between T_s and the relative expanded uncertainty associated with h and Nux	92
Figure 5.1: Outlet (a) streamwise velocity and (b) turbulence intensity profile for a SWJ at $Re = 12000$	96
Figure 5.2: Contour plot of turbulence intensity for single wall jet at $Re = 12000$	97
Figure 5.3: Contour plot of the time-averaged surface temperature (T_s) distribution when $Re = 5500$ for (a) a single wall jet, (b) a single offset jet, (c) a dual jet with $OR = 1$ and (d) a dual jet with $OR = 5$	98
Figure 5.4: Comparison of the Nux profiles corresponding to $Z = 0$ and $Z = \pm w$, along with the Z -averaged Nux profile, for a dual jet with $Re = 5500$ and $OR =$ (a) 1 and (b) 5, where the error bars pertain to the Z -averaged dataset.	99
Figure 5.5: Repeatability of time-averaged dual jet Nux profiles for $Re = 12000$, where $OR =$ (a) 1 and (b) 5.	99
Figure 5.6: Velocity contour and vector plot for single wall jet for $Re =$ (a) 6000 and (b) 12000.	100
Figure 5.7: (a) Decay of local maximum streamwise velocity with downstream distance for a SWJ compared with that of Wang and Tan [19]. (b) Local streamwise velocity profile for a SWJ for $Re = 12000$ at $X = 8, 10$, and 12 compared to that of AbdulNour et al. [68] ($X = 10.16$, $Re = 7700$) and Kim et al. [79] ($X = 10$, $Re = 39000$).	101
Figure 5.8: Time-averaged Nux profiles for a SWJ at various Re compared to the findings of AbdulNour et al. [68] for $Re = 7700$, Mabuchi and Kumada [74] for $Re = 9140$ and Akfirat [73] for both $Re = 5540$ and $Re = 8310$	102
Figure 5.9: Velocity contour and vector plot for a SOJ for (a) $Re = 5500$, $OR = 1$, (b) $Re = 12000$, $OR = 1$, (c) $Re = 5500$, $OR = 3$, and (d) $Re = 12000$, $OR = 3$	103
Figure 5.10: Streamline plot showing recirculation region between SOJ and the solid bounding wall for $Re = 12000$ and $OR =$ (a) 1 and (b) 3.	104
Figure 5.11: Decay of local maximum streamwise velocity with downstream distance for a SOJ with varying Re for (a) $OR = 1$ and (b) $OR = 3$, compared with that of Wang and Tan [19] ($Re =$	

10000 OR = 1), Nasr and Lai [76] (Re = 11000 OR = 0.6) and Pelfrey and Liburdy [93] (Re = 15000 OR = 6).	105
Figure 5.12: Time-averaged Nux profiles for a SOJ at various Re for OR = (a) 1, (b) 3, (c) 5 and (d) 7.	106
Figure 5.13 Time-averaged Nux profiles for a single offset jet at various Re compared to the findings of Kim et al. [45] for Re = 39000 and equivalent offset ratio of 1.5.	107
Figure 6.1: Contour and vector plot of time-averaged velocity field for a dual jet with Re = 10000 and OR = Vr = 1.	110
Figure 6.2: (a) Streamline plot and (b) Close-up of recirculation region for dual jet with OR = Vr = 1 and Re = 10000.	110
Figure 6.3: Identification of the locations of the merge and combined points, upper and lower vortex centres, for a dual jet with OR = Vr = 1 and Re = 10000.	111
Figure 6.4: Shear layer and boundary layer development for a dual jet with Re = 10000 and OR = Vr = 1 compared to that of Wang and Tan [19].	112
Figure 6.5: Comparison of the dual jet vorticity contour plot with jet half-width locations and boundary layer thickness for Re = 10000 and OR = Vr = 1.	113
Figure 6.6: Normalised (a) velocity profiles at various downstream locations and (b) streamwise decay of the maximum velocity for a dual jet with Re = 10000 and OR = Vr = 1 alongside that of Wang and Tan [19].	114
Figure 6.7: Turbulence intensity distribution inside a dual jet flow field for Re = 10000 and OR = Vr = 1.	115
Figure 6.8: Instantaneous velocity contour plots for a dual jet with Re = 10000 and OR = Vr = 1, where (a) t = 0.0035 s, (b) t = 0.004 s, (c) t = 0.0045 s and (d) t = 0.005 s.	116
Figure 6.9 Instantaneous streamline plots for a dual jet with Re = 10000 and OR = Vr = 1, where (a) t = 0.0035 s, (b) t = 0.004 s, (c) t = 0.0045 s and (d) t = 0.005 s.	117
Figure 6.10: (a) Vorticity and (b) swirling strength contour plot (with U-utrans vector plot overlayed) for a dual jet with Re = 10000 and OR = Vr = 1 at t = 0.2045 s, where (b) focusses only on the region marked as 'A' in (a).	118
Figure 6.11: Analysis of the transverse velocity component v at mp for a dual jet with Re = 10000 and OR = Vr = 1. (a) Change in v with respect to time t, (b) Power spectral density (PSD) of v time signal.	118
Figure 6.12: Time-averaged Nux profile for a dual jet with Re = 10000 and OR = Vr = 1 alongside the corresponding streamline plot and decay of maximum streamwise velocity.	120
Figure 6.13: Comparison of the dual jet Nux profile for Re = 10000 and OR = Vr = 1 with that obtained by Vishnuvardhanarao and Das [28], Mondal et al. [22] and Singh and Dewan [24] for identical dual jet flow parameters.	121
Figure 6.14: Velocity contour and vector plots comparing the effect of the confinement for OR = 1 and Vr = 1, where (a, c) Re = 5500 and (b, d) Re = 12000. Top/bottom row shows confined/unconfined cases, respectively.	122
Figure 6.15: Velocity contour and vector plots comparing the effect of the confinement for OR = 3 and Vr = 1, where (a, c) Re = 5500 and (b, d) Re = 12000. Top/bottom row shows confined/unconfined cases, respectively.	123
Figure 6.16: Comparison of the dual jet Nux profiles with and without the smooth enclosure, i.e., confined vs uncovered, for Re = 5500 and 12000 with OR = (a) 1 and (b) 3, where Vr = 1.	124
Figure 6.17: Contour and vector plot of the time-averaged velocity field for a dual jet with OR = 1 and Vr = 1, where Re =(a) 6000, (b) 8000, (c) 10000 and (d) 12000.	126
Figure 6.18: Contour and vector plot of the time-averaged velocity field for a dual jet with OR = 3 and Vr = 1, where Re =(a) 6000, (b) 8000, (c) 10000 and (d) 12000.	126

Figure 6.19: Comparison of dual jet streamline plot for changing Re when $V_r = 1$, where (a) $Re = 6000$, $OR = 1$ (b) $Re = 6000$, $OR = 3$ (c) $Re = 12000$, $OR = 1$ and (d) $Re = 12000$, $OR = 3$	127
Figure 6.20: Shear layer and boundary layer development for $Re = 6000, 8000, 10000$ and 12000 for (a) $OR = 1$ and (b) $OR = 3$ and $V_r = 1$	127
Figure 6.21: Effect of Re on the X-location (left) and Y-location (right) of mp, cp, UVC and LVC, for a dual jet with various OR and $V_r = 1$, where (a, b) $OR = 1$, (c, d) $OR = 2$, (e, f) $OR = 3$	128
Figure 6.22: Decay of the maximum streamwise velocity, normalised with respect to the overall maximum velocity, for various Re and $V_r = 1$, where (a) $OR = 1$ and (b) $OR = 3$	130
Figure 6.23: Comparison of the normalised velocity profiles for each Re at various downstream locations for a dual jet with $OR = 1$ and $V_r = 1$	131
Figure 6.24: Comparison of the normalised velocity profiles for each Re at various downstream locations for a dual jet with $OR = 3$ and $V_r = 1$	132
Figure 6.25: (a) Power spectral density (PSD) and (b) Strouhal number (St) associated with the v time signal in the vicinity of mp for a dual jet with $OR = 1$, $V_r = 1$ and $Re = 6000, 8000, 10000$ and 12000	132
Figure 6.26: Time-averaged Nux profiles for a dual jet for different Re values at $V_r = 1$ for $OR =$ (a) 1, (b) 3, (c) 5 and (d) 7.	133
Figure 6.27: Relationship between the surface-averaged, time-averaged Nusselt number Nux and the dual jet Reynolds number for $V_r = 1$	135
Figure 6.28: Contour plot and streamlines for the time-averaged velocity field of a dual jet with $Re = 12000$ and $V_r = 1$, where $OR =$ (a) 1, (b) 2 and (c) 3.	137
Figure 6.29: Comparison of shear layer and boundary layer development for $OR = 1$ and $OR = 3$ in a dual jet with $Re = 5500$ and $V_r = 1$	137
Figure 6.30: Effect of OR on the (a) X-location and (b) Y-location of mp, cp, UVC and LVC, for a dual jet with $Re = 5500$ and $Re = 12000$ and $V_r = 1$	138
Figure 6.31: Decay of the local maximum streamwise velocity, normalised with respect to the overall maximum velocity, for a dual jet with $V_r = 1$ and varying OR, where (a) $Re = 5500$ and (b) $Re = 12000$, compared to that of Kumar [27] for $Re = 15000$ and $OR = 2$	139
Figure 6.32: Comparison of the normalised velocity profiles for $OR = 1, 2$ and 3 at various downstream locations for a dual jet with $Re = 12000$ and $V_r = 1$	140
Figure 6.33: Instantaneous contour plot and streamlines for the velocity field of a dual jet with $Re = 10000$ and $V_r = 1$ at a sample timestep t, where $OR =$ (a) 2 and (b) 3.	141
Figure 6.34: Instantaneous vorticity contour plot of a dual jet with $Re = 10000$ and $V_r = 1$, where $OR =$ (a) 2 and (b) 3, alongside isolines of constant swirling strength λ_{ci}	141
Figure 6.35: (a) Power spectral density (PSD) and (b) Strouhal number (St) associated with the v time signal in the vicinity of mp for a dual jet with $Re = 10000$ and $V_r = 1$ for $OR = 1, 2$ and 3	142
Figure 6.36: Time-averaged Nux profiles for a dual jet for different OR values and $V_r = 1$ for $Re =$ (a) 6000, (b) 8000, (c) 10000 and (d) 12000.	143
Figure 6.37: Comparison of dual jet Nux profile obtained for $Re = 12000$ and $V_r = 1$ with that of (a) Kumar [83] ($Re = 15000$, $OR = 2$) and Hnaïen et al. [84] ($Re = 10000$, $OR = 2$) for $OR = 2$ and (b) Kumar [83] ($Re = 15000$, $OR = 4$) for $OR = 3$ and $OR = 5$	144
Figure 6.38: Relationship between the surface-averaged, time-averaged Nusselt number Nux and the dual jet offset ratio for $V_r = 1$	145
Figure 6.39: Power law fit describing the relationship between $NuxRe^{0.5}$ and OR for $V_r = 1$	145
Figure 6.40: Effect of V_r on the dual jet flow structure for $OR = 1$, where 2 mass flow rates are displayed horizontally for each V_r case. (a) $V_r = 0.5$, $Re_{tot} = 16000$, (b) $V_r = 0.5$, $Re_{tot} = 18000$,	

(c) $V_r = 0.8$, $Re_{tot} = 13500$, (d) $V_r = 0.8$, $Re_{tot} = 20000$, (e) $V_r = 1.5$, $Re_{tot} = 15000$, (f) $V_r = 1.5$, $Re_{tot} = 20000$, (g) $V_r = 2$, $Re_{tot} = 16000$, (h) $V_r = 2$, $Re_{tot} = 18000$	147
Figure 6.41: Effect of V_r on the dual jet flow structure for $OR = 2$, where 2 mass flow rates are displayed horizontally for each V_r case. (a) $V_r = 0.5$, $Re_{tot} = 16000$, (b) $V_r = 0.5$, $Re_{tot} = 18000$, (c) $V_r = 0.8$, $Re_{tot} = 13500$, (d) $V_r = 0.8$, $Re_{tot} = 20000$, (e) $V_r = 1.5$, $Re_{tot} = 15000$, (f) $V_r = 1.5$, $Re_{tot} = 20000$, (g) $V_r = 2$, $Re_{tot} = 16000$, (h) $V_r = 2$, $Re_{tot} = 18000$	148
Figure 6.42: Effect of V_r on the dual jet flow structure for $OR = 3$, where 2 mass flow rates are displayed horizontally for each V_r case. (a) $V_r = 0.5$, $Re_{tot} = 16000$, (b) $V_r = 0.5$, $Re_{tot} = 18000$, (c) $V_r = 0.8$, $Re_{tot} = 13500$, (d) $V_r = 0.8$, $Re_{tot} = 20000$, (e) $V_r = 1.5$, $Re_{tot} = 15000$, (f) $V_r = 1.5$, $Re_{tot} = 20000$, (g) $V_r = 2$, $Re_{tot} = 16000$, (h) $V_r = 2$, $Re_{tot} = 18000$	149
Figure 6.43: Effect of V_r on the vorticity contour plot for a dual jet with $Re_{tot} = 16000$ and $OR = 2$, where $V_r =$ (a) 0.5 and (b) 2.....	150
Figure 6.44: Effect of V_r on the X-location (left) and Y-location (right) of mp, cp, UVC and LVC, for a dual jet with various OR and $Re_{tot} = 16000$, where (a, b) $OR = 1$, (c, d) $OR = 2$ and (e, f) $OR = 3$	151
Figure 6.45: Effect of decreasing (left) and increasing (right) V_r with respect to $V_r = 1$ on the decay of the maximum streamwise velocity, normalised with respect to the overall maximum velocity, for a dual jet with $Re_{tot} = 16000$ and $OR =$ (a, b) 1, (c, d) 2 and (e, f) 3.	153
Figure 6.46: Comparison of the normalised velocity profiles for $V_r = 0.5$ (left) and $V_r = 2$ (right) at $X = 8$, 10, and 12 for a dual jet with $Re_{tot} = 16000$ to that of dual jet with $Re = 12000$ and $V_r = 1$ for the same OR , where (a, b) $OR = 1$ (c, d) $OR = 3$. The $V_r = 2$ cases are also compared to Kim et al. [79] for a SWJ ($X = 10$, $Re = 39000$).	155
Figure 6.47: Effect of V_r on the power spectral density (PSD) associated with the v time signal for a dual jet with $Re_{tot} = 16000$ for $OR =$ (a) 1, (b) 2 and (c) 3, and (d) the corresponding Strouhal number St trend.	156
Figure 6.48: Effect of changing total mass flow rate m , and hence the total Reynolds number Re_{tot} , for a constant velocity ratio of $V_r = 0.8$, where $OR =$ (a) 1, (b) 3, (c) 5 and (d) 7.....	157
Figure 6.49: Effect of changing total mass flow rate m , and hence the total Reynolds number Re_{tot} , for a constant velocity ratio of $V_r = 1.5$, where $OR =$ (a) 1, (b) 3, (c) 5 and (d) 7.....	158
Figure 6.50: Effect of decreasing V_r for a dual jet of constant mass flow rate, i.e., $Re_{tot} = 16000$, and varying offset ratio, where $OR =$ (a) 1, (b) 3, (c) 5 and (d) 7.	160
Figure 6.51: Effect of increasing V_r for a dual jet of constant mass flow rate, i.e., $Re_{tot} = 16000$, and varying offset ratio, where $OR =$ (a) 1, (b) 3, (c) 5 and (d) 7.	161
Figure 6.52: Variation of time-averaged Nux profile for changing V_r as reported by Vishnuvardhanarao and Das [28] for $Re = 20000$ and $OR = 1$, where the jet inlets velocities, U_w and U_o , are non-dimensionalised with respect to the maximum inlet velocity.	162
Figure 7.1: Typical boundary layer growth for flow past a flat plate.	168
Figure 7.2: Computational domain design for the numerical study, where $w = 7$ mm.	177
Figure 7.3: Boundary conditions imposed on numerical model	179
Figure 7.4: Final mesh design for numerical dual jet study consisting of 1031000 cells.....	180
Figure 8.1: Comparison of the performance of various turbulence models when used to simulate the experimental Nux profile for a dual jet with $Re = 5500$ and $OR = 3$	184
Figure 8.2: Comparison of the output from the numerical model using a blend factor of 0, 0.5, and 1 when used to simulate the experimental Nux profile for a dual jet with $Re = 5500$ and $OR = 3$	185
Figure 8.3: Computational domain to obtain the outlet velocity profile from the contraction inside the airflow conditioning chamber of the dual jet experiment.	186
Figure 8.4: (a) Velocity profile and (b) turbulence intensity profile at the nozzle exit and at a location 35 mm upstream of the nozzle exit as per the standalone contraction simulation.	187

Figure 8.5: Comparison of the Nux profile output from the numerical model for both a uniform velocity inlet profile and the velocity inlet profile obtained using the airflow contraction simulation, for a dual jet with $Re = 5500$ and $OR = 3$	187
Figure 8.6: Effect of varying the turbulence intensity at the velocity inlet on the Nux profile output from the dual jet numerical model for $Re = 5500$ and $OR = 3$	189
Figure 8.7: Effect of varying the turbulent viscosity ratio at the velocity inlet on the Nux profile for $Re = 5500$ and $OR = 3$	189
Figure 8.8: Effect of reducing the intermittency value at the velocity inlet on the Nux profile for $Re = 5500$ and $OR = 3$	190
Figure 8.9: Effect of varying Prt on the Nux profile output for $Re = 5500$ and $OR = 3$	191
Figure 8.10: Analysis of y^+ along solid wall boundary for $Re = 5500$ and $Re = 12000$ at $OR = 3$	192
Figure 8.11: Distribution of cell aspect ratios inside the computational flow domain.	193
Figure 8.12: Comparison of (a) the Nux profiles acquired using the coarse, medium, and fine meshes with the relevant experimental Nux data, and (b) the normalised comparison error with respect to the validation uncertainty interval.	195
Figure 8.13: Contour plot and streamlines representing the steady-state velocity field as per the dual jet numerical model for $V_r = 1$, where (a) $Re = 5500$, $OR = 1$, (b) $Re = 12000$, $OR = 1$, (c) $Re = 5500$, $OR = 3$ and (d) $Re = 12000$, $OR = 3$	199
Figure 8.14: Comparison of numerical and experimental data for the X- (left) and Y- (right) locations of mp , cp , UVC and LVC for varying Re and $V_r = 1$, where $OR =$ (a, b) 1 and (c, d) 3.	201
Figure 8.15: Comparison of numerical and experimental data for the normalised maximum velocity for varying Re and $V_r = 1$, where $OR =$ (a) 1 and (b) 3.	202
Figure 8.16: Comparison of numerical and experimental dual jet Nux profiles for varying Re and $V_r = 1$, where $OR =$ (a) 1 and (b) 3.	203

List of Tables

Table 2.1: Summary of key dimensionless parameters used to define the dual jet flow field.	20
Table 3.1: Summary of main findings of dual jet studies from the published literature. Note that OR and Vr values marked with a * have been converted to align with the definition used in the current study, i.e., OR = dw and Vr = uwuo. (mp = merge point, cp = combined point, LVC = lower vortex centre, UVC = upper vortex centre, PIV = particle image velocimetry, LES = Large eddy simulation, TBC = thermal boundary condition, ADB =adiabatic, UWT = uniform wall temperature, UWF = uniform wall flux)	38
Table 4.1: Relevant Inconel 625® foil properties at 21°C [122]	51
Table 4.2: FLIR A655sc IR camera specifications when used with 25o lens [131]......	54
Table 4.3: Properties of YF3 ZnS anti-reflection coated germanium window, as provided by Shalom EO....	55
Table 4.4: Paint properties adopted for foil energy analysis, as per Raghu and Philip [148].	70
Table 4.5: Summary of uncertainties in thermocouple temperature measurement.	89
Table 4.6: Summary of uncertainties associated with airflow measurements quantities.	90
Table 4.7: Summary of thermal and spatial uncertainty values associated with the IR camera.....	91
Table 4.8: Summary of propagated uncertainties in heat transfer quantities.	91
Table 4.9: Summary of uncertainties in PIV flow quantities for Re = 12000 test case.	94
Table 7.1: Numerical modelling in past dual jet studies, where the model validation approach is specified for each, which includes the chosen validation parameter, the flow type, and the validation data source.	174
Table 8.1: Summary of final set of tuning parameters for each dual jet configuration used as part of the numerical study.....	191
Table 8.2: Verification data for the numerical model, where Nux3, Nux2 and Nux1 correspond to the local Nusselt number data produced by the fine, medium, and coarse meshes, respectively.	196
Table 8.3: Validation data for numerical model using the finest mesh density.	197
Table B.1: Locations of mp, cp, UVC and LVC over the range of Re and OR investigated with Vr = 1. ...	225
Table B.2: Locations of mp, cp, UVC and LVC over the range of Vr and OR investigated with Retot = 16000.	226

Nomenclature

<i>Symbol</i>	<i>Description</i>	<i>Units</i>
A	Area	$[m^2]$
b	Free Jet Half-Width	$[m]$
c	Speed of Sound	$[m/s]$
c_p	Specific Heat Capacity	$[J/kgK]$
C_s	Scattering Cross Section	$[m^2]$
d	Offset Distance	$[m]$
d	Diameter	$[m]$
dx	Streamwise Pixel Dimension	$[m]$
dz	Transverse Pixel Dimension	$[m]$
e_b	Heat Radiation from Black Body	$[W]$
e	Internal Energy	$[J]$
$\dot{e}_{gen}$	Rate of Heat Energy Generation	$[W]$
f_s	Vortex Shedding Frequency	$[Hz]$
g	Gravity	$[m/s^2]$
h	Heat Transfer Coefficient	$[W/m^2 K]$
h	Contraction Height	$[m]$
H	Offset Jet Nozzle Height from Wall	$[m]$
I	Light Intensity Distribution	$[W/m^2]$
I	Current	$[A]$
I_0	Laser Intensity	$[W/m^2]$
k	Thermal Conductivity	$[W/mK]$
k	Turbulent Kinetic Energy per Unit Mass	$[m^2/s^2]$
l	Jet Length	$[m]$

L	Characteristic Length	$[m]$
L	Foil Length	$[m]$
$\dot{m}$	Mass Flowrate	$[kg/s]$
p	Pressure	$[Pa]$
P	Power	$[W]$
q''	Heat Flux	$[W/m^2]$
$\dot{Q}$	Rate of Heat Transfer	$[W]$
R	Thermal Resistance	$[K/W]$
R	Electrical Resistance	$[\Omega]$
t	Time	$[s]$
Δt	Time Delay	$[s]$
T	Temperature	$[K]$
ΔT	Temperature Difference	$[K]$
u	Streamwise Velocity	$[m/s]$
Δu or u'	Streamwise Velocity Fluctuation	$[m/s]$
$u_c(x)$	Combined Uncertainty	$[varies]$
$U_c(x)$	Expanded Uncertainty	$[varies]$
$\mathbf{U}$	Velocity Vector, $\mathbf{U} = u\mathbf{i} + v\mathbf{j} + w\mathbf{k}$	$[m/s]$
v	Transverse Velocity	$[m/s]$
Δv or v'	Transverse Velocity Fluctuation	$[m/s]$
V	Volume	$[m^3]$
V	Volumetric Flowrate	$[m^3/s]$
w	Jet Width	$[m]$
W	Foil Width	$[m]$
x	Streamwise Location	$[m]$
$\mathbf{X}$	Direction Vector, $\mathbf{X} = x\mathbf{i} + y\mathbf{j} + z\mathbf{k}$	$[m]$
y	Wall-Normal Location	$[m]$
z	Spanwise Location	$[m]$

Greek Symbols

<i>Symbol</i>	<i>Description</i>	<i>Units</i>
α	Thermal Diffusivity	$[m^2/s]$
α	Offset Jet Inclination	$[^\circ]$
β	Coefficient of Thermal Expansion	$[K^{-1}]$
β	Screen Porosity	$[-]$
β	Wall Inclination	$[^\circ]$
δ	Thickness	$[m]$
δ_{GRE}^*	Generalised Richardson Extrapolation Grid Error	$[varies]$
ε	Emissivity	$[-]$
ε	Energy Dissipation Rate	$[m^2/s^3]$
γ	Intermittency	$[-]$
μ	Dynamic Viscosity	$[kg/ms]$
λ	Wavelength	$[m]$
λ_{ci}	Swirling Strength	$[s^{-2}]$
θ	Momentum Thickness	$[m]$
ρ	Density	$[kg/m^3]$
ρ	Resistivity	$[\Omega m]$
σ	Stefan Boltzmann Constant	$[W/m^2 K^4]$
τ	Response Time	$[s]$
ν	Kinematic Viscosity	$[m^2/s]$
ω	Vorticity	$[s^{-1}]$
ω	Specific Dissipation Rate	$[s^{-1}]$

Non-Dimensional Numbers

<i>Symbol</i>	<i>Description</i>
AR	Jet Aspect Ratio, l/d
Bi	Biot Number
Br	Brinkman Number
C_p	Wall Pressure Coefficient
CR	Contraction Ratio

ER	Expansion Ratio
Gr	Grashoff Number
K	Pressure Drop Coefficient
I	Turbulence Intensity
k	Coverage Factor
M	Magnification
Ma	Mach number
Nu	Nusselt Number
OR	Offset Ratio, d/w
p	Apparent Order of Accuracy
Pr	Prandtl Number
r_g	Grid Convergence Ratio
Ra	Rayleigh Number
Re	Reynolds Number
Ri	Richardson number
St	Strouhal Number
StK	Stokes Number
μ_t/μ	Turbulent Viscosity Ratio
V_r	Velocity ratio, u_w/u_o
X	Non-Dimensionalised Downstream Location, x/w
Y	Non-Dimensionalised Wall-Normal Location, y/w

Acronyms and Abbreviations

<i>Symbol</i>	<i>Description</i>
CFD	Computational Fluid Dynamics
cp	Combined Point
$DEHS$	Di-Ethyl-Hexyl-Sebacat
DJ	Dual Jet
$DKSM$	Dynamic Kinetic Energy Subgrid-Scale Model
DNS	Direct Numerical Simulation
$DSLM$	Dynamic Smagorinsky-Lily Model

<i>FJ</i>	Free Jet
<i>FOV</i>	Field of View
<i>FS</i>	Factor of Safety
<i>IR</i>	Infrared
<i>LDA</i>	Laser Doppler Anemometry
<i>LES</i>	Large Eddy Simulation
<i>LHS</i>	Left Hand Side
<i>LV</i>	Lower Vortex
<i>LVC</i>	Lower Vortex Centre
<i>mp</i>	Merge Point
<i>NETD</i>	Noise Equivalent Temperature Difference
<i>NUC</i>	Non-Uniformity Correction
<i>PIV</i>	Particle Image Velocimetry
<i>PLA</i>	Polylactic Acid
<i>POD</i>	Proper Orthogonal Decomposition
<i>PSD</i>	Power Spectral Density
<i>RANS</i>	Reynolds Averaged Navier Stokes
<i>RHS</i>	Right Hand Side
<i>RNG</i>	Re-Normalisation Group
<i>RSM</i>	Reynolds Stress Model
<i>SNR</i>	Signal to Noise Ratio
<i>SOJ</i>	Single Offset Jet
<i>SST</i>	Shear Stress Transport
<i>SWJ</i>	Single Wall Jet
<i>UV</i>	Upper Vortex
<i>UVC</i>	Upper Vortex Centre
<i>UWF</i>	Uniform Wall Flux
<i>UWT</i>	Uniform Wall Temperature
<i>VI</i>	Virtual Instrument

Subscripts

<i>Symbol</i>	<i>Description</i>
0.5	Jet Half-Width
1	Layer 1
2	Layer 2
3	Layer 3
<i>ag</i>	Air Gap Properties
<i>air</i>	Air Properties
<i>bl</i>	Boundary Layer
<i>c</i>	Cross Sectional
<i>cap</i>	Capacitive
<i>cond</i>	Conductive
<i>conv</i>	Convective
<i>cp</i>	Combined Point
<i>f</i>	Foil Properties
<i>eff</i>	Effective
<i>Ge</i>	Germanium Window Properties
<i>gen</i>	Generation
<i>jet</i>	Jet
<i>k</i>	Kapton
<i>lc</i>	Lateral Conduction
<i>LVC</i>	Lower Vortex Centre
<i>m</i>	Local Traversal Maximum
<i>max</i>	Local Streamwise Maximum
<i>min</i>	Local Streamwise Minimum
<i>mp</i>	Merge Point
<i>n</i>	Normal
<i>o</i>	Offset Jet
<i>p</i>	Seeding Particle Properties
<i>p</i>	Paint Properties

<i>rad</i>	Radiative
<i>t</i>	Turbulent
<i>th</i>	Theoretical
<i>tot</i>	Total
<i>trans</i>	Translational
<i>UVC</i>	Upper Vortex Centre
<i>s</i>	Settling Chamber
<i>s</i>	Slip
<i>s</i>	Surface Property
<i>st</i>	Stored
<i>w</i>	Wall Jet
∞	Surroundings

Chapter 1

Introduction

This study experimentally and numerically characterises the fluid flow and heat transfer behaviour of a wall-bounded dual jet, where the term ‘dual jet’ refers to any jet configuration encompassing two co-flowing jet streams. When a solid wall boundary is introduced, the, now wall-bounded, dual jet comprises a wall jet flowing immediately parallel to the a solid boundary, alongside a co-flowing offset jet whose inlet is offset from the solid boundary by some distance. This area of study was chosen due to the frequent use of dual jets in industry, such as entrainment processes, enhanced mixing, film-cooling, heat exchangers, and air conditioning, where the use of a dual jet offers an enhanced level of control and optimisation compared to a single jet [1]. One of the most commonly encountered uses of a dual jet arrangement is for wastewater evacuation processes, whereby the enhanced mixing induced by a dual jet arrangement can help to quickly dilute pollutants to a safe level when discharged into a river or lake [2]. This is also an important factor when designing exhaust stacks, where an optimum distance between adjacent exhaust stacks must be identified to obtain the required level of mixing, while controlling the ratio of the exhaust velocities can help control the deflection of the exhaust plumes to avoid certain obstacles [3]. The improved mixing brought about through the use of a dual jet has also proved advantageous in fuel injection systems, where the enhanced dispersion of fuel inside the combustion chamber can help improve the overall engine efficiency [4].

A wall-bounded dual jet is analogous to the setup within a gas turbine, as shown in Figure 1.1, where a cooler wall jet is used to shield the turbine blades from a coflowing stream of high temperature gas, which could be in excess of the melting temperature of the turbine blade itself [5, 6]. The efficiency of a gas turbine has been found to increase with increasing turbine inlet temperature, where an increase of 100 K represents an increase in efficiency of $2 - 3\%$. As a result, it is common to encounter gas temperatures in excess of 1950 K in the vicinity of the turbine blades and it is expected that gas temperatures should exceed 2200 K in the near future [7]. With gas turbines and combined-cycle power stations accounting for approximately 22% of the global electricity production, the topic of turbine blade cooling is currently an important area of research, in particular the combination of film-cooling with air jet impingement.

The use of a co-flowing wall jet to stabilise a much larger offset jet is commonly found in industry, particularly for cooling applications. This can be found in manufacturing processes such a blown film extrusion, which is crucial for the production of thin plastic films from 0.5 mm to $5\text{ }\mu\text{m}$, where the addition of a wall jet to the cooling offset jet elongates the attachment length of the offset jet and significantly reduces fluctuations present

in the flow [8, 9]. This is achieved through the use of a dual lip ring located near the die, as shown in Figure 1.2, which cools the molten tubular film as it exits the die. The goal in this instance is to maximise the rate of solidification of the extruded material, where the output from the production line is generally limited by the cooling capabilities of the airflow [8]. Further to this, as instabilities in the airflow can induce deformities in the extruded product, the co-flowing wall jet plays an important role in ensuring the quality of the final product.

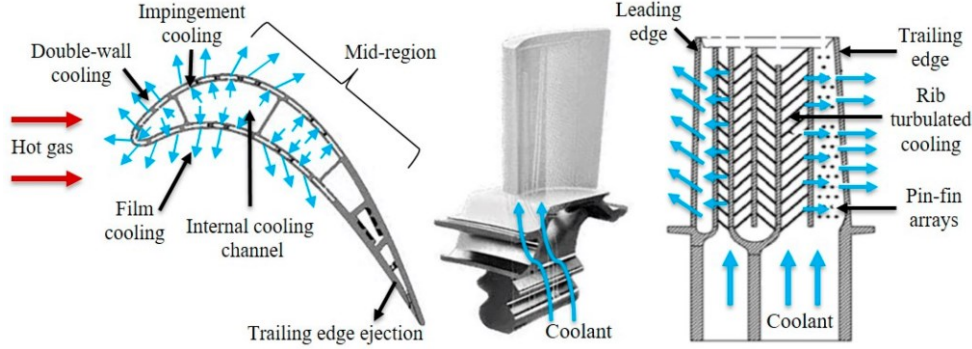


Figure 1.1: Schematic of turbine blade shielding in gas turbine engine [7].

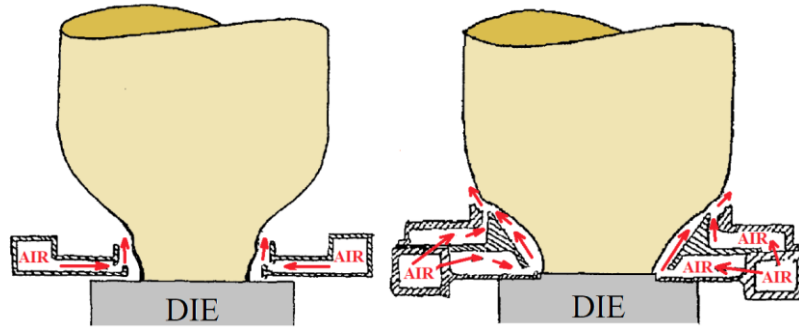


Figure 1.2: Schematic of single (left) vs dual (right) lip ring design used for cooling during blown film extrusion process [8].

The use of dual jets, or a multiple jet arrangement, is often the preferred choice for cooling purposes, not only for the increased rate of heat transfer that they can provide, but also for the noise suppression that can be achieved due to the jet interaction [10, 11]. The use of dual jets as a means of noise suppression is used abundantly throughout the aviation industry, where tight noise budgets imposed on airports have heightened the need for improvement and further research in this area. In addition to this, the Advisory Council for Aeronautics Research in Europe (ACARE) have set out a Strategic Research and Innovation Agenda (SRIA) *i.e.*, Flightpath 2050, to reduce aircraft noise pollution by 65% by 2050, ensuring noise reduction in aviation remains a topical area of research [12]. Examples of the use of dual jets in the suppression of the noise output from an aircraft include the use of a bypass nozzle to induce mixing with the core exhaust gases at the exit of the turbofan engine, as shown in Figure 1.3, or the use of dual planar air curtains on the landing gear or aircraft to reduce airframe noise [11, 13].

The present research is primarily concerned with the heat transfer characteristics of a dual jet flow when used to cool a heated surface, such as that frequently encountered in the electronics cooling industry. The current need for effective thermal management solutions for electronic devices is a well-known issue, where the continuous miniaturisation of electronic components has been found to significantly increase the heat flux from the component during normal operation. To ensure the device is operating within a safe temperature range and to improve the longevity of the device, heat fluxes up to the order of 1 MW/m^2 can require dissipation at

present, which may pose challenges for many of the traditional methods of cooling [14]. There have been significant advancements in this area over recent years, such as the use of impinging jet arrays or minichannels, however, further enhancements are required on a continuous basis as heat fluxes continue to rise [15, 16].

The author therefore proposes the use of wall-bounded dual jet flows as a possible solution in this regard, where the associated enhanced mixing can increase the rate of heat transfer from the heated surface, while an improved level of control would allow for tuning or optimisation to a specific application. This sentiment is reflected across much of the published dual jet studies, all of which cite electronics cooling as a potential application. Dual jet characterisation is clearly a topical area of research, with 50% of all dual jet studies being published since 2019. Having said this, the information available regarding the heat transfer characteristics of a dual jet flow across the published literature is still quite limited and lacks congruency, where predicted trends do not appear to be consistent across multiple studies despite focusing on a similar range of jet parameters. As such, the available literature is insufficient for enabling the control or optimisation of a dual jet flow at present, where the heat transfer capabilities of dual jet flows are still relatively misunderstood.

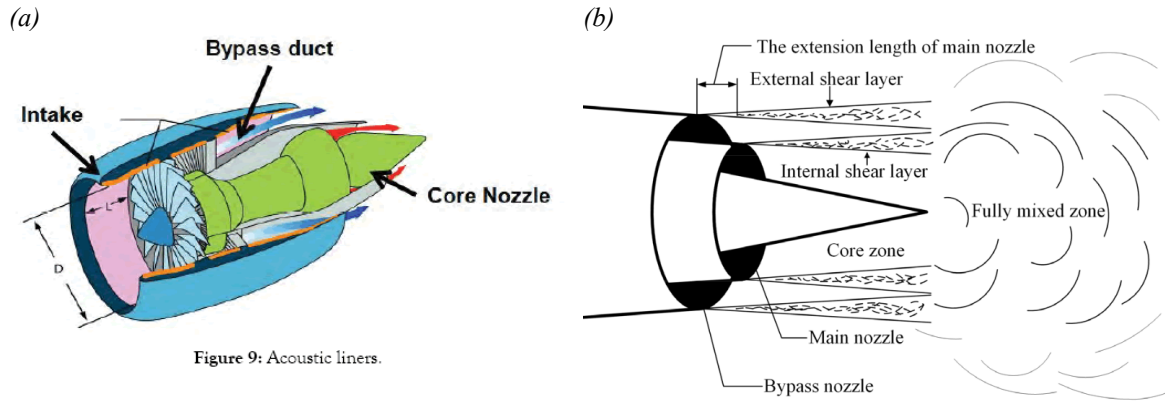


Figure 1.3: Noise suppression at exit of turbofan engine through the use of a co-flowing bypass duct, as provided by (a) Sadeghian [17] and (b) Ji et al. [13]

This chapter is divided into two sections. The first section provides a brief overview of dual jet flow and the work that has been conducted in this area to date, where further information regarding the underlying theory of single and dual jet flow(s) is discussed in Chapter 2. The second section subsequently outlines the objectives of the present study with respect to the gap in the existing literature, which is explored in greater detail in Chapter 3.

1.1. Overview of Dual Jets

For this investigation, the term ‘dual jet’ refers specifically to the combination of a wall jet and an offset jet, where the initial direction of the fluid at the nozzle exit is parallel for both jets, as shown in Figure 1.4. The term ‘wall jet’ is given to a stream of fluid discharged tangentially to a solid wall, whereas an offset jet flows parallel to a wall boundary but offset by some distance [18]. The resulting flow behaviour through the amalgamation of these two jet types is highly complex and distinctly different from that of a wall jet or offset jet individually. As the jets are issued simultaneously from the nozzle exit, a low-pressure region or recirculation zone is created between the wall jet and the offset jet, which causes the offset jet to deflect towards the wall and merge with the wall jet at some downstream distance, *i.e.*, at the merge point (*mp*). Following this, the two jets continue to interact before becoming fully combined, at the combined point (*cp*), beyond

which the dual jet behaves in a similar manner to an equivalent single wall jet. While both wall jets and offset jets have each been studied extensively, the combination of such is severely underrepresented in the literature, meaning the fundamentals of the dual jet flow field are not yet fully understood. Of those studies which have been dedicated to dual jet flow, all except one have been entirely numerical in nature, therefore only a single experimental investigation regarding dual jets has been conducted to date. This experimental study, however, focussed only on acquiring fluid flow measurements, meaning the heat transfer characteristics associated with dual jet flows are yet to be explored through experimentation.

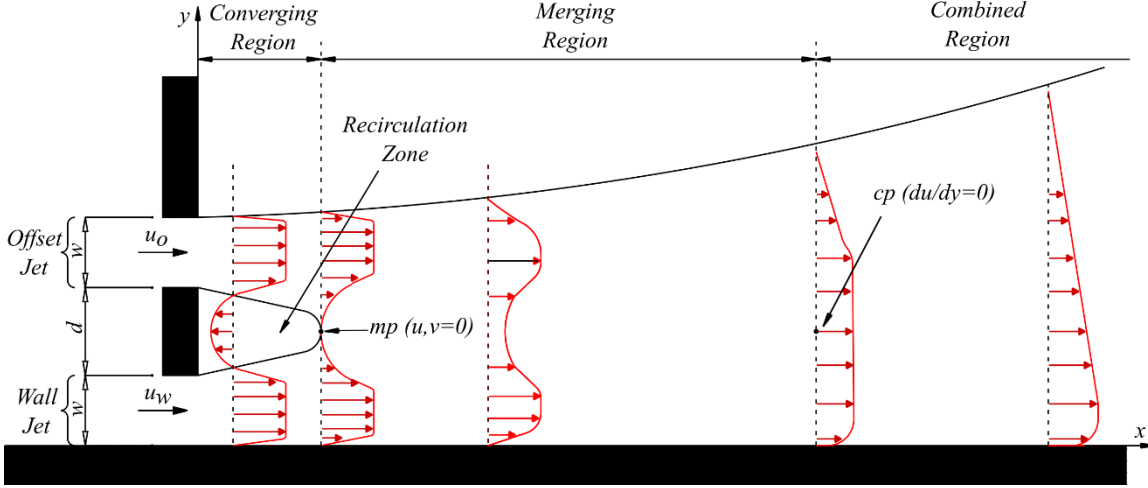


Figure 1.4: Dual jet flow schematic.

The first study which examined the flow behaviour of a combined wall and offset jet was conducted by Wang and Tan [19] for a single Reynolds number and jet offset ratio, *i.e.*, $Re = 10,000$ and $OR = 1$. Using particle image velocimetry (PIV) techniques, Wang and Tan [19] reported on the unstable nature of dual jet flows, where the occurrence of periodic vortex shedding was observed in the near jet region, similar to a bluff body in crossflow. The work of Wang and Tan [19] sparked the interest in dual jets flows as an important area of research, and since then multiple studies have been published which investigate the dual jet flow field through numerical means, such as Kumar and Das [1] and Mondal *et al.* [20]. In each case, the numerical flow data was validated using that of Wang and Tan [19], typically a comparison of the resulting velocity profile in the self-similar region, however, it was noted that most numerical studies predicted self-similarity would occur much further downstream compared to Wang and Tan [19], *i.e.*, $X \geq 30$ compared to $X \geq 15$, where $X = x/w$. This slower rate of flow development noticeably indexed the respective positions of the most prominent flow features further downstream in the numerical dual jet flow field, where studies such as Mondal *et al.* [21] obtained a combined point at $X_{cp} = 16.72$ for the same set of flow parameters as Wang and Tan [19], who themselves obtained $X_{cp} = 6.4$.

Given the obvious discrepancies obtained when simulating the dual jet flow field for a set of jet parameters for which credible experimental data was readily available, it incites obvious doubt in any published trends obtained through varying certain jet parameters, most notably the jet Reynolds number ($Re = u_{w/o}w/\nu$), the offset ratio ($OR = d/w$ as per Figure 1.4), or the ratio of the jet velocities ($V_r = u_w/u_o$). This doubt is further validated through the stark differences in the trends observed, such as the position of the merge point with varying Re , where the value of X_{mp} was reported to increase, decrease and remain constant with increasing Re by Mondal *et al.* [22], Kumar *et al.* [23] and Singh and Dewan [24], respectively. A similar level of uncertainty surrounds the effect of OR , where both Hnaïen *et al.* [25] and Assoudi *et al.* [26] noted a continuous increase

in X_{cp} with increasing OR , which implied a worsening of the mixing characteristics of the dual jet with raising the height of the offset jet. Conversely, Kumar [27] observed initial decline of X_{cp} until $OR = 5$, beyond which X_{cp} subsequently increased, which suggested that the mixing between the jets is at a maximum for $OR = 5$. With regard to varying V_r , only a single study has reported on the change in the locations of the key dual jet flow features with altering V_r , *i.e.*, Hnaïen *et al.* [25], meaning the published findings are yet to be corroborated.

The heat transfer characteristics induced in the solid wall boundary when subject to a dual jet flow still remain relatively unknown, despite the multiple numerical studies which focussed on acquiring this knowledge. The first study to investigate the heat transfer characteristics of a dual jet flow through numerical means was conducted by Vishnuvardhanarao and Das [28], who analysed the effect of the Reynolds number and the velocity ratio on the local Nusselt number Nu_x profile in the solid wall. In the absence of experimental heat transfer data, Vishnuvardhanarao and Das [28] validated their numerical model based on flow-only data using that of Wang and Tan [19]. In subsequent studies such as Farooq and Das [29] and Mondal *et al.* [22], experimental heat transfer data relating to single offset jet flows was used to validate their numerical findings, however, the flow field associated with a single offset jet is distinctly different from that of a dual jet, meaning their modelling requirements may differ greatly. The lack of appropriate validation is clearly evident across all the published results, where the Nu_x profiles produced by multiple studies vary greatly for the same set of jet parameters. For example, Vishnuvardhanarao and Das [28] noted a highly fluctuating Nu_x profile for the same set of jet parameters as Wang and Tan [19], while Mondal *et al.* [22] reported the occurrence of a single dominant peak and Singh and Dewan [24] obtained a smooth profile that decayed steadily with streamwise distance. In each case, little information is provided on the factors which drove the reported Nu_x profile, which also casts doubt over any trends observed with respect to Re , OR , and V_r by these or any other published studies. The heat transfer characteristics of a dual jet are therefore yet to be explored in depth, where the acquisition of credible trends is required if intending to control or optimise the dual jet cooling characteristics towards a specific application.

1.2. Research Aims and Objectives

Based on these observations, the present investigation is concerned with characterising the fluid flow and heat transfer behaviour associated with dual jet flows through experimental means with respect to the Reynolds number, the offset ratio, and the velocity ratio. It is subsequently the intention to develop an improved numerical model that can be successfully validated with respect to the experimental flow and heat transfer data, which can be used in further studies to investigate dual jet flows over an expanded range of parameters, far beyond the realms of experimentation.

The primary focus of this investigation is the acquisition of a wide database of experimental data relating to the heat transfer aspects of dual jet flow, where the provision of verified, repeatable experimental heat transfer measurements currently represents a significant gap in the published literature. An extensive study is conducted which focusses on the induced heat transfer characteristics of a dual jet flow when subject to a uniform heat flux in the bounding wall for a moderate Reynolds range of $5500 \leq Re \leq 12000$. This range was chosen as it successfully encapsulates the experimental parameters of Wang and Tan [19], while also being typical of that used for electronics cooling applications and realistically achievable in the laboratory environment. The offset ratio is varied from $1 \leq OR \leq 7$ to remain consistent with the values generally encountered throughout

the published literature, which allows for a direct comparison with the trends acquired. Finally, the velocity ratio is varied from $0.5 \leq V_r \leq 2$ as this is the maximum allowable range based on the aforementioned Reynolds number limitations. In doing so, it is hoped to identify a characteristic Nu_x profile in the solid wall boundary, where the influence of Re , OR , and V_r can be defined with a reasonable level of confidence, which can subsequently be used as a credible means of numerical model validation and introduce consistency across all published dual jet studies. This research has led to one journal publication to date, where Murphy *et al.* [30] focusses on the heat transfer characteristics associated with a dual jet for varying Re , where $OR = V_r = 1$ and $5500 \leq Re \leq 12000$. The results of this heat transfer study also feature in two conference proceedings, where Murphy *et al.* [31] and Murphy *et al.* [32] examine the effect of varying Re and V_r , respectively, for a dual jet with $OR = 3$.

This study subsequently aims to affiliate the derived heat transfer trends with specific flow features inside the dual jet flow field. This is achieved using a particle image velocimetry (PIV) investigation, where instantaneous dual jet velocity fields are acquired at a frequency of 2000 Hz, from which the average flow properties can be attained. This allows for both the time-resolved and time-averaged analysis of the dual jet flow field with respect to the same Re and V_r range as the heat transfer study, however, the offset ratio range is limited to $1 \leq OR \leq 3$ due to the reduced field-of-view of the high-speed camera. In doing so, the effect of Re , OR , and V_r on the relative positions of the merge point and combined points can be obtained and subsequently related to the observed changes in the heat transfer behaviour.

The final objective of this study is to conduct a numerical investigation as a means to corroborate that the experimental trends can successfully be simulated to a reasonable level of accuracy. This is conducted for a subset of the Re and OR values, for which careful consideration is given towards the numerical inputs to achieve a successfully validated model. The intention is to improve on the methodology for dual jet modelling compared to that currently available in the literature, leading to simulation results which can be considered repeatable, accurate and consistent with physical reality. This model can subsequently be used and modified for further studies to investigate dual jet flow over an expanded range of Re , OR , and V_r values beyond the scope of the experimental investigation, as well as adjusting alternative parameters, such as the working fluid, the nozzle dimensions/shape, and the jet temperature ratio.

Following the brief overview of dual jet flow and its various industrial applications in Chapter 1, the remainder of this thesis is structured as follows; Chapter 2 provides the basic jet flow theory required for the present study, which includes free jets, plane parallel jet, wall jets and singles offset jets, as well as an in-depth discussion on the dual jet flow field and all the relevant terminology required. Chapter 3 subsequently explores the topic of wall-bounded dual jets throughout the published literature as a means to discuss the key findings to date, highlight the existing discrepancies and, hence, identify the gap in the literature which the present investigation aims to address. The setup and processes required for the experimental investigation are discussed in Chapter 4, which includes the experimental apparatus, methodology, data reduction, calibration of instrumentation and uncertainty analysis. The output from the experimental facility is validated in Chapter 5, following which the results of the experimental dual jet study are discussed in Chapter 6, where results of both the heat transfer study and PIV investigation are discussed in tandem. Chapter 7 introduces the numerical aspect of the present study, where the present approach towards simulating the dual jet flow is discussed with reference to both the relevant theory and past dual jet studies. The effectiveness of the dual jet numerical model

with respect to the experimental data is hence discussed in Chapter 8. Finally, Chapter 9 presents the main conclusions from the present investigation and discusses the prospect of future work in the field of dual jet flow.

Chapter 2

Dual Jet Flow

The purpose of this chapter is to provide an overview of the dual jet flow structure and introduce the relevant terminology. While the purpose of the present study is to investigate the flow behaviour and heat transfer characteristics associated with a dual jet, a general understanding of other jet flow configurations is required to achieve this. This chapter therefore first outlines the important features relating to both free jets and various wall-bounded jet flow configurations, and subsequently incorporates the relevant findings into the dual jet discussion to provide context for certain flow features. For the sake of brevity, the discussion is limited to fully turbulent jet flows only, as their use is considerably more prevalent across industrial applications. This is due to the enhanced mixing and superior heat transfer that can be acquired compared to a jet flow within the laminar or transitional flow regime. For this reason, the range of jet Reynolds numbers for the present investigation were chosen such that they existed mostly within the turbulent flow regime, *i.e.*, $5500 \leq Re \leq 12000$, where the critical Reynolds number for a free jet has been previously found to exist in the region $2000 \leq Re_{cr} \leq 3000$, depending on whether a planar or round jet was considered [33-35]. This Reynolds number range is also consistent with many previously published dual jet studies, the majority of which focussed on the range $10000 \leq Re \leq 20000$, therefore allowing for a direct comparison of the results acquired with a wide number of studies. Further information of the range of dual jet parameters examined in previous dual jet studies is summarised in Table 3.1.

2.1. Turbulent Free Jets

The topic of free jets has been reported on extensively throughout the published literature, with notable studies such as Miller and Comings [36], Wygnanski and Fiedler [37], Westerweel *et al.* [38] and Gutmark and Wygnanski [39]. A free jet is formed when a stream of fluid is discharged from a nozzle into quiescent surroundings, where no solid boundaries exist downstream of the jet nozzle exit to influence the jet development [40]. A distinction does exist however between free jets which emerge into an ambient fluid of similar density, termed a submerged jet, or that of lesser density, *i.e.* a free surface jet [41]. As this investigation is only concerned with the properties associated with a submerged jet, the use of the term ‘jet’ for the remainder of this literature review refers exclusively to submerged jets.

The category of turbulent free jets can be further subdivided into axisymmetric jets, rectangular jets or plane jets, depending on the nozzle shape at the jet exit. In the case where a jet is issued from a round nozzle, the jet is considered round or axisymmetric, while a rectangular jet is acquired through the use of a rectangular nozzle,

where it has been shown that a rectangular jet can possess much higher rates of entrainment and enhanced mixing when compared to an axisymmetric jet due to the presence of higher order instability modes [42, 43]. In the case where the length (z –dimension) of the rectangular nozzle compared to its width (y –dimension), *i.e.*, the aspect ratio, is sufficiently large, 3D effects arising from the presence of the shorter edge become negligible, resulting in a flow field that is statistically two-dimensional in the streamwise x - and transverse y -directions, but independent of variations along the spanwise z -direction [44]. The resulting jet flow is often referred to as a slot jet when analysed as a three-dimensional flow structure, however, when idealised as a two-dimensional flow field along the xy –plane to simplify the flow analysis, it is termed a plane jet. The remainder of this discussion therefore focuses on the free plane jet, as the flow features are more closely aligned with those of the two-dimensional dual jet flow field that is the primary focus of this investigation.

2.1.1. Single Plane Jet

The time-averaged flow structures associated with a free single plane jet issuing from a slot nozzle of width $2b_0$ can be seen in Figure 2.1 [45]. As the fluid issues from the nozzle exit, a thin region of high vorticity, or shear layer, is formed between the high velocity jet fluid and the quiescent surroundings [40]. This arises as result of Kelvin-Helmholtz instabilities in the fluid flow, where Kelvin-Helmholtz instabilities are defined as flow instabilities which arise as a result of a velocity or density difference across a finite thickness within a fluid domain, *i.e.*, a discontinuity in the fluid velocity or density field [46, 47]. This leads to the formation of roll-up vortices at the nozzle exit, which, when convected downstream, can pair-up with like-signed regions of vorticity to form much larger vortical structures [48, 49]. In doing so, surrounding ambient fluid is drawn into the jet flow which acts to slow the average jet velocity to maintain the momentum flux as the mass flow rate of the jet increases [40, 50]. This process is referred to as entrainment and it is responsible for the spreading of the jet in the vertical dimension and its eventual demise further downstream.

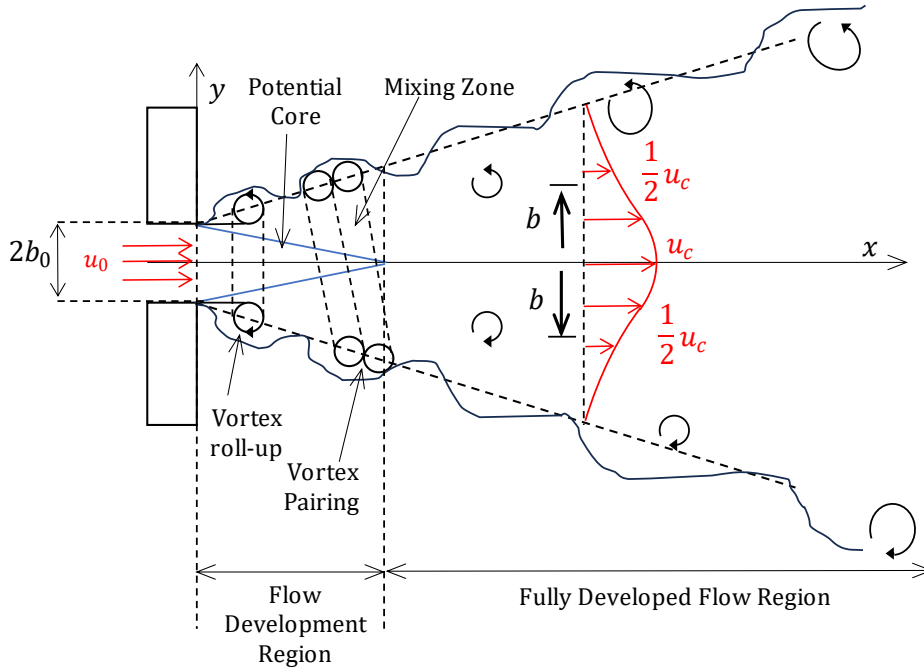


Figure 2.1 General flow structure of a free plane jet [40]

This leads to the formation of two distinct downstream regions in the jet flow field, *i.e.*, the flow development region and fully developed flow region [45]. The flow development region, which occurs closest to the nozzle

exit, is characterised by the presence of the potential core, a region where the initial jet exit velocity u_0 is conserved, assuming the velocity profile is uniform across the jet inlet. At the nozzle exit, the potential core extends across the entire jet width, however, as the thickness of the shear layer grows with streamwise distance, the width of the potential core decreases until its depletion at some point downstream. The potential core length is hence defined as the streamwise distance along the jet centreline from the jet exit to the point at which the local fluid velocity is $u = 0.95u_0$, which is expected to be at approximately $12b_0$ for a plane jet [45, 51].

In the fully developed region, the velocity profile describing the variation of the streamwise velocity u along the transverse y dimension acquires a distinct geometrical shape that remains relatively consistent at each streamwise position. That is, at each downstream location, the maximum streamwise velocity u_c exists at the jet centreline, *i.e.*, $y = 0$ m, and decays continuously towards 0 m/s with increasing y [45]. Due to the effects of entrainment with the ambient fluid, the jet flow increasingly spreads along the vertical direction with increasing streamwise distance, which subsequently reduces the value of u_c with increasing x . As the actual jet width is quite difficult to identify experimentally, the streamwise jet development or jet spreading is generally monitored in terms of the jet half-width, b , defined at each x – location as the vertical distance from the jet centreline to a point in the flow at which the local streamwise velocity is half of the jet centreline velocity, *i.e.*, $u_b = u_c/2$ [40, 45].

When analysing the development of the local velocity profile, it is common to non-dimensionalise the velocity values with respect to the centreline velocity u_m , while the transverse dimension is generally scaled using the jet half-width. In plotting u/u_c against y/b , it has been noted the velocity distributions begin to collapse along a common curve after a certain streamwise distance, which is often referred to as a jet flow reaching self-similarity [40, 45, 52]. In this region, the non-dimensionalised velocity profile does not appear to vary with streamwise distance, meaning the flow behaviour is considerably more predictable in this region and the mean flow statistics are somewhat independent of the jet conditions at the nozzle exit [52, 53]. After some streamwise distance, the self-similarity associated with a jet flow will begin to break down as u_c is slowed significantly and the jet reaches its eventual end.

2.1.2. Plane Parallel Jets

While single free jets have an abundance of uses in industry, it is also common to encounter multiple plane jets in close proximity due to the enhanced mixing or control that may bring to a specific application, for example fuel injection systems, thrust augmenting ejectors for vertical and/or short take-off and landing (V/STOL) aircraft, and dilution of thermal pollution [3, 54, 55].

A plane parallel jet is formed when two free slot jets are ejected from the same vertical plane and the initial inlet direction of both jets are parallel to one another. The flow schematic can be viewed in Figure 2.2, which was first investigated experimentally in work carried out by Miller and Comings [56]. In this schematic, two slot jets of width w are separated by a distance d , or non-dimensionalised separation ratio of d/w , where the x – axis acts as a plane of symmetry. Upon ejection from their respective nozzle exits, a shear layer forms along the inner and outer edge of each jet, where ‘inner’ refers to the side closest to the opposing jet. The mutual entrainment of both jets with the central column of fluid leads to the creation of a low-pressure zone, or recirculation zone, between them, which causes the jets to deflect towards each other and eventually merge at some downstream location. The location at which the inner shear layers of both jets first meet is termed the merge point, *mp*, at which point the mean velocity flow is 0 m/s and the pressure reaches a local maximum

value [57, 58]. The downstream region extending from the jet exit plane to the merge point, encapsulating the full recirculation zone, is termed the converging region, the length of which can readily be controlled through adjusting the jet separation d . In doing so, it has been observed that a specific range of separation values exist which induce an unsteady flow regime inside the recirculation zone, where vortices are shed in a periodic manner similar to flow past a bluff body. Outside of this range of jet separation values, two stable counter-rotating vortices are found to exist within the recirculation zone [3].

Downstream of the merge point, the interaction between the jets begins, continuing over the entire length of the merging region [3]. Within the merging region, the velocity along the centreline between the two jets, u_c , increases steadily towards a maximum value u_{max} at the combined point cp , at which point the jets are considered to be fully combined and the local deflection in the local velocity profile no longer exists [57]. Downstream of the combined point, within the combined region, the two jets behave as an equivalent single free jet [59]. In this region, the combined jet continues to spread in a linear fashion with downstream distance, where the rate at which this occurs has been shown to increase with increasing jet separation. Eventually, the combined jet flow reaches a self-similar state. While this flow structure is typical of the plane parallel jet behaviour, research has shown that the extent of the jet mixing or jet deflection can be further controlled through altering the ratio of the jet inlet velocities or the respective sizes of the nozzle inlets, which can be advantageous when tuning to a specific application. For example, Fujisawa *et al.* [60] found that through adjusting the ratio of the jet inlet velocities, the extent of the deflection of the weaker jet is increased, which biases the transverse positions of both mp and cp towards the stronger jet and away from the jet centreline. More recently, Nassira and Amina [61] noted an enhanced mixing between jets of higher temperatures due to increased buoyancy effects, whereas cooler jets were notably less prone to merge.

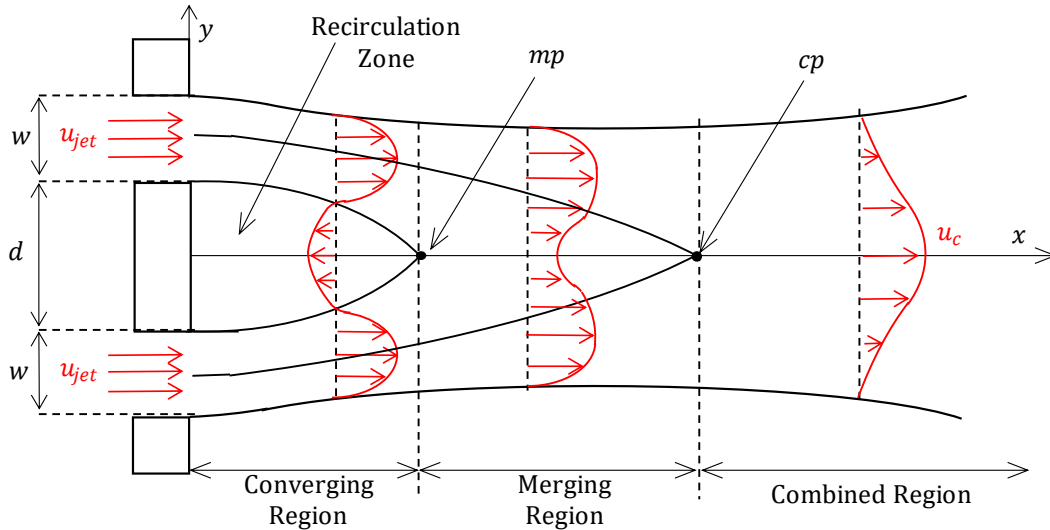


Figure 2.2: Fluid flow schematic for plane parallel jets.

2.2.Wall-Bounded Turbulent Jets

The use of turbulent jets as an effective means of cooling generally involves the impingement of the jet flow onto a heated solid surface, thus introducing a wall-bounded aspect to turbulent jet flow. However, the use of wall bounded jet flows is not exclusive to heat transfer applications. The influence of the wall on the jet flow structure depends on the jet proximity to the wall, as well as the direction of impingement. In the case of direct impingement, the jet is classified as an impinging jet which possesses a distinct flow structure. Impinging jets

have been reported on extensively throughout the published literature in studies such as Lytle and Webb [62], Donaldson and Snedeker [63] and Jambunathan *et al.* [64]. The present investigation is concerned, however, with jets which flow parallel to the solid surface, either directly tangential, *i.e.*, a wall jet, or offset from the wall boundary by some distance, *i.e.*, an offset jet. The following sections thus discuss the flow and heat transfer characteristics associated with wall and offset jets.

2.2.1. Wall Jets

A wall jet is formed when a jet, usually a slot jet, is discharged tangentially to a solid wall surface. The flow characteristics of a wall jet have previously been a popular area of research throughout the published literature due to their wide range of industrial applications, for example Eriksson [65], Craft and Launder [66], Nizou [67] and AbdulNour *et al.* [68]. The basic flow structure of a wall jet is shown in Figure 2.3, where the downstream fluid flow development is distinctly different to that of a free jet. The presence of the wall dampens the growth of large-scale eddies, which thus reduces the rate of entrainment and the downstream decay of maximum velocity compared to the free jet [69].

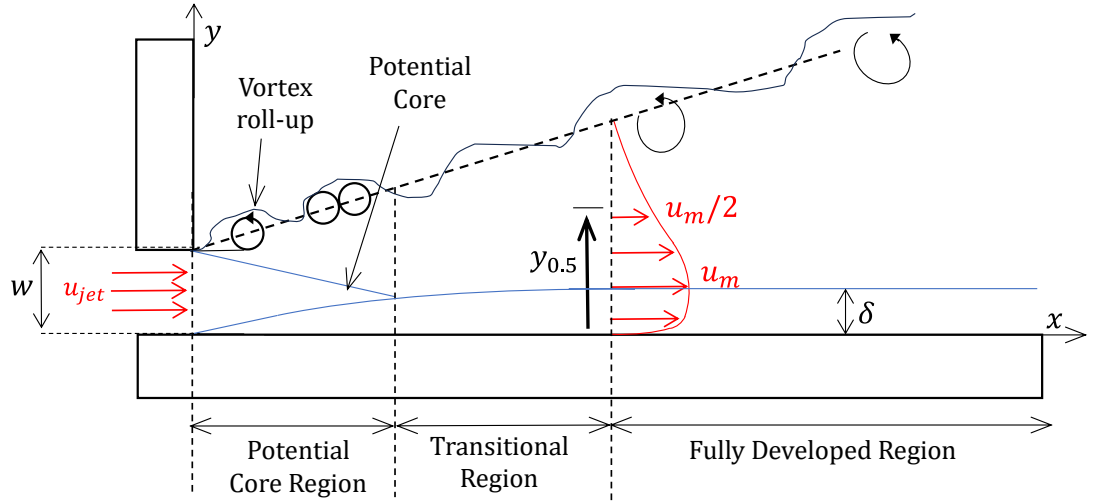


Figure 2.3: Fluid flow schematic of a plane wall jet

Within the wall jet flow structure exists two separate shear layers: the inner shear layer and the outer shear layer. The inner shear layer corresponds to the development of the viscous boundary layer along the solid wall boundary, where the no-slip condition ensures the fluid particles directly adjacent to the surface remain stationary. The boundary layer thickness δ_{bl} is hence defined as the vertical distance from the solid wall to the point in the flow at which $u = 0.99u_m$, where u_m is the local maximum streamwise velocity [70]. Conversely, the outer shear layer is free to develop in a similar manner to a free jet, whereby the existence of a velocity difference between the flowing jet and the surrounding quiescent fluid leads to the formation of large-scale vortices due to the occurrence of Kelvin-Helmholtz instabilities. The development of the outer shear layer is monitored by means of the jet half-width, $y_{0.5}$, which corresponds to the vertical distance from the solid wall to the point in the flow at which the local streamwise velocity is half the local maximum velocity, *i.e.*, $u = u_m/2$ [45].

The fluid structure of a wall jet is divided into three downstream regions. Similar to a free jet, the length of the region closest to the jet exit is defined by the existence of the potential core, *i.e.*, the *potential core region*. Beyond this exists the *transitional region*, where, due to the prior demise of the potential core, the inner and outer shear layers are now free to interact. Within this region, the wall jet transitions from laminar to turbulent

flow as the large-scale vortical structures within the outer shear layer pair with the smaller scale vortices in the inner shear layer, which are of opposing sign [70, 71]. The intensity of the vortical structures within the inner shear layer has been shown to grow in strength with downstream distance, such that their associated strength is much greater than that of the outer shear layer, which enables the wall jet to cling to the surface over the total downstream distance [70]. The wall jet finally reaches a fully turbulent state within the *fully developed region*, at which point the jet flow achieves self-similarity when u/u_m is plotted against $y/y_{0.5}$ at consecutive streamwise locations, where examples of a typical self-similar velocity profiles for a single wall jet are presented in Figure 2.4 [69].

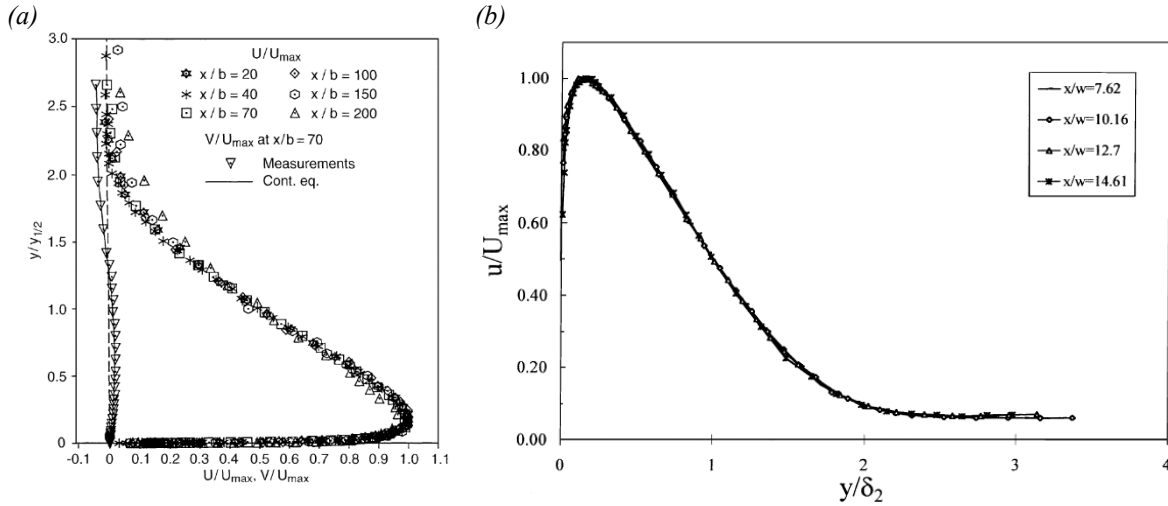


Figure 2.4: Typical streamwise velocity profiles obtained within the self-similar region of a planar wall jet, as reported by (a) Eriksson *et al.* [72] and (b) AbdulNour *et al.* [68].

The characteristic heat transfer profile associated with a wall jet flow is presented in Figure 2.5(a) and Figure 2.5(b) in terms of the heat transfer coefficient h , as obtained by AbdulNour *et al.* [68] and Akfırat [73], respectively. In each case, h immediately decreases upon exiting the jet nozzle towards a minimum value, the position of which is relatively similar for both studies. This decline has previously been attributed to the growth of the wall boundary and degradation of the potential core, where the local minimum point coincides with the potential core length. Beyond this point, the interaction between the outer shear layer and the wall boundary layer induces a sudden transition to turbulence in the wall boundary layer, which subsequently causes a rise in the heat transfer coefficient along this region [73].

The growth of h due to the increasingly turbulent boundary layer reaches a limiting point, however, where h appears to reach a maximum value and subsequently declines over the remaining downstream distance. In each case, this was attributed to the continuous thickening of the wall boundary layer, as well as the slowing of the overall mean jet velocity as entrainment effects began to dominate [68]. In similar work carried out by Mabuchi and Kumada [74], it was noted that the respective positions of the local minimum and maximum values in the heat transfer coefficient profile remain relatively unchanged for varying Reynolds number, however, an enhancement of the rate of heat transfer at each downstream location was observed for increasing Re , which was consistent with that presented by Akfırat [73].

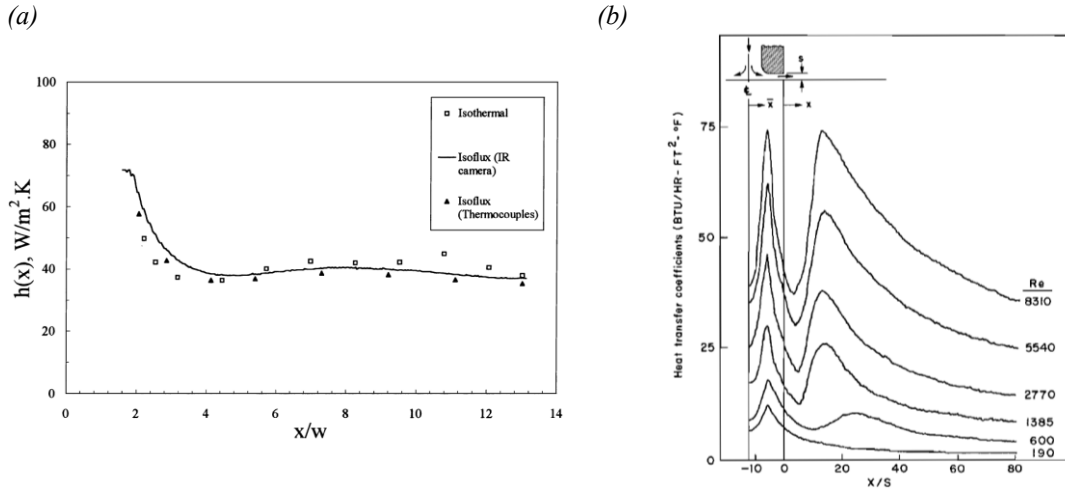


Figure 2.5: Variation in the local heat transfer coefficient in the solid boundary when subject to a plane wall jet flow, as provided by (a) AbdulNour et al. [68] and (b) Akfirat [73]

2.2.2. Offset Jets

An offset jet is created when a slot jet is discharged from a vertical plane into the ambient fluid near a solid wall, where the wall is parallel to the jet inlet direction [75]. The fluid flow schematic for an offset jet can be seen in Figure 2.6, which is often considered similar to that of a backward facing step [76]. In the schematic, the jet nozzle is offset from the solid wall by a distance H , which is often non-dimensionalised by means of the offset ratio, *i.e.*, $OR = H/w$. Similar to previous jet setups, the jet flow in the downstream direction is divided into three distinct regions based on the flow behaviour: the recirculation region, the impingement region, and the wall region [77].

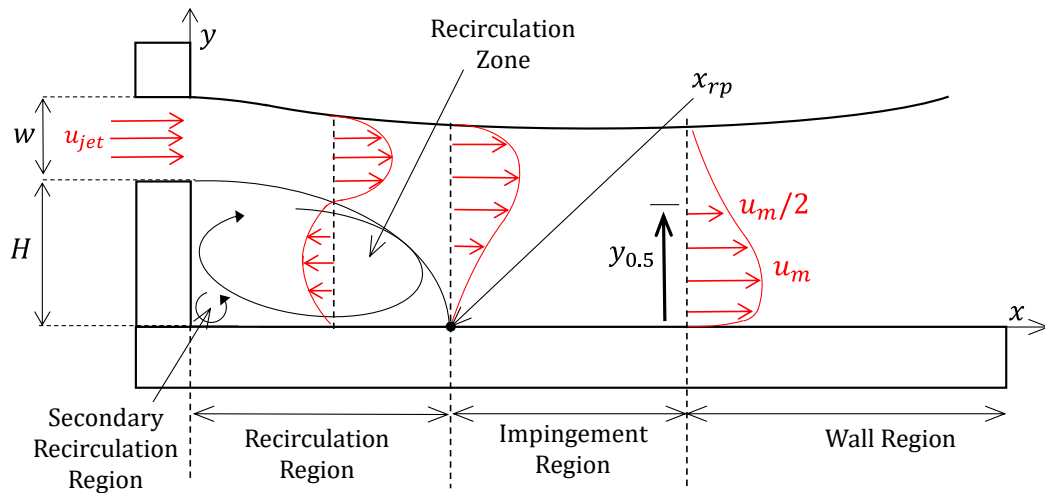


Figure 2.6: Fluid flow schematic for an offset jet.

The recirculation region occurs closest to the nozzle exit and is characterised by the presence of the recirculation zone. Just downstream of the nozzle exit, the jet flow is momentum dominated, meaning the jet develops in a similar manner to that of a free jet [78]. However, the presence of the wall boundary induces unequal rates of entrainment above and below the jet, leading to the creation of a low-pressure zone directly below the jet exit, termed the recirculation zone. This low-pressure region causes the jet to deflect towards the wall and eventually impinge on the boundary at some downstream location, *i.e.*, the Coanda effect. At the point where the jet flow attaches to the wall, *i.e.*, the reattachment point x_{rp} , part of the inner shear layer is deflected

upstream due to the presence of an adverse pressure gradient, thus contributing to the creation of a large vortical structure within the recirculation zone [76, 77]. While the recirculation zone is primarily dominated by the presence of a single recirculation bubble, a secondary recirculation region has been found to exist in close proximity to where the solid wall meets the jet exit plane [79]. The offset ratio has been shown to heavily influence the extent of the recirculation zone, whereby an increase in the offset ratio increases the length of the recirculation zone and the reattachment point is pushed further downstream [80]. Beyond the reattachment point exists the impingement region, within which the jet centreline continues to deflect towards the wall. Within this region, the growth of a boundary layer at the wall surface, and subsequent interactions between the boundary layer and the outer shear layer, enable the jet to cling to the surface [76]. As the jet flow progresses, it has been found that at some downstream location, *i.e.*, within the wall region, the flow resembles that of an equivalent wall jet and will continue to develop in a similar manner.

The characteristic local Nusselt number Nu_x profile associated with a single offset jet flow is presented in both Figure 2.7(a) and Figure 2.7(b), which correspond to that reported by Kim *et al.* [79] and Song *et al.* [81], respectively. In each case, the sharp decline in Nu_x is observed at the nozzle exit due to the presence of the secondary recirculation zone. Beyond this point, the dominating recirculation bubble induces a steady rise in Nu_x towards a local maximum value, the position of which coincides with the reattachment point [79]. Therefore, as the flow continues to develop in a similar manner to the single wall jet with further downstream distance, Nu_x decreases accordingly, where the absence of a secondary peak in the Nu_x profile along this region implies that the boundary layer is fully turbulent as a result of the jet impingement, where a natural laminar-to-turbulent transition in the boundary layer does not occur. The effect of OR on the Nu_x profile is clear in each case, where the downstream movement of x_{rp} with increasing OR consequentially increases the streamwise position of the maximum Nu_x value. In doing so, the value of the maximum Nu_x value decreases incrementally as the offset jet velocity at the impingement location becomes increasingly less. Downstream of x_{rp} , a close alignment is observed between the respective Nu_x profiles at each OR as the mass flow rate remains consistent throughout.

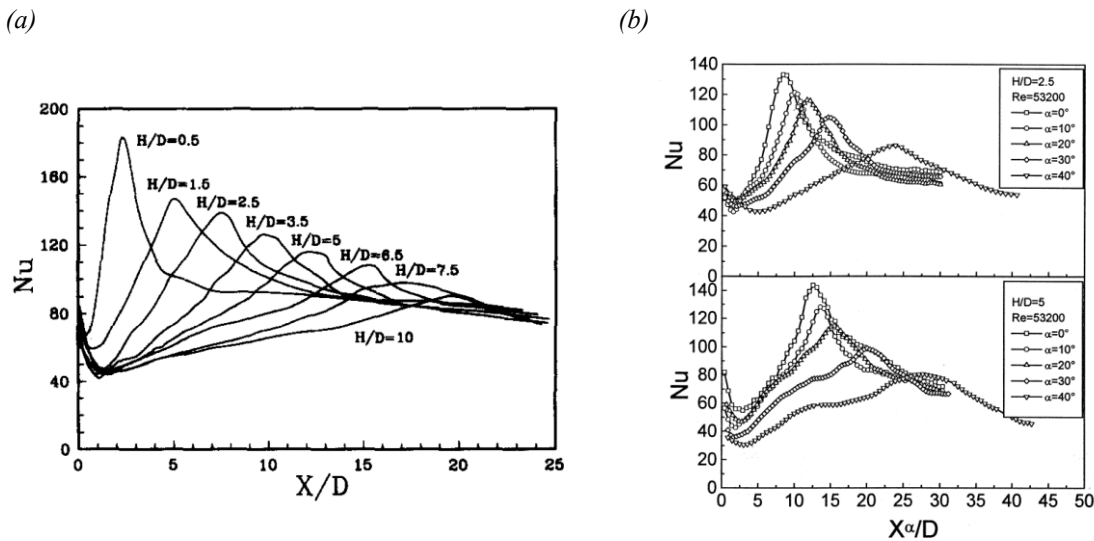


Figure 2.7: Variation in the local Nusselt number in the solid boundary when subject to a plane offset jet flow, as provided by (a) Kim *et al.* [79] ($Re = 39000$) and Song *et al.* [81].

2.3. Dual Jet Flow

The term ‘dual jet’ throughout the published literature generally refers to the combination of a wall jet and an offset jet, where the nozzle exits for both jets are aligned in the same vertical plane. The restriction of the fluid flow in such a manner inhibits the jet development, which results in a highly complex flow structure quite different to a conventional plane parallel jet setup. The use of dual jets is common for many industrial applications, such as wastewater evacuation processes and heat exchangers. While this terminology is somewhat vague considering the distinct wall-bounded nature of this particular jet flow configuration, where the label of a ‘dual jet’ could technically be applied to any double jet flow configuration, it is used liberally throughout the published literature to refer specifically to the wall and offset jet combination. For that reason, this terminology is adopted for the present investigation, where the use of the term ‘dual jet’ for the remainder of the study refers directly to the wall-bounded wall jet and offset jet combination.

Much of the information presently available regarding dual jets has been derived from a limited number of numerical studies, while only a single experimental study has been conducted to date, a complete summary of which is presented in Table 3.1. The generally accepted time-averaged two-dimensional dual jet flow structure can be seen in Figure 2.8, which was originally devised by Wang and Tan [19] based on the published work of Nasr and Lai [82] for plane parallel jets. The schematic shows two rectangular slot jets, a wall jet and an offset jet, emerging into the quiescent ambient fluid from the same vertical plane, with initial parallel streamwise mean velocities of u_w and u_o at their respective nozzle exits and a common working fluid. The velocity ratio is hence defined as the ratio of the jet mean streamwise velocities, such that $V_r = u_w/u_o$.

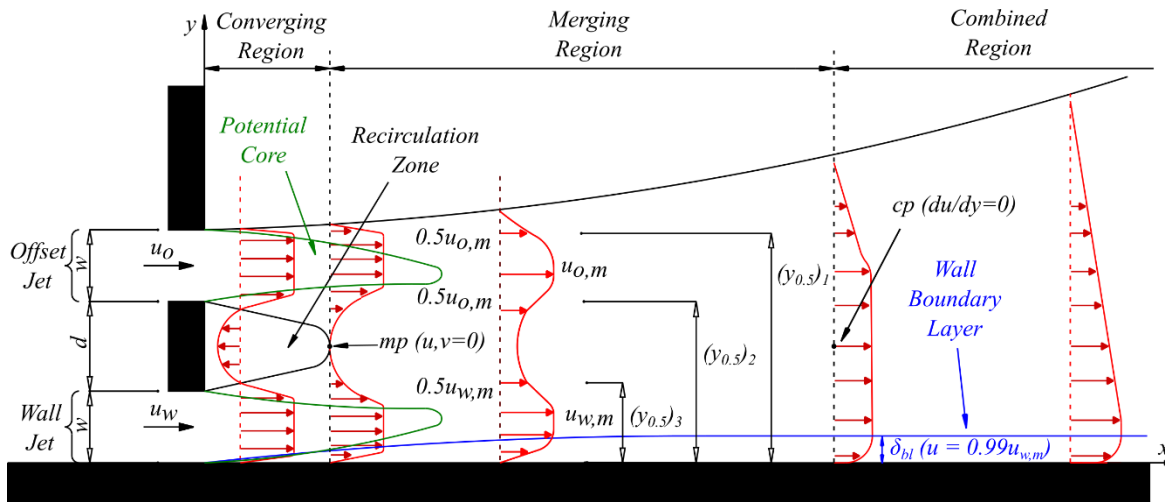


Figure 2.8: Flow structure of a dual jet, adapted from Wang and Tan [19].

Both jets are initially issued in the positive x -direction, where the origin of the coordinate system is located at the intersection between the jet exit plane and the bottom wall. In Figure 2.8, the characteristic jet width, denoted by w , refers to the jet dimension in the y -direction, and is identical for both jets in this case. All locations within the flow domain are generally expressed in a non-dimensional form with respect to the jet width, such that $X = x/w$ and $Y = y/w$. The Reynolds number of the flow for each jet in the dual jet configuration is also based on the jet width, such that

$$Re_{jet} = \frac{\rho u_{jet} w}{\mu} = \frac{u_{jet} w}{\nu} \quad (2.1)$$

where u_{jet} is the corresponding jet velocity at the nozzle exit, either u_w or u_o , and ν is the kinematic viscosity of the working fluid. In the specific case where $u_w = u_o$, the specified Reynolds number for a dual jet flow Re represents that calculated based on just one jet, as opposed to the summation of both Reynolds number contributions, such that $Re = Re_w = Re_o$. The two-dimensional plane at $z = 0$ is assumed to lie at the midpoint of the jet nozzle length l , *i.e.*, the jet dimension in the z -direction (not shown the schematic) spanning the region $-l/2 \leq z \leq l/2$. The jet aspect ratio, AR , is hence defined as $AR = l/w$, which must be sufficiently large to mitigate edge effects along the jet centre plane, where Nasr and Lai [76] recommends $AR \geq 30$. To verify this recommendation with respect to a wall-bounded dual jet flow, Wang and Tan [19] used particle image velocimetry (PIV) to obtain velocity measurements along the centre plane of a dual jet of width $w = 10 \text{ mm}$ and aspect ratio 30. Additional velocity measurements were acquired along the planes located at $z = \pm 10 \text{ mm}$, *i.e.*, $z = \pm w$, thus revealing almost identical flow behaviour along $z = -w, 0$ and $+w$. This finding implied good 2D character along the centre plane of the jet, therefore verifying the use of an aspect ratio of 30 to ensure sufficient planarity within the dual jet flow field.

The wall jet and offset jet are separated by an offset distance of d , where the jet separation is more commonly described in the literature in terms of a dimensionless offset ratio value for comparison purposes, such that $OR = d/w$. However, it should be noted that discrepancies exist in the literature regarding the definition of the offset ratio. A common definition of the offset ratio for dual jets is $OR = H/w$, which was first introduced by Kumar [83] and subsequently adopted by Refs. [10, 25, 27, 84-88]. This definition was created as means of comparison between a dual jet and a single offset jet for the purpose of numerical model verification, where H refers to the vertical distance between the bounding wall and the lower edge of the offset jet nozzle. Alternatively, Rathore *et al.* [6] defines H as the vertical distance between the bounding wall and the offset jet centreline at the nozzle exit. Finally, Kumar and Das [1] and Zhao and Sun [89] each use a definition for the offset ratio of $OR = D/w$, where D is the distance between the jet centrelines at the nozzle exit. However, for this investigation it was decided to adopt the original offset ratio definition, such that $OR = d/w$, as this is consistent with the majority of dual jet studies, including Wang and Tan [19], *i.e.*, the first and only experimental dual jet study, as well as Vishnuvardhanarao and Das [28], Li *et al.* [2], Mondal *et al.* [20], Farooq and Das [29], and Assoudi *et al.* [26].

Upon entering the fluid domain, both jets possess a potential core region within which the associated jet exit centreline velocity is maintained for some distance downstream, eventually degrading due to ambient fluid entrainment. Due to the specific configuration of a dual jet, four shear layers are created within the flow domain when the jet flow is issued into the ambient conditions, three of which are free shear layers that arise as a result of Kelvin-Helmholtz instabilities. The development of the three free shear layers is visualised using the jet half-width locations, $(y_{0.5})$, which correspond to the vertical distances from the solid wall to the points in the flow where the fluid velocity is half the local maximum velocity, *i.e.*, where $u = u_m/2$. Layers 1 and 2 respectively refer to the outer and inner shear layers of the offset jet, denoted by the jet half-widths $(y_{0.5})_1$ and $(y_{0.5})_2$, calculated based on the maximum offset jet velocity $u_{o,m}$, while Layer 3 is the inner shear layer of the wall jet, denoted by the jet half-width $(y_{0.5})_3$ calculated using the maximum wall jet velocity $u_{w,m}$. The final shear layer corresponds to the boundary layer that develops along the solid boundary, the thickness (δ_{bl}) of which is defined at each downstream location as the vertical distance from the solid boundary to the point in the flow where the local streamwise velocity is 99% of the local maximum velocity, *i.e.*, where $u = 0.99u_m$ [19, 22].

The dual jet flow development can be divided into three distinct regions; the converging region, the merging region, and the combined region. The *converging region* occurs closest to the nozzle exit, where the mutual entrainment of both jets with the central column of fluid leads to the development of a sub-atmospheric pressure zone directly downstream of the jet exit plane, assuming both jets are issued initially into atmospheric pressure conditions [20]. This thus causes the jets to deflect towards each other, before both jets eventually meet at a stagnation point in the flow, *i.e.*, the merge point mp , the location of which is described by the coordinates $(X_{mp}, Y_{mp}) = (x_{mp}/w, y_{mp}/w)$. As per Figure 2.8, the low pressure region created between the jets is termed the recirculation zone, inside which flow reversal and complex vortical structures exist [1]. The extent of the recirculation zone diminishes with increasing downstream distance, the termination of which coincides with the merge point, at which point the mean flow velocity is 0 m/s , *i.e.* $u = v = 0 \text{ m/s}$ [19]. As the converging region is defined by the existence of the recirculation zone, the length of the converging region hence equates to X_{mp} .

The merge point marks the beginning of the *merging region*, where the jets continue to interact until becoming fully combined at the combined point cp [22]. This point, located at $(X_{cp}, Y_{cp}) = (x_{cp}/w, y_{cp}/w)$ is identified as the downstream location beyond which only a singular maximum velocity value is present in the flow domain for each X location and the deflection in the local velocity profile no longer exists. This is signified by a singular instance of $du/dy = 0$ for a given X location [1]. Beyond the combined point, the dual jet continues to develop along the *combined region* in a manner similar to a single equivalent wall jet and eventually reach a self-similar state [19].

While the flow structure presented in Figure 2.8 refers to that typically obtained using two slot jet nozzles as the wall jet and the offset jet, a similar flow field also been observed for an axisymmetric dual jet flow configuration, such as that created through the use of two concentric annular jet nozzles. An example of where this setup is regularly encountered is a dual lip air ring, as previously discussed in Chapter 1, which is used during blown film extrusion. The resulting flow field from a dual lip air ring is investigated in both Sidiropoulos and Vlachopoulos [90] and Gao *et al.* [91], where distinct consistencies are present when compared with the streamwise flow development of a typical planar dual jet flow, and obvious differences can be attributed to the use of varying jet widths and a velocity ratio less than 1.

2.4. Summary

The purpose of this chapter was to provide an overview of the dual jet flow field and introduce the relevant terminology prior to reviewing the relevant literature. In order to do so, it was necessary to discuss some of the more basic concepts associated with free plane jets, plane parallel jets, wall jets, and offset jets, all of which enhance the acquired understanding of the dual jet flow field. The time-averaged flow structure associated with a dual jet was hence discussed in detail and the flow features of importance were identified. Finally, the key dimensionless parameters commonly used to define the dual jet flow field were discussed, as summarised in Table 2.1, the variation of which is of specific interest to the present investigation.

Table 2.1: Summary of key dimensionless parameters used to define the dual jet flow field.

<i>Dimensionless Parameters</i>	<i>Symbol</i>	<i>Formula</i>
Reynolds Number	Re	$u_{w/o}w/\nu$
Offset Ratio	OR	d/w
Velocity Ratio	V_r	u_w/u_o

Chapter 3

Literature Review

While both wall jets and offset jets have each been studied extensively, the combination of such is severely underrepresented in the published literature. Of those studies which have been dedicated to dual jet flow, the majority have been entirely numerical in nature. Excluding publications linked to this PhD thesis, such as Murphy *et al.* [30], only a single experimental investigation regarding dual jets has been conducted to date, that of Wang and Tan [19]. However, this study considered only the fluid flow behaviour, meaning the heat transfer characteristics associated with dual jet flows are yet to be explored through experimentation. It should be noted, however, that an experimental investigation carried out by Gao *et al.* [21] focussed on a jet configuration similar to a dual jet, though the scope of the study did not align with the characterisation of dual jet flow. In this case, a thin co-flowing wall jet was used to stabilise a much larger offset jet, meaning the range of jet parameters explored and, hence, the insight provided into dual jet flow was quite limited. Therefore, dual jet flow remains largely unexplored through experimental means.

The remaining studies dedicated to dual jet flow focus only on numerical simulations, where the lack of credible experimental data for the purpose of model validation has led to a great degree of disparity across the published numerical results. The following sections discuss the reported findings of the published dual jet studies, where the published studies which focus primarily on the flow field characteristics of a dual jet are reviewed initially, followed by studies which analyse the transfer of heat to a dual jet flow. A summary of the publications discussed alongside their main findings can be found in Table 3.1 in section 3.3.

3.1. Hydrodynamic Studies of Dual Jet Flows

The first study to analyse the fluid flow characteristics of a dual jet was carried out by Wang and Tan [19]. With water as a working fluid, Wang and Tan [19] used particle image velocimetry (PIV) techniques to analyse the flow field associated with a dual jet configuration for $OR = 1$ and jet Reynolds number of $Re = 10,000$. For jet widths of 10 mm and jet aspect ratio of 30, the authors recorded 360 instantaneous velocity fields at a maximum frequency of 15 Hz using an 8-bit charge-coupled device (CCD) camera with a 1024×1008 pixel resolution. Ensemble averaging of the images yielded the time-averaged velocity field shown in Figure 3.1(a), from which the locations of the merge point and the combined point were identified at $X_{mp} = 0.75$ and $X_{cp} = 6.4$, respectively. The maximum jet velocity was observed to decay rapidly with downstream distance, much faster than that of a single wall jet, a single offset jet, or two parallel plane jets, which the authors attributed to the strong jet interaction in the merging region. Self-similarity was hence observed for $X \geq 15$. Most notably,

the authors identified the presence of a periodic von Kármán-like vortex street in the inner shear layer regions when analysing the time-resolved flow behaviour, which indicated a periodic interaction between the two jets. A schematic of the flow behaviour is shown in Figure 3.1(b). In essence, the presence of an offset wall between the two jets behaved in a similar manner to a bluff body in crossflow, whereby roll-up vortical structures were shed in the wake region in a periodic fashion, with a Strouhal number of $St = 0.25$, where

$$St = \frac{f_s w}{u_{jet}} \quad (3.1)$$

and f_s is the vortex shedding frequency. This behaviour was found to also affect the development of the outer free shear layer, *i.e.*, Layer 1, where a vortex pairing mechanism was observed inside flow field between the von Kármán-like vorticity and nearby patches of vorticity with opposing sign.

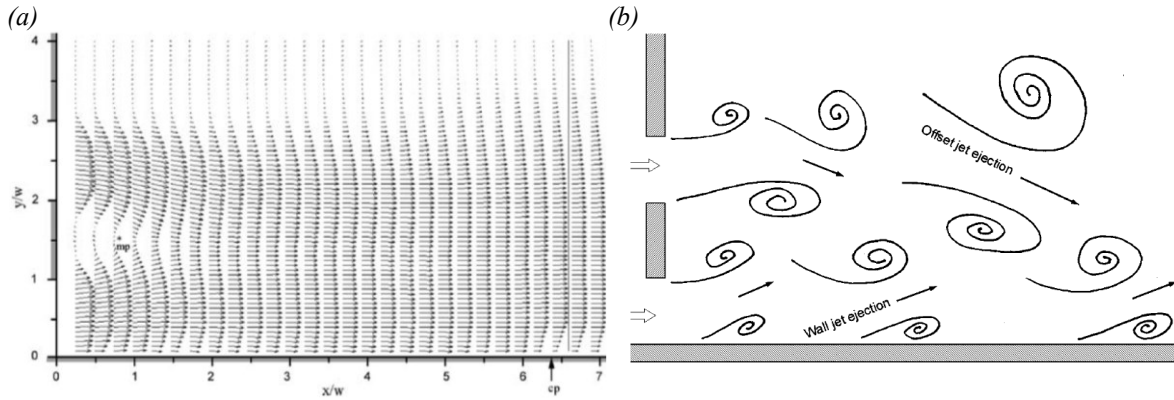


Figure 3.1: (a) Mean velocity vector plot and (b) von Kármán vortex interaction as acquired by Wang and Tan [19] using particle image velocimetry.

Li *et al.* [2] subsequently investigated an identical dual jet configuration through numerical means using large eddy simulation (LES), where the primary intention was to examine the dilution characteristics with respect to the flow behaviour, inspired by the extensive use of dual jet flows for wastewater evacuation purposes. A reasonable agreement was acquired when validating their computational fluid dynamics (CFD) model against the experimental data of Wang and Tan [19]. Discrepancies were found, however, regarding the point at which the dual jet velocity profile reaches self-similarity, where Li *et al.* [2] noted that was achieved by $X = 10$ but Wang and Tan [19] found this did not occur until at least $X = 15$. Other reported findings included trends associated with the streamwise velocity and turbulence intensity along the centreline between the two jets, as presented in Figure 3.2, as well as the respective downstream development of the individual shear layers.

In a follow up study, Li *et al.* [92] analysed the effect of changing the offset ratio ($OR = 0.5, 1, 2, 3.25, 5, 8$) and the velocity ratio $V_r = u_w/u_o$ numerically using the $k - \varepsilon$ turbulence model. To alter the velocity ratio, u_w was initially maintained at a constant value and u_o was varied to produce $V_r = 0.25, 0.5, 0.75$ and 1 , following which u_o was maintained at a constant value and u_w was varied to achieve $V_r = 1, 1.33, 2$ and 4 , thereby varying the total mass flow rate consistently throughout. In doing so, the flow behaviour shifted from that of an offset jet to a wall jet, as shown in Figure 3.3, where the extent of the recirculation zone gradually diminished with increasing V_r . For $V_r < 1$, the rate of decay of the maximum flow velocity u_m increased with increasing V_r as the entrainment of the wall jet became stronger. On the contrary, for $V_r > 1$, the rate of decay of maximum velocity decreased with increasing V_r in the region $X < 35$ as the offset jet entrainment weakens. For a constant velocity ratio, the rate of decay of maximum velocity was found to increase with increasing OR .

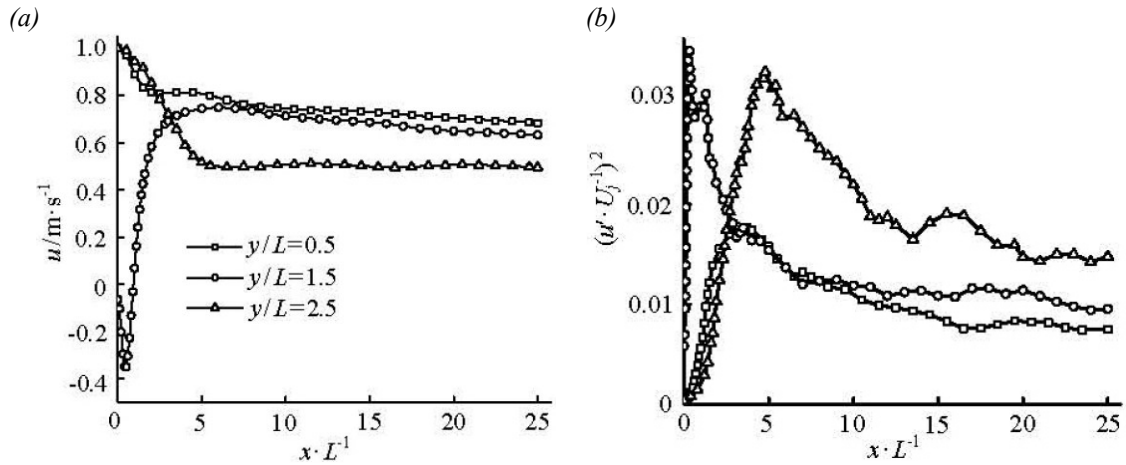


Figure 3.2: Change in (a) streamwise velocity and (b) turbulence intensity with downstream distance, as reported by Li et al. [2] by means of LES, where L is analogous to the jet width w , i.e., $L = w$.

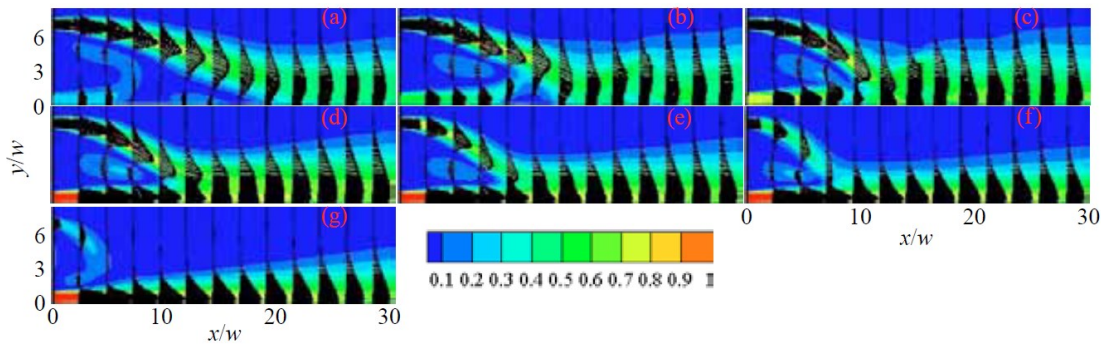


Figure 3.3: Contours and vector plot of velocity magnitude for $OR = 5$ as per Li et al. [92] by means of the $k - \varepsilon$ turbulence model, where $V_r =$ (a) 0.25, (b) 0.5, (c) 0.75, (d) 1, (e) 1.33, (f) 2, and (g) 4.

In a later numerical study carried out by Kumar and Das [1] using the $k - \varepsilon$ turbulence model, it was noted that, unlike Wang and Tan [19], two stable, counter-rotating vortices existed within the recirculation region, shown in Figure 3.4. Due to their respective positions within the recirculation region, these vortices were labelled individually as the upper vortex (UV) and the lower vortex (LV), and their corresponding centres were termed the upper vortex centre (UVC) and lower vortex centre (LVC). Due to an absence of experimental dual jet data at higher OR values, the numerical model in this case was validated using results from Pelfrey and Liburdy [93], Tanaka [94], and Spall *et al.* [95] for $Re = 20,000$ and an equivalent OR of 8. However, these references pertain to experimental flow data for either a single offset jet or plane parallel jets, both of which have quite different flow characteristics to a dual jet. When compared to the results of a single offset jet, a greater deflection of the offset jet was observed in the case of the dual jet due to the existence of a lower pressure inside the recirculation zone.

Kumar and Das [1] also devised a means to accurately identify the locations of the merge point, the combined point and the upper and lower vortex centres, which took into account both the streamwise and transversal position. As demonstrated in Figure 3.5(a), the locations of UVC , LVC and mp were identified by means of the intersection between the $u = 0$ and $v = 0$ contour lines, where mp coincides with the intersection point located furthest downstream. Conversely, the combined point corresponded to the point where, at a given downstream location, the velocity gradient $\partial u / \partial y$ has a singular zero value, as shown in Figure 3.5(b). By this method, (X_{mp}, Y_{mp}) and (X_{cp}, Y_{cp}) were identified at (10.46, 2.42) and (18.38, 2.78), respectively, where it was noted

that neither Y coordinate corresponded to the centreline between the two jets (approximately $Y = 4.5$), which had been assumed to be the case in all previous studies up until this point.

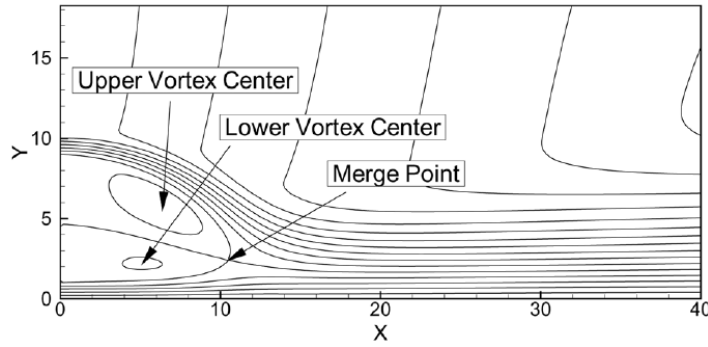


Figure 3.4: Streamline plot of a dual jet indicating the presence of two counter-rotating vortices [1]

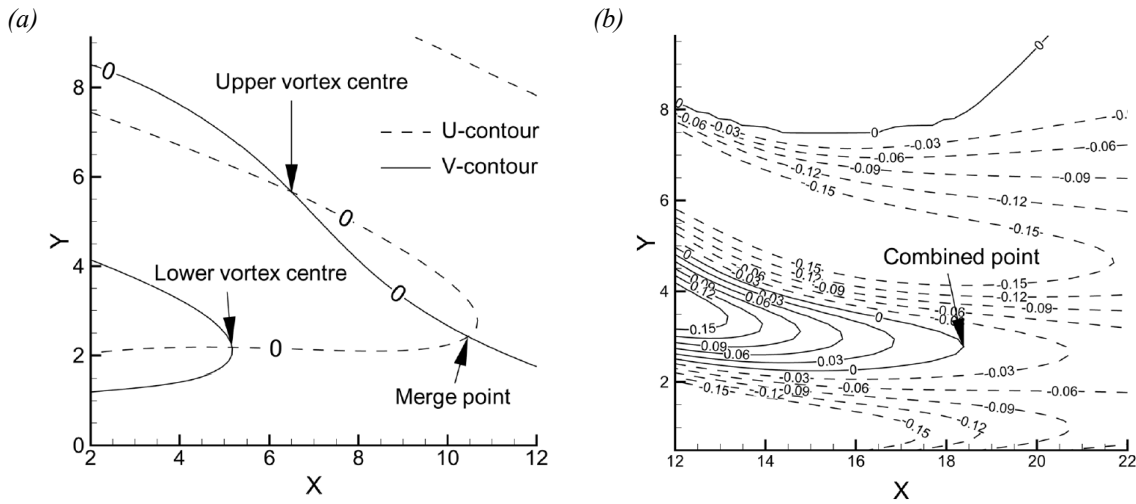


Figure 3.5: Identification of (a) UVC, LVC and mp and (b) cp as per Kumar and Das [1] by means of the $k - \varepsilon$ turbulence model.

In a numerical study carried out by Mondal *et al.* [20], which again adopted the $k - \varepsilon$ turbulence model, the influence of the offset ratio on the flow behaviour near the jet nozzle exit of a dual jet with $Re = 10,000$ was investigated. In doing so, it was found that within the range $0.7 \leq OR \leq 2.1$, flow in the near jet region was characterised by von Kármán-like shedding of vortices, as shown in Figure 3.6(a), where similar to the findings published by Wang and Tan [19]. However, outside this range, *i.e.*, $OR \leq 0.6$ or $OR \geq 2.2$, two counter-rotating vortices were observed in this region, as demonstrated in Figure 3.6(b), consistent with that previously noted by Kumar and Das [1]. This thus suggested that a range of dual jet offset ratio values exist within which periodic vortex shedding occurs. Outside this range, the flow displays more stable behaviour in the form of two counter-rotating vortices. Within this unsteady region, the Strouhal number associated with the vortex shedding was found to decrease with increasing OR , where the obtained value of $St = 0.22$ when $OR = 1$ was comparable to that of Wang and Tan [19]. Mondal *et al.* [20] attributed the decrease in the Strouhal number with increasing OR to the increased distance between the two inner shear layers, meaning an increased time was taken for the roll-up vortex from each shear layer to grow sufficiently in size and strength to draw in the opposing vortex, hence delaying the shedding process. In addition to this, the results revealed that, within the unsteady range of OR , the time series of u and v exhibited sinusoidal oscillatory behaviour, where both signals possessed identical frequencies corresponding to the vortex shedding frequency f_s . A similar finding was reported more recently by Singh and Dewan [24] when analysing the time-resolved flow behaviour of a dual

jet, who noted a slightly higher Strouhal number of $St = 0.294$ for the same jet parameters using the SST $k - \omega$ turbulence model. In addition, for a fixed OR of 1, Singh and Dewan [24] noted that the value of X_{mp} remained relatively constant when Re was varied in the range $10,000 \leq Re \leq 40,000$, and self-similarity was observed in each case for $X \geq 30$, which was considerably further downstream than that previously reported by Wang and Tan [19].

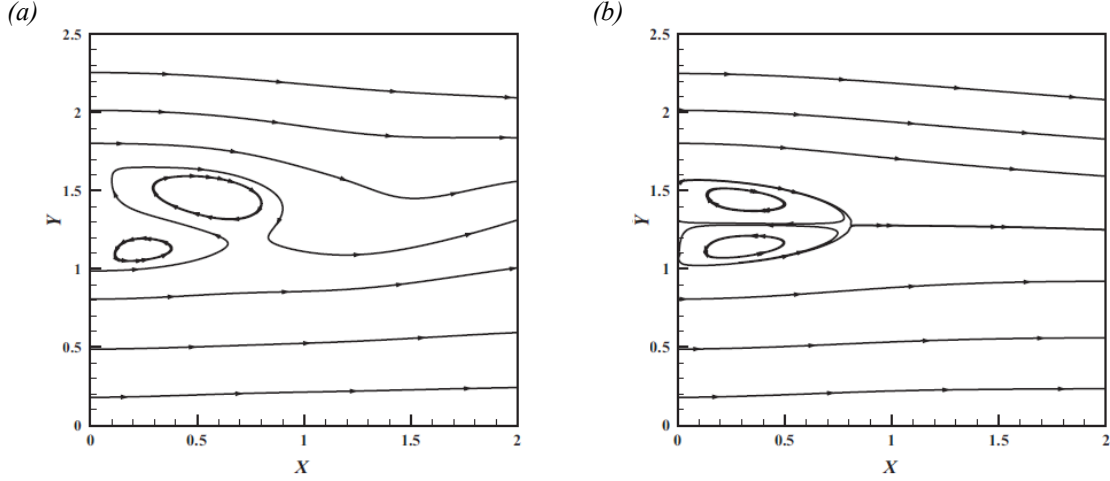


Figure 3.6: Streamline plot acquired by Mondal et al. [20] using the $k - \varepsilon$ turbulence model for a dual jet with $OR = (a) 0.7$ and $(b) 0.6$, demonstrating the stable and unstable flow regimes, respectively.

The effect of the offset ratio on dual jet flow was further investigated using the $k - \varepsilon$ turbulence model by Kumar [83] for $Re = 15,000$, where OR was varied from 2 to 14 when adopting the definition of OR used in the present study. In this case, two-counter-rotating vortices were observed for all OR examined, as per Figure 3.7. As is to be expected, increasing OR was found to increase the size of the recirculation zone, which moved the respective positions of UVC and LVC further downstream, while UVC was pushed further from the solid wall by the lower vortex. While the size of both vortices consequentially increased, the growth in the upper vortex was considerably more in comparison to the lower vortex, which lead to an increased suppression of the lower vortex. This enabled the strength of the lower vortex to grow with increasing OR , the value of which always exceeded the strength of the upper vortex. The offset ratio was found to impact the decay of maximum velocity with downstream distance, however different behaviours were observed for $OR = 2$ and $OR > 2$, as shown in Figure 3.8. For $OR = 2$, it was noted that u_m remained at a constant value until the potential core was consumed, before decreasing rapidly towards a local minimum value, beyond which u_m exhibited a slight increase and subsequently decayed over the remaining downstream distance. For all other OR investigated, this local peak in the u_m profile was not observed.

Following this, Kumar [27] further investigated this topic for the same range of dual jet parameters, where the locations of the merge point, the combined point, and the upper and lower vortex centres were identified using the method outlined by Kumar and Das [1] for the purpose of deriving appropriate correlations. In doing so, it was noted that while X_{mp} , Y_{mp} and Y_{cp} all increased consistently with increasing OR , the value of X_{cp} appeared to initially decrease towards a minimum value at $OR = 5$, beyond which it subsequently increased. An explanation for this behaviour was derived from the analysis of the minimum pressure value within the recirculation zone, where a linear decrease in the minimum pressure was observed for decreasing OR when $OR > 5$, however, for $OR \leq 5$, a sudden drop in pressure was noted. These considerably lower pressures for $OR \leq 5$ enabled the jets to deflect towards each other and collide at a much higher intensity, thus delaying the

onset of the combined point. For $OR > 5$, however, inertial effects dictated the location of X_{cp} . In a subsequent study by Kumar *et al.* [23] for a similar set of flow parameters, it was noted that increasing OR increased the intensity of the upper vortex, while increasing Re reduced X_{mp} . This finding was in contrast to that reported by Singh and Dewan [24].

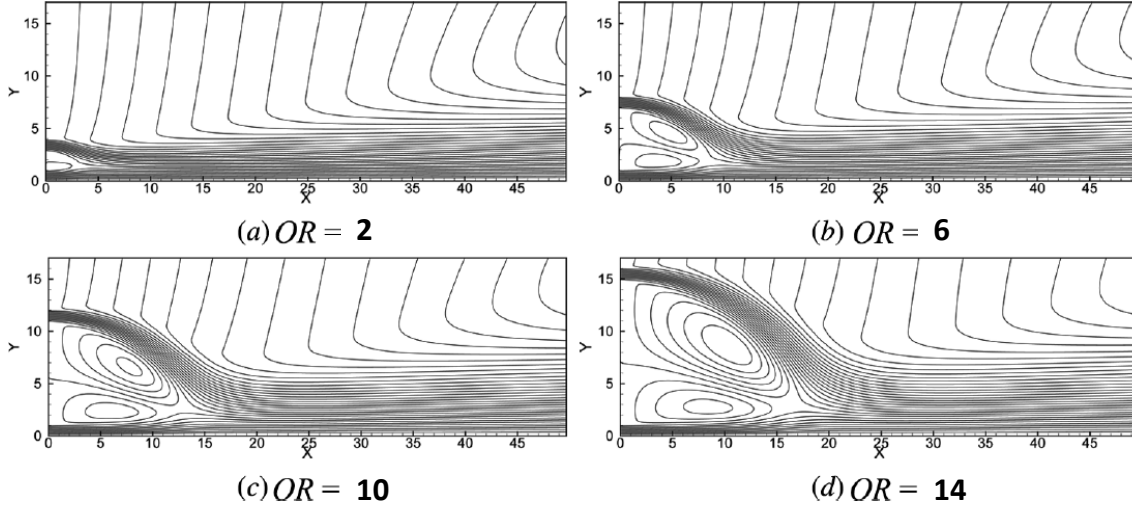


Figure 3.7: Streamline plots representing the effect of varying OR on the dual jet flow field, as acquired by Kumar [83] using the $k - \varepsilon$ turbulence model.

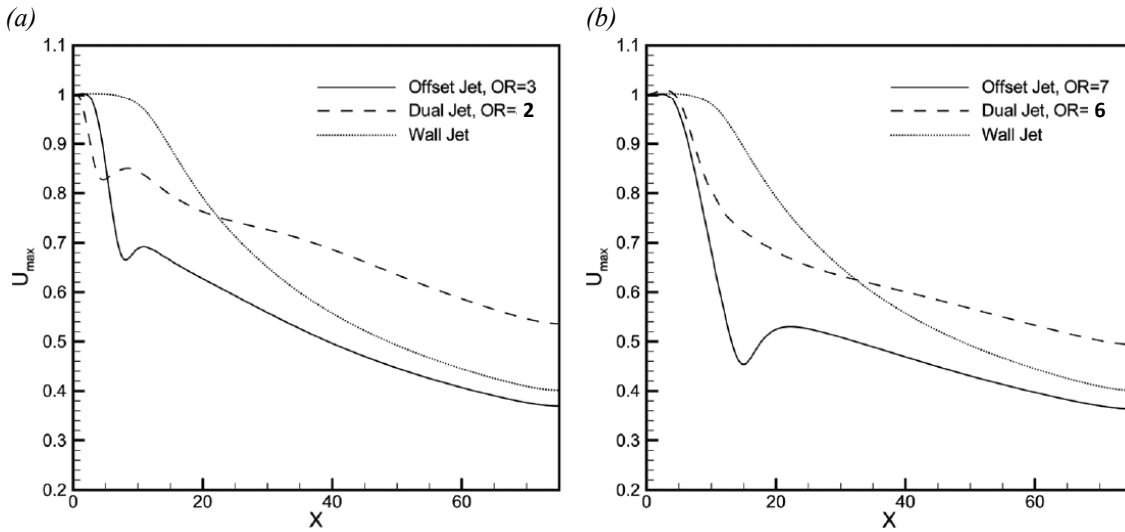


Figure 3.8: Decay of maximum velocity with downstream distance for a dual jet with $Re = 15,000$ for $OR = (a) 2$ and $(b) 6$, as acquired by Kumar [83] using the $k - \varepsilon$ turbulence model.

The first numerical study to focus on the effect of V_r on the onset of periodic flow instabilities in dual jet flow was conducted by Mondal *et al.* [96] for $OR = 1$ and $Re = 10,000$ using the $k - \varepsilon$ turbulence model. The velocity ratio was varied by maintaining u_o at a constant reference value and subsequently adjusting u_w to achieve $0.5 \leq V_r \leq 2$, therefore changing the total mass flowrate throughout. An unsteady interaction between the two jets was observed for $0.78 \leq V_r \leq 1.34$, where the shed vortices were convected downstream to produce an organised von Kármán-like street. Outside of this V_r sub-range, two counter-rotating vortices were noted in the recirculation zone. Increasing V_r was found to increase St inside the unsteady range of V_r values. The transient behaviour of a dual flow field was further investigated in Mondal *et al.* [21] and Mondal *et al.* [97], where a unsteady range of jet width values were, respectively, identified for varying just the wall jet width (identified as h) and both jet widths simultaneously for $Re = 10,000$. Mondal *et al.* [21] identified periodic

vortex shedding for $0.3 \leq h/d \leq 2$, where St decreased with increasing h/d before reaching a constant value for $1 \leq h/d \leq 2$, while Mondal *et al.* [97] made a similar observation for $0.6 \leq w/d \leq 2$, where St decreased with increasing w/d for $0.6 \leq w/d < 1.6$ but remained constant thereafter.

The onset of flow instabilities in the dual jet flow field through varying OR was also investigated by Zhao and Sun [89], however, for a much lower Reynolds number flow of $Re = 56$ using the lattice Boltzmann BGK model. Periodic vortex shedding was reported to occur for a much higher equivalent offset ratio range of $8 \leq OR \leq 9$, while two counter-rotating vortices were observed for $OR \leq 7$. The corresponding Strouhal number was found to have a value of 0.012 when $OR = 8$, which indicates a much lower shedding frequency compared to previous studies. In addition to the transient characteristics, Zhao and Sun [89] noted a downstream movement of both mp and cp with increasing OR while $OR \leq 7$, however, a slight decrease in their respective streamwise locations was observed when $OR = 8$.

Hnaïen *et al.* [25] further analysed the positions of mp , cp , UVC and LVC with respect to varying OR and V_r using the $k - \omega$ turbulence model and a series of correlations were hence proposed, the validity of which were never verified in any of the subsequent dual jet studies. A sample of the results acquired can be seen in Figure 3.9, which presents the contours of the streamwise velocity and static pressure fields acquired for $Re = 15,000$ and $OR = 8$. Similar to many previous studies, through increasing OR for a constant V_r value, the positions of mp , cp , UVC and LVC were found to move downstream and further from the solid wall, which became less apparent for higher V_r . For a fixed offset ratio, however, the positions of mp , cp , UVC and LVC appeared to move upstream with increasing velocity ratio, while mp and LVC moved closer to the solid wall, the effect of which was again more prominent for higher OR values. In addition, the longitudinal displacement of UVC with increasing velocity ratio appeared to be more pronounced compared to LVC , where the presence of the solid wall boundary appeared to impose a greater resistance on the displacement of LVC compared to that exerted by the surrounding fluid on UVC .

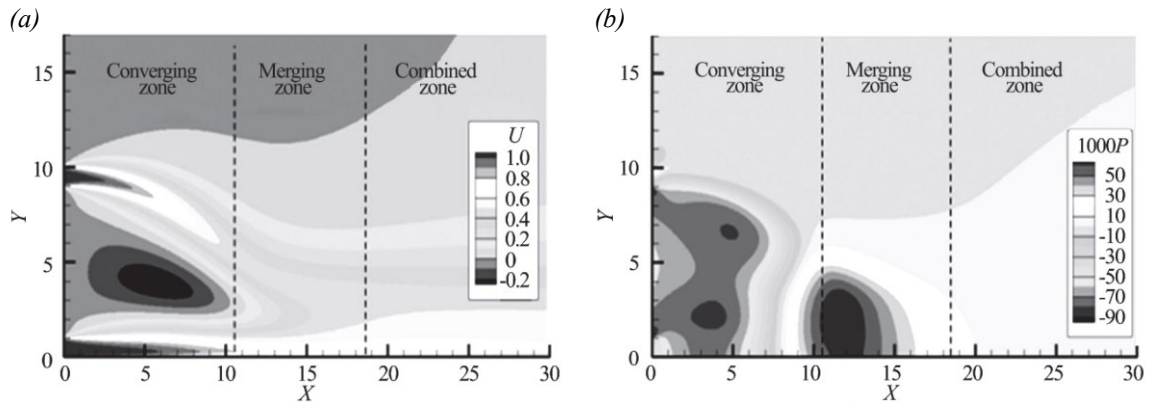


Figure 3.9: Contours of (a) streamwise velocity and (b) static pressure for $Re = 15,000$ and $OR = 8$ obtained by Hnaïen *et al.* [25] using the $k - \omega$ turbulence model.

Assoudi *et al.* [26] also investigated the effect of the offset ratio and the velocity ratio on the flow behaviour within a dual jet using the parameter ranges $1 \leq OR \leq 10$ and $0.57 \leq V_r \leq 1$ for $Re = 15,000$ using the Reynolds stress model (RSM). Unlike previous studies which examined a similar set of dual jet parameters, the occurrence of periodic vortex shedding was not reported for $OR = 1$. Instead, the steady, double-vortex phenomenon was observed throughout. Similar to Kumar [83], an increase in OR was found to cause a growth in the size of the upper vortex, which acted to further suppress the lower vortex, as shown in Figure 3.10. As a

result, the value of X_{mp} increased, thus increasing the extent of the recirculation zone. The values of X_{mp} identified during this study were shown to be in good agreement with that provided by Kumar [27] and Mondal *et al.* [20]. The streamwise position of the combined point was also seen to increase with increasing OR , similar to previous studies, however, it was noted by Assoudi *et al.* [26] that the values obtained in this study were less than that reported by earlier investigations. With regard to the velocity ratio, it was noted that a decrease in V_r allowed the flow field to shift from that of a dual jet to a single offset jet, which caused a subsequent growth of the recirculation zone. For $V_r = 1$, it was observed that, although both jets possessed comparable strengths, the entrainment effects of the wall jet appeared to be considerably more intense due to the presence of the boundary. Conversely, for $V_r \approx 0.57$, the offset jet possessed stronger entrainment effects, thus dragging the wall jet away from the boundary and into the recirculation zone.

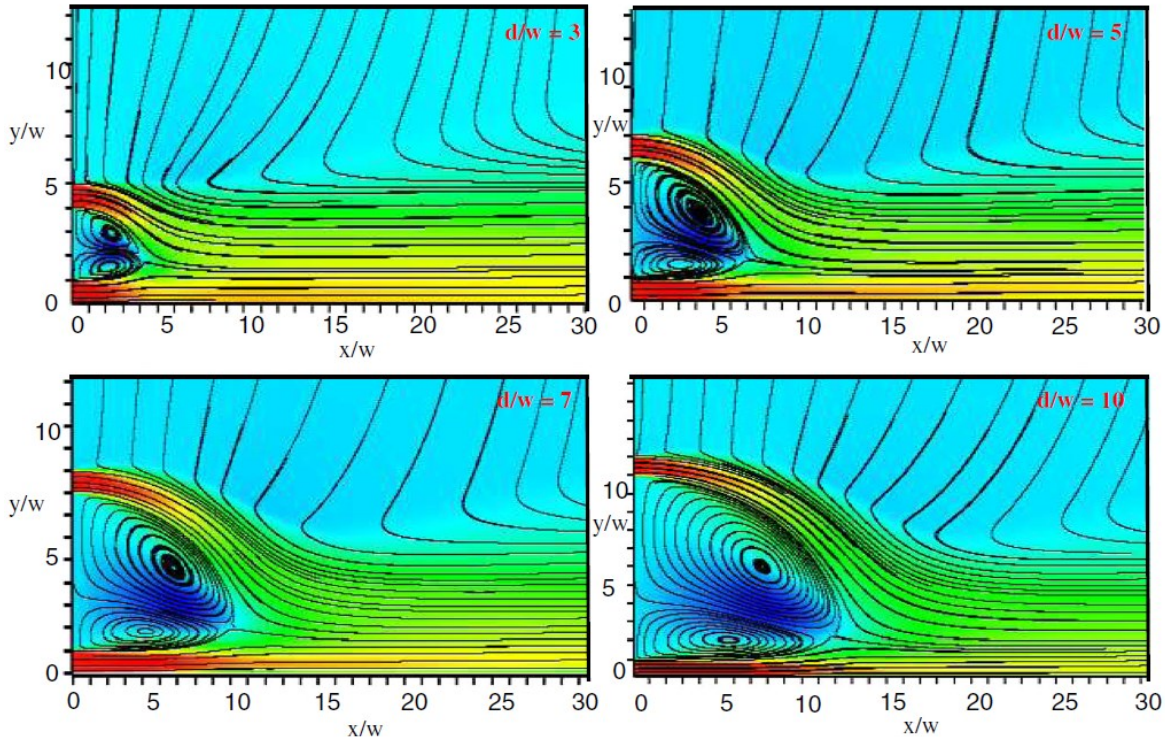


Figure 3.10: Streamline plots for a dual jet with varying OR and $Re = 15000$ as per Assoudi *et al.* [26], acquired using the Reynolds stress model (RSM).

The remaining studies which focussed on the fluid flow aspects of dual jet flow introduced variable parameters which would be considered outside of the scope of the present investigation, which is primarily concerned with the variation of Re , OR , and V_r . They do, however, present some interesting findings, where a certain set of parameters may overlap with the present study, and are therefore discussed in brief. For example, Hnaïen *et al.* [85], and subsequently Hnaïen *et al.* [10], investigated the effect of the wall inclination β on dual jet flow behaviour, defined as per Figure 3.11(a), with respect to the maximum velocity decay and spreading of the inner and outer shear layers using the $k - \omega$ turbulence model. For $Re = 15,000$ and $OR = 8$, Hnaïen *et al.* [85] reported the existence of a critical downstream location, X_{cr} , before and after which the wall jet and offset jet, respectively, dictate the value of u_m , where X_{cr} was found to vary with β . Hnaïen *et al.* [85] also noted a slight increase in the jet velocity after the nozzle exit, the value and streamwise location of which decreased with increasing β . An increase in the jet spreading rate within the outer shear layers was also observed for increasing β . Hnaïen *et al.* [86] later investigated the effect of the offset jet ejection angle α , as defined in

Figure 3.11(b), where, for a fixed OR , increasing α displaced mp , cp , UVC and LVC further upstream and towards the solid wall, thus indicating an acceleration of the merging process.

A series of studies also focussed of the effect of replacing the solid smooth wall boundary with the wavy surface, the first of which was conducted by Singh *et al.* [87] using the $k - \varepsilon$ turbulence model. For this investigation, the Reynolds number and equivalent OR were fixed at 15000 and 6, respectively, and the non-dimensionalised amplitude of the wavy surface was varied from 0.1 to 0.7 for a constant number of cycles N , where $N = 10$. It was observed that the introduction of a wavy surface brought about a sudden decrease in the pressure inside the recirculation zone, which continued to decrease with the increasing amplitude of the surface. This subsequently moved mp , cp further upstream, while also increasing the maximum streamwise velocity in the flow field. The effect of the wavy wall on the flow field characteristics of a dual jet was further investigated by the authors in [98-101].

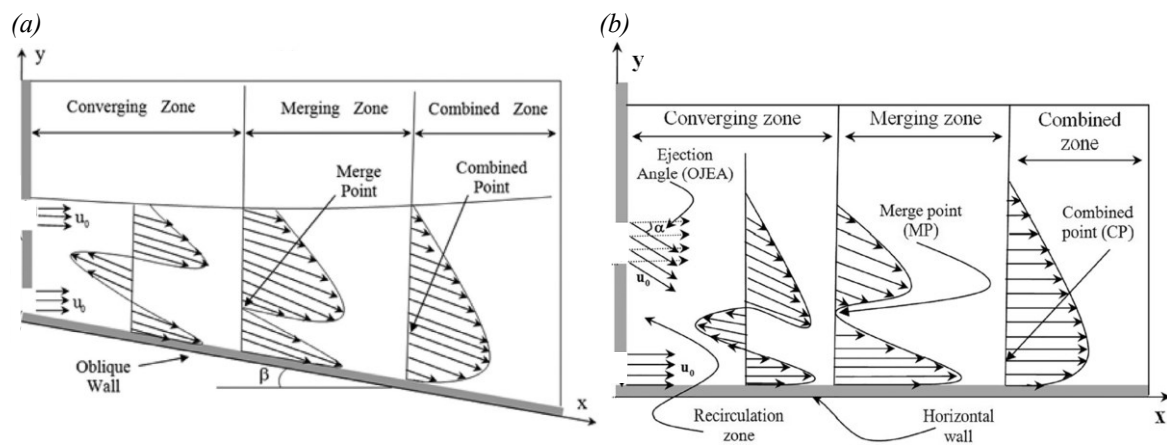


Figure 3.11: Definition of (a) wall inclination angle β and (b) offset jet ejection angle α in the context of the dual jet flow configuration, as per Hnaien *et al.* [10] and Hnaien *et al.* [86], respectively.

A series of numerical studies carried out by Mondal *et al.* [102], Hnaien *et al.* [103] and Hnaien *et al.* [104] using the $k - \omega$ turbulence model introduced a parallel co-flow to the fluid domain into which the dual jet is injected, therefore replacing the initial quiescent condition inside the fluid domain with a steady flow velocity, where all three studies varied the co-flow velocity for a constant Re of 15,000. In doing so, Mondal *et al.* [102] noted that mp and cp moved downstream with increasing co-flow velocity, the effect of which was greater at higher OR , while Hnaien *et al.* [103] reported a reduction in the size of the upper and lower vortex with increasing co-flow ejection angle. Hnaien *et al.* [104] observed a decrease the size of the upper vortex towards that of the lower vortex with increasing co-flow velocity, while moving both UVC and LVC further downstream. Finally, the concept of a dual offset jet was explored in both Mondal and O'Shaughnessy [105] and Mondal *et al.* [106], where the wall jet was replaced with a second offset jet to produce a distinctly different flow field containing two separate recirculation regions, one bounded by the solid interface and the other existing between the two jets.

3.2.Heat Transfer Characteristics of Dual Jet Flows

The heat transfer characteristics associated with dual jet flow have been studied to a much lesser extent, the published results from which are considerably less agreeable in the absence of verified experimental data. The first numerical study to focus on the heat transfer aspects of dual jet flow was conducted by Vishnuvardhanarao and Das [28] using the $k - \varepsilon$ turbulence model, who examined the effect of varying the jet Reynolds number

in the range $10,000 \leq Re \leq 40,000$ and the velocity ratio on the downstream local Nusselt number (Nu_x) profile along the heated wall for $OR = 1$. Similar to many studies previously discussed, the velocity ratio was varied through altering one jet velocity and maintaining the other at a constant value, where u_w was adjusted incrementally to achieve $V_r = 0.25, 0.5, 0.75$ and 1 and u_o was subsequently varied to produce $V_r = 1, 1.33, 2$ and 4 . Examples of the Nu_x profiles acquired are presented in Figure 3.12.

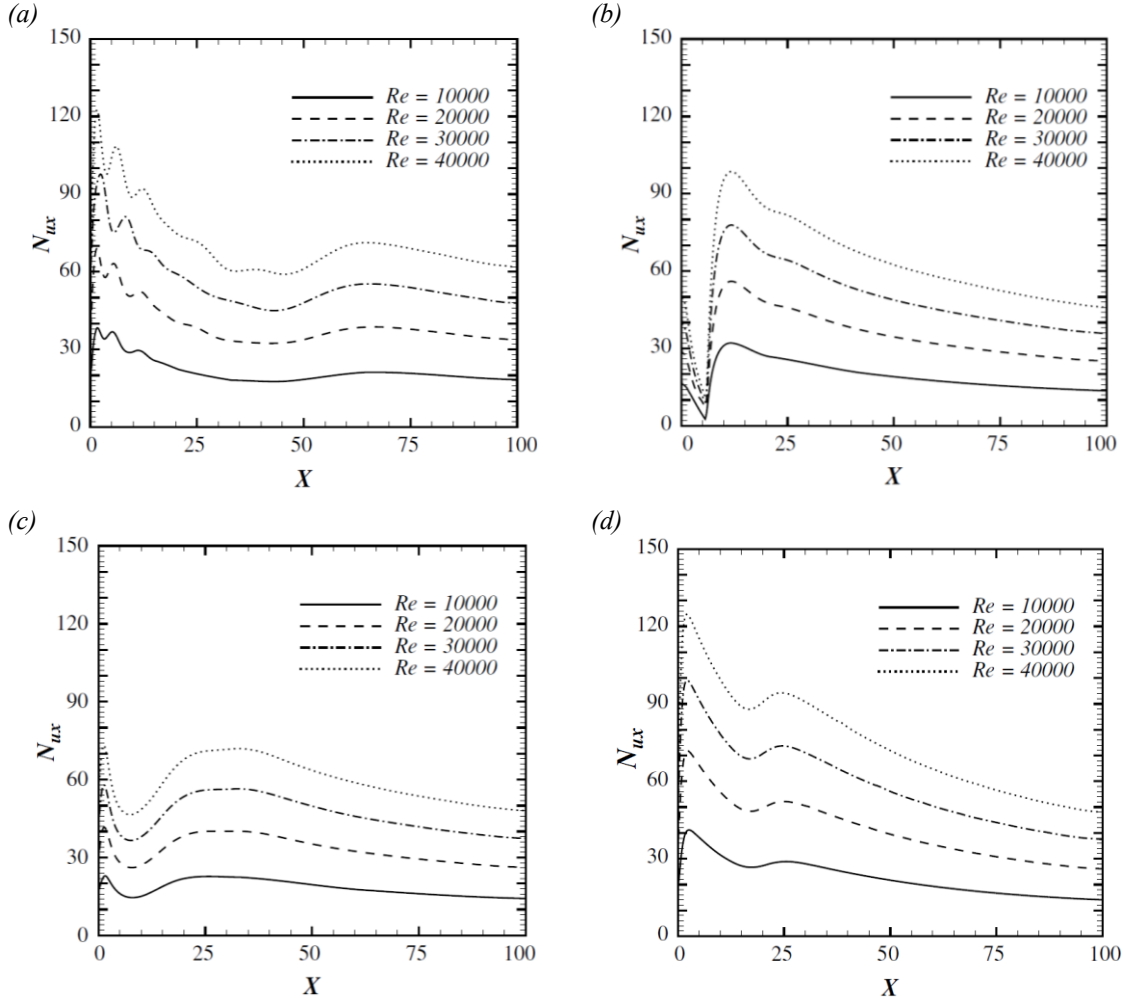


Figure 3.12: Nu_x distribution along the solid wall, as per Vishnuvardhanarao and Das [28], for a dual jet with varying Re and $OR = 1$, acquired using the $k - \varepsilon$ turbulence model, where (a) $V_r = 1$ (b) $V_r = 0.25$ (c) $V_r = 0.5$ and (d) $V_r = 4$.

For all jet velocity combinations examined, Vishnuvardhanarao and Das [28] reported that an increase in Re resulted in an increase in Nu_x , while the overall profile trends appeared to remain unchanged. For $V_r = 1$, significant fluctuations in Nu_x were observed close to the nozzle exit, which were attributed to the occurrence of vortex shedding as per Wang and Tan [19], beyond which a general decay in Nu_x was evident. The profile altered greatly for $V_r \neq 1$, where, for $V_r \geq 0.5$, a maximum value of Nu_x was noted near the jet nozzle exit, as well as the existence of a secondary peak further downstream due to the mixing of the two jet streams. For this subset of jet velocity combinations, the effect of varying u_o was negligible near the jet exit and up to $X = 20$, however, beyond this point, the secondary peak appeared to both widen and move further downstream with increasing u_o . For $V_r < 0.5$, the maximum value of Nu_x instead existed at some point further downstream, beyond which Nu_x continued to decay with streamwise distance. In this case, u_w was observed to have considerable influence over Nu_x near the jet exit, where Nu_x was significantly enhanced in this region when $u_w > u_o$ compared to interchanging the jet velocities for the same mass flow rate. Finally, the surface averaged

Nusselt number ($\overline{Nu_x}$) was found to increase linearly with Re and a maximum value was achieved when $u_o/u_w = 0.25$.

Farooq and Das [29] subsequently conducted a numerical study to investigate the influence of the offset ratio in the range $2 \leq OR \leq 3$ on the heat transfer characteristics of a dual jet for $Re = 15,000$ and $Pr = 0.71$, where, again, the $k - \varepsilon$ turbulence model was adopted. Similar to Vishnuvardhanarao and Das [28], Farooq and Das [29] obtained quite an oscillatory Nu_x profile, however, a prominent peak was observed in the location of the merge point, which appeared to move downstream with increasing OR , as shown in Figure 3.13(a). The magnitude of Nu_x at the peak location was also reported to increase with increasing OR , which acted to increase $\overline{Nu_x}$ somewhat linearly. Kumar [83] subsequently investigated the effect of the varying offset ratio over the much wider range of $2 \leq OR \leq 14$ for the same jet Reynolds number of $Re = 15,000$ and, in doing so, obtained a very different Nu_x profile, as presented in Figure 3.13(b). In this case, while being considerably less oscillatory, two distinct peaks were identified in the Nu_x profile for $OR \geq 4$, i.e., a primary peak located close to the nozzle exit and a secondary peak at some distance downstream. The primary peak was reported to remain relatively consistent with changing OR , but the secondary peak appeared to move downstream with increasing OR , as well as grow in magnitude. Conversely, the Nu_x profile obtained for $OR = 2$ was found to contain only a single peak close to the nozzle exit. The authors provided no explanation is provided for this behaviour.

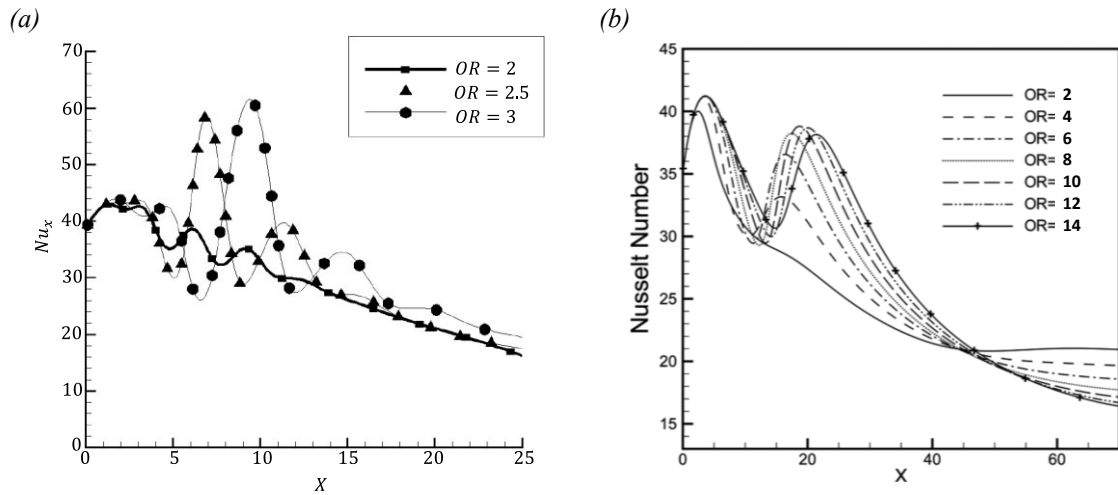


Figure 3.13: Variation in Nu_x profile with respect to changing OR for $Re = 15000$ as per (a) Farooq and Das [29] and (b) Kumar [83] by means of the $k - \varepsilon$ turbulence model.

Mondal *et al.* [22] examined the impact of Reynolds number in the range $10,000 \leq Re \leq 20,000$ and Prandtl number in the range $1 \leq Pr \leq 4$ on Nu_x and $\overline{Nu_x}$ for $OR = 1$, while also varying the thickness and conductivity of the bounding wall. Similar to Farooq and Das [29], a single peak was identified in the Nu_x profile just downstream of the nozzle exit, where the value of Nu_x continued to decay thereafter, as shown in Figure 3.14(a). The occurrence of this peak was again attributed to the position of the merge point, which appeared to move downstream with increasing Re . Aside from the slight change in the peak position, increasing Re was found to cause a proportional enhancement of Nu_x at each downstream location, due to a rise in the convective heat transfer, while preserving the overall trend. As a result, a linear rise in $\overline{Nu_x}$ was noted for increasing Re .

Following this, Hnaien *et al.* [84] further investigated the effect of varying Re on the dual jet induced Nu_x profile using the $k - \omega$ turbulence model, in addition to varying the offset ratio, where $10,000 \leq Re < 40,000$ and $4 \leq OR \leq 19$. The results obtained are presented in Figure 3.14(b), where, similar to Mondal *et al.* [22], a singular prominent peak existed just downstream of the nozzle exit. While the plots shown in Figure 3.14(b) indicate a potential secondary peak for higher OR values, which becomes considerably more apparent at higher Re values, this was not addressed in the discussion provided by Hnaien *et al.* [84]. With regard to changing Re , Hnaien *et al.* [84] obtained the same trend as Mondal *et al.* [22], such that increasing Re enhanced Nu_x at all downstream locations, which subsequently increased in $\overline{Nu_x}$. Conversely, while increasing OR caused a rise in Nu_x along the converging region, thus approaching that of a single wall jet, a subsequent decrease was observed along the merging and combined regions, therefore causing an overall reduction in $\overline{Nu_x}$, which was inconsistent with Farooq and Das [29]. This effect became more apparent for higher Re values.

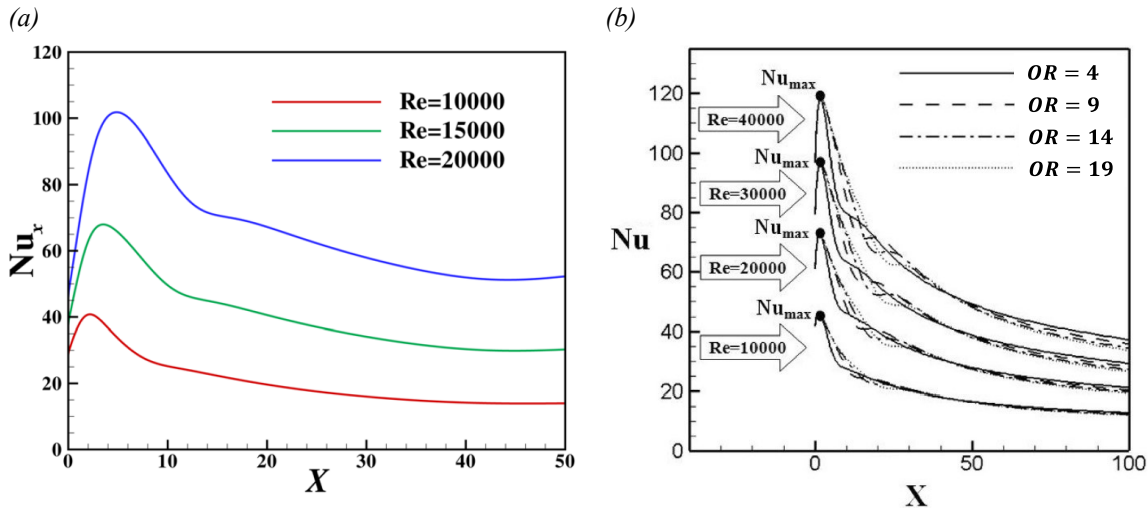


Figure 3.14: Variation in the Nu_x profile with respect to Re for a dual jet as per (a) Mondal *et al.* [22] ($OR = 1$) and (b) Hnaien *et al.* [84] ($4 \leq OR \leq 19$) by means of the $k - \varepsilon$ and $k - \omega$ turbulence models, respectively.

The effect of the jet Reynolds number on the Nu_x profile for a dual jet flow was also analysed by Singh and Dewan [24] using the SST $k - \omega$ turbulence model, with a view to gaining a better understanding of the transient heat transfer characteristics associated with the unsteady flow field. Despite this, the local Nusselt number data was presented as time-averaged quantities only, where the Nu_x profile acquired for each Re is shown in Figure 3.15(a). Similar to previous studies, increasing Re was found to increase Nu_x at all downstream locations, however, the trend associated with the Nu_x profile was distinctly different from that of any other study previously discussed. In this case, the maximum Nu_x value was obtained at the nozzle exit, which proceeded to decline over the remaining downstream distance. The Nu_x profiles were compared to that of Vishnuvardhanarao and Das [28] in Figure 3.15(a), where poor agreement between the datasets was attributed to the use of a different turbulence model in each case, *i.e.*, SST $k - \omega$ and standard $k - \varepsilon$ respectively.

The remaining investigations which examine the heat transfer characteristics of dual jet flows vary parameters which lie outside the scope of this study, which is similar to that previously observed during the review of the hydrodynamic literature. They are therefore just discussed in brief to highlight any potential findings of interest. First, as aforementioned in section 3.1, Hnaien *et al.* [10] investigated the effect of varying the wall inclination angle β , where a sample set of Nu_x profiles acquired are presented in Figure 3.15(b) for $Re = 15,000$. Similar

to Mondal *et al.* [22], Nu_x and $\overline{Nu_x}$ were found to increase linearly with increasing Reynolds number for all β examined, where the rate of increase appeared to decrease with higher β values, and a single peak was observed in the Nu_x profile for $\beta = 0$. As β was increased, a general decline was observed in Nu_x , however, the peak value of the Nu_x profile appeared to remain constant for $\beta \geq 15$. Furthermore, increasing β beyond 15 brought about the emergence of a local minimum value in advance of the Nu_x peak, the severity of which was greater for higher Reynolds numbers.

The study of Rathore *et al.* [6] was the first to introduce a temperature difference between the wall jet and the offset jet. For jets with equal velocities and $OR = 2$, the Richardson number ($Ri = Gr/Re^2$) was varied from 0.0028 to 0.875 by modifying the offset jet temperature, simulating Grashof numbers (Gr) equal to 1000, 10,000 and 35,000 for $Re = 200, 400$ and 600. Mixing and heat transfer between the offset jet and wall jet was seen to increase with an increase in Ri as buoyancy effects become more apparent, while the size of the recirculation region decreased. Assoudi *et al.* [26] subsequently examined the effect of varying this temperature ratio with respect to changing OR and V_r using the RSM turbulence model. In doing so, the locations of the merge point and combined points were found to increase with both OR and V_r due to the growth of the recirculation region, while adjusting the temperature ratio did not affect X_{mp} . Assoudi *et al.* [26] also noted that the rate of streamwise temperature decay increased with decreasing OR . The temperature variation within the dual jet flow field was also examined by Kumar *et al.* [23] when subject to a constant heat flux of $10,000 \text{ W/m}^2$ in the bounding wall, where OR was varied from 3 to 9 and $5,000 \leq Re \leq 20,000$. In doing so, the fluid temperature within the recirculation region was found to increase with increasing OR .

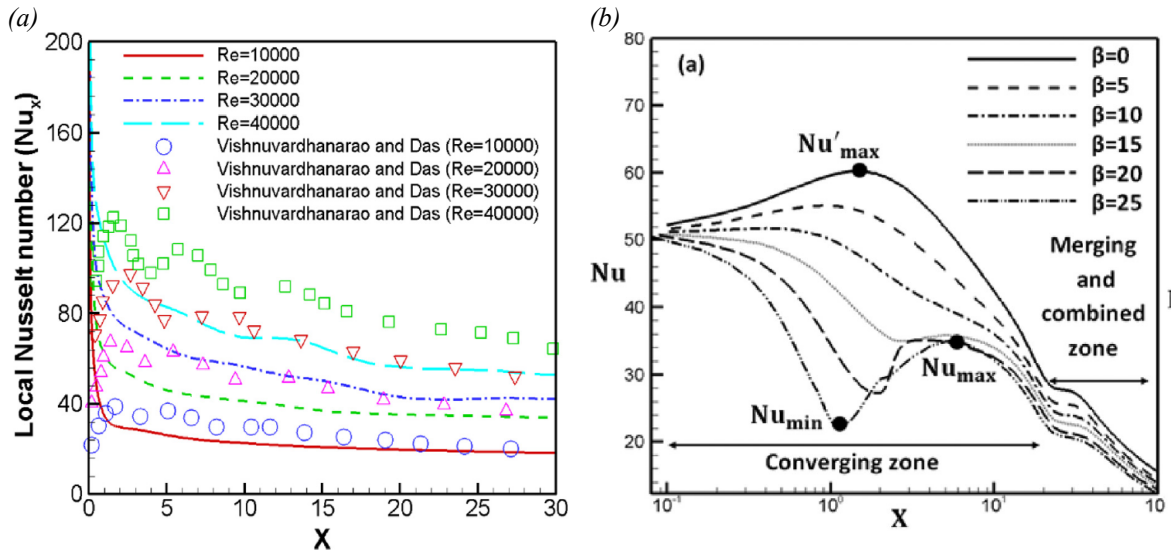


Figure 3.15: Variation in the Nu_x profile for a dual jet with respect to (a) Re , as per Singh and Dewan [24] by means of the $k - \varepsilon$ turbulence model for $OR = 1$ and (b) β , as per Hnaien *et al.* [10] by means of the $k - \omega$ turbulence model for $Re = 15,000$ and $OR = 8$.

As previously discussed, Singh *et al.* [87] analysed the heat transfer enhancement acquired through replacing the bounding wall with a wavy surface. The resulting Nu_x profile is presented in Figure 3.16(a), where a series of peaks were observed to coincide with the high points of the wavy surface. Two more distinct peaks were evident near the jet exit, even for lower wall amplitudes, which was similar to that reported by Kumar [10]. The amplitude of these peaks was found to increase with the amplitude of the wavy surface, while the streamwise location of both peaks decreased accordingly. Further downstream, the Nu_x profile followed the wavy surface. The average Nusselt number across the wall was found to increase with increasing wavy surface

amplitude until a dimensionless value of 0.5, beyond which it appeared to decrease. The maximum value for $\overline{Nu_x}$ was hence observed at an amplitude of 0.5, which equated to a 12% increase from that achievable using a plane wall. The enhancement in the heat transfer through the use of a wavy bounding wall is further investigated in studies [88, 98, 99, 101, 107], where a wider parameter range is explored, such as the effect of OR , Re , amplitude and the number of cycles.

Mondal *et al.* [102] and Hnaïen *et al.* [103] also considered the effect of the addition of a parallel co-flow on the heat transfer characteristics of a dual jet flow. A sample Nu_x profile acquired by Mondal *et al.* [102] is shown in Figure 3.16(b), where Nu_x was observed to decrease along the converging region, rise towards a maximum value inside the merging region, and decline over the remaining downstream distance in the combined region. The co-flow velocity was to only affect the Nu_x values inside the merging region, wherein an increase in the co-flow velocity negatively impacted Nu_x . Hnaïen *et al.* [103] also noted that an increase in the co-flow ejection angles increased Nu_x values primarily along the merging and combined regions.

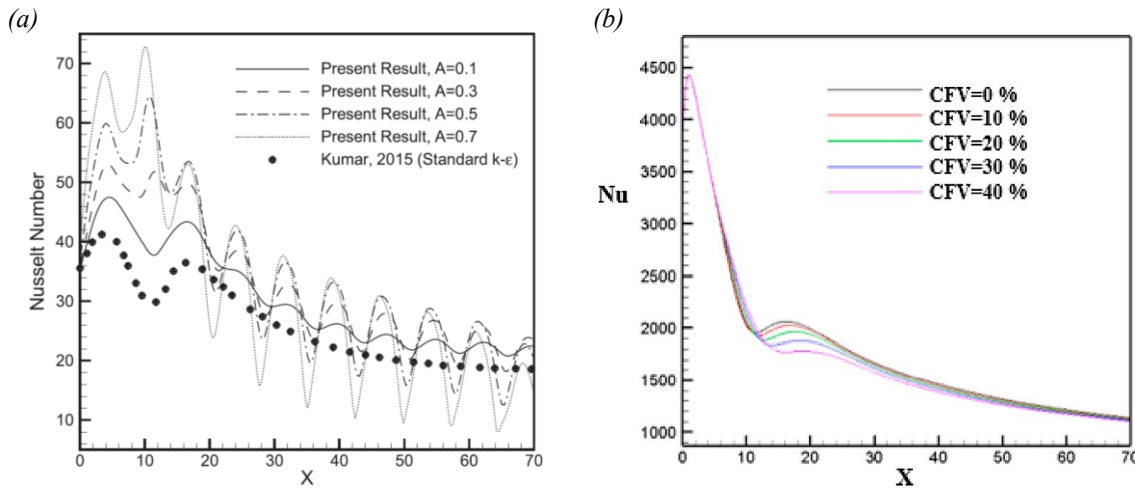


Figure 3.16: Variation in the Nu_x profile for a dual jet with $Re = 15000$ with respect to (a) varying amplitude, as per Singh *et al.* [87] by means of the $k - \epsilon$ turbulence model for $OR = 6$ and (b) varying co-flow velocity, as per Mondal *et al.* [102] by means of the $k - \omega$ turbulence model for $OR = 8$.

Finally, Rathore and Verma [108] studied the effect of the solid wall motion on the thermal characteristics of a dual jet for $4 \leq OR < 12$ and $10,000 \leq Re \leq 25,000$ using the $k - \epsilon$ turbulence model. The resulting Nu_x profiles for a stationary wall contained two peaks, similar to Kumar [10], but reduced to a single peak for certain wall velocities. Increasing the solid wall boundary reduced $\overline{Nu_x}$ while the wall velocity was less than $u_{jet}/2$, however further increasing the solid wall velocity beyond this point causes $\overline{Nu_x}$ to rise.

3.3.Summary

This literature review discussed the reported findings from all relevant dual jet studies published to date, a summary of which can be found in Table 3.1 alongside the range of parameters investigated. As a collective, the results show that the behaviour of a dual jet is strongly influenced by Re , OR , and V_r , while other parameters can also be used to alter the resulting flow field. While consistencies can be identified between specific studies regarding the overall trends observed, a number of discrepancies are also evident across the published literature, as well as the scope to further explain some of the observed behaviour.

With regard to the time-averaged flow structure of a dual jet, all of the reviewed studies appear to be in agreement regarding the effect of OR on the positioning of mp , UVC and LVC , where the growth of the

recirculation region with increasing OR moved all three flow features further downstream. The position of the combined point with increasing OR is considerably less defined, where a continuous downstream movement was noted in studies such as Hnaïen *et al.* [25] and Assoudi *et al.* [26], however, Kumar [27] noted an initial decline in X_{cp} until $OR = 5$, beyond which it began to increase. With regard to changing Re , both Kumar *et al.* [23] and Mondal *et al.* [22] noted a change in X_{mp} with varying Re , but Singh and Dewan [24] appeared to suggest that these values were uncorrelated. In addition to this, the locations of X_{mp} and X_{cp} appear to vary considerably throughout the literature for a similar set of jet parameters, where Assoudi *et al.* [26] noted the observed values for X_{cp} that were noticeably less than those provided by both Kumar [27] and Mondal *et al.* [20]. Further to this, the values obtained across all numerical studies are consistently located further downstream when compared to the experimental data of Wang and Tan [19], particularly in the case of the combined point. Finally, as the transverse locations of each flow feature were often not considered, insufficient data is available to corroborate the trends observed.

While a few studies investigated the effect of the velocity ratio on the locations of various dual jet flow features, it is interesting to note that these studies focussed on the use of a single Reynolds number, meaning the observed trends are not verified for a wider range of flow rates. However, the results regarding the velocity ratio seemed consistent across the relevant studies; as V_r increases, the flow behaviour shifted from that of an offset jet to a wall jet and the recirculation zone gradually diminished. Having said this, only Hnaïen *et al.* [25] investigated the effect of the velocity ratio on the locations of the key dual jet flow features. In a separate study, Hnaïen *et al.* [85] documented the presence of a critical downstream distance before which the wall jet dictated the maximum jet velocity, whereas the offset jet primarily influenced the value of U_m beyond this point. However, no further dual jet investigations sought to verify these findings.

Many of the published studies examined the decay of the maximum velocity with downstream distance. It was noted that the decay of u_m began when the potential core was consumed and the rate of decay in the case of a dual jet was greater than that of any other jet type discussed in this report. However, Kumar [83] observed a local peak in the maximum velocity just beyond the potential core when $OR = 2$. A similar finding was also evident in that published by Mondal *et al.* [96] using the same offset ratio. However, no explanation was provided for such behaviour, and no subsequent studies reiterated this finding.

One of the most interesting findings regarding dual jet flow was the existence of an unsteady range of OR and V_r values which produce a von Kármán-like shedding of vortices and a periodic interaction between both jets. Outside of this range, two counter-rotating vortices were found to exist within the recirculation region, the size of which generally depended on the offset ratio. Having said this, very few studies focussed on the onset of such behaviour, and, of those which did, only a single Reynolds number was examined. For example, Mondal *et al.* [20] observed that for $Re = 10,000$, von Kármán-like vortex shedding occurred when $0.7 \leq OR \leq 2.1$. However, Zhao and Sun [89] noticed that this type of behaviour occurred when $8 \leq OR \leq 9$ for $Re = 56$, indicating a clear dependence on the Reynolds number of the flow and the onset of instabilities, where Zhao and Sun [89] was the only study which to concern itself with a Reynolds number less than 1000.

Significantly fewer studies were concerned with the heat transfer characteristics of dual jet flow, all of which presented their respective heat transfer findings in terms of time-averaged quantities, which is interesting considering the potential instabilities associated with dual jet flows. In doing so, the Nu_x profiles acquired for a dual jet flow varied considerably throughout the relevant studies, with regard to both the overall trend and

the observed Nu_x magnitudes, making it difficult to draw comparisons between the respective datasets. While some studies such Mondal *et al.* [22] and Hnaïen *et al.* [84] noted the occurrence of a single peak in the Nu_x profile, both Kumar [83] and Singh *et al.* [87] acquired two distinct Nu_x peaks and Singh and Dewan [24] obtained none. In each case, the cause of the Nu_x peak is vaguely attributed to the merge point. However, no further explanation or description of the near wall flow physics at this point that could induce this heat transfer behaviour has ever been provided. Without plausible explanation, it is not possible to ascertain which Nu_x profile is the best representation of the actual dual jet heat transfer characteristics.

While there is a general agreement between all studies regarding an increase in both Nu_x and $\overline{Nu_x}$ with an increase in the Reynolds number, the effect of OR is somewhat contested. While Farooq and Das [29] noted a consistent increase in Nu_x , both Kumar [83] and Hnaïen *et al.* [84] found Nu_x only increased in the region closest to the jet exit, while a decline was observed further downstream. The effect of V_r on the heat transfer characteristics was only conducted by Vishnuvardhanarao and Das [28], meaning the reported findings are yet to be verified, however, little explanation was provided for the trends observed. In addition, as the mass flow rate of the dual jet was not maintained at a constant value while varying V_r , the Nu_x profiles acquired do not necessarily allow for a fair comparison. In this case, it is difficult to differentiate between changes to the Nu_x profile that arise as a result of varying V_r or inevitable changes due to increasing mass flow rate. Further investigation is therefore required regarding the effect of V_r on the Nu_x profile induced by a dual jet.

For the most part, the discrepancies in the published literature discussed thus far can be attributed to a severe lack of credible experimental data relating to dual jet flows to validate the numerical findings. As a result, the suitability of the turbulence models used to conduct each numerical dual jet investigation is brought into question, where the use of numerical modelling in previously published dual jet studies is discussed in further detail in section 7.2. While only a single experimental study has been conducted to date focussing on the flow behaviour associated with dual jets, where only a single set of dual jet flow parameters were considered, there have been no experimental investigations directed towards the heat transfer aspect, aside from the published studies linked to this thesis. This is an interesting finding, especially considering the extensive use of dual jets in industry for cooling purposes, yet their behaviour still remains relatively unknown. To accommodate for this, many numerical investigations have validated their numerical models through comparison with the experimental results from single offset jet studies. However, as mentioned by many of the published studies discussed, the flow field associated with a dual jet varies greatly from that of a single offset jet. The ability of a numerical model to produce results consistent with that of a single offset jet does not necessarily imply its suitability for dual jet flows. A gap therefore exists in the published literature regarding the provision of verified, repeatable experimental and numerical data relating to the flow and heat transfer behaviour of a dual jet over a wide range of jet parameters.

3.3.1. Research Objectives

Based on these findings, the following objectives were outlined for the present investigation:

- To conduct an experimental investigation which focusses on the heat transfer aspects of dual jet flow when subject to a uniform wall flux (UWF) boundary condition for varying Reynolds number, offset ratio, and the velocity ratio, *i.e.* $5500 \leq Re \leq 12000$, $1 \leq OR \leq 7$ and $0.5 \leq V_r \leq 2$.
- To use particle image velocimetry (PIV) techniques to affiliate the heat transfer findings with specific dual jet flow features through experimental means.

- To develop a validated numerical dual jet model that produces results consistent with the experimental findings and improves on the dual jet numerical modelling techniques available in the literature.

Table 3.1: Summary of main findings of dual jet studies from the published literature. Note that OR and V_r values marked with a * have been converted to align with the definition used in the current study, i.e., $OR = d/w$ and $V_r = u_w/u_o$. (mp = merge point, cp = combined point, LVC = lower vortex centre, UVC = upper vortex centre, PIV = particle image velocimetry, LES = Large eddy simulation, TBC = thermal boundary condition, ADB =adiabatic, UWT = uniform wall temperature, UWF = uniform wall flux)

Authors	OR	V_r	Re ($\times 10^3$)	Method	TBC	mp	cp	LVC	UVC	Main Findings
Present Study	1 – 7	0.5 – 2	5.5 – 12	IR Therm- ography & PIV	UWF	–	–	–	–	
Published Experimental Studies										
Wang and Tan [19]	1	1	10	PIV	ADB	(0.75, –)	(6.4, –)	–	–	- von Kármán vortex shedding with $St = 0.25$. - Self-similarity at $X = 15$.
Published Numerical Studies										
Li <i>et al.</i> [2]	1	1	10	LES	ADB	–	–	–	–	- Self-similarity at $X = 10$.
Li <i>et al.</i> [92]	0.5 – 8	0.25 – 4*	10	$k - \varepsilon$	ADB	–	–	–	–	- As V_r increases, flow pattern shifts from offset jet to wall jet and recirculating region gradually diminishes.
Kumar and Das [1]	8*	1	20	$k - \varepsilon$	ADB	(10.5, 2.4)	(18.4, 2.8)	–	–	- Two stable, counter-rotating vortices in recirculation region.
Mondal <i>et al.</i> [20]	0.5 – 2.5	1	10	$k - \varepsilon$	ADB	(1.4, –) – (1.9, –)	(8.3, –) – (36.3, –)	–	–	- Von Kármán vortex shedding for $0.7 \leq OR \leq 2.1$, two stable, counter-rotating vortices outside of this range. - St decreases with OR .
Singh and Dewan [24]	1*	1	10 – 40	SST $k - \omega$	UWT	(1.5, 1.4) – (1.4, 1.5)	–	–	–	- X_{mp} independent of Re . - Self-similarity at $X \geq 30$.
Kumar [83] & Kumar [27]	2 – 14*	1	15	$k - \varepsilon$	ADB & UWT	(3, 1.7) – (16.3, 3.1)	(16.3, 1.7) – (24.1, 3.8)	(1.2, 1.4) – (8.7, 2.9)	(1.3, 2.4) – (10.6, 8.6)	- As OR increases, UV size increases with increasing OR and LV pushes UVC up. Both UVC and LVC move axially downstream - Secondary Nu_x peak for $OR > 2$.
Kumar <i>et al.</i> [23]	3 – 9	1	5 – 20	$k - \varepsilon$	UWF	–	–	–	–	- Increasing OR increases intensity of UV . - Increasing Re reduces X_{mp} .

Mondal <i>et al.</i> [96]	1	0.5 – 2	10	$k - \varepsilon$	ADB	(1.3, 1.6) – (1.2, 1.3)	–	–	–	- <i>Periodic vortex shedding for $0.78 \leq V_r \leq 1.34$, two stable, counter-rotating vortices outside of the range.</i> - <i>St decreases with V_r.</i>
Mondal <i>et al.</i> [21]	1	1	10	$k - \varepsilon$	ADB	(1.5, –) – (1.6, –)	(11.5, –) – (18.7, –)	-	-	- As wall jet nozzle width h increased, St decreases for $0.3 \leq h/d \leq 1$ and constant for $h/d > 1$.
Mondal <i>et al.</i> [97]	0.5 – 3.33*	1	10	$k - \varepsilon$	ADB	(1.4, –) – (1.9, –)	(8.3, –) – (36.3, –)			- For $w/d \leq 0.5$, two steady counter rotating vortices. - For $0.6 \leq w/d < 1.6$, periodic vortex shedding, St decreasing. St constant for $w/d \geq 1.6$.
Zhao and Sun [89]	3 – 9*	1	56×10^{-3}	BGK	ADB	(3.9, 2.4) – (5.7, 5.6)	(9, 1.9) – (12.7, 5.5)	–	–	- For $OR \leq 8$, X_{mp} and X_{cp} increase with increasing OR but decrease at $OR = 9$. - Evidence of vortex shedding at $OR \geq 9$.
Hnaïen <i>et al.</i> [25]	4 – 14	0.5 – 1	15	$k - \omega$	ADB	(3.8, 2.4) – (14.9, 16)	(4.9, 7.4) – (15, 24.5)	(3.4, 2.2) – (7.5, 3.3)	(4.3, 5) – (9.8, 9.1)	- Increasing OR increases both the transverse and longitudinal positions of mp , cp , UVC and LVC , less obvious for higher V_r . - Increasing V_r moves mp , cp , UVC and LVC upstream, while bringing just mp and LVC closer to the solid wall.
Assoudi <i>et al.</i> [26]	1 – 10	0.57 – 1	15	RSM	ADB	(1.4, –) – (12.3, –)	(11.2, –) – (16, –)	–	–	- Dual jet decays faster than SOJ in the near region. - Increasing OR increases UV and X_{mp} while suppressing LV .
Hnaïen <i>et al.</i> [85]	8	1	15	$k - \omega$	ADB	–	–	–	–	- Offset jet and wall jet, respectively, dictate u_m before and after critical location X_{cr} that depends on wall inclination. - u increase just downstream of the nozzle exit, magnitude of increase reduces with increasing wall inclination.
Hnaïen <i>et al.</i> [86]	8*	1	15	$k - \omega$	ADB	(4.8, 2.1) – (17.1, 3.2)	(13.9, 2.1) – (23.5, 3.6)	(1.9, 1.8) – (7.2, 3.2)	(2.3, 3.7) – (9.1, 9.3)	- Increasing the offset jet ejection angle moves mp , cp , UVC and LVC upstream and accelerates the merging process.

Singh <i>et al.</i> [87]	6*	1	15	$k - \varepsilon$	ADB & UWT	(8.2, 2.1) – (7.7, 2.2)	(15.8, 2.5) – (14.4, 2.6)	(4, 1.9) – (5.4, 1.3)	(5.1, 4.5) – (4.6, 4.3)	- Wavy surface increases pressure gradient. - Increasing amplitude increases Nu_x to a certain point, decreases thereafter.
Mondal <i>et al.</i> [102]	4 – 10*	1	15	$k - \omega$	UWT	(5.3, –) – (16.3, –)	(13.4, –) – (37.7, –)	(3, 2.1) – (3.6, 2.2)	(3.6, 5) – (4, 5.3)	- Increasing co-flow velocity negatively affects $\overline{Nu_x}$, with most noticeable influence in the merging region.
Hnaïen <i>et al.</i> [103]	1	4	15	$k - \omega$	UWT	–	–	–	–	- Increasing co-flow ejection angle reduces size of UV and LV and increases Nu_x in merging region.
Hnaïen <i>et al.</i> [104]	4 – 10*	1	15	$k - \omega$	ADB	(7.6, 2.4) – (9.6, 2.7)	(15.4, 2.4) – (24.3, 3)	(3, 2.1) – (3.6, 2.22)	(3.6, 5) – (4, 5.3)	- Increasing co-flow velocity decreases size of UV and moves mp , cp , UVC and LVC downstream and further from the wall.
Vishnuvardhanarao and Das [28]	1	0.25 – 4*	10 – 40	$k - \varepsilon$	UWT & UWF	–	–	–	–	- Evidence of periodic vortex shedding in Nu_x profile. - Singular prominent Nu_x peak.
Farooq and Das [29]	2 – 3	1	15	$k - \varepsilon$	UWT & UWF	–	–	–	–	- Nu_x peak at mp that moves downstream with OR .
Mondal <i>et al.</i> [22]	1	1	10 – 20	$k - \varepsilon$	UWT	–	–	–	–	- mp moves downstream with Re - Nu_x affected by Re and Pr
Hnaïen <i>et al.</i> [84]	4 – 19*	1	10 – 40	$k - \omega$	UWF	–	–	–	–	- Increasing OR increases Nu_x towards that of SWJ near jet exit, decreases thereafter. Effect more obvious for higher Re .
Hnaïen <i>et al.</i> [10]	8 *	1	15 – 40	$k - \omega$	UWT & UWF	–	–	–	–	- Increasing the wall inclination decreased Nu_x .
Rathore and Verma [108]	4 – 12*	1	10 – 25	$k - \varepsilon$	UWT & UWF	–	–	–	–	- Increasing solid wall boundary velocity U_b decreases Nu_x if $U_b/U_{jet} < 1$. Further increasing U_b increases Nu_x . - U_m increases at each X for increasing U_b if $U_b/U_{jet} \geq 1$

Chapter 4

Experimental Investigation

This chapter provides an in-depth discussion of the experimental setup and methodologies employed to investigate the transfer of heat to a dual jet flow, along with a description of the required calibration procedures and uncertainty analysis of the measured parameters. The experimental investigation consists of a two-part study; an initial heat transfer study of a dual jet flow subject to a uniform wall flux (UWF) boundary condition, followed by a flow visualisation study to understand the underlying flow phenomenon driving the heat transfer behaviour. This required the design and manufacture of a purpose-built experimental rig.

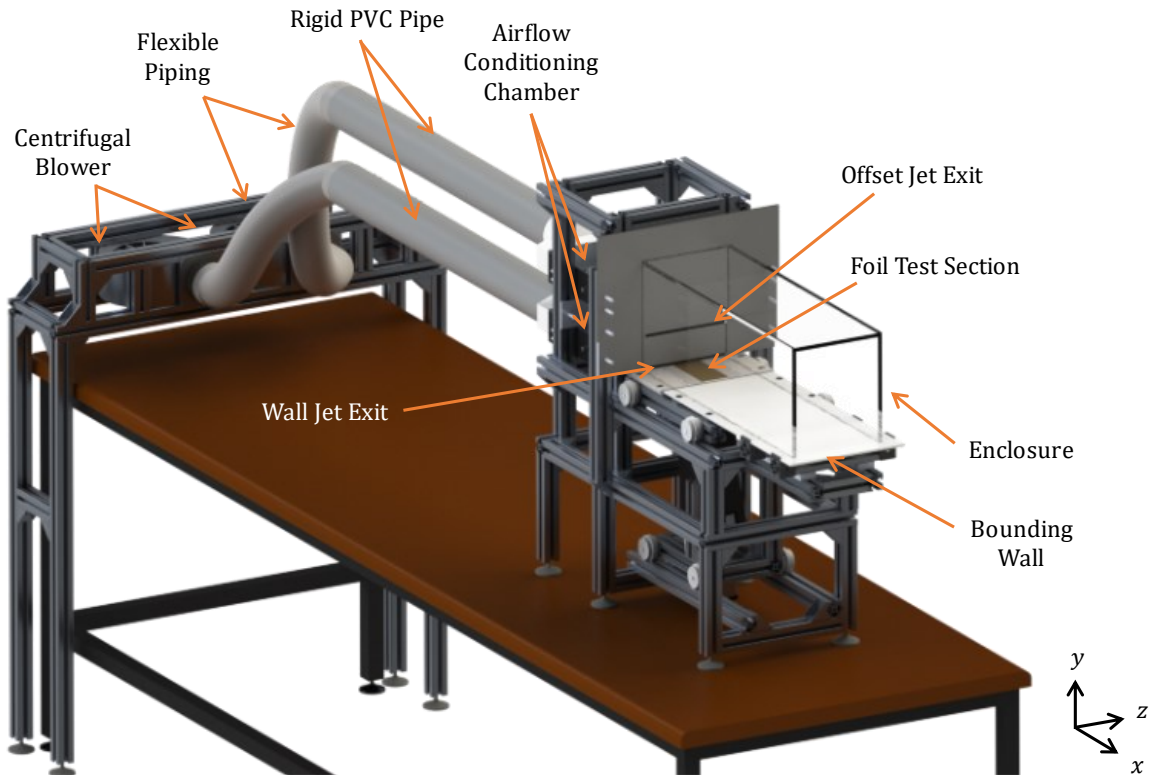
4.1. Experimental Rig Setup

The experimental rig can be seen in Figure 4.1 and consists of a wall jet and an offset jet alongside a solid bounding wall. The downstream flow is contained within a smooth, transparent enclosure. The base experimental setup, which includes the rig support structure, the jet configuration, the airflow supply and its associated flow conditioning, remained consistent for both the heat transfer and flow visualisation studies. The solid bounding wall was adapted for each study to suit the chosen measurement technique; however, it was ensured that the surface smoothness and dimensions remained constant throughout. The rig design is hence discussed in three stages in this section: 1) the jet nozzle design and airflow conditioning, 2) the thermography system for the heat transfer study, and 3) the particle image velocimetry (PIV) setup for the flow visualisation study.

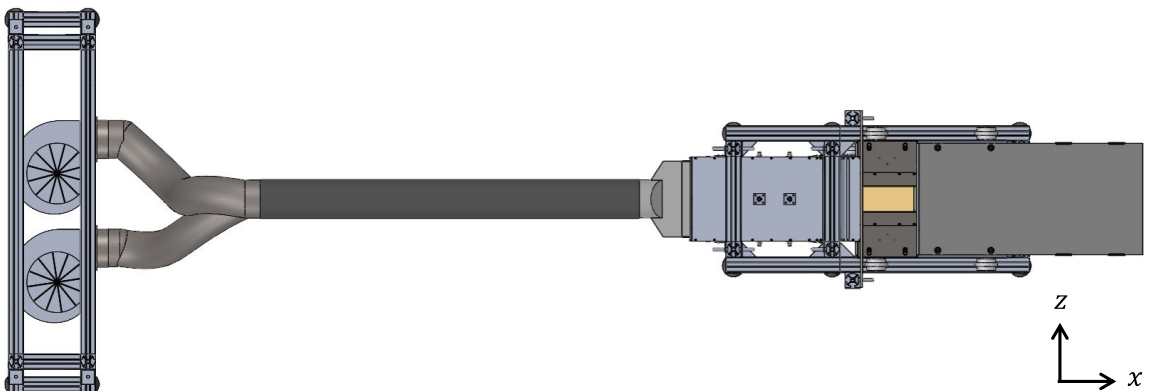
4.1.1. Airflow Measurement and Conditioning

This section outlines the nozzle design and necessary flow conditioning for the experimental dual jet setup, along with the chosen method of flowrate measurement, where air was adopted as the working fluid. As shown in Figure 4.1, airflow to each jet is provided by separate AXAIR GREG9 160x62R 230/1/50/60 centrifugal forward-curved single-inlet blowers, where the use of centrifugal blowers were recommended by Mehta and Bradshaw [109] due to their efficiency over a wide range of flow conditions and relatively low vibration generation. To mitigate the transmission of vibrations throughout the experimental setup, the blowers were mounted on a standalone structure and connected to the downstream components by means of flexible hosing. The insertion of an additional 1 m length of rigid PVC pipe of 100 mm inner diameter further diminished instabilities induced to the flow by the blowers in advance of the test region and ensured that the downstream flow remained unaffected by upstream conditions.

(a)



(b)



(c)

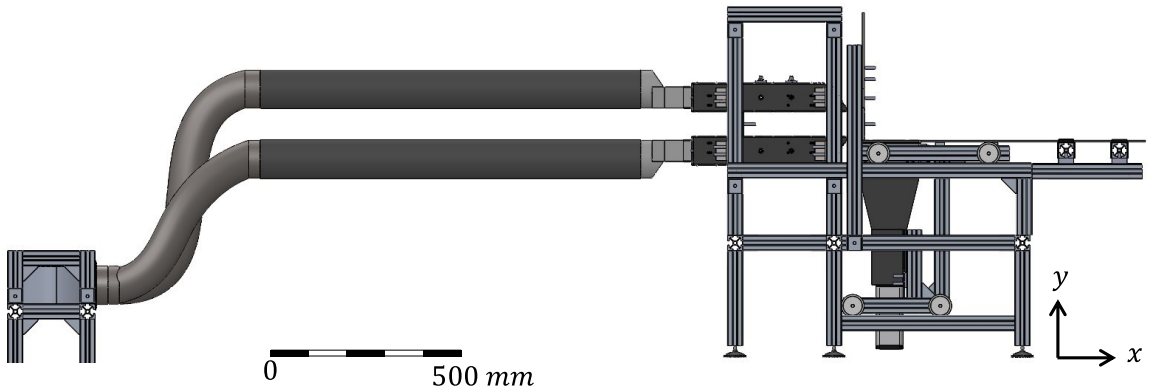


Figure 4.1 Experimental Rig Setup. (a) Isometric view, (b) plan view, and (c) front elevation view.

Similar to several published studies of dual jets and parallel jets, a rectangular slot nozzle shape was adopted for each jet. A jet width of $w = 7 \text{ mm}$ was chosen as this was considered to be the maximum allowable size when the resulting propagated rig dimensions were considered with respect to laboratory space constraints. Each jet nozzle was incorporated in the airflow conditioning chamber design, as shown in in Figure 4.1, inside which the necessary flow conditioning operations took place. Each airflow chamber had a rectangular cross-section and was connected to the PVC pipe using a round-to-rectangular diffuser.

The airflow conditioning chamber design, and arrangement as per in Figure 4.1, allowed for a minimum offset ratio of 1 when the material dimensions were accounted for. The offset ratio was hence made adjustable by means of interchangeable offset plate inserts manufactured from 6 mm thick aluminium plate, as shown in Figure 4.2. Each nozzle plate insert was designed to completely fill the space between the jets, plus an additional 121 mm on either side of the jets to prevent abnormal recirculation behaviour at the jet edges. Similarly, an additional plate of height 250 mm was inserted above the offset jet to make contact with the upper surface of the transparent enclosure. In total, 4 distinct offset plate inserts were made to achieve the experimental offset ratio range of $1 \leq OR \leq 7$. A 1 mm aluminium sheet provided sufficient spacing in the case of $OR = 1$ due to the 3 mm thickness contribution from each airflow conditioning chamber.

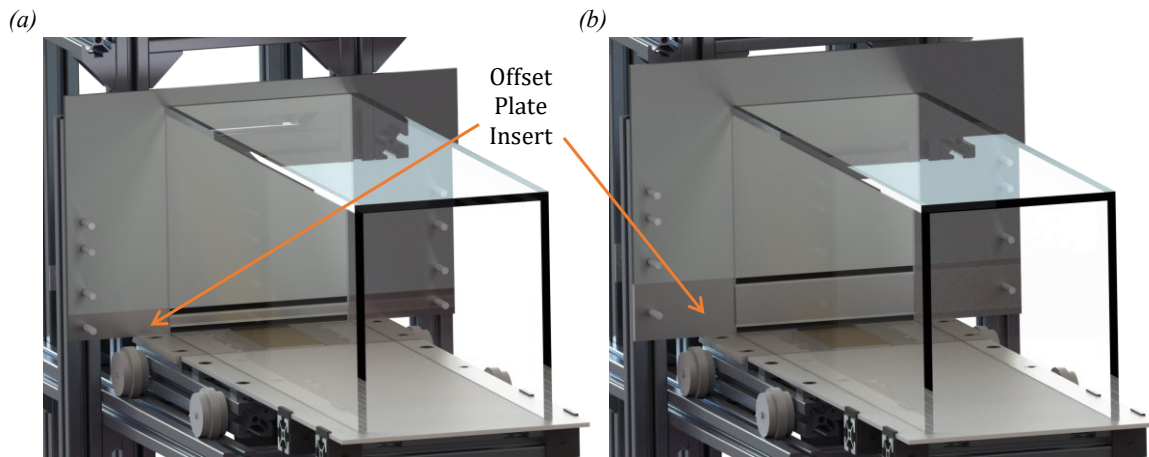


Figure 4.2: Dual jet offset ratio changed from (a) $OR = 2$ to (b) $OR = 7$ by means of interchangeable offset plate inserts

The depth of the airflow conditioning chamber, which corresponds to the jet length l , directly affects the planarity of the jet flow. The ideal flow scenario would ensure 2D flow behaviour along the test region of interest about the jet centreline, *i.e.*, $z = \pm 3w$, and be minimally affected by the presence of the jet side walls. The jet aspect ratio, defined as $AR = l/w$ and used in previous experimental planar jet studies, varies throughout the literature, however both Nasr and Lai [76] and Wang and Tan [19] show that a jet aspect ratio of 30 is sufficient to ensure predictable 2D behaviour along the mid-plane of the flow domain and negate edge effects in this region. A jet aspect ratio of 30 was thus adopted for the present investigation, resulting in a jet length of $l = 210 \text{ mm}$. In addition to this, Nasr and Lai [76] reported that the use of side walls inside the jet flow domain can help mitigate jet spreading effects and further promote a planar flow regime. As the focus of Nasr and Lai [76] was single offset jets, the use of side walls allowed the pressure inside the recirculation region to reach its intended minimum value and prevented pressure ‘leakage’ at the slot jet length extremities. A similar approach was hence adopted in the current study through the addition of a smooth-walled transparent enclosure over the flow domain. As shown in Figure 4.1, the side walls of this enclosure were aligned with

side walls of the jet nozzle exits. The internal enclosure height of 250 mm was selected based on Wang and Tan [19], who adopted a water level height of $35w$, while ensuring that it did not influence the flow field in the vicinity of the dual jet.

4.1.1.1. Flow Conditioning

The primary intention of the airflow conditioning chambers was to ensure a uniform jet velocity and minimise turbulent fluctuations across the entire cross section of the nozzle exit; hence any resulting flow behaviours can be attributed to the interaction of the offset and wall jets rather than flow abnormalities emerging from the nozzle itself. In addition, encouraging uniform flow at the nozzle exit would allow for the data to be collected and processed from a 2-dimensional perspective, thus reducing the overall complexity of the postprocessing procedures. The airflow conditioning chambers were designed using the approach outlined by Mehta and Bradshaw [109], who provided an extensive list of rules for the design of low speed wind tunnels, where the airflow chambers used in this investigation are comparable to the open circuit blower tunnel described. This approach was also adopted by Jackson [110], whose flow conditioning and nozzle contraction design provided much of the basis for the present investigation. The final flow conditioning configuration is presented in Figure 4.3, where the airflow conditioning chamber design associated with both the wall jet and the offset jet were identical but inverted with respect to one another. The final singular airflow chamber design can be seen in Figure 4.4, which provides both a complete view of the airflow chamber and an internal view of the airflow conditioning components.

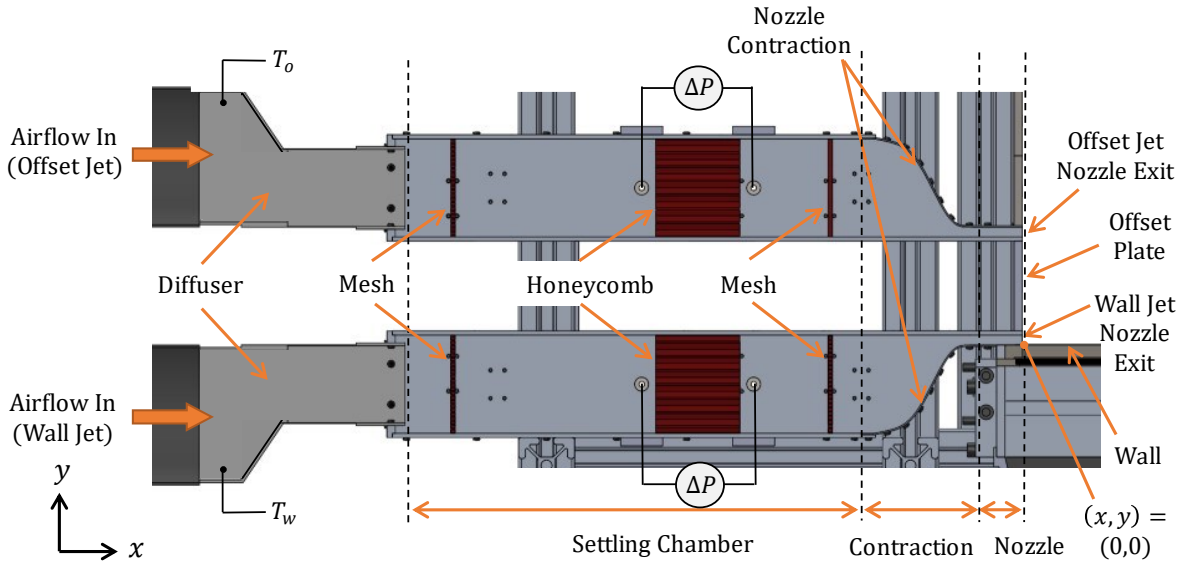


Figure 4.3: Schematic of airflow chamber design

The airflow conditioning chamber design consisted of 3 streamwise regions: a settling chamber, a contraction region, and the nozzle exit. The purpose of a settling chamber is to create a low velocity region upstream of the working section, *i.e.*, the nozzle exit, wherein the flow conditioning components can be placed while minimising their induced pressure drop [109]. This is achieved by increasing the cross-sectional area of the flow domain, where the use of a contraction between the settling chamber and the working section can subsequently decrease the cross-sectional area and accelerate the flow to the required test velocity. The use of a contraction in advance of the working section also acts to further reduce the mean velocity fluctuations present in the flow emerging from the settling chamber. The contraction ratio, CR , is the ratio of the settling

chamber cross sectional area to that of the working section. The use of a larger contraction ratio can prove advantageous through a greater reduction in velocity fluctuations, thus producing a better flow quality in the working section, however this can be accompanied by higher running costs and therefore prudence is advised [109]. For the present investigation, a contraction ratio of $CR = 10$ was chosen for the settling chamber, which should lead to a $\times 10$ times acceleration in the mean streamwise velocity compared to that of the settling chamber, which is accompanied by a corresponding decrease in both the streamwise and transverse velocity variation. For a 2D contraction, the contraction ratio is simplified to the ratio of the settling chamber height h_s to that of the working section, which, for this investigation, is the jet width w , such that $CR = h_s/w$. Therefore, for the chosen jet width of $w = 7 \text{ mm}$, a settling chamber height of $h_s = 70 \text{ mm}$ was required to ensure $CR = 10$.

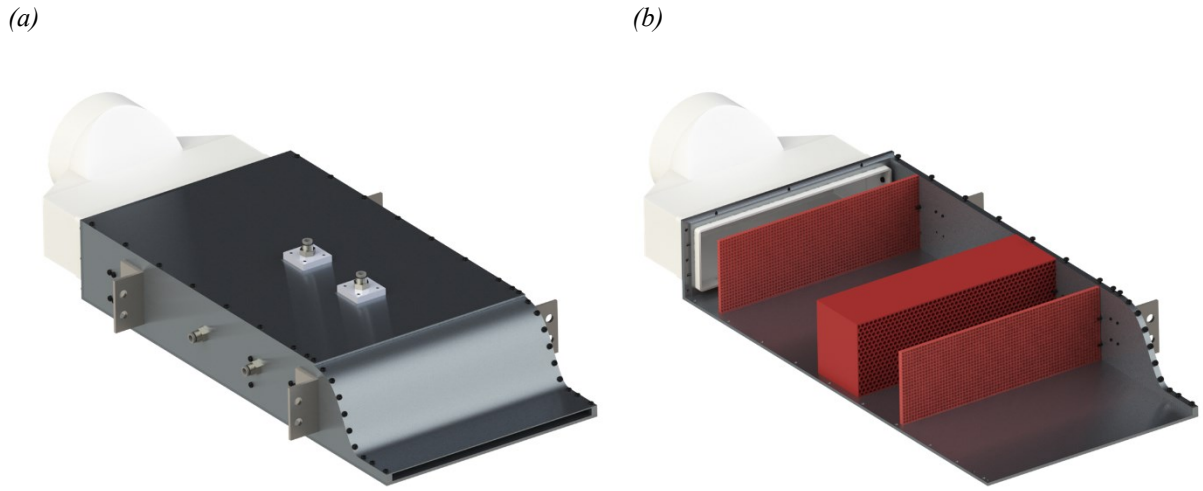


Figure 4.4: Airflow chamber design for each jet, (a) complete view (b) internal view of airflow conditioning components

Regarding the flow conditioning components, it is generally recommended to use a honeycomb in combination with a series of mesh screens which act to reduce the turbulence intensity in the flow by means of an intentional pressure drop [110]. A honeycomb flow straightener is a device which consists of series of passageways, *i.e.*, cells, aligned with the axial direction of the bulk fluid flow, where the cross-sectional shape of each cell is generally circular, square, or hexagonal [111]. The use of honeycombs has been shown to effectively reduce swirl, turbulent length scales, and lateral mean velocity variations, as the cells suppress cross velocity components in the flow and break down large scale eddies [111, 112]. It was decided to adopt a hexagonal cross-sectional cell shape for the present investigation, as Amirnordin *et al.* [113] demonstrated that a hexagonal cell can result in a pressure drop that is up to 43% less when compared to its square counterpart. The length of each cell side was established as 2 mm with a wall thickness of 1 mm , resulting in a hydraulic diameter of 3.46 mm . The downstream length of the honeycomb also plays a significant role in the resulting fluid flow, where, although a deeper honeycomb will perform better, increasing the cell length leads to an increased pressure drop across the honeycomb [112]. A downstream length of 60 mm was thus chosen for the honeycomb flow straightener, as this value induced sufficiently measurable induced pressure drop, which is the intended means of flow rate measurement, without exacerbating downstream turbulence conditions.

However, honeycomb flow straighteners do not effectively reduce the longitudinal components of turbulence in the flow and, due to the development of a shear layer along the cell length, can shed small scale turbulence

downstream of their insertion point [114]. The use of a mesh screens in conjunction with a honeycomb flow is hence recommended, which can improve the uniformity of the velocity profile and reduce the boundary layer thickness by means of inducing a static pressure drop proportional to the square of the flow speed [109]. Mesh screens are primarily characterised by their pressure drop coefficient K , which is dependent upon a mesh screen's associated open area ratio or porosity β , given by Wieghardt [115] and equation 4., as well as the Reynolds number of the flow.

$$\beta = \frac{\text{area of holes}}{\text{total area}} = \left(1 - \frac{d}{l}\right)^2 \quad (4.1)$$

In equation 4., d is the diameter of the wire which forms the mesh and l is the width of a single square in the mesh. The process by which K is determined is dependent on the ratio of the Reynolds number to the screen porosity, where if $Re/\beta \leq 600$, the value of K decreases with increasing Re such that [115]

$$K = \frac{6.5(1 - \beta)}{\beta^2} \times \left(\frac{Re}{\beta}\right)^{-\frac{1}{3}} \quad (4.2)$$

However, when $Re/\beta > 600$, the value of K is independent of the flow speed, and

$$K = \frac{0.9(1 - \beta)}{\beta^2} \quad (4.3)$$

as shown by Collar [116], which can overestimate K for higher Re [109]. It was decided to therefore use 2 mesh screens in the current airflow conditioning setup; the first with a porosity of 0.25 ($d = 1 \text{ mm}$, $l = 2 \text{ mm}$) was inserted just upstream of the honeycomb flow straightener, where the very low porosity was required to adequately reduce additional turbulence and swirl in the flow at the exit of the diffuser due to the presence of the centrifugal blowers. The second mesh screen was inserted downstream of the honeycomb flow straightener to further reduce the longitudinal velocity variations in the flow and suppress turbulent structures shed from the honeycomb. Mehta and Bradshaw [109] suggests that the screen located at the most downstream position should have a porosity of no less than 0.57 if a 2D boundary layer is required in the working section. Below this value, instabilities can be introduced to the flow. A porosity of 0.6 ($d = 0.5 \text{ mm}$, $l = 2.5 \text{ mm}$) was therefore chosen for the present investigation.

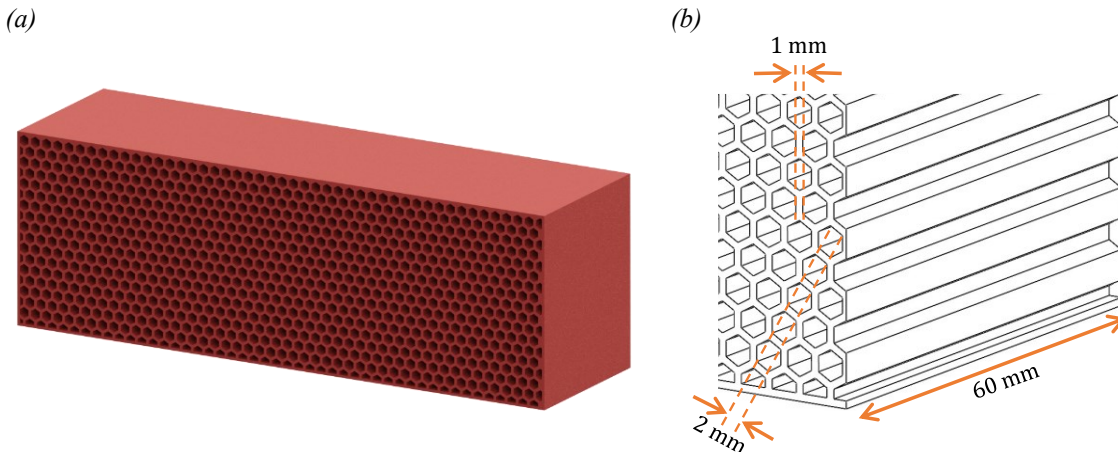


Figure 4.5: Final honeycomb flow straightener design

The length and shape of the contraction in advance of the nozzle exit were designed using the procedure outlined by Morel [117] for the design of a two-dimensional wind tunnel contraction. The optimum contraction length aims to minimise the boundary layer thickness and promote flow uniformity at the nozzle exit, while

avoiding flow separation at the chamber walls. Mehta and Bradshaw [109] suggest that the exact dimensions of the contour are not important if the profile is smooth, especially where the contraction joins the parallel sections at either end, however Morel [117] provides more specific instruction for the design of a symmetric 2D contraction. For this investigation, the design of a one-sided contraction was found not to be available in the literature, which is unsurprising considering the potential asymmetry that could be induced in the resulting velocity profile. However, as this one-sided contraction was required for the present investigation to achieve the full range of offset ratio values, while ensuring identical airflow conditioning procedures are applied to both the wall jet and offset jet, it was decided to follow the recommendations of Morel [117] for a standard two-dimensional contraction but design for a lower turbulence intensity than required at the nozzle exit to allow for an additional factor of safety. In doing so, it was also ensured that the nozzle length remained sufficiently short to avoid excessive boundary layer growth along the flat bottom wall.

The contour design method created by Morel [117] allows the user to choose a contraction ratio and, as the occurrence of separation can be assessed from the pressure distribution along the contour wall, the user can also estimate values for the wall pressure coefficient at the locations of minimum and maximum velocity using the separation criteria devised by Stratford [118], *i.e.*, C_{pi} at the contraction inlet and C_{pe} the nozzle exit, respectively. The chosen value of C_{pe} is also indicative of the desired velocity non-uniformity at the nozzle exit, u'_{jet} . Morel [117] noted a direct proportionality between C_{pe} and u'_{jet} , such that

$$u'_{jet} = 0.19C_{pe} \quad (4.4)$$

For the present investigation, values of 0.4 and 0.015 were selected for C_{pi} and C_{pe} , respectively, which indicated a desired non-uniformity at the nozzle exit of 0.3%. While this value is unrealistically low, it was deemed prudent to design for a non-uniformity value much less than that actually required to allow for a suitable factor of safety over the total flow velocity range. As a turbulence intensity of up to 5% would be considered appropriate for the nozzle exit, it was decided that $C_{pe} = 0.015$ would suffice. Following the selection of CR , C_{pi} and C_{pe} , Morel [117] provides a series of formulae to first calculate the required contraction length and hence obtain a parametric equation to describe the contraction contour. In doing so, a contraction length of $L = 77.3 \text{ mm}$ was determined and a contour shape as described by equation 4.5, where h is the contraction height at each location x along the contraction length and the limiting value X is the non-dimensionalised match point between the adjoining contours with respect to L , such that $X_m = 0.647$.

$$h = \begin{cases} 0.063(1 - 5200x^3) + 0.007, & x/L \leq X \\ 0.108 \left(1 - \frac{x}{0.0773}\right)^3 + 0.007, & x/L > X \end{cases} \quad (4.5)$$

The final dimensions of the airflow conditioning chamber and inside layout is presented in Figure 4.6, which was created using a MATLAB code adopted from Jackson [110]. In this schematic, the final contraction length was rounded to 80 mm for convenience and an additional 35 mm was added to the nozzle exit to allow for the non-uniformities exiting from the contraction to further decrease below an acceptable level in advance of the working section, as per Mehta and Bradshaw [109]. The spacing of 21 mm between the contraction entry and the first mesh screen was also calculated based on Mehta and Bradshaw [109], who proposed that 0.2 times the settling chamber hydraulic diameter would be the optimum distance since, below this value, the flow can become distorted or, above this value, excessive boundary layer development can occur. The 63 mm spacing between the this mesh screen and the honeycomb flow straightener was calculated using 0.6 times the settling chamber hydraulic diameter, based on Jackson [110], to enable the static pressure to fully recover in advance

of the mesh screen and thus ensure the pressure drop across the honeycomb was independent of the mesh screen [109]. Upstream of the honeycomb flow straightener, a further 188 mm was added to the airflow chamber to bring the total length to 450 mm , as this distance, along with the additional mesh screen located 145 mm upstream of the honeycomb, should allow the non-uniformities in the flow exiting from the diffuser to reduce sufficiently.

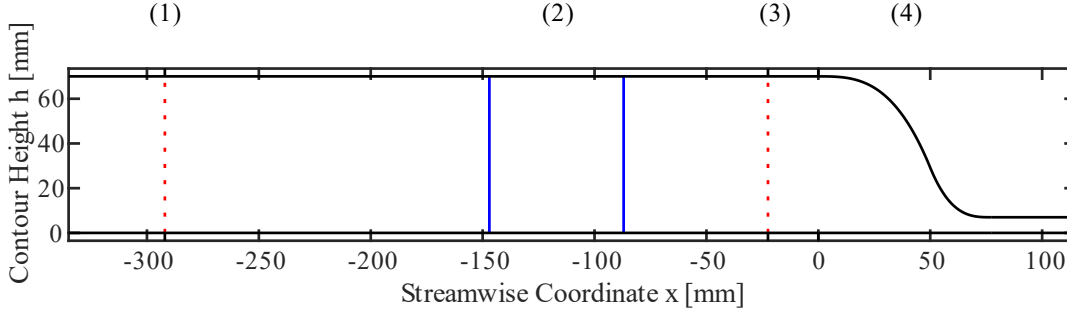


Figure 4.6: Arrangement of flow conditioning components inside the airflow chamber, where $x = 0$ coincides with the contraction entry. (1) indicates the first mesh screen location, (2) marks the position of the honeycomb, its associated depth represented by the blue lines, (3) indicates the second mesh screen location and (4) is the nozzle contraction.

The side walls of the airflow conditioning chamber were manufactured from 6 mm aluminium plate, where the desired contraction contour was recreated using CNC machining. The bottom wall was produced as a single 3 mm aluminium plate section for each chamber, while the top wall was manufactured in two sections. The first section, consisting of the portion located within the settling chamber region, was manufactured from 3 mm aluminium plate, while the remaining portion in contact with the contraction and the nozzle exit was produced using a Phrozen Transform Resin 3D Printer, which ensured the cross-sectional consistency of the contraction section and smoothness of the top wall. In the case of the wall jet, the upper wall of the contraction was shortened by 35 mm from the nozzle exit to allow for a seamless connection with the solid wall boundary. All flow conditioning components inside the settling chamber consisted of 3D printed polylactic acid (PLA) at 100% density, as this allowed for the devices to be customised to the cross section of the airflow chamber, where each of the mesh screens were given a 3 mm thickness in the streamwise direction.

4.1.1.2. Flow Measurement and Control

The relationship between the volumetric flowrate and the induced pressure drop across the honeycomb flow straightener was used as the primary means of flowrate measurement in the experimental setup. Through establishing the calibration equation relating both quantities, the volumetric flowrate to each jet could hence be inferred by means of the live differential pressure readings across the respective honeycomb flow straightener, where the full calibration procedure is described in section 4.4.2. The final streamwise length of the honeycomb flow straightener of 60 mm induced a pressure drop in the range of 10 Pa to 50 Pa for $5500 \leq Re \leq 12000$. The pressure drop across each honeycomb was thus measured at the locations indicated in Figure 4.3 using 2 Sensirion SDP1000 differential pressure sensors, which have a full-scale measurement range of 0 to 62 Pa . The $0 - 5\text{ V}$ DC output from each sensor was acquired using an NI 9215 analogue input DAQ module.

As previously mentioned, airflow was provided to each jet using separate AXAIR GREG9 160x62R 230/1/50/60 centrifugal forward-curved single-inlet blowers. These blowers were chosen based on the

calculated maximum expected pressure drop across the system for the intended volumetric flowrate, *i.e.*, 22.89 Pa to 110.56 Pa for $5500 \leq Re \leq 12000$. These values were obtained through summing the individual pressure drop contributions from the honeycomb flow straightener and the mesh screens, calculated by multiplying their associated pressure drop coefficient value K by the dynamic pressure, as well as accounting for the skin friction and exit pressure loss as per Mehta and Bradshaw [109]. This blower model also allowed for the flowrate to each jet to be controlled independently by means of a 0 – 10 V input DC signal, which was supplied using an NI 9263 analogue output DAQ module.

The airflow to each jet is controlled and monitored using a bespoke LabVIEW virtual instrument (VI) which implemented a basic negative feedback control loop, as described by the flowchart presented in Figure 4.7. The LabVIEW VI allowed the user to set a target Reynolds number value for each jet, where the output signal to each blower could be adjusted incrementally based on the real-time pressure readings from the honeycomb flow straightener. The program converted the live pressure measurements to volumetric flowrate values using the calibrated relationship, and subsequently obtained the corresponding jet Reynolds number. It should be noted that as the Reynolds number is a temperature dependent property, the kinematic viscosity value ν used to calculate the current jet Reynolds number was interpolated in real-time using the simultaneously measured jet temperature. A T-type thermocouple located inside the entrance to each settling chamber ascertained the wall jet and offset jet temperatures independently, the approximate locations of which are indicated in Figure 4.3 and labelled T_w and T_o respectively. Both thermocouples were of the welded bead type with a 0.3 mm tip diameter and were recorded by the LabVIEW VI using an NI 9211 DAQ module. The typical temperature range for both jets was 18°C to 23°C.

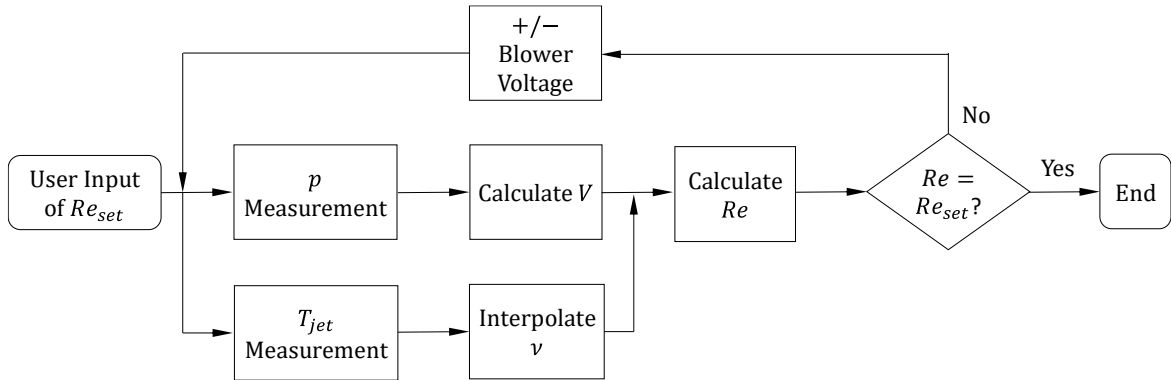


Figure 4.7: Flowchart of flow control function implemented in the LabVIEW VI.

4.1.2. Heat Transfer Study: Thermography System

This section outlines the design and technical details of the thermography system used to conduct the transient heat transfer analysis of the dual jet. The discussion is divided into two sections: 1) the implementation of the uniform wall flux boundary condition and 2) the chosen method of temperature acquisition.

4.1.2.1. Wall Test Section

The uniform wall flux (UWF) boundary condition refers to the application of a known heat flux, q'' , across the test section of interest. By measuring the surface temperature distribution (T_s) and the jet bulk air temperature (T_{jet}), the local heat transfer coefficient h can be inferred through Newton's law of cooling, *i.e.*,

$$q'' = h(T_s - T_{jet}) \quad (4.6)$$

The use of a UWF boundary condition can be found in many studies such as Lytle and Webb [62], AbdulNour *et al.* [68], Nizou [67] and Kim *et al.* [79], however the method of implementation tends to vary widely. In general, the constant heat flux is created through ohmically heating a substrate by means of an electrical current, where past studies have used substrates such as a thin coating of indium tin oxide on a glass surface [68], a sheet of gold film Intrex [79], or stainless steel shim stock [62]. However, the most commonly used substrate is a thin foil, tensioned to prevent warping at higher temperatures, which can be seen in studies such as Gibbons *et al.* [119] and Brown *et al.* [120]. The use of thin foil is generally preferable due to its low thermal inertia and thus relatively high response rate with respect to the temperature fluctuations [121]. In addition to this, a thinner foil yields less lateral conduction, meaning strong temperature gradients within the foil surface can more readily be preserved. This allows for the local convective heat flux at each location to be estimated with greater accuracy and improves on the spatial resolution of the final results. Therefore, for the present study, it was decided to adopt the tensioned foil approach to achieve a UWF boundary condition in the bounding wall. The chosen foil material was Inconel 625® Corrosion Resistant Alloy (Ni61/Cr22/Mo 9/Fe 5) manufactured by Goodfellow. Due to its high tensile strength of 950 MPa, this foil allowed for a large tensioning force to be applied. Additionally, the high electrical resistivity of 129 $\mu\Omega\text{mcm}$ allowed for the desired heat flux to be achieved using a lower amperage current when compared to a similar sized section of stainless-steel foil.

Consideration was also given to the final dimensions of the foil section, which also dictated the downstream extent of the test region in the solid wall. A number of factors had to be taken into account, including capturing important flow features, feasibility, cost of materials, and the field-of-view of the infra-red camera to be used. While the ideal system would encapsulate the complete downstream region, this was obviously not feasible and it was decided to focus on the primarily capturing the heat transfer properties of the near jet region, where the flow features are unique to dual jet flow. The test region hence needed to span a downstream region that included the locations of both the merge point and the combined point for the range of jet parameters investigated, as the dual jet should behave similarly to a single wall jet beyond this, which has been studied extensively in the published literature. The location of cp , however, varies widely in the literature, and varies greatly with both OR and Re . In the study carried out by Wang and Tan [19] for $Re = 10000$ and $OR = 1$, X_{cp} had a reported value of 6.4. Kumar and Das [1] subsequently found $X_{cp} = 18.38$ for $Re = 20000$ and $OR = 8$, which is a similar jet configuration to the upper limit for the present investigation. Mondal *et al.* [20] recorded a similar X_{cp} value of 18.54, however this was for a much smaller Re and OR combination of 10000 and 2.5, respectively. Kumar [27] reported a value of $X_{cp} = 16.25$ for similar jet parameters to Mondal *et al.* [20], and increasing the offset ratio from 2 to 10 increased X_{cp} to 20.3. As the published trends regarding X_{cp} were quite inconsistent, it was decided to choose an approximate lower limit of $18w$ for the length of the test region. The final foil dimensions were ultimately chosen based on available stock sizing that could sufficiently span the test region of interest, where mitigating the need to trim the foil reduced the possibility of failure under the applied tensile load due to induced irregularities at the edges. The final chosen foil had square dimensions $140 \times 140 \text{ mm}$ and a thickness of $25 \mu\text{m}$ which was considered sufficiently thin to minimise lateral conduction. Further foil properties are provided in Table 4.1.

The final wall test section design can be seen in Figure 4.8(a), which included the heated foil section, a spring-loaded tensioning mechanism to prevent warping at higher temperatures and wall cover inserts on either side

of the foil ensure the upper surface remained flush. As indicated in the figure, the x-axis was aligned with the centre of the foil, *i.e.*, the jet centreline, and the intended jet airflow was in the positive x-direction. The placement of the wall test section in the complete experimental setup is shown in Figure 4.8(b), where it can be seen that the test section makes up the final 35 mm of the lower wall of the wall jet nozzle to ensure the wall jet emerges directly tangential to the solid wall without an offset. The foil, however, only extends into the wall jet nozzle by 5 mm, where a thin layer of Kapton tape was used to insulate this 5 mm foil strip from the jet flow, preventing any flow to the underside of the foil and ensuring the transition between surfaces in this region is smooth. Downstream of the test region, the remaining solid wall consisted of a smooth Perspex sheet, the length of which brought the total downstream extent of the solid bounding wall to 100w.

Table 4.1: Relevant Inconel 625® foil properties at 21°C [122]

Description	Unit	Value
Dimensions	$[mm \times mm \times mm]$	$140 \times 140 \times 0.025$
Resistivity ρ_f	$[\mu\Omega cm]$	129 (132 @ 93°C)
Thermal conductivity k_f	$[W/mK]$	9.8 (10.8 @ 93°C)
Density ρ_f	$[kg/m^3]$	8.44×10^3
Volumetric expansion coefficient	$[K^{-1}]$	1×10^{-6}

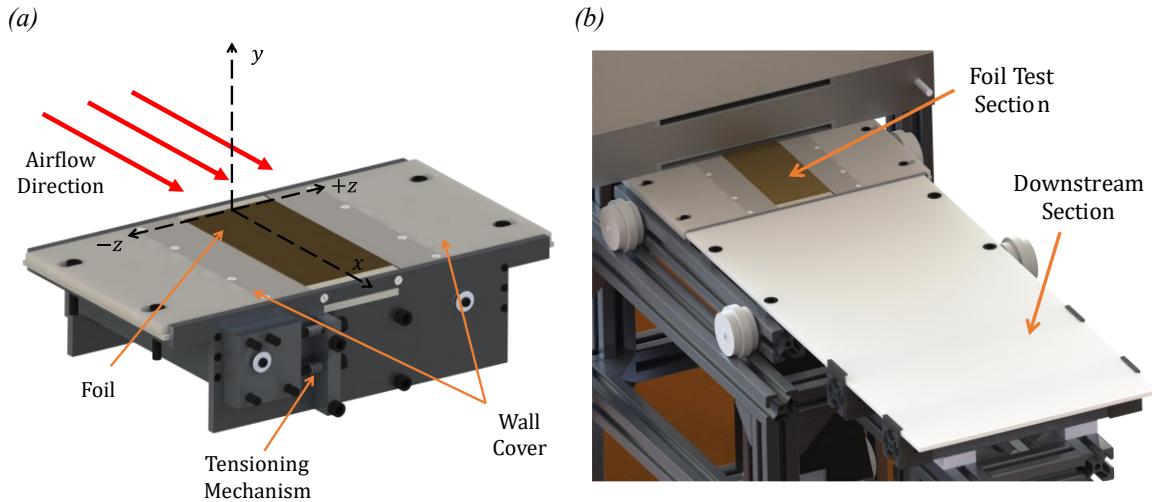


Figure 4.8: a) UWF wall test section design, b) Placement of wall test section in experimental setup with full downstream insert.

The foil tensioning mechanism can be seen in Figure 4.9, the design of which was based on Gibbons *et al.* [119]. As shown in the schematic, the foil was clamped between two copper busbars which deliver the electrical current from a Kikusui PWR1600L DC power supply, where the free length of foil between the busbars was 78 mm and silver conductive grease was used to ensure a good contact between the foil and the busbars. To ensure the foil top surface remains as smooth as possible, the busbars were located beneath the surface, and the contact area between the wall cover inserts and the foil was minimised through the inclusion of a knife-edge on the wall cover inserts. The busbars were mounted on a hinged surface, where the location of one hinge was fixed relative to the support structure and the other was spring loaded for the purpose of applying a tensile force to the foil, as shown in Figure 4.10. The hinge compensated for the varying angle of the busbar as the

foil elongated with increasing temperature. The foil was tensioned across a central housing section, which established a 10 mm stagnant air gap between the foil and a germanium window, which acted as a thermal barrier to constrain heat losses to the upper surface of the foil. Heat loss through the air gap was monitored by a welded-bead T-type thermocouple, with 0.3 mm tip diameter, located at a perpendicular distance of 9.5 mm from the underside of the test foil, the typical temperature range for which was 27°C to 32°C. The central housing section, along with the hinged surfaces and the wall cover inserts, were manufactured from polyether ether ketone (PEEK) due to its insulative properties, high tensile strength, and dimensional stability at higher temperatures.

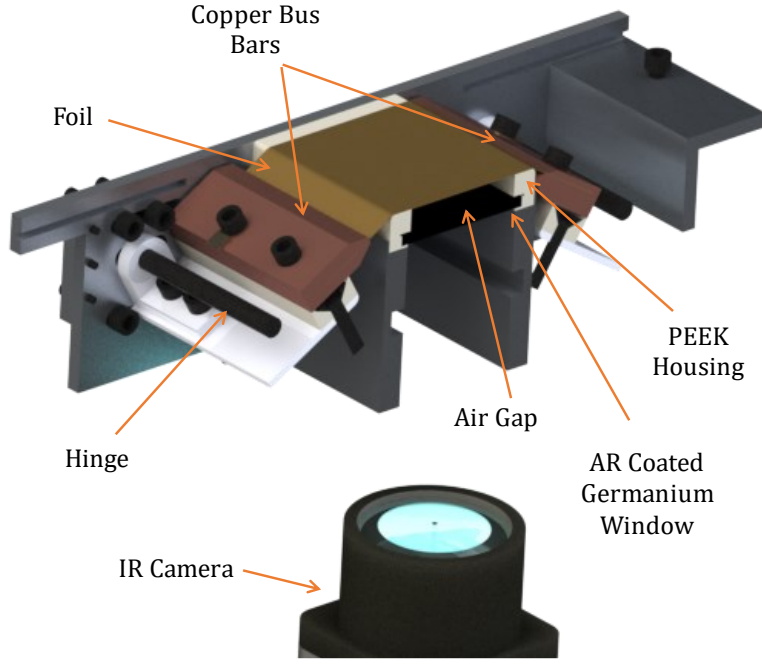


Figure 4.9: Section view of foil tensioning design for UWF test section.

Current to test foil was supplied in constant current mode and monitored through the power supply's digital display. For a constant current of $I = 25.2 \text{ A}$ the surface heat flux was maintained at approximately $q_s'' = 1670 \text{ W/m}^2$. It should be noted that the power to the foil, and hence the surface heat generation, was originally calculated using $Q = VI$, where V was a measured voltage drop between the copper busbars. This method, however, was found to slightly overpredict the voltage drop, which had a magnitude of $< 1 \text{ V}$, and had a significant effect on the calculated surface final heat flux. The exact location for the voltage drop measurement was the subject of debate, and the method was considered overly sensitive to any error. It was decided to adopt $Q = I^2R$ to estimate the power supplied to the foil, where R was the resistance of the test foil. The resistance of the foil was estimated using $R = \rho_f L / A_c$, where L was the free length of foil between the copper busbars, A_c was the cross-sectional area of the foil and ρ_f was the foil resistivity as per Table 4.1. By this method, as only the free length of the Inconel foil required consideration, any losses due the resistance of the busbars themselves, the resistance of the connecting wires or the contact resistance between the foil and the busbars were eliminated, thus improving the accuracy of the final power value acquired. Using the technical data supplied for Inconel 625®, it was noted that the calculated resistance of the foil should vary by a maximum of 1% over a 100 K temperature change, which accounts for both the thermal expansion and changes in resistivity of the foil, which was considered adequately low for the present investigation.

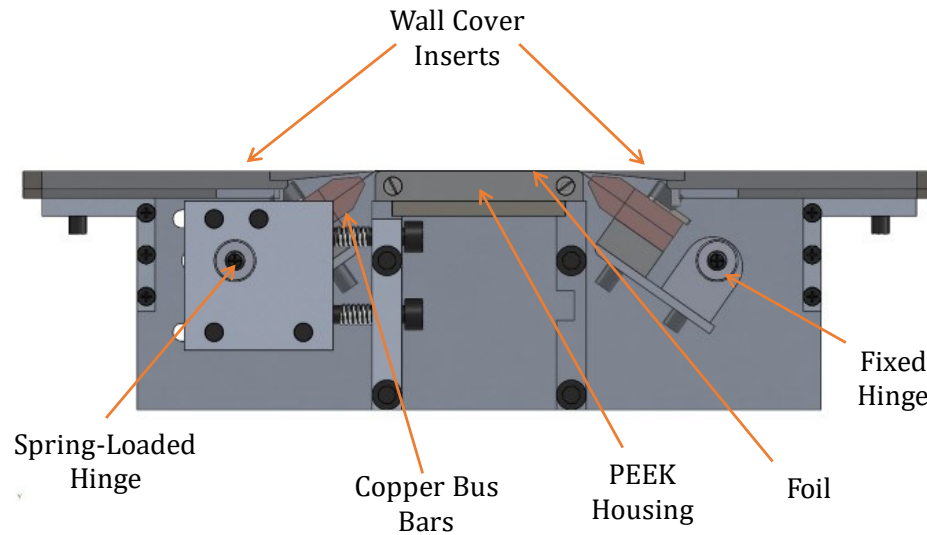


Figure 4.10: Foil tensioning mechanism.

4.1.2.2. Temperature Acquisition

Following the establishment of a UWF boundary condition, the local surface heat transfer coefficient distribution can be acquired through the acquisition of the surface temperature profile. The published literature presents a variety of means to achieve this. For example, investigations such as Kim *et al.* [79] and Lee *et al.* [123] used thermionic liquid crystals to provide a surface temperature map, however, this method is generally not preferred due to their narrow operating temperature range and low response rate, and is therefore somewhat outdated [124]. An alternative method seen throughout the literature is the use of a series of thermocouples, such as Nizou [67], however, as only discrete temperature measurements at specific locations can be acquired, sharp temperature gradients cannot be captured accurately. The use of high speed infrared thermal imaging is therefore the more popular option throughout the published literature, due to its superior response rate and much higher spatial resolution, and this method has been used extensively in studies such as Brown [125], Gibbons *et al.* [119], McGuinn [126], Roux *et al.* [127] and Chang *et al.* [128].

Infrared thermal imaging operates by measuring the emission of infrared thermal radiation from a test surface of known emissivity and inferring the surface temperature. For the present investigation, a FLIR A655sc thermal camera with a 25° lens was used for thermal imagery. Some relevant camera specifications are provided in Table 4.2. The FLIR A655sc thermal camera uses a 14-bit uncooled-microbolometer infrared focal plane array (IRFPA) to detect temperature variations in the test surface, which is a popular choice across many industrial applications due to the relatively high spatial resolution and sensitivity that can be acquired without the use of external cooling [129]. The IR camera was mounted below the wall test section and focussed on the underside of the test foil, as shown in Figure 4.11, with a distance of approximately 330 mm between the lower surface of the foil and the front of the IR camera lens, which is within the camera's focal limitations. To avoid induced noise due to the Narcissus effect, *i.e.*, the detection of IR camera sensor reflection in the test surface [130], the IR camera was rotated by 5.5° from the vertical xz -plane to offset the region of interest from the centre of the focal array. To further reduce external reflections and minimise induced errors from the surrounding environment, a viewing column was 3D printed from PLA to block disruptions along the focal path and all visible surfaces to the IR camera were painted with a matte black layer [128].

Table 4.2: FLIR A655sc IR camera specifications when used with 25° lens [131].

<i>Description</i>	<i>Unit</i>	<i>Value</i>
Noise Equivalent Temperature Difference (NETD)	[mK]	< 30
Field of View (FOV)	[°]	25×19 (31 diagonal)
Spectral Range	[μm]	7.5 – 14
Minimum Focus Distance	[m]	0.25
Resolution	[pixels]	640 × 480
Recording frequency	[Hz]	25

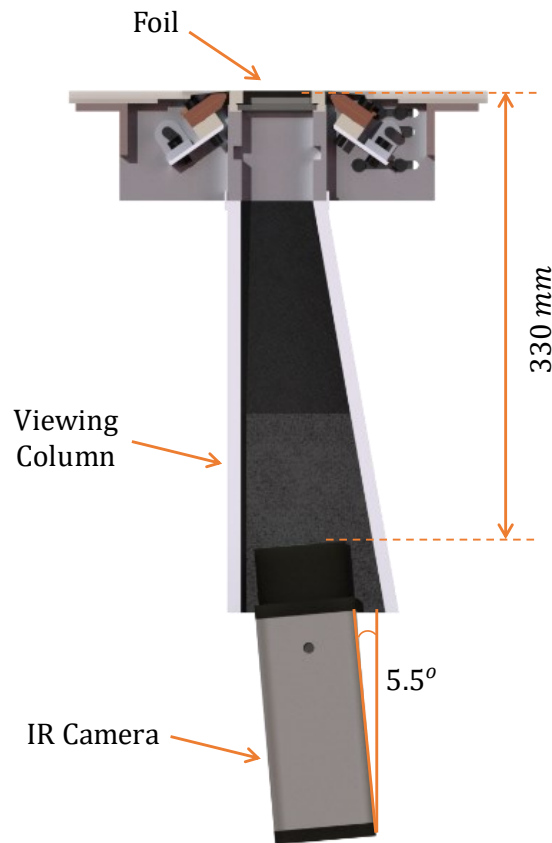


Figure 4.11: Position of thermal camera with respect to heated foil.

Using a spray gun, the underside of the foil was painted with a thin layer of matte black paint which had a known emissivity of 0.95. The thickness of the paint layer was determined using a Leica DM LM Microscope fitted with a ZEISS Axiocam 208 Color 4K microscope camera after testing was complete, which used an integrated Labscope software to acquire accurate small-scale measurements. A sample image, provided in Figure 4.12(a), shows the foil sample oriented parallel to the view direction, allowing for visibility of both the foil thickness and the paint layer, where the foil sample was cut carefully using a scalpel to minimise any damage to the paint layer. The paint layer thickness was determined through averaging a series of discrete paint layer thickness measurements along the foil sample, as per the annotated image presented in Figure 4.12(b). The paint layer thickness was estimated as $43.2\ \mu\text{m}$, which was subsequently verified using a Mitutoyo micrometer.

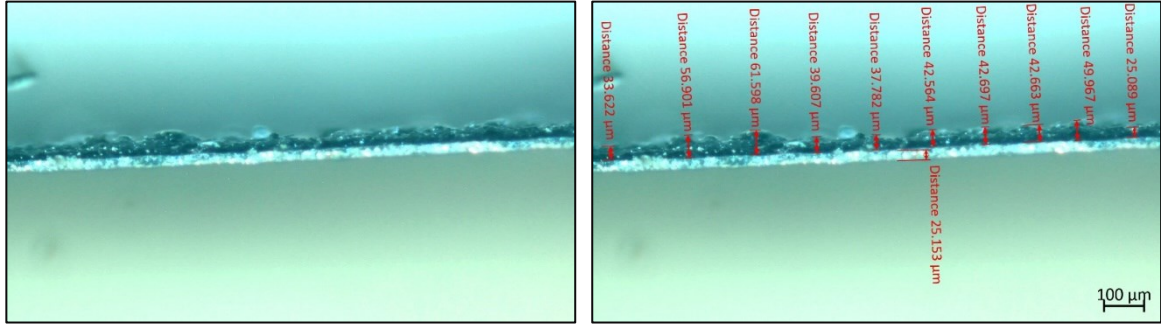


Figure 4.12: Paint layer thickness image acquired from Labscope software. (a) raw image, and (b) annotated image

As discussed previously, the presence of an insulative air gap beneath the foil reduced heat losses from the underside of the foil. The 10 mm air gap was established using the PEEK housing and an anti-reflection (AR) coated germanium window, as shown in Figure 4.9, which was chosen due to its high transmittance within the sensitive wavelength range for the FLIR A655sc thermal camera, *i.e.*, 7.5 to 14 μm . The custom size 140 $\times$ 58 $\times$ 4 mm germanium window was manufactured by Shalom EO and provided with YF3 ZnS anti-reflection coating to improve transmittance specifically within the desired wavelength range, as shown in Table 4.3. Further relevant properties of the AR coated germanium window are also contained in Table 4.3.

Table 4.3: Properties of YF3 ZnS anti-reflection coated germanium window, as provided by Shalom EO.

Description	Value	Transmittance Curve
Dimensions	140 $\times$ 58 $\times$ 4 mm	
AR Coating Material	YF3 ZnS	
Transmission Range	1.8 to 23 μm	
Refractive Index	4.0026 at 11 μm	
Reflection Loss	53% at 11 μm	
Absorption Coefficient	0.027 cm^{-1} at 10.6 μm	
Density ρ_{Ge}	5330 kg/m^3	
Thermal Conductivity k_{Ge}	58.61 W/mK	
Specific Heat Capacity C_{Ge}	310 J/kgK	

When placed in the wall test section, the field-of-view offered by the germanium window to the thermal camera of the foil underside allowed for the heat transfer analysis of 18.5w in the downstream x-direction and $\pm 3.5\text{w}$ in the transverse z-direction, as per the coordinate system in Figure 4.8(a). A recording frequency of 25 Hz was adopted for the present investigation, which allowed for the use of the full window resolution of the IR camera of 640 $\times$ 480 pixels. Instantaneous data was recorded over a 20 s time interval, which was read into the integrated FLIR ResearchIR software using an ethernet connection. The IR camera was triggered to record externally by the LabVIEW VI to synchronise the thermal acquisition with the airflow measurements, where an NI 9263 analogue output DAQ module was used to generate the 5 V trigger signal. The test function implemented in the LabVIEW VI is described by the flowchart presented in Figure 4.13, where the recorded test data was outputted from the LabVIEW VI and Research IR software as .xlsx and .seq format, respectively.

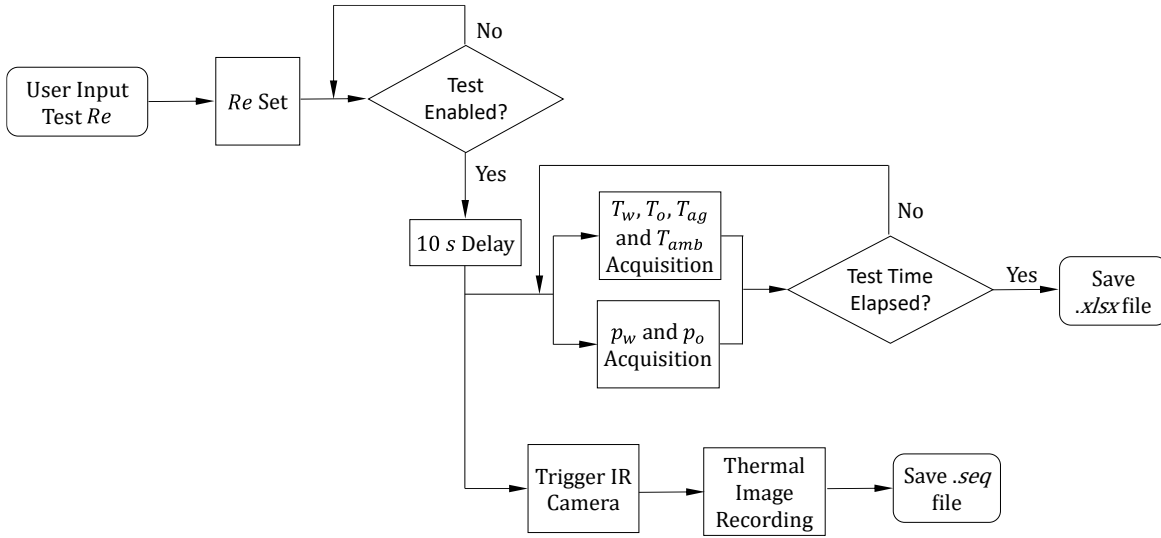


Figure 4.13: Flowchart of testing function implemented in the LabVIEW VI for the heat transfer study.

4.1.3. Flow Visualisation Study: Particle Image Velocimetry Setup

This section outlines the experimental setup for the flow visualisation aspect of the present investigation, the aim of which was the transient analysis of the dual jet flow field, with the objective to understand the unique flow features that drive the measured heat transfer behaviour. Many methods of flow field acquisition can be seen throughout the published literature, such as shadowgraphy, Laser-Doppler Anemometry (LDA) or hot wire anemometers, however, it was decided for the current study to use Particle Image Velocimetry (PIV) as it allowed for the analysis of a relatively large flow field and the extraction of velocity information at multiple locations with high spatial resolution [132]. While the temporal resolution associated with PIV may not be on par with some single point velocity measurement techniques, technological advancements have enabled PIV to attain sampling rates in the kHz region, which was considered sufficient for the present investigation. In addition, the use of PIV in conjunction with jet flows is commonplace in the published literature, including studies such as McGuinn [126], Patte-Rouland *et al.* [133], and Assoudi *et al.* [134], as it provided both qualitative and quantitative data for the entire flow field and allowed for interesting flow features to be easily identified [132, 135]. More specifically, as Wang and Tan [19] previously used PIV techniques to analyse dual jet flow, the use of PIV for the current study maintained consistency and allowed for a direct comparison of the results acquired.

The basic underlying principle of particle image velocimetry is the interpretation of the fluid velocity from the measured displacement of the fluid over a known time interval. To achieve this, an appropriate seeding particle must be introduced to the flow, the motion of which throughout the flow field is subsequently captured in successive images using sufficient illumination and high-speed camera recording equipment. The particle displacement is interpreted based on the spatial calibration of the image plane and, as the time delay between successive images is known, the particle velocity can be readily calculated [132]. This principle is reflected in Figure 4.14, which presents the typical experimental PIV setup for the acquisition of two-component velocity data inside a two-dimensional flow field, *i.e.*, 2D-2C PIV. In this setup, a thin laser light sheet is used to illuminate the tracer particles along the plane of interest, which is pulsed twice to produce successive images at time t_0 and $t_0 + \Delta t$. For evaluation purposes of analysis, the visible flow field is divided into a series of

smaller interrogation windows and a statistical cross-correlation method determines the average particle displacement inside each window to produce a single 2D velocity vector, the compilation of which comprises a complete vector flow field [132, 135]. Other more advanced PIV methods currently exist which can be used to analyse a 3-dimensional flow field, such as stereoscopic PIV, however, the use of a more complex PIV method was deemed unnecessary for the present investigation as a planar flow was expected along the jet centreline. Therefore, a simple 2D-2C PIV approach was adopted for the analysis of the dual jet flow, similar to Wang and Tan [19].

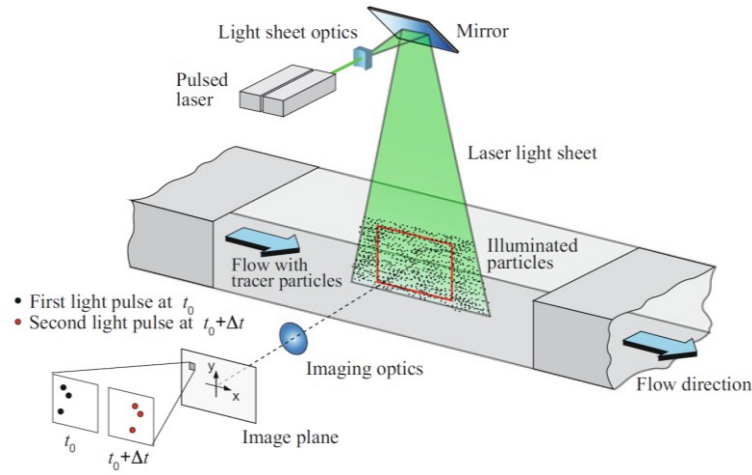


Figure 4.14: Typical arrangement for 2C-2D PIV experiment, as per Raffel et al. [132].

A schematic showing the incorporation of the PIV system with the dual jet experimental setup is presented in Figure 4.15, which enabled the transient analysis of the dual jet flow field. In order to achieve this, modifications were required in both the solid bounding wall and the surrounding enclosure to allow the laser sheet to successfully enter and leave the test region with minimal distortion. The PIV experimental setup is discussed in greater detail in the succeeding sections regarding 1) flow seeding, 2) particle illumination, and 3) flow field image capture.

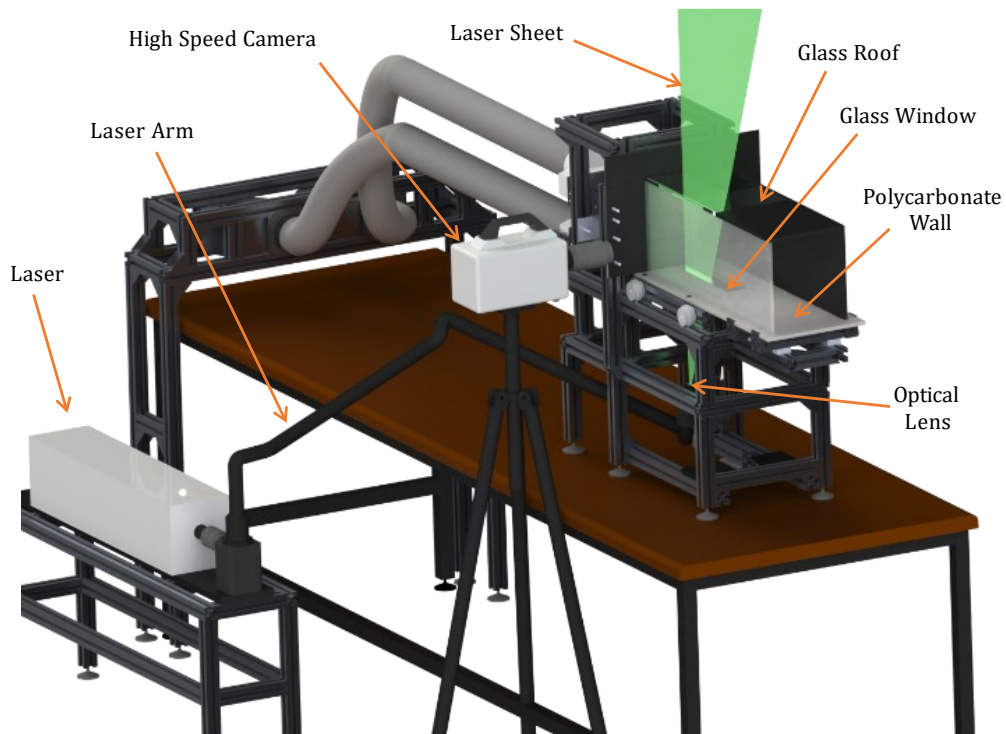


Figure 4.15: Particle image velocimetry setup

4.1.3.1. Flow Particle Seeding

The chosen seeding particle for a given flow generally dictates the overall success and accuracy of the subsequent PIV study, as PIV relies on the assumption that the tracer particle motion closely follows the true path of the fluid. This may not always be the case, as gravitational forces or slip between the tracer particles and the adjacent fluid can affect the perceived fluid displacement. The ideal seeding particle should therefore remain neutrally buoyant within the working fluid, which can be achieved through altering either the particle density or the particle diameter. According to Raffel *et al.* [132], the gravitational effect of the seeding particle velocity can be derived from Stokes law, as described by equation 4.7, which assumes spherical particles suspended in a viscous fluid moving with a low Reynolds number.

$$U_g = d_p^2 \frac{(\rho_p - \rho)}{18\mu} g \quad (4.7)$$

In equation 4.7, U_g is the gravitationally induced particle velocity, d_p is the seeding particle diameter, ρ_p is the seeding particle density, ρ is the fluid density, μ is the fluid dynamic viscosity, and g is acceleration due to gravity. Similarly, Raffel *et al.* [132] derived an expression for the slip velocity U_s between the tracer particle velocity U_p and the fluid velocity U , subject to acceleration a , such that

$$U_s = U_p - U = d_p^2 \frac{(\rho_p - \rho)}{18\mu} a \quad (4.8)$$

Equations 4.7 and 4.8 indicate that neutral buoyancy can be achieved through the choice of a seeding particle of similar density to the working fluid, hence eliminating the induced fluid velocity discrepancy from both gravity and velocity slip. While this may easily be achieved for liquid flows, gaseous flows present an added complexity, as the seeding particle density is generally much greater than that of the working fluid. The seeding particle diameter must therefore be significantly reduced to achieve a desired neutral buoyancy, however, this compromises the light scattering capability of the tracer particles and subsequently reduces the signal-to-noise ratio (SNR) [136]. The light scattering properties of the seeding particles therefore play an important role in the choice of seeding particle for gaseous flows, which was relevant to the present investigation.

In cases where $\rho_p \gg \rho$, Raffel *et al.* [132] recommends the use of the Stokes number St_K to gauge the seeding particle's ability to virtuously trace the true fluid motion. The Stokes number can be calculated using equation 4.9, where τ_p is the particle response time to a step deceleration of the flow velocity, and τ_f is the characteristic time scale of the flow, usually interpreted as the ratio between the characteristic length scale and characteristic velocity of the flow field. The particle response time is given by equation 4.10 and, if subject to a constant flow deceleration, the particle velocity at time t can be calculated using equation 4.11, which follows a simple exponential law. While a constant acceleration or Stokes law may not be applicable to all flows, the use of τ_p is still recommended as a convenient means to gauge a seeding particle's ability to reach velocity equilibrium with the fluid flow. The choice of seeding particle is therefore considered adequate, with regard to accurately tracing the fluid motion, if $St_K < 10^{-1}$ [132, 137].

$$St_K = \frac{\tau_p}{\tau_f} \quad (4.9)$$

$$\tau_p = d_p^2 \frac{\rho_p}{18\mu} \quad (4.10)$$

$$U_p(t) = U \left[1 - \exp\left(-\frac{t}{\tau_p}\right) \right] \quad (4.11)$$

With regard to the light scattering capability of the tracer particles, this is often measured in terms of the scattering cross section C_s , as per equation 4.12, where P_s is the total scattered power and I_0 is the laser intensity incident on the particle [136]. For PIV investigations, the scattering of the laser light with wavelength λ is maintained within the Mie-scattering regime, *i.e.*, $d_p/\lambda > 1$, as it produces much large C_s values that increase with particle diameter, thus requiring lower laser power [132, 138]. It should be noted, however, that the scattered light intensity in the Mie scattering has a directional dependency, and increasing d_p reduces the intensity of light scattered 90° to the laser direction, *i.e.*, towards the camera recording equipment in 2C-2D PIV [132, 136]. Careful consideration must therefore be given to the diameter of the tracer particles, where d_p must be sufficiently low as to maximise the seeding particles tracing ability and 90° scattering, while also large enough to produce an adequate C_s value without requiring excessive laser power. For gaseous flows, a value of $d_p = 1 \mu m$ is generally recommended.

$$C_s = P_s/I_0 \quad (4.12)$$

For the present investigation, it was decided to use atomised Di-Ethyl-Hexyl-Sebacat (DEHS) fluid to seed the dual jet flow, which has a particle density of $\rho_p = 0.91 \text{ g/cm}^3$ and dynamic viscosity of $1.8 \times 10^{-5} \text{ Pa.s}$. The use of droplet seeding for gaseous flow is generally preferable as it can be applied to a wide range of flows and is relatively harmless if inhaled by the user, where other options include helium-filled soap bubbles, which are limited to low-speed flows, or solid powders, which can be very harmful if inhaled by the user. The use of DEHS fluid in this manner can, however, lead to significant fouling of the test region or other exposed surfaces, and may pose issues for fan-cooled electronic equipment in the vicinity. While glycol-based seeding can offer a solution in this regard, the associated particle size and light scattering properties were found to be insufficient for the present experimental setup, therefore DEHS was identified as the most suitable option despite the possible disadvantages. The DEHS fluid was atomised using a LaVision Aerosol Generator in conjunction with the laboratory compressed air supply, which produces seeding particles of approximately $d_p = 1 \mu m$ that can remain neutrally buoyant in air for a sufficient time period according to the manufacturer's specifications [139]. Using equation 4.10, a theoretical particle response time of $\tau_p = 2.8 \mu s$ was hence identified, and, assuming the maximum characteristic velocity of $u_{jet} = 25.59 \text{ m/s}$ for $Re = 12000$ and a characteristic length of w , the fluid response time was approximated as $\tau_f = w/u_{jet} = 274 \mu s$. This produced a maximum Stokes number value of approximately $St_k = 0.01$, which indicated that the seeding particle motion should accurately represent the dual jet flow field in the experimental setup. To ensure uniform seeding density throughout the total fluid domain, which included both the dual jet flow and the entrained ambient fluid, the entire laboratory space was filled with DEHS seeding particles until the desired seeding density was attained, during which time airflow was supplied to the jets. A seeding density of approximately 10 particles per interrogation window was preferable, however the environment was continuously seeded on a trail-and-error basis until acceptable PIV measurements were produced prior to testing.

4.1.3.2. Flow Field Particle Illumination

For the present experimental setup, the laser sheet entered the flow domain from below and exited through the enclosure roof, as shown in Figure 4.15. To facilitate this, it was necessary to modify both the solid wall

boundary and the flow enclosure to minimise distortion and reflection of the laser sheet, while all other components in the experimental setup remained unchanged. Firstly, the solid bounding wall was replaced with a smooth 10mm thick polycarbonate sheet, the streamwise length of which equated to $100w$ from the nozzle exit. A clear, low iron, plane glass window measuring $300 \times 60 \times 6 \text{ mm}$ was included in the polycarbonate sheet to allow the laser sheet to pass into the flow domain in the region of interest with minimal distortion. The first edge of this window was located 20 mm upstream of the nozzle exit, to allow the laser sheet to access the origin of the flow domain, *i.e.*, the intersection between the jet exit plane and the bottom wall, and its length was centred along the mid plane of the flow domain. The glass window was recessed into polycarbonate sheet to ensure the upper surfaces were flush inside the flow domain, with any remaining gaps filled, as this enabled a smooth transition between both materials and minimised disruptions to the flow. Prior to insertion, the 6 mm edges of the glass window were coated with a matte black paint layer to prevent the lateral refractions of the laser sheet inside the glass from illuminating the polycarbonate sheet. Similarly, to enable the laser sheet to exit to flow domain and reduce the reflection of the laser sheet back into the flow domain, part of the enclosure roof closest to the jet exit plane was replaced with a $400 \times 230 \times 10 \text{ mm}$ glass window. Finally, all surfaces within the enclosure where transparency was not required were covered a layer of adhesive matte black vinyl film and any remaining reflective surfaces were painted matte black to reduce reflections.

The laser system used for the PIV study was a Quantronix Darwin-Duo Nd:YLF Laser, which had a nominal dual-beam output of 35 mJ per beam and a 527 nm wavelength. The laser output was transmitted to the testing facility using a LaVision Laser Guiding Arm and the laser sheet was created using a LaVision Divergent Sheet Optics and a $f = -20$ cylindrical divergent lens. The sheet optics was attached to an adjustable mount located beneath the glass window in the solid wall and directed upwards towards the flow domain, as shown in Figure 4.16. This generated a thin laser sheet with an approximate thickness of 1.5 mm, where minimising laser sheet thickness was required to improve the light intensity and reduce both background noise and noise due to out-of-plane motion [140].

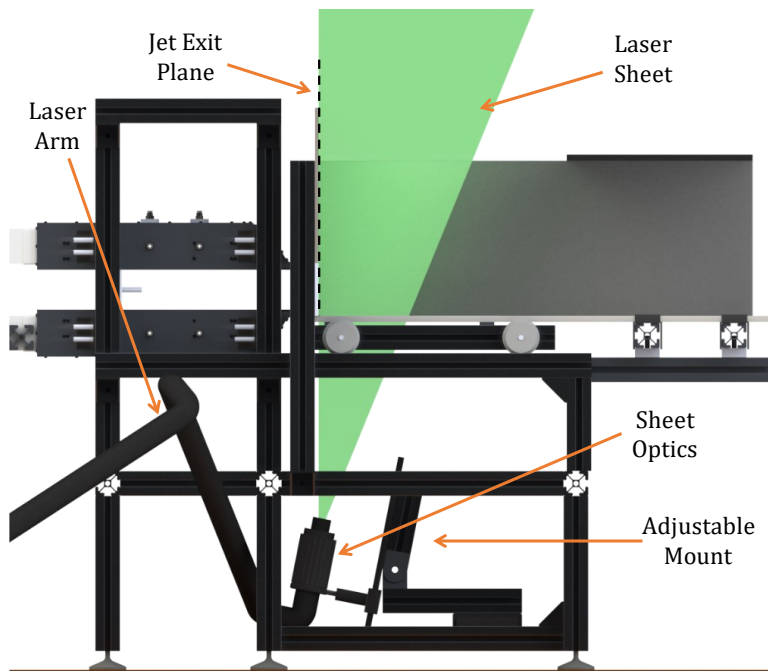


Figure 4.16: Alignment of the laser sheet in the PIV experimental setup.

Consideration had to also be given to the exact orientation of the laser sheet to maximise light intensity in the region of interest while not inducing unnecessary reflection. The ideal setup would allow the laser sheet to illuminate the entire dual jet centre plane, beginning at $(X, Y) = (0, 0)$ and extending across the full field-of-view, without any dark spots or shadows, while also ensuring that the laser sheet was not directly incident on the jet exit plane. This was achieved using the laser sheet orientation shown in Figure 4.16, where the diagonal edge of the laser sheet was aligned with the jet exit plane allowing it to barely skim the surface. To ensure the laser sheet was directly vertical in the dual jet centre plane, a thin line was drawn along the centre of the solid wall and the enclosure roof in the streamwise direction with a florescent orange marker. During setup, the laser sheet was considered vertically aligned if both target lines were illuminated along their length by direct incidence from the laser. In addition, it was ensured that the partial reflection of the laser beam off the enclosure roof was incidental along the solid wall target line also. These target lines were removed prior to testing.

4.1.3.3. Flow Field Particle Imaging

The flow field was captured using a Photron FASTCAM SA1.1/HSS 6 high speed CMOS camera, which had a sensor resolution of 1024×1024 pixels and 12-bit greyscale output. When used in conjunction with a 100 mm lens, an approximate non-dimensional field of view (FOV) of $12.5w \times 12.5w$ ($x \times y$) was attained in the flow field, with the origin of the coordinate system located at the intersection between the jet exit plane and the bottom wall. To maximise the light intensity reaching the CMOS sensor, the lens was set to the largest aperture, which also allowed for the use of a higher frame rate. Careful consideration was also given to the camera viewing angle, as it was important not to oversaturate the CMOS sensor with the visible laser reflections from the bottom wall and the jet exit plane. The camera was therefore positioned at a slight angle that aligned the bottom wall and jet exit plane with the left and bottom extremities of the FOV, and any distortion of the flow field image could thus be removed through proper spatial calibration. To prevent the occurrence of ‘peak-locking’, which can occur when seeding particles are less than 3 pixels in diameter, the camera image was defocused slightly from the jet centre plane [132].

The high-speed camera and laser system were triggered using a LaVision PTU high-speed controller and controlled using the LaVision DaVis 10 software. The flow field is captured using the double-pulse double-frame mode, meaning that at each sampling point, each beam within the laser system is pulsed separately with a time delay of Δt , resulting in two images from which the flow velocity can be interpreted. The chosen value of Δt dictates the dynamic velocity range for the PIV system, *i.e.*, the range of particle displacement values that can be adequately resolved by the system [141]. Generally, the quarter window rule is advised, where the particle displacement over Δt should be no more than one quarter of the length of the interrogation window in the streamwise direction [132, 142]. However, for jet flows the choice of a suitable value for Δt is more complex as the velocity range can be quite large throughout the flow field [141]. For the present investigation, where the flow velocity was expected to vary from practically stationary to in excess of 30 m/s, the value of Δt was iteratively modified until meaningful PIV results were produced across the flow field, resulting in a final Δt value of 25 μs . Further to this, a sampling rate of 2000 Hz was adopted for the present investigation, as this would allow for vortex shedding with a frequency > 700 Hz to be detected, which was expected based on the Strouhal number of 0.25 provided by Wang and Tan [19]. While the chosen sampling rate was still relatively low compared to the expected vortex shedding frequency, increasing the sampling rate would have resulted in a reduction of the total flow time over which images could be captured, therefore 2000 Hz was

deemed an appropriate compromise between a sufficiently high sample rate and adequately long flow time to enable the identification of periodic structures in the flow. For each test case, 2000 images were captured, which equated to 1 second of flow time. Prior to each test, an intensity correction was performed with the lens cap on to account for the dark current noise inherent to the CMOS sensor [132].

4.2.Experimental Procedure

The following sections outline the separate methodologies adopted for the heat transfer and the flow visualisation investigations, respectively. In both cases, the process was carried out for an experimental Reynolds number range of $5500 \leq Re \leq 12000$, where the Reynolds number to each individual jet was adjusted separately to achieve an experimental velocity ratio range of $0.5 \leq V_r \leq 2$. For the heat transfer study, the offset ratio was adjusted from $1 \leq OR \leq 7$, while the range of values for the flow visualisation study was limited to $1 \leq OR \leq 3$ due to the camera FOV.

4.2.1. Heat Transfer Investigation

The following describes the experimental procedure followed throughout the heat transfer study.

- 1) Prior to testing, the Inconel foil was prepared by spraying a thin layer of black paint to the underside of the foil within the IR camera FOV, where it was ensured the clamped ends of the foil remained free of any paint residue. The thickness of the paint layer was subsequently established using a microscope when all testing was completed for all test cases.
- 2) The painted foil was placed inside the copper bus bars, with a thin layer of silver conductive paste, and the tensile load was applied. It was ensured that the foil was perfectly smooth when tensioned and no wrinkles were evident on the foil surface. The wall cover inserts were thus slid into place until they were just in contact with the foil.
- 3) The wall test section was inserted into the experimental rig and the IR camera was focussed on the underside of the test foil.
- 4) The offset ratio was set to the desired height using the appropriate offset plate insert. It was ensured that the offset plate was flush with each of the nozzle exits, and that both airflow chambers were level following adjustment.
- 5) The desired jet Reynolds number was set using the LabVIEW control VI and sufficient time was allowed for the volumetric flow rate to reach a steady state value.
- 6) A current flow of 25.2 A was supplied to the foil to produce a uniform heat flux of $1670\text{ W/m}^2\text{K}$. The system was given sufficient time to reach a thermal steady state, indicated by a constant air gap temperature.
- 7) Data was recorded over a 20 second time interval, where the thermal images were captured at a rate of 25 Hz , resulting in 500 images. Thermocouple and pressure readings were recorded at 2.5 Hz and 10 Hz respectively during this interval.
- 8) The acquired thermal images were processed in MATLAB, where the appropriate temperature calibration was applied. A foil energy balance was subsequently performed, from which the surface heat transfer coefficient and Nusselt number profiles were established.
- 9) The experimental procedure was repeated from step 5 for subsequent flow configurations.

4.2.2. Flow Visualisation Investigation

The following outlines the experimental procedure for used for flow visualisation study.

- 1) The experimental rig was set up as per Figure 4.15 and the offset ratio was set to the desired height using the appropriate offset plate insert. It was ensured that the offset plate was flush with each of the nozzle exits, and that both airflow chambers were level following adjustment.
- 2) The high-speed camera was switched on approximately 2 hours in advance of testing to allow it to stabilise at a constant temperature.
- 3) The atomiser was connected to the laboratory compressed air supply and the valve was opened to allow DEHS seeding particles to enter the environment.
- 4) The laser was switched on and allowed to reach the set temperature. The laser current was thus set to approximately 31 A.
- 5) The desired jet Reynolds number was set using the LabVIEW control VI and sufficient time was allowed for the volumetric flow rate to reach a steady state value.
- 6) Sufficient time was allowed for the entire laboratory space to be filled with seeding particles and reach the desired density. This was monitored by taking sample recordings and post-processing the data until meaningful results were acquired.
- 7) 2000 flow field images were captured with a sampling rate of 2000 Hz and $\Delta t = 25\mu s$ using the DaVis 10 software. Separately, the airflow thermocouple and pressure drop readings were recorded at 25 Hz and 10 Hz using the LabVIEW VI.
- 8) The procedure was repeated from step 6 for subsequent flow configurations.

4.3.Data Reduction

The following sections outline the data reduction techniques employed to produce meaningful insights from the raw data collected during heat transfer and PIV experiments. Each section defines the relevant formulae used, how useful data was identified, and any assumptions made.

4.3.1. Heat Transfer Study

The purpose of the data reduction was to acquire the local Nusselt number distribution Nu_x across the surface of interest based on the thermal images of the foil test region. The thermal images were exported as .seq files from the ResearchIR software, which could be read directly into MATLAB using a FLIRMovieReader.m script provided by FLIR. This converted each thermal image into a matrix with each cell containing the number of counts recorded for each pixel. From here, the data postprocessing was carried out in a number of stages; first, the thermal camera calibration was applied to convert each counts reading to a corresponding temperature value. From here, the foil edges were detected and the test region was isolated from the total field-of-view. A foil energy balance was then performed on the foil on a per-pixel basis to identify the convective heat loss to the working fluid. Finally, the Nu_x values associated with each pixel could be calculated. The following sections outline the mathematical processes used to acquire Nu_x from the calibrated foil temperature readings.

4.3.1.1.Assumptions

To carry out the necessary calculations to identify the surface Nu_x distribution, a number of assumptions were required. The first assumption was the adoption of the lumped capacitance method, whereby a constant temperature was assumed through the thickness of the foil and paint layer, meaning the temperature measurements recorded of the painted foil underside should correspond to the upper surface of the foil inside a specified pixel. The lumped capacitance method implies a negligible temperature gradient through the thickness of the foil and paint layer, which is considered valid if the resistance to conduction within the solid is small compared to the convective resistance between the solid and the working fluid [51]. This is indicated by the Biot number Bi , calculated using equation 4.13, where the use of the lumped capacitive method is valid if $Bi < 0.1$ [51]. In equation 4.13, h_{conv} is the convective heat transfer coefficient, k is the conductivity of the solid and L_c is a characteristic length value, usually taken as the ratio of the solid's volume V_s to its surface area A_s , i.e., $L_c = V_s/A_s$. For a plane wall of thickness δ , the value for L_c can be simplified to half the wall thickness, such that $L_c = \delta/2$ [51]. For the present investigation, Bi was calculated for the foil and paint layers separately, using the maximum h_{conv} value obtained over a series of test cases. This resulted in $Bi = 3.71 \times 10^{-4}$ and $Bi = 1.1 \times 10^{-2}$ for the foil and paint layers, respectively, indicating that the lumped capacitance assumption could be considered valid.

$$Bi = \frac{h_{conv}L_c}{k} \quad (4.13)$$

Regarding heat loss from the underside of the foil, it was assumed that the thickness of stagnant air gap beneath the foil was suitably thin to ensure that heat transfer through this region was via conduction only, thus allowing for natural convection effects to be neglected. This assumption was validated using the Rayleigh number Ra , which indicates the relative dominance of buoyancy to viscous forces in the flow. For a rectangular cavity comprising two opposing walls of temperatures T_1 and T_2 , respectively, separated by an air gap of thickness δ_{ag} , the Rayleigh number can be obtained using equation 4.14, where β is the thermal expansion coefficient,

ν is the kinematic viscosity and α is the thermal diffusivity. For this case, the critical Rayleigh number is defined as $Ra_c = 1708$, below which viscous forces dominate and advection cannot occur, thus resulting in heat transfer by conduction only [51]. For the current study, a sample set of Ra values were calculated for $OR = 1$, where T_1 was equated to the average surface temperature of the foil, T_2 was the measured temperature inside the air gap and the air gap thickness was 10 mm. This resulted in a range of Rayleigh number values from $876.77 \leq Ra \leq 1253.47$, all of which were less than the critical Rayleigh number, thus inducting that advection could not occur within the air gap. In addition to this, Bergman *et al.* [51] stated that for a horizontal rectangular cavity heated from above, in the absence of radiation, heat transfer from the top to the bottom surface occurs exclusively by conduction, regardless of Ra , thus further validating the assumption.

$$Ra = \frac{g\beta(T_1 - T_2)\delta_{ag}^3}{\nu\alpha} \quad (4.14)$$

The final assumption related to the predicted mode of heat transfer from the heated surface and the working fluid. As there was an external flow over a heated wall, it was assumed that forced convection was the primary mode of heat transfer, however, the possibility of additional natural convection effects needed to be addressed. This was achieved using the Richardson number Ri , as per equation 4.15, where convective heat transfer can be classified as natural, mixed, or forced if $Ri \gg 1$, $Ri \approx 1$ or $Ri \ll 1$, respectively [51, 143]. Hence, by taking T_s as the foil surface temperature, T_∞ as the jet temperature, L as the foil length, and U as the jet velocity, the Richardson number was found to vary in the range $2.76 \times 10^{-5} \leq Ri \leq 1.48 \times 10^{-4}$ for $5500 \leq Re \leq 12000$, thus indicating forced convection only to the working fluid.

$$Ri = \frac{Gr}{Re^2} = \frac{g\beta(T_s - T_\infty)L}{U^2} \quad (4.15)$$

4.3.1.2. Foil Edge Detection

Prior to commencing the heat transfer calculations, it was necessary to isolate the test region of interest from the total thermal image to minimise the computational effort required. This was achieved using a bespoke MATLAB programme which operated on the basis of detecting sudden changes in the temperature gradient at the foil edges. A sample thermal image is shown in Figure 4.17(a), where the test foil shows significantly higher temperatures compared to the captured surroundings. Figure 4.18 subsequently plots the downstream temperature gradient along the centreline of the test region shown in Figure 4.17(a), where a significant drop-off is clearly evident at the foil extremities. As a quite complex flow behaviour, and hence heat transfer trend, was expected near the jet exit, it was difficult to identify the exact location of the starting foil edge based on the temperature gradient change. The downstream edge therefore presented as a more reasonable option, as much smoother temperature gradients were measured in this region, meaning a sudden change in the temperature gradient was much more obvious. The downstream foil edge could therefore be easily identified using this method, following which the upstream foil leading edge was detected based on the known foil length of $18.5w$ and calibrated streamwise pixel dimension dx . The transverse foil edges were detected in a similar manner, with an added simplicity as the temperature was approximately constant along the foil's width due to the planarity of the flow field. The transverse foil edges were hence identified using the location of the sudden temperature gradient for the first edge, and the second edge using the known foil width of $\pm 3.5w$ and calibrated transverse pixel dimension dz . The results acquired using this method for the original image shown in Figure 4.17(a) is shown in Figure 4.17(b), where it is clear to see that the test foil was successfully isolated from

its surroundings. For each test case, this foil edge locations were detected for the first frame and assumed constant for the remaining frames in the sequence.

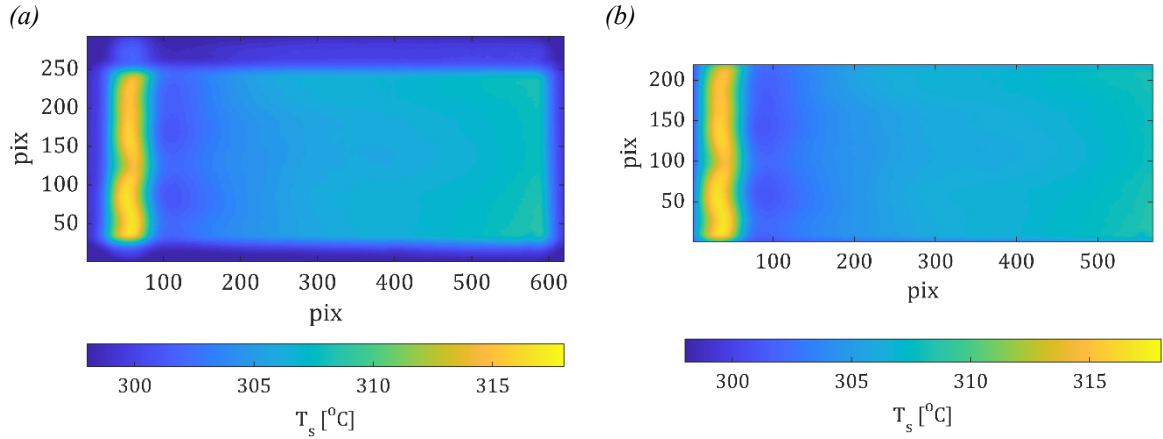


Figure 4.17: Surface temperature contour plot of test foil (a) before and (b) after the foil edge detection, where the jet exit is located along the left side of the image and the flow direction is from left to right .

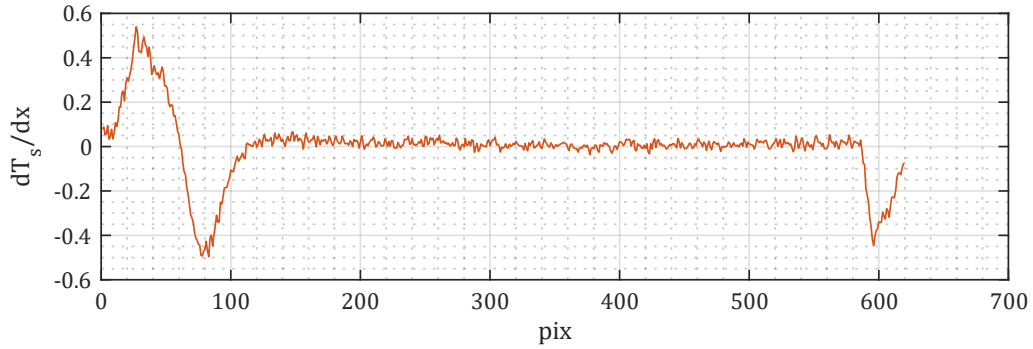


Figure 4.18: Downstream temperature gradient along centreline of test foil before foil edge detection.

4.3.1.3. Foil Energy Balance

The purpose of the foil energy balance was to calculate the convective heat transfer from the foil to the working fluid through the subtraction of the various identified heat loss components from the uniform heat generation within the foil. The approach used in the present investigation was originally developed by Donoghue [144], and subsequently adopted in many subsequent studies such as Donnelly [145], Gibbons *et al.* [146], and Meehan [147]. This method involved the application of the principle of energy conservation to each pixel within the test foil region, where each pixel had dimensions $dx \times dz$ [mm] in the streamwise and transverse directions, respectively. To achieve this over all frames in a given test case, all calculations were carried out using a custom MATLAB script, which was adapted from that of Gibbons *et al.* [146] to suit the current flow configuration.

As aforementioned, the foil energy balance applied the basic principle of energy conservation to each $dx \times dz$ foil element, as described by equation 4.16, where the rate of change of the stored energy within the element $\dot{E}_{st}$ is equated to the rate of energy transport into the element $\dot{E}_{in}$, minus the rate of energy transport out of the element $\dot{E}_{out}$, plus the rate of energy generation within the element itself $\dot{E}_{gen}$ [51].

$$\dot{E}_{in} - \dot{E}_{out} + \dot{E}_{gen} = \dot{E}_{st} \quad (4.16)$$

The assumed flow of thermal energy into the foil element is shown in Figure 4.19, where the foil and paint layers are depicted as having independent heat flow terms, denoted by the subscripts f and p , respectively, and thickness δ . In Figure 4.19, q''_{conv} is the convective heat flux to the working fluid, q''_{cond} is the conductive heat flux loss through the insulative air gap and q''_{rad} is the radiative heat flux exchange between the paint layer and the germanium window, where it was assumed that radiative heat loss from the upper surface is negligible due to the surface's relatively low temperature and low emissivity value of < 0.2 . The terms q''_x and q''_z represent the lateral conduction into the foil element in the x and z directions at locations x and z , respectively. On the opposing side, the lateral conduction terms out of the foil element are denoted by q''_{x+dx} and q''_{z+dz} , resulting in a net lateral heat flux conduction into the foil element of $(q''_x - q''_{x+dx})$ and $(q''_z - q''_{z+dz})$ in the x and z directions, respectively. The thermal energy balance on the foil element is therefore described by equation 4.17, which equates the change in the stored energy within the element, *i.e.*, the capacitive heat flux q''_{cap} , plus any heat loss terms, to the sum of heat flux generation within the foil q''_{gen} and the lateral conduction terms, the flow direction of which are not yet known. By rearranging equation 4.17 and replacing all the lateral conduction terms with a single heat flux value q''_{lc} , the convective heat flux from the test foil to the working fluid can be calculated using equation 4.18, as summarised in Figure 4.20 in the context of the experimental setup.

$$q''_{conv} + q''_{cond} + q''_{rad} + q''_{cap} = q''_{gen} + (q''_{x,f} - q''_{x+dx,f}) + (q''_{z,f} - q''_{z+dz,f}) + (q''_{x,p} - q''_{x+dx,p}) + (q''_{z,p} - q''_{z+dz,p}) \quad (4.17)$$

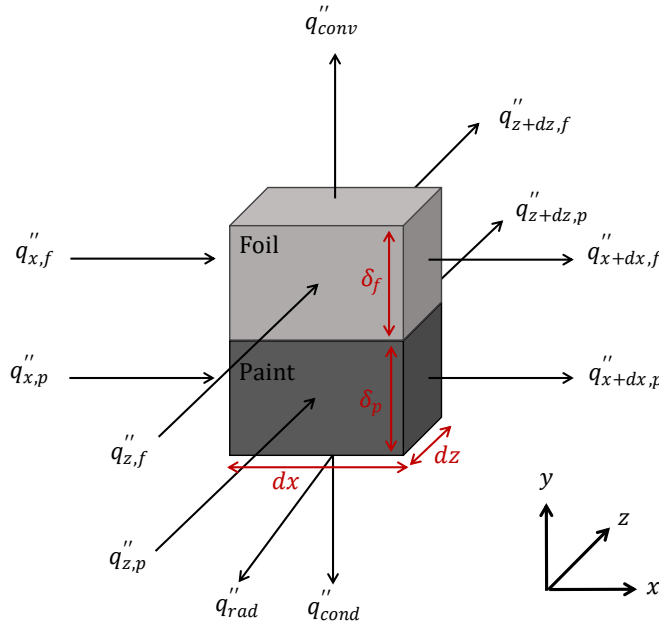


Figure 4.19: Flow of thermal energy into the foil and paint layers of a $dx \times dz$ foil and paint element.

$$q''_{conv} = q''_{gen} - q''_{cond} - q''_{rad} - q''_{cap} - q''_{lc} \quad (4.18)$$

To obtain an expression for the heat flux generation q''_{gen} , it was assumed that heat was generated in a uniform manner across the total foil surface between the busbars, therefore producing a constant q''_{gen} value for all foil elements. The ohmic heat generation within the foil was calculated using equation 4.19, where I is the constant current supply of $I = 25.2 \text{ A}$ and A_f is the total foil area between the busbars, *i.e.*, $A_f = L \times W$,

where L is the free foil length of 78 mm and W is the original foil width of 140 mm . The foil resistance R_f is defined by equation 4.20, where ρ is the foil resistivity, as per Table 4.1, and A_c is the foil cross-sectional area, given by $A_c = W \times \delta_f$.

$$q''_{gen} = \frac{I^2 R_f}{A_f} \quad (4.19)$$

$$R_f = \frac{\rho L}{A_c} \quad (4.20)$$

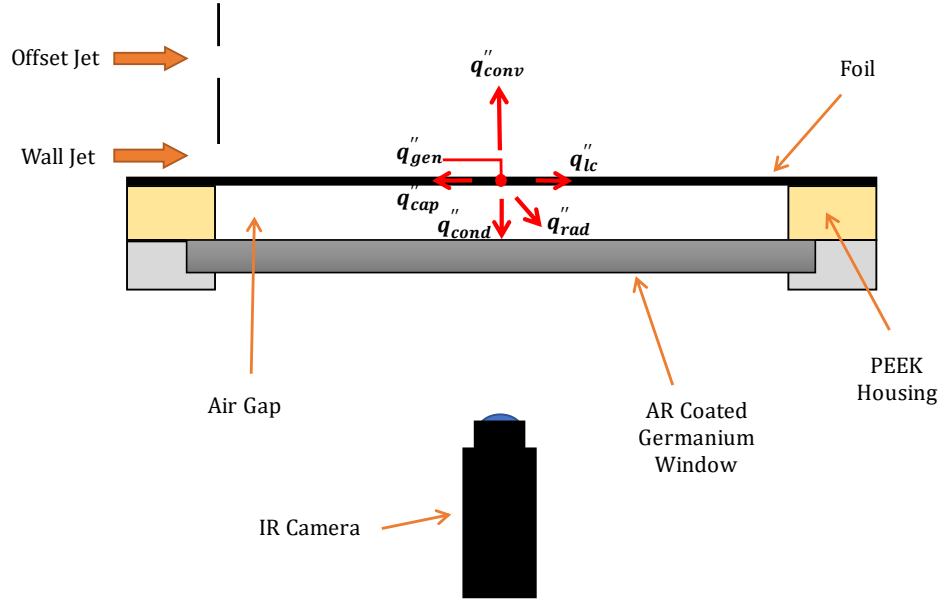


Figure 4.20: Foil energy balance as per equation 4.18 in the context of the experimental setup.

The remaining heat flux terms in equation 4.18 were calculated on a per-pixel basis as temperature dependent properties, where T_s was the measured surface temperature of the pixel in question. Firstly, Fourier's law was employed to determine the heat loss via conduction through the insulating air gap, q''_{cond} , as given by equation 4.21. In equation 4.21, T_{ag} was the measured temperature inside the air gap and δ_{ag} was the thickness of the air gap from the underside of the foil to the point of temperature measurement, *i.e.*, $\delta_{ag} = 9.5 \text{ mm}$.

$$q''_{cond} = -\frac{k_{air}}{dx dz} \frac{(T_{ag} - T_s)}{\delta_{ag}} \quad (4.21)$$

The radiative heat transfer between the foil and the germanium window was calculated based on the infinite parallel plate assumption, as per Bergman *et al.* [51], which assumes diffuse, grey surface properties. While diffuse grey properties may not have been applicable to the surface of the germanium window, the radiative thermal exchange between both surfaces was found to be relatively minor as a result of the germanium window's low emissivity, therefore the infinite parallel plate assumption was still assumed to be an appropriate approximation in this case. The radiative heat transfer was hence calculated using equation 4.22, where ε_p and ε_{Ge} denote the emissivity of the matte black paint layer and the germanium window, respectively, T_{Ge} was the inferred temperature of the germanium window based on T_{ag} and σ was the Stefan Boltzmann constant. For this equation, the germanium window was assumed to have a low emissivity value of $\varepsilon_{Ge} = 0.2$ based on the minimum transmittance value provided for the germanium of approximately 0.8.

$$q''_{rad} = \frac{\sigma(T_s^4 - T_{Ge}^4)}{1/\varepsilon_p + 1/\varepsilon_{Ge} - 1} \quad (4.22)$$

The capacitive heating of the foil and paint layers were calculated using equation 4.23, where ρ was the density of the substrate, c was the corresponding specific heat capacity and $T_s/\partial t$ represented the change in surface temperature with respect to time, calculated by means of the MATLAB gradient function. While the heating of the nearby air layers could also have been included in this term, it was noted that the sensitivity of the convective heat flux to the capacitive heating of the foil and paint layers was extremely low, therefore the added complexity through the inclusion of the air layer heating was deemed unnecessary considering its relatively minor influence. The paint properties used for the foil energy balance calculations were adopted from Raghu and Philip [148] and are listed in Table 4.4.

$$q''_{cap} = \frac{(\rho_f c_f \delta_f + \rho_p c_p \delta_p) \partial T_s}{dx dz} \frac{\partial T_s}{\partial t} \quad (4.23)$$

Table 4.4: Paint properties adopted for foil energy analysis, as per Raghu and Philip [148].

Description	Unit	Value
Density ρ_p	$[kg/m^3]$	1162
Specific Heat Capacity c_p	$[J/kgK]$	2835
Conductivity k_p	$[W/mK]$	0.57
Emissivity ε_p	$[-]$	0.99
Thickness δ_p	$[\mu m]$	43.2

Finally, the lateral conduction heat flux term in equation 4.18 combines the effect of all the lateral conduction terms from equation 4.17, such that

$$q''_{lc} = (q''_{x,f} - q''_{x+dx,f}) + (q''_{z,f} - q''_{z+dz,f}) + (q''_{x,p} - q''_{x+dx,p}) + (q''_{z,p} - q''_{z+dz,p}) \quad (4.24)$$

Using Fourier's law, the lateral heat into the element in the x and z directions can be calculated using equation 4.25 and equation 4.26 where the negative sign in each case is indicative of the heat flow direction towards the lower temperature value. On the opposite side, lateral heat conduction out of the element can be derived using equations 4.27 and 4.28, resulting in a net lateral conduction to the element as described by equation 4.29 for the foil and paint layers combined.

$$q_x = -k\delta dz \frac{\partial T_s}{\partial x} \quad (4.25)$$

$$q_z = -k\delta dx \frac{\partial T_s}{\partial z} \quad (4.26)$$

$$q_{x+dx} = q_x + \frac{\partial q_x}{\partial x} dx \quad (4.27)$$

$$q_{z+dz} = q_z + \frac{\partial q_z}{\partial z} dz \quad (4.28)$$

$$q_{lc} = -(k_f \delta_f + k_p \delta_p) \left(\frac{\partial^2 T_s}{\partial x^2} + \frac{\partial^2 T_s}{\partial z^2} \right) dx dz \quad (4.29)$$

To remain consistent with Donoghue [144], and to obtain a comparable lateral heat flux value for use in equation 4.18, equation 4.29 was rewritten to assume an equivalent uniform lateral heat conduction over the combined surface area of the 4 vertical sides of the element, *i.e.*, $2\delta(dx + dz)$ for a given layer. The final value for q''_{lc} was hence obtained using equation 4.30, which divided the total lateral heat conduction by the pixel area $dx dz$. To solve equation 4.29, the Laplacian temperature field $\left(\frac{\partial^2 T_s}{\partial x^2} + \frac{\partial^2 T_s}{\partial z^2} \right)$ was obtained using the MATLAB `del2()` operator.

$$q''_{lc} = -\frac{(k_f \delta_f^2 + k_p \delta_p^2)}{dx dz} \left(\frac{\partial^2 T_s}{\partial x^2} + \frac{\partial^2 T_s}{\partial z^2} \right) \times 2(dx + dz) \quad (4.30)$$

4.3.1.4. Local Nusselt Number

Following the acquisition of q''_{conv} through the foil energy balance, the local Nusselt number distribution on the foil surface was established using equation 4.31 for each pixel, where h was the local heat transfer coefficient, k_{air} was the air thermal conductivity and the jet temperature T_{jet} was obtained through averaging the wall jet and offset jet temperatures. The Nu_x values for each frame in a given test case were hence calculated using a custom MATLAB script, where air properties were assumed to vary with temperature and were thus calculated at the air film temperature, *i.e.*, $(T_s + T_{jet})/2$. While the MATLAB `sgolayfilt()` function was used to initially apply a third order Savitzky-Golay finite impulse response smoothing filter to the raw surface temperature data to dampen the inherent fluctuations, a significant reduction in the noise present in the final Nu data was primarily realised through averaging the Nu data across the entire foil width. This was made possible through the acquisition of an adequately two-dimensional flow field, further details of which are discussed in section 5.1.2

$$Nu = \frac{hw}{k_{air}} = \left(\frac{q''_{conv}}{T_s - T_{jet}} \right) \times \frac{w}{k_{air}} \quad (4.31)$$

4.3.2. PIV Study

The purpose of the data post-processing procedures with regard to the PIV study was to derive a velocity vector field describing the dual jet flow behaviour from the original captured images of the tracer particle motion. This was primarily achieved inside the DaVis 10 software, following which some additional flow properties were determined using MATLAB. The post-processing of the raw data occurred in several stages, discussed in subsequent sections; firstly, the necessary image pre-processing was applied to improve the quality of the measurements that could be obtained. The PIV post-processing cross-correlation procedure was subsequently carried out to obtain the particle displacement and velocity vector field, following which a process known as proper orthogonal decomposition (POD) was used to eliminate extraneous flow data and identify the coherent structures inside the flow field. Finally, the data was exported in .csv format for further processing in MATLAB, which included the identification of important flow features, such as the merge point.

4.3.2.1. Image Pre-processing

The use of image pre-processing is beneficial to the outcome of a PIV study. For an ideal PIV setup, all seeding particles with the flow field of interest would be illuminated in a uniform manner, and the difference between the particle illumination for successive laser pulses would be negligible, meaning the locations of particles can

simply be identified as the correlation peaks in the light intensity field. This, however, is rarely the case, and noise is often introduced to the PIV setup due non-uniform illumination of the tracer particles, either due to irregularities in the light sheet, pulse-to-pulse variations, or reflections [132]. As a result, brighter particles tend to dominate during the cross-correlation process, while duller particles possess less of an influence. Image pre-processing enhances the image quality by increasing the contrast between the particles and the background, improving the particle shape, and redistributing the particle signal intensities to ensure a similar influence is acquired from each particle during cross-correlation [132, 149].

Three primary image pre-processing techniques were applied in the present investigation using the DaVis 10 software, namely the *max-min filter for intensity normalisation*, the *Gaussian smoothing* linear filter and the *subtract sliding minimum* non-linear filter. As the name suggests, the purpose of the *max-min filter for intensity normalisation* was to alter the brightness of all images in a recording sequence based on the mean brightness of the first image in segmented regions referred to as tiles, the size of which should exceed the particle diameter but be small enough to eliminate background variations [132, 150]. This step improved the uniformity of the illumination across the flow field for a tile size of 5 pixels, which was particularly necessary to improve the contrast of particles near the jet exit, where illumination was weakest.

The *Gaussian smoothing* linear filter extracts high frequency noise from an image by averaging over the neighbouring pixels, where the term ‘Gaussian’ indicates that the centre pixel was given more weighting [150]. In addition, the use of a low-pass filter in this regard also acted to widen correlation peaks in the flow field, thus reducing the occurrence of peak locking, allowing for particle centres to be more easily identified and improving the performance of the peak fitting algorithm [132]. For this study, a 3×3 averaging window was specified.

The *subtract sliding minimum* non-linear filter acts as a high pass filter to remove the low frequency signal associated with the background, over a user-specified number of pixels, leaving only local high-frequency fluctuations associated with the particles [149, 150]. This helped to remove background reflections and enhance the signal-to-noise ratio for the present setup for specified a 5 pixel window, thus improving the accuracy of the system’s particle tracking capabilities.

4.3.2.2. PIV Data Processing

The aim of the PIV data processing procedure is to obtain the instantaneous particle displacement field, and hence velocity field, based on an image pair obtained at subsequent time instances, t and $t + \Delta t$. To achieve this, the flow field is divided into a series of interrogation windows, within which the particle motion is locally evaluated [151]. Each interrogation window is characterised by its light intensity distribution I and I' at time t and $t + \Delta t$, respectively, onto which the statistical particle tracker operator can be applied, *i.e.*, cross-correlation [151]. This produces a correlation map, $R(x, y)$, which can be interpreted to produce the particle displacement. This displacement can be further refined using a Gaussian sub-pixel interpolation scheme, from which the particle velocity is obtained through division by Δt [151]. For the present investigation, this step was carried out inside the DaVis 10 software package, the details of which are specified below.

The cross-correlation procedure is essentially a measure of the similarity between I and I' , where I is the light intensity distribution for an interrogation window W of size $2K \times 2L$ pixels at time t , and I' is the light intensity

distribution for an interrogation window at time $t + \Delta t$, the size of which is greater than or equal to that of W , *i.e.*, $2(K + M) \times 2(L + N)$, where $M, N \geq 0$. The discrete correlation function is given by

$$R(x, y) = \sum_{i=-K}^K \sum_{j=-L}^L I(i, j) I'(i + x, j + y) \quad (4.32)$$

which provides a single numerical value representing the sum of the products of all over-lapping pixel intensities of I inside I' , where I is ‘shifted’ inside I' by some amount (x, y) . For this process, the extremities of I do not extend beyond the edges of I' for each sample shift, such that (x, y) is limited by $-M \leq x \leq M$ and $-N \leq y \leq N$, hence producing a final correlation plane of size $(2M + 1) \times (2N + 1)$ [132]. In doing so, a greater overlap of pixel intensities, or closer alignment of particles, between I and I' for a given shift (x, y) result in a greater value of $R(x, y)$, thus implying a greater similarity. The cross-correlation value $R(x, y)$ is therefore a statistical indication of the degree of match between both windows, and the particle displacement can therefore be assumed to coincide with the (x, y) shift that produces the largest $R(x, y)$ value in the cross-correlation plane [132]. This process is hence repeated for all interrogation windows within the fluid domain, thus producing the displacement vector field corresponding to a time t .

This technique does have limitations associated with it, however. Due to the nature of this technique, the cross-correlation procedure cannot provide any information on the exact path a particle has taken or the particle acceleration, but assumes a straight line displacement between t and Δt [132], *i.e.*, a first order displacement. The size of integration window must therefore be carefully chosen such that it is small enough to allow for higher order displacement terms to be neglected. This, however, leads to the second issue, whereby decreasing the interrogation window size or increasing the flow domain significantly increases the computational effort required, where a significant amount of multiplication is required during the cross-correlation procedure when implemented as per equation 4.32 [132]. This is solved through the use of the Fourier transforms of I and I' , *i.e.*, $\hat{I}$ and $\hat{I}'$, respectively, which reduce equation 4.32 to a simple complex-conjugate multiplication, the result of which is inversely Fourier transformed to obtain the actual cross-correlation plane [132].

The final limitation associated with equation 4.32 is it achieves, at best, ± 0.5 pixel accuracy when obtaining the location of the peak in the correlation plane [152]. This is improved using a sub pixel interpolation scheme, for which a large number of algorithms exist [152]. For the present study, DaVis 10 employs a three-point interpolation scheme to achieve 0.05 pixel accuracy, which assumes 1) that the spread of the light intensity from an individual particle can be described by an Airy function and 2) that the distribution of the peak in the correlation plane can be described by the Gaussian function, (x, y) , such that

$$f(x, y) = C \exp \left[-\frac{(x - x_0)^2}{k} \right] \quad (4.33)$$

For this process, the correlation plane is initially determined using equation 4.32 and the peak location is identified at (i, j) with cross correlation value $R(i, j)$. The cross correlation values of the neighbouring pixels are subsequently extracted, *i.e.*, $R(i - 1, j)$, $R(i + 1, j)$, $R(i, j - 1)$ and $R(i, j + 1)$, following which the exact location the peak (x_0, y_0) can be calculated using equations 4.34 and 4.35 [132, 153]. Finally, the distribution of the peak inside the correlation plane is described by 4.36, where R_0 is the maximum cross correlation value at (x_0, y_0) , d_{tx} and d_{ty} are the widths of the correlation peaks in the x and y directions, respectively, and k_{xy} is the peak’s ellipticity, all of which are acquired through non-linear regression methods [132]. This method

performs best for narrow correlation peaks, approximately 2 – 3 pixels in width, as wider peaks can lead to greater uncertainty, but narrower peaks can become non-discernible from the background noise [153].

$$x_0 = i + \frac{\ln(R(i-1, j)) - \ln(R(i+1, j))}{2 \ln(R(i-1, j)) - 4 \ln(R(i, j)) + 2 \ln(R(i+1, j))} \quad (4.34)$$

$$y_0 = j + \frac{\ln(R(i, j-1)) - \ln(R(i, j+1))}{2 \ln(R(i, j-1)) - 4 \ln(R(i, j)) + 2 \ln(R(i, j+1))} \quad (4.35)$$

$$f(x, y) = R_0 \exp \left[\frac{-(x - x_0)^2}{(1/8)d_{tx}^2} - \frac{(y - y_0)^2}{(1/8)d_{ty}^2} - \frac{k_{xy}(x - x_0)(y - y_0)}{d_{tx}d_{ty}} \right] \quad (4.36)$$

The DaVis 10 software allows for the user to specify whether or not the PIV calculations are carried out in a single pass or over multiple passes. When selected, the multiple pass option allows for the vector field computed in pass to be used as a reference for the next pass, which helps to inform the specified shift of the interrogation window [153]. In doing so, a better correlation can be acquired and the signal-to-noise ratio is increased, thus increasing the overall solution accuracy. In addition, the user can specify a decreasing interrogation window size with each pass, therefore achieving a significantly smaller interrogation window size than what may have been possible otherwise. This aids in improving the resolution of the final displacement vector field and reduces the instance of erroneous vectors [153]. To further enhance the spatial resolution, and thus increase the final number of displacement vector acquired, the degree of overlap between neighbouring interrogation windows can be selected.

For the present investigation, the data was processed using multiple passes with decreasing interrogation window size. The initial and final interrogation window sizes were chosen based on a trial-and-error approach, where it was noted that larger interrogation window sizes failed to produce good results due to the presence of strong velocity gradients in the flow field, which was of particular importance close to the nozzle exit. Beginning with the initial pass, the interrogation window size was decreased in an incremental fashion until the results produced on the first pass could be considered an adequate interpretation of the expected flow field and the flow velocity could be readily resolved at all locations of interest. The integration window size in the subsequent passes were hence chosen based on the relative improvement on the flow field accuracy compared to the previous pass, where the window size was again decreased in an incremental fashion, however, a certain degree of prudence was exercised when compromising on the accuracy of the final results compared to the computational effort required. In doing so, the initial interrogation window size was finally specified as 48×48 pixels with a 75% overlap, which was maintained for 4 passes. For the final pass, the interrogation window size was set to reduce to 16×16 pixels in 2 passes, with an 87.5% overlap.

As previously mentioned in section 0, the pulse separation time of $\Delta t = 25 \mu s$ was chosen in a similar manner, where the quarter window rule was used as an initial guide, however, further fine-tuning of the final value was required to ensure the flow velocity could be adequately resolved across the entire flow field despite the presence of strong velocity gradients. The optimisation of the pulse separation time and the interrogation window size were therefore carried out concurrently due to the co-dependence of both parameters on the accuracy of the final results. Having said this, resolving the velocity vectors located immediately at the nozzle exit proved difficult due to the high seeding density in this region, in addition to amplified laser reflections and

a physical misalignment between the laser pulses. As a result, a certain degree of gap filling was introduced to the post-processing procedure in this region for display purpose only, where the most upstream position of flow field analysis was fixed to 1 mm downstream of the nozzle exit to eliminate the interpolated data. Beyond this 1 mm point, all remaining velocity vectors in the flow field could readily be resolved and included in further analysis.

4.3.2.3. Proper Orthogonal Decomposition

The proper orthogonal decomposition (POD) function in the DaVis 10 software was used to remove spurious, unrelated vector data from the flow field, while also identifying the presence of coherent structures inside the flow domain. This is achieved through a decomposition of the time-resolved flow field data into a series of eigenfields, each of which contain information regarding a dominant flow behaviour. Each eigenfield is assigned a corresponding energy value, which represents the fraction of the total energy in the original flow field attributed to a given eigenfield [153]. Therefore, by ordering the eigenfields based on decreasing energy value, it is possible to identify the dominant physical flow features, *i.e.*, the higher energy value eigenfields [153]. The lower energy value eigenfields are thus considered a representation of random errors in the flow domain and assumed to be uncorrelated with the overall flow features [132]. The final approximated flow field is obtained through the superposition of the important eigenfields, which better represents the overall flow behaviour and replaces instances of random noise with a more realistic estimate.

For the present investigation, POD was used sparingly during the time-resolved analysis only to ease in the identification of the dominant flow structures and verify the presence of periodic vortex shedding. A sample PIV dataset post-processed by means of POD is shown in Figure 4.21, where the u contour plot obtained using the raw PIV data is displayed alongside its POD counterpart. In this case, 500 eigenfields were superimposed to achieve 90% of the total energy, which results in a slightly smoother flow field and visibly less instances of random noise, while preserving the overall flow features. This was not required for the time averaged analysis, however, as much of the noise present in the relevant datasets was removed by virtue of the averaging procedure in this case.

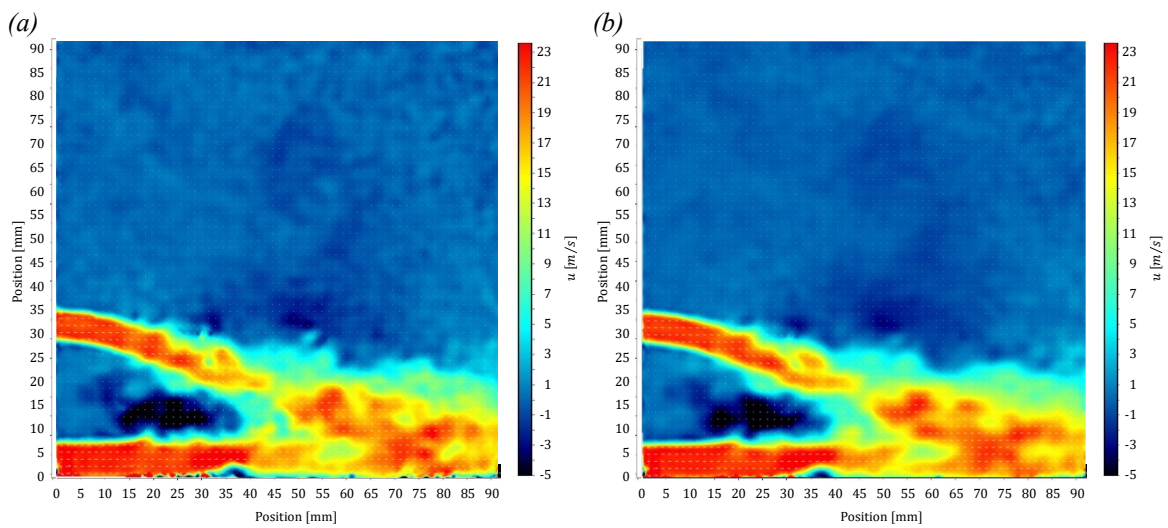


Figure 4.21: Comparison of sample instantaneous flow field (a) before and (b) after POD post-processing.

4.3.2.4. Data Post-Processing Flow Quantities

Following the acquisition of the displacement field in DaVis 10, the data was exported in .csv format to be further analysed in MATLAB. The data was exported as x and y displacement values for each pixel location, which were first converted to their corresponding the velocity values, u and v , through division by Δt . The velocity magnitude U was hence obtained using $U = \sqrt{u^2 + v^2}$. From the velocity field, a number of flow features could be identified, depending on the nature of the input data, *i.e.*, time-averaged or time resolved. In the case of time-averaged flow data, the locations of the merge point, the upper vortex centre, and the lower vortex centre were identified based on the intersection of the $u = 0$ and $v = 0$ contour lines. These contour lines were extracted from the velocity field using the MATLAB *contour()* function, and the intersection locations were found using *polyxpoly()*. In a similar manner, the combined point was identified as the furthest downstream distance at which multiple clear instances of $du/dy = 0$ existed. The du/dy field was obtained using the MATLAB *gradient* function, from which the $du/dy = 0$ contour lines could be extracted again using *contour()*.

Continuing with the time-averaged analysis, the development of the shear layers and the wall boundary layer were next identified from the $x -$ velocity vector field. At each X location, the maximum u value(s), u_m , were obtained using *max()*, following which the jet half-width locations were identified by searching the pixels vertically above or below the location of u_m until the first instance of $u = 0.5u_m$ is obtained. This location was further refined through the use of interpolation where necessary. In a similar manner, the extremity of the boundary layer was identified as $u = 0.99u_m$.

Finally, the vorticity fields for both the time-averaged and time resolved cases were obtained through the calculation the curl of the velocity field, where

$$\omega = \nabla \times U = \frac{dv}{dx} - \frac{du}{dy} \quad (4.37)$$

This was completed using the MATLAB *curl()* function, which provided values for both the vorticity magnitude and the angular velocity.

4.4. Calibration of Instrumentation

The following sections describe the calibration procedures implemented in the current investigation to ensure the accuracy and credibility of the output from the instrumentation used in the experimental setup. This included the calibration of the various thermocouples throughout the investigation, verifying the method of volumetric flowrate measurement, the thermal and spatial calibration of IR camera output, and the spatial calibration for the PIV study.

4.4.1. Thermocouple

Prior to testing, it was necessary to verify the accuracy of the thermocouples used over the course of the experimental investigation. To ensure consistency, all thermocouples were calibrated against an ASL F100 Precision Handheld RTD thermometer, the calibration documentation for which is included in Appendix C. This was achieved using an isothermal water bath, temperature controlled by means of a Techne FTE10DPC TE-10D Digital Thermoregulator, into which all thermocouples of interest and the RTD probe were placed. The temperature of the water bath was increased from 10°C to 80°C in 5°C increments, where the output from each thermocouple was recorded over 20 s at 2.5 Hz for each setpoint temperature, as well as noting the RTD probe reading. The temperature output from each the thermocouples was separately plotted against the RTD probe readings and the individual calibration curve associated with each thermocouple was derived. A sample of the results acquired from this process can be seen in Figure 4.22 for the thermocouples representing the T_w and T_o measurements, where a clear linear relationship is evident. The application of a linear regression model produced the respective calibration equations 4.38 and 4.39, which were used to correct the thermocouple temperature output during testing within the LabVIEW VI. The remainder of the thermocouple calibration curves and corresponding equations are presented in Appendix A.

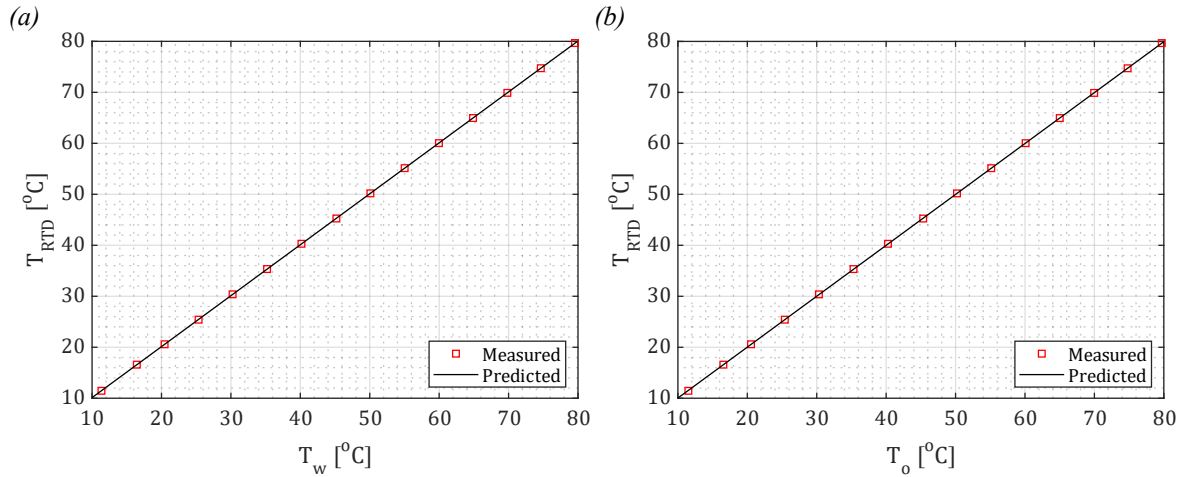


Figure 4.22: Thermocouple calibration curves for thermocouples representing (a) T_w and (b) T_o .

$$T_{RTD} = 0.9993T_w + 0.1337 \quad (4.38)$$

$$T_{RTD} = 0.9987T_o + 0.0499 \quad (4.39)$$

4.4.2. Volumetric Flowrate

For this investigation, the primary means of volumetric flowrate measurement was based on differential pressure readings across the honeycomb flow straighteners for the wall jet and the offset jet. To achieve this, a

reliable calibration curve was required to relate the differential pressure input to an output volumetric flowrate. The required a bespoke calibration procedure which subjected the in-situ honeycomb flow straightener to a series of known volumetric flowrates while simultaneously measuring the corresponding differential pressure. The modified experimental setup required for this calibration procedure is shown in Figure 4.23, where an additional 3 m of rigid PVC piping was inserted between the flexible piping and the original 1 m length of PVC pipe. The section of 1 m PVC pipe directly adjacent to the round-to-rectangle diffuser was modified to allow for the insertion of an AVT-D-R Air Velocity Transmitter at its lengthwise midpoint, *i.e.*, 0.5 m from one end of the pipe. This device operated using hot-wire technology and provided simultaneous airflow velocity and temperature readings. The primary advantage of this flow velocity sensor was its adjustable ‘plunge’ depth feature, meaning the flow velocity could be measured at various points across the pipe diameter in order to acquire the flow velocity distribution across the pipe cross section. The flow velocity sensor had a 0 – 10 V DC output signal, which could correspond to a flow velocity range of 0 – 2 m/s, 0 – 10 m/s or 0 – 20 m/s, depending on the arrangement of the connection jumpers inside the central housing. For the experimental Reynolds number range of $5500 \leq Re \leq 12000$, a maximum flowrate of approximately 5 m/s was expected within the PVC pipe for a 100 mm diameter, therefore the flow velocity sensor was configured for the 0 – 10 m/s output measurement range. The output signal was read into the LabVIEW VI using a NI 9215 analogue input DAQ module, which was calibrated externally by the manufacturer. The calibration certificate is provided in Appendix C.

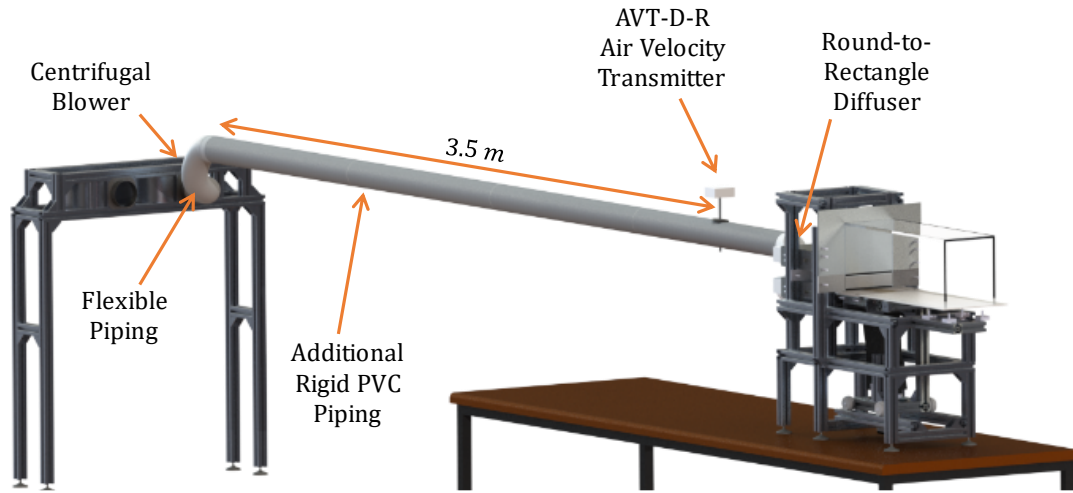


Figure 4.23: Experimental setup for volumetric flowrate calibration.

To ensure the flow velocity sensor remained aligned along the pipe diameter in the chosen cross-sectional plane, a hole was drilled on either side of the pipe diameter, and it was ensured that the flow velocity sensor was through both holes at all times. To achieve this, it was necessary to lengthen the sensor, which was completed using a round section of aluminium turned down to the same diameter. The mount that was supplied with the flow velocity sensor included a rubber seal lining which prevented air leakage from one side of the pipe diameter. For the opposite side, an additional mount was 3D printed from PLA at 100% density, which included a slot for the insertion of a rubber O-ring. In doing so, it was noted that the physical size of the AVT-D-R Air Velocity Transmitter may impose a blockage in the system, thus affecting the flow velocity profile in this region. The resulting velocity profile measured in this region, however, displayed no obvious signs of distortion caused by the sensor itself, such as that presented in Figure 4.24, whereby the shape and symmetry

of the profile were as expected. As such, it was assumed that the blockage effect of the sensor itself was minimal and could safely be disregarded for the remainder of the calibration process.

Due to the position of the temperature sensor on the AVT-D-R Air Velocity Transmitter, it was not possible to acquire flow velocity measurements within 15 mm of the wall boundary on one side. It was therefore necessary to infer the flow velocity in this region based on the assumption that the velocity profile was axisymmetric, which is valid for a fully developed flow. For the experimental Reynolds number range, a minimum entry length of $l_e = 2.18 m$ was required in advance of the measurement point to achieve a fully developed flow in this region, as per equation 4.40, where d is the pipe diameter of 100 mm and Re_d is the calculated Reynolds number using d as the characteristic length, where a uniform velocity profile is assumed at the pipe inlet [154]. As the length of PVC piping upstream of the flow velocity measurement location was 3.5 m, the additional length allowed for the flow to be considered approximately fully developed at the measurement location despite the additional swirl at the pipe inlet, and therefore validate the axisymmetric velocity profile assumption. A sample velocity profile acquired in the manner is presented in Figure 4.24, which highlights the directly measured points and those acquired through the axisymmetric assumption. The combination of the data points shows a relatively flat velocity profile, which suggests the flow in this region is approaching a fully developed state, however, it has not yet been achieved. Having said this, as a relatively good agreement exists between the points measured at repeat radial locations, the axisymmetric assumption was assumed to be sufficiently validated for the purpose of the calibration procedure. To further validate this assumption, the jet velocity at the nozzle exit was acquired using a handheld hot wire anemometer to ensure it aligned with that expected for the given volumetric flowrate calculated based on the axisymmetric assumption. It should be noted that AVT-D-R Air Velocity Transmitter presented limitations when acquiring flow velocity measurements at the solid boundary, where stationary fluid was expected due to the no-slip condition. As shown in Figure 4.24, a non-zero flow velocity value was acquired at this point, however, this value was instead assumed to be a reasonable estimate of the average flow velocity in the vicinity near the solid wall boundary, within which a steep velocity gradient exists. It was therefore included in the calculation of the average flow velocity, which was generally in good agreement with that expected for a given Re .

$$\frac{l_e}{d} = 4.4Re_d^{\frac{1}{6}} \quad (4.40)$$

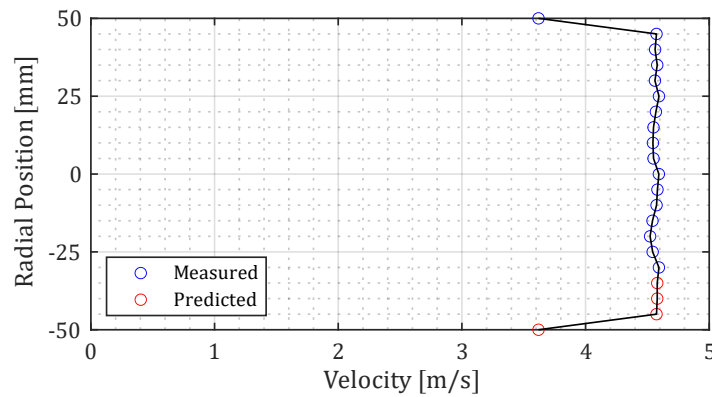


Figure 4.24: Sample velocity profile acquired inside the PVC inlet pipe leading to the wall jet at the maximum blower output, where the measured and predicted data points are highlighted.

Using this experimental setup, it was possible to infer the volumetric flowrate through the system using the average flow velocity across the pipe diameter, while measuring the differential pressure across the honeycomb

flow straightener in tandem. It should be noted a specific calibration was not conducted for the output of the pressure sensors and the sensitivity coefficient provided by the manufacturer was deemed sufficient. This was due to the fact that, ultimately, it was the calibrated relationship between the pressure sensor output and the volumetric flowrate that was important, and actual differential pressure readings were not required outside of this function. Prior to multiplication by the manufacturer's sensitivity coefficient, a constant voltage offset was subtracted from the measured signal inside the LabVIEW VI, the value of which was determined regularly by averaging the pressure sensor output over 20 seconds when subject to zero flow. In addition to this, a first order filter was applied to the pressure sensor output signal using LabVIEW to reduce the inherent fluctuations present in the data due to noise, where the filter parameters were adjusted to ensure that no information was lost from the resulting signal. The output from the differential pressure sensors were verified using a calibrated Digitron 2000 Series Manometer. This device, however, had a minimum resolution of 1 Pa , and therefore provided just an approximate means of comparison to ensure the output from the differential pressure sensors was reasonably representative.

The calibration procedure was conducted separately for each jet, during which the control signal to the respective blower was increased from 1.5 V to 10 V in approximate 2 V intervals. For each resulting flowrate, the time-averaged flow velocity was measured at 5 mm increments across the pipe diameter. Pressure and flow velocity data was collected at 10 Hz for 20 seconds for each radial location. Once each attainable radial position was complete, the flow velocities at the remaining radial positions were estimated based on the axisymmetric assumption. The average flow velocity across the pipe diameter was then calculated by summing the individual velocity measurements and dividing by the total number of measured values. To subsequently acquire the corresponding volumetric flowrate, the average flow velocity was multiplied by the pipe cross-sectional area, which could hence be plotted against the corresponding time-averaged differential pressure reading to obtain the calibration curve. At each volumetric flowrate, the jet exit velocity was spot-checked using a handheld hot wire anemometer to ensure it was comparable to that expected for the measured volumetric flowrate. Each measurement point was completed twice to confirm repeatability.

The calibration curves acquired using this method for the wall and offset jets are presented in Figure 4.25(a) and Figure 4.25(b), respectively. In each case, a clear linear relationship was acquired and the close alignment between repeated test points indicated sufficient repeatability. While a slight negative curvature was present in both datasets, suggesting perhaps a more quadratic relationship, the high r^2 value of 1 suggested that the relationship was sufficiently linear inside the narrow range of Re values of interest to allow for the application of a linear calibration curve without compromising on accuracy. Following the application of linear fit to the measured data points, the respective calibration equations 4.41 and 4.42 were acquired, which relate the measured differential pressure across the honeycomb flow straightener to a corresponding volumetric flowrate, where the subscripts w and o refer to the wall and offset jet respectively. These calibration equations were hence adopted during testing to facilitate the volumetric flowrate acquisition and to control the jet Reynolds number.

$$V_w = 7.3301 \times 10^{-4} p_w + 4.265 \times 10^{-3} \quad (4.41)$$

$$V_o = 9.436 \times 10^{-4} p_o + 4.292 \times 10^{-3} \quad (4.42)$$

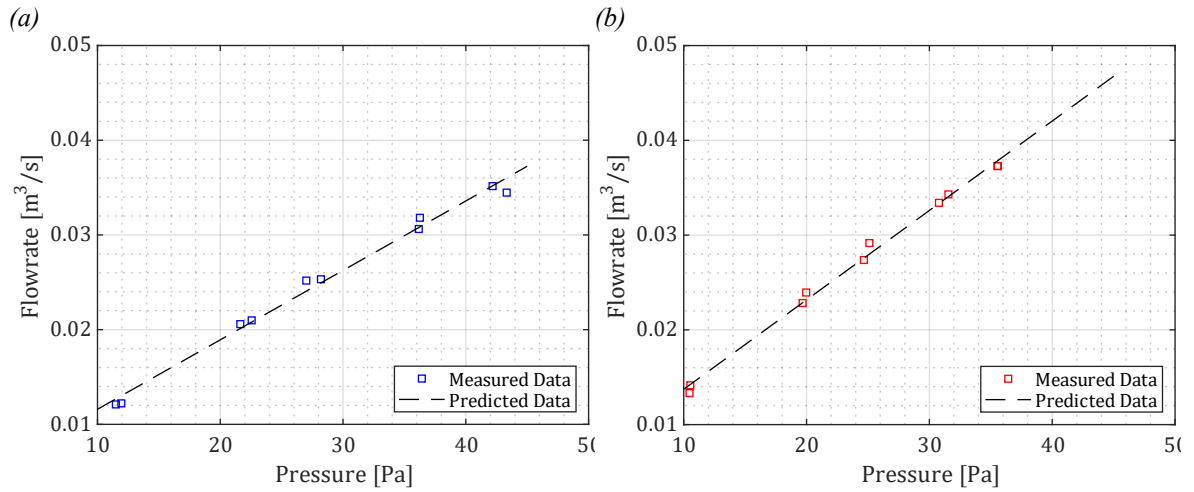


Figure 4.25: Differential pressure to volumetric flowrate calibration curves for (a) the wall jet and (b) the offset jet.

4.4.3. IR Camera

4.4.3.1. Temperature Calibration

The following sections outline the necessary procedures taken to acquire a reliable calibration curve to enable the translation of the counts output C from the IR camera to accurate foil surface temperature readings T_s during testing.

Non-Uniformity Correction

To account for the differences in the offset and gain requirement by each individual detector contained within the IRFPA inside the IR camera, it was necessary to perform a non-uniformity correction (NUC). Under normal circumstances, the offset and gain can vary from detector to detector in the IRFPA due to variability during the manufacturing process, the sensor temperature, the measurement temperature and noise; all of which contribute to the occurrence of a fixed pattern noise (FPN) in the recorded image [155]. In addition, detector differences may be introduced as a result of the reflection of the detector itself from the internal surface of the lens, *i.e.*, the Narcissus effect, leading to the measurement of fictitious temperature gradients across an otherwise isothermal surface. The NUC therefore adjusts the offset and gain of each individual detector, or pixel, to remove this induced temperature non-uniformity, as well as the detection and replacement of bad pixels [155].

The non-uniformity calibration was performed using the built-in NUC calibration wizard in the ResearchIR software, which employed the two-point correction technique. This required the exposure of the IRFPA to two isothermal blackbody surfaces, where each has a different temperature, from which the gain and offset requirements can be estimated and compensated for [155]. To achieve this, a blackbody calibration plate at room temperature (approximately 20°C) was placed against the IR camera lens, thus spanning the total field-of-view, of which a series of 16 thermal images were captured. Following this, the blackbody calibration plate was heated uniformly to 50°C using a hot plate and the thermal image acquisition process was repeated. From these images, the IRFPA non-uniformity and bad pixel instances could be inferred and hence corrected for in subsequent thermal measurements.

Blackbody Calibration

An initial counts-to-temperature calibration was carried out using the blackbody calibrator, however, this was for comparison purposes only and the effect of the germanium window would be taken into account at a later stage. The blackbody calibrator used for this process was an Omega Black Point BB701, the target plate of which had a known emissivity of 0.95 and a controllable surface temperature function. The blackbody calibration was carried out using the built-in calibration wizard in the Research IR software, which obtained an average count reading for a series of target plate temperatures and subsequently generated an appropriate calibration curve. For this investigation, the IR camera was focussed on the target plate and its surface temperature was increased from 20°C to 80°C in 5°C intervals, where sufficient time was allowed between test points to ensure the surface temperature reached a steady state. The ResearchIR software hence produced the calibration equations 4.43 and 4.44, where the counts output was first converted to radiance values r using equation 4.43 and subsequently translated to surface temperature values using equation 4.44. The calibration curve obtained using this method is presented in Figure 4.28.

$$r = 5.819 \times 10^{-7} C - 2.696 \times 10^{-3} \quad (4.43)$$

$$T_s = -8.255 \times 10^{12} r^6 + 5.953 \times 10^{11} r^5 - 1.821 \times 10^{10} r^4 + 3.109 \times 10^8 r^3 - 3.345 \times 10^6 r^2 + 2.973 \times 10^4 r - 74.887 \quad (4.44)$$

In-Situ Counts to Temperature Calibration

To account for the transmittance of the germanium window, it was necessary to perform an in-situ calibration of the IR camera to relate the counts output C to the actual measured surface temperature T_s . This involved heating the foil to a series of uniform surface temperatures and comparing with the corresponding counts data to derive an appropriate relationship. The uniform wall temperature was achieved using the heater block shown in Figure 4.26, which consisted of a $150 \times 80 \times 60 \text{ mm}^3$ aluminium block covered on 5 sides by a 10 mm thick layer of insulation to prevent heat loss and promote better temperature uniformity. The block was heated using $3 \times 200 \text{ W}$ cartridge heaters with a 10 mm diameter, all connected electrically in parallel. The block was placed on the test foil, with the non-insulated side in contact with the foil surface. The temperature uniformity inside the block near the foil surface was monitored using 3 probe type T-type thermocouples of 1 mm diameter.

To monitor the foil temperature, 4 T-type bead thermocouples were fixed to the underside of the foil surface using a thermally conductive glue and previously calibrated as per Section 4.4.1. The arrangement of the thermocouples on the underside of the foil are shown in Figure 4.27(a), where care was taken to maintain an undisrupted region of interest near to foil centre, from which the corresponding counts values were to be obtained using the IR camera. Figure 4.27(b) presents a sample thermal image acquired for the calibration setup at an approximate surface temperature of 40°C. It is clear to see that an adequate surface temperature uniformity was achieved across the foil surface, and the region of interest remained suitably clear to obtain a reliable counts measurement. During the calibration process, the foil was heated from 20°C to 85°C in approximate 5°C intervals, where at least 30 mins was allowed between test points to allow the system to settle to a steady-state surface temperature. For each surface temperature, thermal measurements were acquired over a 20 second period at sampling rates of 25 Hz and 2.5 Hz for the IR camera and thermocouples, respectively.

The calibration process was repeated twice, therefore producing two sets of experimental data for each surface temperature to ensure consistency.

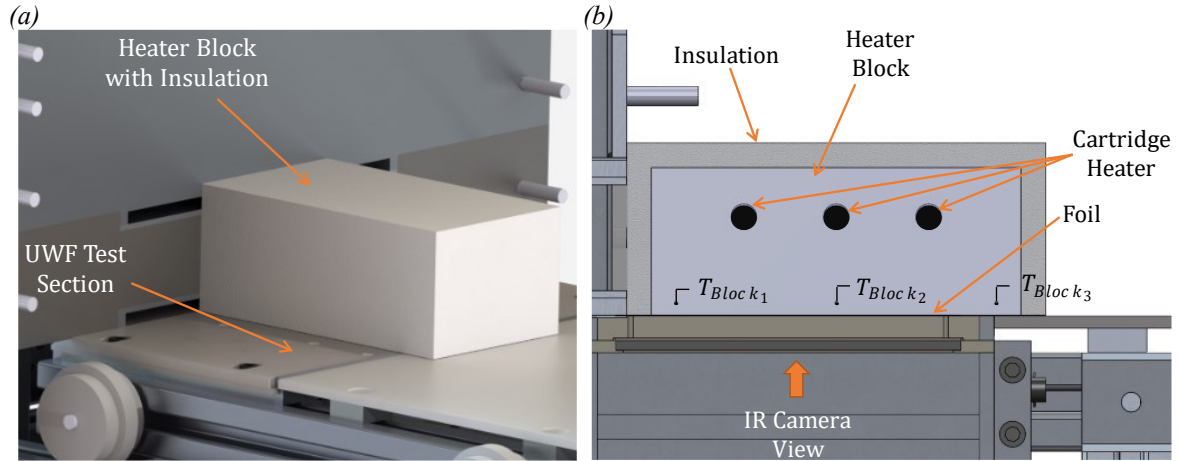


Figure 4.26: (a) Position of heater block for in-situ counts-to-temperature calibration. (b) Schematic of calibration heater block instrumentation.

The results acquired using this method are shown in Figure 4.28, where the average thermocouple temperature value was plotted against the counts data averaged over the entire region of interest. A fourth order polynomial was fitted to the data, thus producing equation 4.45, which was used to convert the counts data from the IR camera to surface temperature measurements for the remainder of this investigation. Compared to the blackbody calibration, it is interesting to note that a very similar trend was obtained, however, the blackbody calibration curve appears to underestimate the surface temperature at each counts value, where the disparity between the datasets increases with increasing temperature. This was to be expected, as the transmittance of the germanium window should result in a lower counts reading by the IR camera for a given surface temperature, meaning the temperature value provided by the IR camera using the blackbody calibration would correspond to this lower counts value. The in-situ calibration hence associates each surface temperature measurement with a higher counts reading to account for the presence of the germanium window.

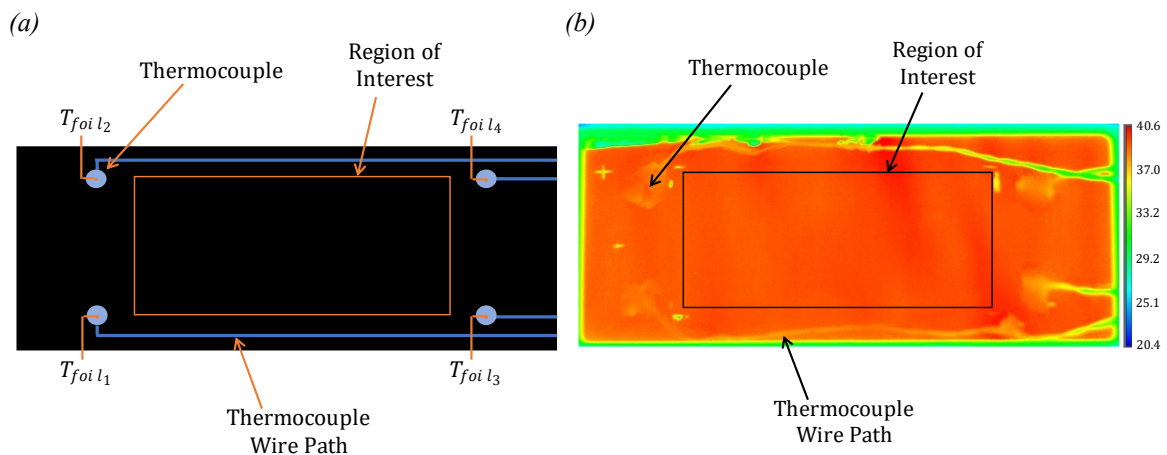


Figure 4.27: (a) Schematic of thermocouple placement on underside of foil. (b) Thermal image captured of foil during counts-to-temperature calibration [°C].

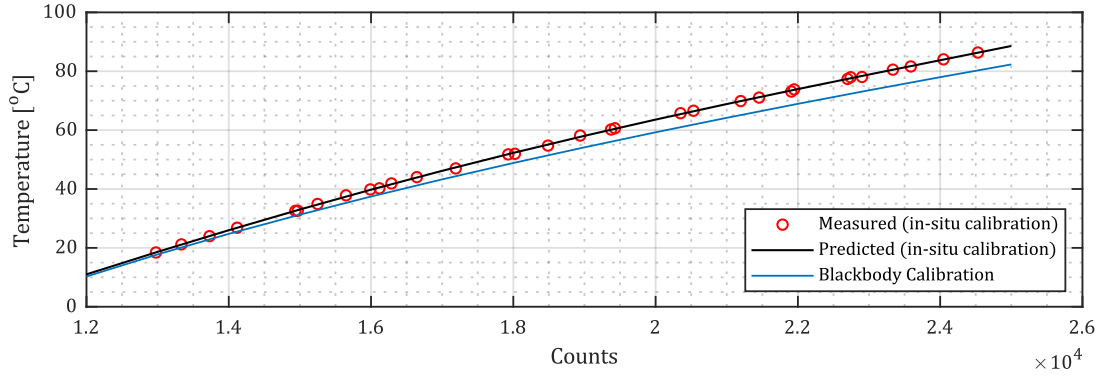


Figure 4.28: Comparison of counts to temperature calibration curves from in-situ calibration and blackbody calibration.

$$T_s = 6.958 \times 10^{-16} C^4 - 4.728 \times 10^{-11} C^3 + 1.039 \times 10^{-6} C^2 - 0.002 C - 50.097 \quad (4.45)$$

4.4.3.2. Spatial Calibration

It was necessary to devise a means to spatially calibrate the IR camera view to obtain a value for the pixel length terms dx and dz . The spatial calibrator shown in Figure 4.29(a) was therefore designed specifically for the experimental setup, which consisted of a series of tensioned, evenly-spaced heater element wires that, when placed on the foil surface as per Figure 4.29(b), produced discrete, high-temperature lines across the foil surface, from which the pixel dimensions could be calculated. For this design, the central wire tensioner section was 3D printed from resin for strength and dimensional stability at the higher temperatures, and current flow was supplied to the wires by means of the copper busbars. The heater element wire was 0.127 mm diameter aluminium wire and a 10 mm spacing was ensured between each wire. Due to the tangential contact between the heated wires and the foil surface, as well as the small wire diameter, the thickness of the high temperature lines were sufficiently minimised and the longitudinal, discrete temperature peaks were constrained to a single pixel. This allowed for the wire locations to be easily identified based on the thermal image and reduced the associated uncertainty. To perform the IR camera spatial calibration, the spatial calibrator was placed in contact with the foil, as per Figure 4.29(b), and the focus of the IR camera was adjusted until the high temperature lines produced were well-defined and sufficiently sharp. IR surface temperature data was thus recorded over a 20 second period and subsequently processed using a MATLAB script to obtain the downstream pixel dimension dx . This IR camera spatial calibration was performed regularly prior to testing to correct for any drift in the camera focus over time.

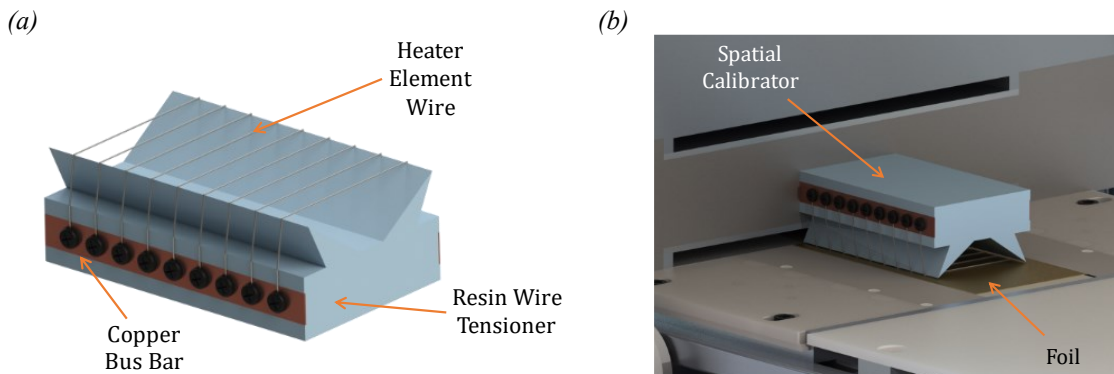


Figure 4.29: (a) IR camera spatial calibrator (b) placement of spatial calibrator in experimental setup.

A sample thermal image acquired is presented in Figure 4.30(a), where the existence of high temperature lines is clear. Using the MATLAB *findpeaks()* function, the temperature peaks were identified, as outlined in Figure 4.30(b), and the constant wire spacing of 10 mm was divided by the average number of pixels between longitudinal temperature peaks to obtain a value for dx , such that $dx \approx 2.27 \times 10^{-4} \text{ m}$. The transverse pixel dimension dz was subsequently calculated based on the IR camera mount angle of 5.5° to account for the slight distortion induced, *i.e.*, $dz = dx \times \cos(5.5) \approx 2.26 \times 10^{-4} \text{ m}$. This method of spatial calibration also allowed for any distortion induced as a result of the camera lens itself to be identified, however the regular longitudinal spacing of the temperature peaks in Figure 4.30(b) over the total distance implied that this effect was negligible and could therefore be disregarded.

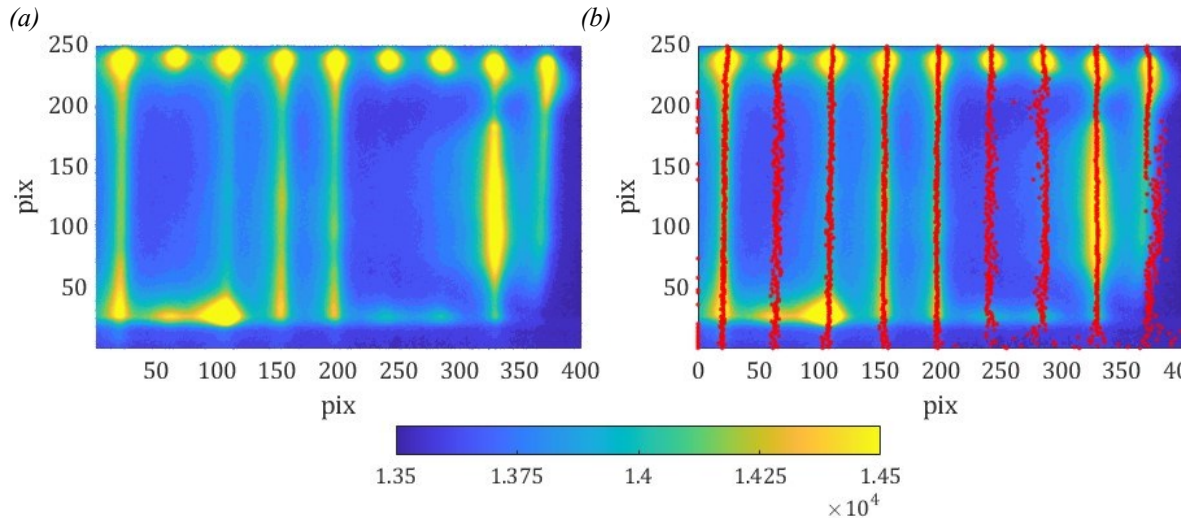


Figure 4.30: (a) Sample surface temperature contour plot acquired during the IR camera spatial calibration and (b) location of temperature peaks as per the MATLAB script.

4.4.4. PIV Spatial Calibration

Prior to the acquisition of any PIV data, it was necessary to perform a spatial calibration of the image plane to identify the displacement of the seeding particles, as well as to remove any induced distortion resulting from the viewing angle of the high-speed camera. To achieve this, a custom calibration target was constructed based on the standard LaVision calibration plate and its dimensions occupied the entire xz -plane along the jet centreline, where a series of spacers ensured the calibration target remained aligned in the correct plane. The front face of the calibration target consisted of a rectangular array of 2 mm diameter dots, the horizontal and vertical spacing between which was 10 mm. In addition, the location front and bottom edges were highlighted using arrows, which was helpful during the camera alignment process. An image of the portion of the spatial calibration target inside the final FOV is shown in Figure 4.31(a), where the tip of each arrow is just inside the FOV as a means to minimise the appearance of the laser reflections from the solid boundaries. The spatial calibration was conducted using the in-built calibration wizard in the DaVis 10 software, which required the user to input information regarding the array spacing on the front face of calibration target. Following this, a live image was captured and three points on the calibration target were selection. The software hence used this user input to identify the remaining points on the calibration target, derive a spatial calibration curve to calculate the pixel length, and remove the image distortion. The final calibrated FOV is presented in Figure 4.31(b), which resulted in a pixel length of 0.02 mm.

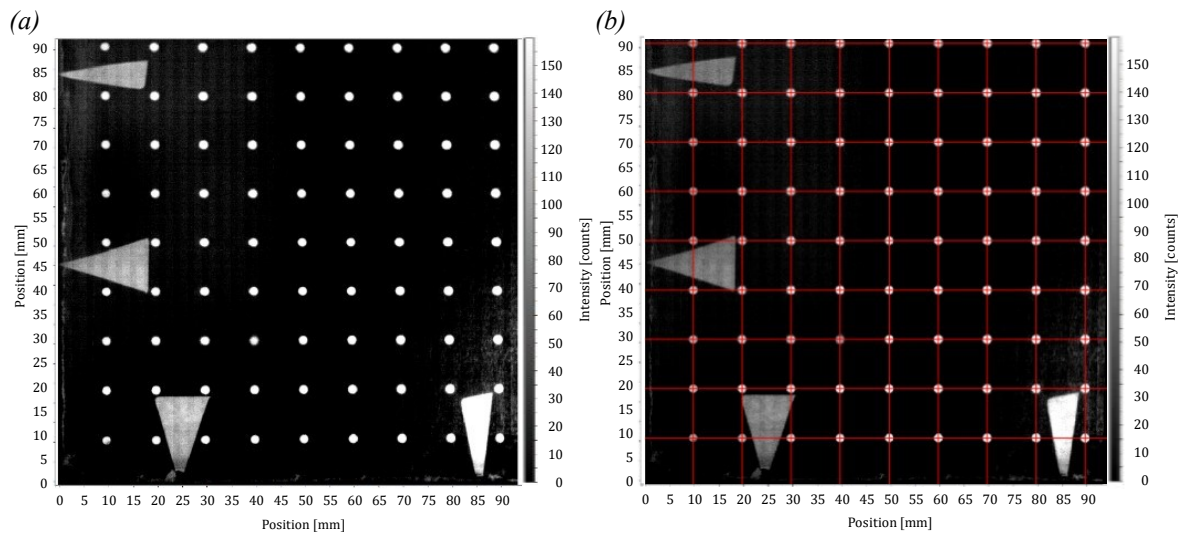


Figure 4.31: (a) PIV spatial calibration target (b) Final spatial calibration and distortion correction of PIV field-of-view.

4.5. Experimental Uncertainty Analysis

The following sections describe the procedures undertaken to quantify the uncertainty associated with all quantities used as part of this investigation, which includes parameters that were both directly measured or derived through appropriate post-processing. The uncertainty analysis was conducted based on that outlined in the ‘Guide to the Expression of Uncertainty in Measurement’ [156], where the combined influence of both Type A and Type B uncertainties for each variable of interest were considered.

Type A uncertainties are understood as uncertainties identified through statistical means based on repeat measurements by the experimenter [157]. In this instance, the probability of attaining a certain value is assumed to be normally or Gaussian distributed about the true value, where the best estimate of the true value corresponds to the arithmetic mean of the repeat measurement. The true value for a directly measured variable x is provided by equation 4.46, where n repeat measurements of x were acquired, denoted by $x_1, x_2, \dots, x_n$. The standard deviation of the set of measured values from the true value is hence calculated using equation 4.47, from which the standard uncertainty $u_a(x)$ can be derived using equation 4.48. To obtain the expanded uncertainty value for the measurand, which specifies an uncertainty interval for the measurand for which a certain probability can be assumed, the standard uncertainty is multiplied by a suitable coverage factor k , such that $U(x) = ku_a(x)$. For a normally distributed uncertainty interval calculated based on $n \geq 30$ statistically independent samples, a 95% level of confidence can be attained using a coverage factor of 1.96.

$$\bar{x} = \frac{1}{n} \sum_{i=1}^{i=n} x_i \quad (4.46)$$

$$\sigma_x = \sqrt{\frac{1}{n-1} \sum_{i=1}^{i=n} (x_i - \bar{x})^2} \quad (4.47)$$

$$u_a(x) = \frac{\sigma}{\sqrt{n}} \quad (4.48)$$

Conversely, Type B uncertainties refer to the use of third-party information to derive a standard uncertainty value, such as books, calibration certificates, data tables or manufacturers specifications [157]. However, as the quoted level of accuracy $\pm a$ from a third-party source is generally not accompanied by additional information outlining the basis of its calculation or an associated level of confidence, an assumption must be made regarding the probability distribution within the stated upper and lower limits for the measured value, denoted by a_+ and a_- respectively. For this case, the true value is accepted as the midpoint between a_+ and a_- , such that

$$\bar{x} = \frac{a_+ + a_-}{2} \quad (4.49)$$

and, in the absence of information otherwise, each value inside a_+ and a_- can either be considered equally probable (rectangular distribution) or more probable towards the midpoint compared to the outer limits (triangular distribution). The step function discontinuity associated with a rectangular probability distribution is generally unphysical, the choice of a triangular probability distribution can be more realistic [156]. The standard deviation, *i.e.*, the standard uncertainty, for a triangular probability distribution is hence calculated

using equation 4.50, which can be multiplied by a coverage factor of 2 to obtain a 95% confidence level with the uncertainty interval.

$$u_b(x) = \frac{a}{\sqrt{6}} \quad (4.50)$$

The combined uncertainty $u_c(x)$, which accounts for the combined effect of all Type A and Type B standard uncertainties for a given measured variable, is calculated using the root sum of squares (RSS) method, such that

$$u_c(x) = \sqrt{u_1(x)^2 + u_2(x)^2 + \dots + u_n(x)^2} \quad (4.51)$$

where the measurand x has n contributing sources of uncertainty of either Type A or B [158]. For variables of interest not acquired through direct measurement, and instead derived by the formula $y = f(x_1, x_2, \dots, x_n)$ for which the standard uncertainty of each input variable is known, the uncertainties are propagated to obtain a standard uncertainty value $u_c(y)$ using equation 4.52, such that

$$u_c(y) = \sqrt{\sum_{i=1}^n \left(\frac{\partial f}{\partial x_i} \right)^2 u^2(x_i)} \quad (4.52)$$

where $\frac{\partial f}{\partial x_i}$ are referred to as the sensitivity coefficients for each input variable [156]. Equation 4.52, however, assumes no correlation exists between uncertainties $u(x_1), u(x_2), \dots, u(x_n)$, and, in the instance where $u(x_1)$ and $u(x_2)$ are correlated, an additional covariance term must be included equation 4.52, *i.e.*,

$$u_c(y) = \sqrt{\sum_{i=1}^n \left(\frac{\partial f}{\partial x_i} \right)^2 u^2(x_i) + 2\rho(x_1, x_2) \frac{\partial f}{\partial x_1} \frac{\partial f}{\partial x_2} u(x_1)u(x_2)} \quad (4.53)$$

In equation 4.53, $\rho(x_1, x_2)$ is the correlation coefficient which describes the mutual dependence of variables x_1 and x_2 , where $-1 \leq \rho(x_1, x_2) < 1$. A value of $\rho(x_1, x_2)$ close to 0 therefore indicates no correlation exists between the measured variables, while $\rho(x_1, x_2) = 1$ or $\rho(x_1, x_2) = -1$ indicates a strong positive or negative correlation, respectively [156]. The value of the correlation coefficient can be estimated using equation 4.54, where the measured variables x_1 and x_2 were each sampled K times [156].

$$\rho(x_1, x_2) = \frac{\frac{1}{K-1} \sum_{j=1}^K (x_1 - \bar{x}_1)(x_2 - \bar{x}_2)}{\sigma_{x_1} \sigma_{x_2}} \quad (4.54)$$

In many cases, the variable of interest y may be derived from direct measurements of a variable x using a previously established calibration curve, *i.e.*, equation 4.55. The uncertainty y is obtained using equation 4.56, where $u_c(x)$ is the standard uncertainty associated with the measurand x , $u_{reg}^2(y)$ is the uncertainty of the fit of the calibration curve regression to the original calibration dataset (x_{cal}, y_{cal}) , and $u_{cal}(y)$ is the uncertainty in y_{cal} , where $x_{cal} = x_1, x_2, \dots, x_n$ and $y_{cal} = y_1, y_2, \dots, y_n$. For simplicity, the regression uncertainty is calculated based on the standard deviation of the datapoints in (x_{cal}, y_{cal}) about the curve fit, *e.g.*, equation 4.57, where $\hat{y}_i$ is the predicted value using each value of x_{cal} in equation 4.55 and n is the number of datapoints. It should be noted that the inclusion of the $n - 2$ term in equation 4.57 indicates that no regression uncertainty exists for a calibration dataset consisting of 2 or less data points [156, 158]

$$y = a_0 + a_1x + a_2x^2 + \dots + a_Nx^N \quad (4.55)$$

$$u_c(y) = \sqrt{u_{cal}(y)^2 + u_{reg}^2(y) + \left(\frac{\partial y}{\partial x} u_c(x)\right)^2} \quad (4.56)$$

$$u_{reg}(y) = \sqrt{\frac{1}{n-2} \sum_{i=1}^{i=n} (y_i - \hat{y}_i)^2} \quad (4.57)$$

The following sections therefore discuss the use of the aforementioned uncertainty principles in the context of the present investigation to determine a suitable uncertainty value for each parameter of interest. In each case, to include an additional factor of safety, a coverage factor of 2 was used to obtain the expanded uncertainty value relating to a 95% confidence interval.

4.5.1. Thermocouple Uncertainty

The uncertainty associated with each thermocouple used in the experimental setup is summarised in Table 4.5. In each case, the regression uncertainty $u_{reg}(T)$ associated with the respective calibration curves were determined using 4.57, and combined with the remaining Type A and Type B uncertainties using equation 4.56. The Type A uncertainty for all thermocouples was derived from the statistical variation of the input temperature measurement using equations 4.46 to 4.48, and the Type B uncertainties consisted of the NI DAQ module accuracy of $\pm 0.05^\circ\text{C}$, the quoted uncertainty of the RTD calibration probe of $\pm 0.05^\circ\text{C}$, and the standard uncertainty of $\pm 0.5^\circ\text{C}$ associated with a T-type thermocouple manufactured to a class 1 tolerance. For each class B uncertainty, a triangular probability distribution was assumed, and hence the standard uncertainty value was interpreted from the manufacturer's specifications using equation 4.50.

Table 4.5: Summary of uncertainties in thermocouple temperature measurement.

<i>Uncertainty</i>	T_w [°C]	T_o [°C]	T_{ag} [°C]	T_{amb} [°C]	T_{foil1} [°C]	T_{foil2} [°C]	T_{foil3} [°C]	T_{foil4} [°C]	T_{block1} [°C]	T_{block2} [°C]	T_{block3} [°C]
$u_{reg}(T)$	0.03	0.04	0.05	0.05	0.03	0.05	0.06	0.08	0.06	0.04	0.06
$u_{DAQ}(T)$	0.02	0.02	0.02	0.02	0.02	0.02	0.02	0.02	0.02	0.02	0.02
$u_{RTD}(T)$	0.02	0.02	0.02	0.02	0.02	0.02	0.02	0.02	0.02	0.02	0.02
$u_{tot}(T)$	0.20	0.20	0.20	0.20	0.20	0.20	0.20	0.20	0.20	0.20	0.20
$u_{stat}(T) (\times 10^4)$	8.04	7.21	9.73	35.46	13.71	13.34	13.59	13.40	13.97	14.04	13.90
$u_c(T)$	0.21	0.21	0.21	0.21	0.21	0.21	0.22	0.22	0.21	0.21	0.22
$U_c(T)$	0.42	0.42	0.42	0.43	0.42	0.42	0.43	0.44	0.43	0.42	0.43

4.5.2. Airflow Uncertainty

The uncertainty accompanying each airflow parameter used as part of the current investigation is summarised in Table 4.6. The uncertainty was initially determined for the differential pressure measurement associated with the wall jet and the offset jet, which consisted of the Type A uncertainty derived from statistical variation in the measured values $u_{stat}(p)$ and relevant Type B uncertainties. These included the induced uncertainty from the NI DAQ module of $\pm 0.02\%$ of the full-scale output (%FS) and the differential pressure sensor uncertainty as per the manufacturer's specifications, *i.e.*, the sensor accuracy of $\pm 0.5\%$ FS, the sensor repeatability of 0.3% FS, and resolution uncertainty of $\pm 0.1 \text{ Pa}$. The uncertainty associated with the volumetric

flowrate was hence derived using equation 4.57 to include the contribution from differential pressure uncertainty $u_p(V)$, the regression uncertainty from the calibration curve $u_{reg}(V)$, and the uncertainty of the volumetric flowrate measurement during the calibration process $u_{cal}(V)$. The value of $u_{cal}(V)$ was propagated from the flow velocity sensor uncertainty using equation 4.52, which had an average uncertainty of 0.7%FS based on the flow velocity sensor calibration cert.

Table 4.6: Summary of uncertainties associated with airflow measurements quantities.

Pressure			Volumetric Flowrate			Propagated Airflow Uncertainties		
Uncertainty	p_w [Pa]	p_o [Pa]	Uncertainty ($\times 10^4$)	V_w [m ³ /s]	V_o [m ³ /s]	Variable y	$u_c(y)$	$U_c(y)$
$u_{manf}(p)$	0.153	0.153	$u_{reg}(V)$	1.12	1.45	U_w [m/s]	0.74	1.47
$u_{DAQ}(p)$	0.009	0.009	$u_p(V)$	8.68	6.58	U_o [m/s]	0.63	1.27
$u_{stat}(p)$	0.005	0.007	$u_{cal}(V)$	6.32	6.42	Re_w	341.07	682.14
$u_c(p)$	0.153	0.153	$u_c(V)$	10.80	9.31	Re_o	293.79	587.57
$U_c(p)$	0.307	0.307	$U_c(V)$	21.59	18.61	V_r	0.12	0.24

The uncertainty associated with all remaining airflow parameters were propagated using equation 4.52 based on the corresponding formula from which they were derived, which included the uncertainty of the jet velocity at the nozzle exit $u_c(U_{jet})$, the jet Reynolds number $u_c(Re_{jet})$ and the jet velocity ratio $u_c(V_r)$. It should be noted that $u_c(V_r)$ value shown in Table 4.6 relates to the maximum achievable V_r of 2.18, for which the uncertainty was greatest. All other uncertainty values were found to remain relatively consistent over the investigated airflow range.

4.5.3. IR Camera Uncertainty

The uncertainty of the IR camera was considered separately in terms of the uncertainty in the temperature measurement and the spatial uncertainty associated with the pixel dimensions, where the quantities acquired are summarised in Table 4.7. First, the surface temperature uncertainty $u_c(T_s)$ was derived using equation 4.51, which accounted for the regression uncertainty of the calibration curve $u_{reg}(T_s)$, the temperature uncertainty of the foil during the calibration process $u_{cal}(T_s)$, where $u_{cal}(T_s) = u_c(T_{foil1})$ as per Table 4.5, and the statistical uncertainty associated with the temperature fluctuations $u_{stat}(T_s)$. The accuracy of the IR camera as per the manufacturer's specifications was also included in the calculation of $u_c(T_s)$, i.e., $\pm 2^\circ\text{C}$, which was as per the Type B recommendation.

The spatial uncertainty was interpreted as the uncertainty associated with the detection of the heater wire locations by the IR camera during the spatial calibration process, termed $u_c(x_{wireIR})$ in Table 4.7. This value was again derived using equation 4.51, and included the uncertainty associated with the specified spatial resolution of the IR camera of 0.68 mrad and the statistical uncertainty due to the variation in the wire location $u_{stat}(x_{wireIR})$. It should be noted that the statistical uncertainty value accounted for both the fluctuation of the wire location with time, i.e., $\pm 0.18 \text{ pix}$, as well as the variation in the detection of the wire location along itself in the z -direction, i.e., $\pm 0.08 \text{ pix}$. The uncertainty associated with the pixel dimensions dx and dy , as well as the pixel area dA , were propagated using equation 4.52, where a tolerance of $\pm 0.1 \text{ mm}$ was assumed for the actual spacing between the heater element wires.

Table 4.7: Summary of thermal and spatial uncertainty values associated with the IR camera.

IR Camera Uncertainty Sources			Propagated IR Camera Uncertainties			
<i>Uncertainty</i>	<i>Unit</i>	<i>Value</i>	<i>Variable y</i>	<i>Unit</i>	$u_c(y)$	$U_c(y)$
$u_{manf}(T_s)$	[°C]	0.82	T_s	[°C]	0.89	1.78
$u_{reg}(T_s)$	[°C]	0.29	x_{wireIR}	[pix]	0.45	0.90
$u_{cal}(T_s)$	[°C]	0.21	dx	[m]	3.96×10^{-6}	7.92×10^{-6}
$u_{stat}(T_s)$	[°C]	2.47×10^{-3}	dz	[m]	3.94×10^{-6}	7.88×10^{-6}
$u_{stat}(x_{wireIR})$	[pix]	0.20	dA	[m ²]	1.26×10^{-9}	2.53×10^{-9}
$u_{res}(x_{wireIR})$	[pix]	0.40				

4.5.4. Uncertainty in Heat Transfer Measurements

The propagated uncertainties associated with each heat transfer quantity of interest are summarised in Table 4.8. The uncertainty associated with each term in the foil energy balance was initially determined, where the values of $u_c(q''_{rad})$, $u_c(q''_{cond})$, and $u_c(q''_{gen})$ were each derived using equation 4.52. For the lateral and capacitive terms, however, the correlation between the surface temperatures of the adjacent pixels had to be considered, as well as the temperature correlation with time for the pixel of interest. The required correlation coefficients were derived using equation 4.54 for a sample dataset, following which the values for $u_c(q''_{cap})$ and $u_c(q''_{lc})$ were calculated using equation 4.53. The uncertainty of the convective heat transfer $u_c(q''_{conv})$ could hence be derived using equation 4.52, which was found to remain relatively consistent over the experimental surface temperature range.

Table 4.8: Summary of propagated uncertainties in heat transfer quantities.

Uncertainties in Foil Energy Balance			Propagated Uncertainties in Heat Transfer quantities				
<i>Uncertainty</i>	<i>Unit</i>	<i>Value</i>	<i>Variable y</i>	<i>Unit</i>	$u_c(y)$	$U_c(y)$	$U_c(y)/ y $
$u_c(q''_{gen})$	[W/m ²]	51.48	$h_{(T_s \approx 30^\circ C)}$	[W/m ² K]	14.31	28.61	19.75%
$u_c(q''_{cond})$	[W/m ²]	2.56	$h_{(T_s \approx 45^\circ C)}$	[W/m ² K]	2.96	5.92	10.69%
$u_c(q''_{rad})$	[W/m ²]	2.01	$h_{(T_s \approx 60^\circ C)}$	[W/m ² K]	1.54	3.09	9.39%
$u_c(q''_{cap})$	[W/m ²]	3.38	$Nu_{x(T_s \approx 30^\circ C)}$	[-]	3.82	7.64	19.75%
$u_c(q''_{lc})$	[W/m ²]	15.6	$Nu_{x(T_s \approx 45^\circ C)}$	[-]	0.77	1.55	10.70%
$u_c(q''_{conv})$	[W/m ²]	53.99	$Nu_{x(T_s \approx 60^\circ C)}$	[-]	0.40	0.80	9.40%
$U_c(q''_{conv})$	[W/m ²]	107.99					

Finally, the uncertainty associated with the surface heat transfer coefficient $u_c(h)$ and the local Nusselt number $u_c(Nu_x)$ were propagated using equation 4.52. Unlike previous quantities, the values of $u_c(h)$ and $u_c(Nu_x)$ were noted to vary considerably with temperature, as shown in Table 4.8, where 30°C, 45°C and 60°C represent the respective minimum, average, and maximum surface temperatures encountered over experimental testing. For both quantities, the relative expanded uncertainty, $U_c(y)/|y|$, was greatest for lower surface temperatures and decreased with increasing T_s , as expected. The values of $U_c(h)/|h|$ and $U_c(Nu_x)/|Nu_x|$ are each plotted in Figure 4.32 for a range of T_s values, where it is clear to see that the decrease in uncertainty with increasing T_s is somewhat logarithmic. This means that the high uncertainty values associated with $T_s = 30^\circ C$ decays

rapidly as T_s is increased, leaving the uncertainty associated with the remaining surface temperature values inside acceptable limits. As the majority of pixels were found to have T_s values between 40°C to 50°C for a typical test case, the uncertainty in the derived heat transfer quantities for the present experimental were considered adequate.

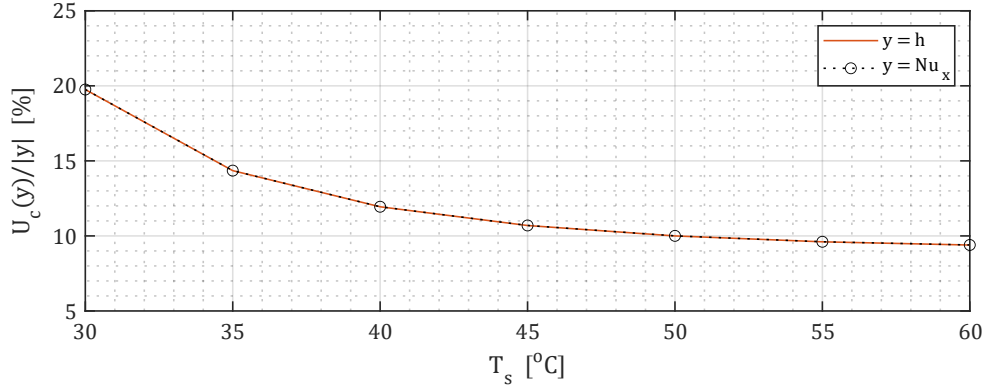


Figure 4.32: Relationship between T_s and the relative expanded uncertainty associated with h and Nu_x .

4.5.5. Uncertainty in Particle Image Velocimetry

Due to the complexity of the setup required for a PIV study, many different potential sources of error are introduced to the experimental facility. These errors can be due to system components errors, flow errors, or evaluation errors, many of which have been previously discussed in sections 4.1.3 and 4.2.2. Careful consideration is therefore required at each phase of the experimental setup and data post-processing procedure to minimise the contribution from each.

The first system component error generally considered is the misalignment of the laser plane. In this scenario, a disparity in the alignment of the laser plane with respect to the flow direction can result in the acquisition of flow velocity data of particles not along the plane of interest. Additional perspective errors can also arise from out-of-plane motion of the seeding particles, which can be minimised by reducing the FOV with an appropriate lens, but cannot be completely omitted without the introduction of a second camera [132, 159]. Differences between the alignment or light intensity distribution of successive laser pulses also contributes towards the error in PIV measurement and must be corrected through appropriate image pre-processing. Further component errors can be derived from timing and synchronisation issues, where delays in the laser pulses of the same order as Δt can be a cause for concern. This was not the case for the present investigation, however, where jitter for the Nd:YLF laser was expected to be of the order of 1 ns [140].

Numerous induced component errors can be derived from the choice of seeding particle, including particle response or lag, particle size, and/or seeding density. As previously discussed, the existence of a slip velocity between the fluid flow and the tracer particle can contribute towards a significant error in the measured flow velocity, however, the response time of the chosen seeding particle with respect to the flow field can be deemed acceptable if $St_K < 10^{-1}$ [132]. In addition to this, inconsistencies in the seeding density and particle size can lead to an uneven illumination of the flow field, which again must be accounted for using image pre-processing techniques. Finally, the phenomenon of ‘peak-locking’ can occur when the resulting particle image is less than 1 pixel, which can introduce a bias to the particle displacement towards the closest integer pixel value [159].

Flow errors relate the causation of the error in the velocity measurement to the flow regime itself. The existence of high velocity gradients, fluctuations or streamline curvature in the flow field may exacerbate the velocity slip of the seeding particles, or could be beyond the scope if the PIV solving algorithm [159]. The final source of error is therefore evaluation errors, where the selection of an appropriate interrogation window size, shape, overlap *etc.* is crucial and must be tailored to suit the flow field in question, as decreasing the interrogation window size enhances the resolution of the result while increasing the uncertainty. [159].

The quantification of the errors discussed above, in terms of an uncertainty value, is difficult and much focus has therefore been given in recent times to the advancement of PIV uncertainty evaluation techniques [159]. The velocity components in particle image velocimetry are derived using equation 4.58, such that

$$u = \frac{\Delta x}{\Delta t} = \frac{\Delta X}{M \Delta t} \quad (4.58)$$

where the magnification factor M relates the particle displacement in the object plane Δx to displacement in the image plane ΔX . As the uncertainty associated with M and Δt are relatively small compared to the total uncertainty, the uncertainty in the velocity measurements in a PIV system are primarily derived from the uncertainty in the measured particle displacement [159]. For the present investigation, the uncertainty in the particle displacement was outputted directly from the DaVis 10 software, which quantifies the displacement uncertainty based on the difference between the two time-successive interrogation windows from which the displacement vector was derived [153]. This difference is identified as an asymmetry in the correlation peak, which would otherwise be symmetrical for a set of perfectly matched images. The uncertainty of the displacement vector in question is thus derived following a statistical analysis of the fluctuating contribution of each pixel to the correlation peak shape, the repetition of which allows for the generation an uncertainty field for the total time-averaged field-of-view [153, 160]. As additional fluctuations were expected to be introduced to the flow field through the use of the dual jet, it was decided that more representative depictions of the PIV uncertainties associated with the experimental setup would be attained using the results for a single wall jet flow, the outcome of which are summarised in Table 4.9. As such, the uncertainty associated with u , v , x and y were exported directly for the Davis 10 software, while the respective uncertainties associated with $|U|$, X and Y were derived using equation 4.52, where the uncertainty associated with u and v were comparable to the 1.6% previously obtained by Wang and Tan [19] during their PIV investigation of dual jet flow. The uncertainty in the flow vorticity was calculated using equation 4.59, which was originally derived by Wieneke [140] from equation 4.37 using equation 4.53, where d refers to the grid spacing and the cross correlation coefficient $\rho(2d)$ has an approximate value of 0.725 for an 87.5% overlap.

$$u_c(\omega) = \sqrt{2 \left(\frac{1}{2d} \right)^2 (1 - \rho(2d))(u_c(u)^2 + u_c(v)^2)} \quad (4.59)$$

Table 4.9: Summary of uncertainties in PIV flow quantities for $Re = 12000$ test case.

<i>Variable y</i>	<i>Unit</i>	$u_c(y)$	$U_c(y)$	$U_c(y)/ y $
u	[m/s]	0.11	0.22	0.87%
v	[m/s]	0.08	0.16	4.37%
$ U $	[m/s]	0.13	0.26	1.02%
ω	[s ⁻¹]	2.52×10^{-2}	5.04×10^{-2}	$2.52 \times 10^{-4}\%$
x	[m]	2.8×10^{-6}	5.6×10^{-6}	0.08%
y	[m]	2×10^{-6}	4×10^{-6}	0.06%
X	[–]	4×10^{-4}	7.84×10^{-4}	0.08%
Y	[–]	2.86×10^{-4}	5.71×10^{-4}	0.06%

Chapter 5

Experimental Setup Validation

This chapter assesses the quality of the output from the experimental setup to ensure the experimental findings relating to dual jet flow can be considered accurate and credible. This is achieved, first, by evaluating the implemented flow conditioning measures with respect to the desired flow regime at the nozzle exit and inside the flow domain, *i.e.*, uniform jet exit velocity profile and two-dimensional flow. In addition to this, the repeatability of results output from the experimental test facilities is analysed to ensure reliability. Finally, the measured results relating to a single wall jet (SWJ) and single offset jet (SOJ) are compared with the relevant literature to ensure their behaviour is consistent with the findings of previous studies for similar flow conditions. In doing so, subsequent findings relating to dual jet flow can be appropriately attributed to the interaction of the two jets, as opposed to anomalies induced to the flow by the experimental setup.

5.1. Effectiveness of Airflow Conditioning

The effectiveness of the implemented flow conditioning measures is assessed based on the flow quality at the nozzle exit, as well the two-dimensionality of the flow inside the flow domain. The following sections discuss evidence of the aforementioned flow features in the measured data and evaluate whether the flow conditioning instruments in the experimental test facility were sufficient. Firstly, the flow profile at the nozzle exit acquired from the PIV data is examined with respect to the velocity and turbulence intensity distribution. The planarity of the flow domain is subsequently assessed based on the heat transfer data collected for a series of sample cases.

5.1.1. Flow Quality at Jet Nozzle Exit

As previously discussed in section 4.1.1, the purpose of the flow conditioning system in the experimental setup is to ensure a uniform streamwise velocity profile across the jet width and to minimise the turbulence intensity at the nozzle exit to an appropriate value. As the flow associated with both single offset jets and dual jets is naturally oscillatory, measured fluctuations or flow anomalies just downstream of the nozzle exit due to inherent jet behaviour cannot be discerned from possible shortcomings of the flow conditioning system, and are therefore not an adequate means of assessment. Conversely, the single wall jet exhibits much more stable behaviour, meaning any velocity perturbations present in the flow at the nozzle exit are likely attributed to flaws in the flow conditioning system. The resulting flow field can therefore be used to conduct a more reasonable assessment of the flow quality at the nozzle exit.

Figure 5.1 presents the time-averaged velocity profile, normalised with respect to the maximum velocity, acquired 1 mm downstream of the nozzle exit for the single wall jet at $Re = 12000$, alongside the corresponding turbulence intensity (I) profile. Overall, a relatively flat velocity profile with a slight asymmetry is noted in Figure 5.1(a), which is in line with that expected when designing the one-sided contraction in advance of the nozzle exit. Evidence of boundary layer growth exists in the near-wall region and the steep streamwise velocity gradient between $0.8 \leq Y \leq 1$ is indicative of both the residual effects of boundary layer growth from inside the jet nozzle, as well as the initial stages of a free shear mixing layer with the quiescent fluid. It should be noted, however, that despite the use of a pixel length of 0.02 mm , the resolution within the boundary layer is still quite limited due to the amplified laser reflections along the solid wall boundary, in addition to error induced as a result of the viewing angle of the CMOS camera. In the flattest region of the velocity profile, approximately $0.1 \leq Y \leq 0.8$, a maximum streamwise velocity variation of 0.7 m/s is noted, which is equivalent to approximately 2.15% of the approximate average velocity of 32.5 m/s along this region. In the experimental dual jet study carried out by Wang and Tan [18], a similar velocity profile at the jet exit was noted, which was described as fairly flat, with a maximum variation of 2%, and a boundary layer thickness of 2.5 mm at the solid wall. For the present investigation, as the variation in the streamwise velocity across the nozzle exit is minor compared to the average streamwise velocity and in line with that obtained by Wang and Tan [18], the outlet velocity profile is considered sufficiently uniform for testing purposes.

For $Re = 12000$, a theoretical average streamwise velocity of $u_w = 27.36\text{ m/s}$ is expected at the nozzle exit, which accounts for the measured wall jet temperature at the time of testing. According to Figure 5.1(a), the average wall jet velocity at the nozzle exit, as per the PIV measurements, is 28.76 m/s , which is inclusive of both the boundary layer and mixing layer regions. As the corresponding jet Reynolds number for $u_w = 28.76\text{ m/s}$ is within the expected uncertainty interval for the jet Reynolds number ascertained by means of the differential pressure measurement, the chosen method of flowrate measurement in the experimental test facility, and the corresponding differential pressure calibration, can be considered credible and accurate.

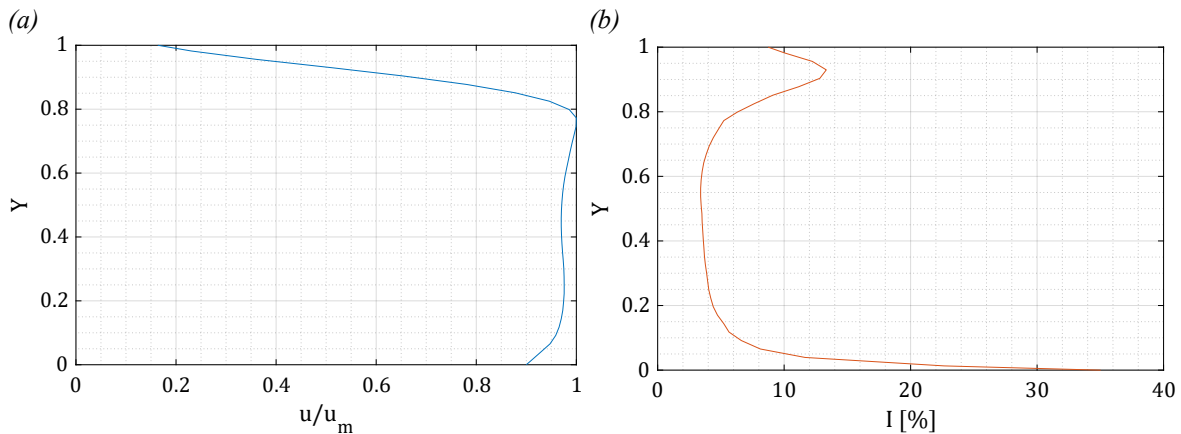


Figure 5.1: Outlet (a) streamwise velocity and (b) turbulence intensity profile for a SWJ at $Re = 12000$.

The turbulence intensity profile at the jet outlet shown in Figure 5.1(b), calculated based on the variation of the velocity magnitude, presents an almost inverse trend compared to the streamwise velocity profile, where the turbulence intensity is lowest in the central region ($0.2 \leq Y \leq 0.7$) and grows towards a maximum value near both the upper and lower boundaries of the nozzle exit. The increased turbulence intensity near the upper boundary of the nozzle exit is to be expected as a result of the mixing inside the free shear layer, however, the higher turbulence intensity towards the solid wall boundary is attributed to both the sheared flow inside the

boundary layer, as well as the increased uncertainty of the PIV measurements in the near-wall region due to limited resolution inside the boundary layer and amplified laser reflections. This assumption is further substantiated by the turbulence intensity distribution across the total flow domain, as described by the contour plot in Figure 5.2, where the progression of the turbulence intensity with downstream distance appears to remain consistent with the growth of the outer shear layer as entrainment with the quiescent fluid increases. Near the solid wall boundary, the thin layer of higher turbulence intensity appeared to thicken somewhat with downstream distance; its presence, however, is relatively minor compared to the overall contour plot.

Based on this assumption, the instance of small regions of high turbulence intensity near the nozzle exit, as per Figure 5.1(b), can be considered unrelated to the upstream flow conditioning, but rather an inherent jet flow feature. Therefore, to assess the performance of the flow conditioning components, focus is given to the central region of low turbulence intensity between $0.2 \leq Y \leq 0.7$, which indicates an exit turbulence intensity of approximately 4%. As Wang and Tan [18] did not report on the turbulence intensity profile or maximum value at the nozzle exit, a direct comparison cannot be made. However, as Wang and Tan [18] adopted water as the working fluid, it is reasonable to assume that the turbulence intensity would have been significantly less than the present investigation. However, as the maximum turbulence intensity value from the present investigation is lower than the target outlet turbulence intensity of 5%, it can be concluded that the turbulence present in the flow at the jet exit has successfully been reduced below an acceptable limit and the flow conditioning system in the experimental setup can be considered adequate, where it should be noted that a turbulent intensity of 5% has been implemented across the majority of published numerical dual jet studies, such as Kumar and Das [1] and Mondal *et al.* [96]. Any resulting flow phenomena in the fluid domain thereafter can confidently be attributed to the downstream development of the jet, as opposed to flow abnormalities emerging from the jet exit.

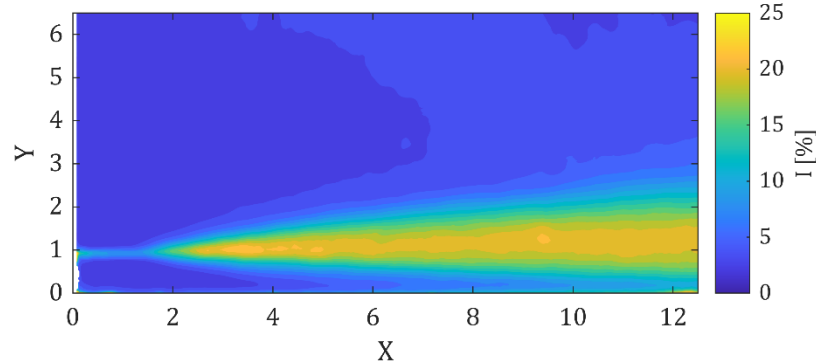


Figure 5.2: Contour plot of turbulence intensity for single wall jet at $Re = 12000$.

5.1.2. Two-Dimensionality

The planarity of the flow is assessed based on the foil surface temperature (T_s) distribution acquired for a set of sample test cases. To ascertain the two-dimensionality in the flow field, it is necessary to establish that T_s is a function of X only and remains constant at a given X location for all Z values inside the test region of interest, *i.e.*, $-3.5 \leq Z \leq 3.5$. Figure 5.3 presents the time-averaged surface temperature distribution of the underside of the test foil for $Re = 5500$ obtained using a single wall jet, a single offset jet and a dual jet for $OR = 1$ and $OR = 5$. In each T_s contour plot, clear bands of constant temperature exist parallel to the nozzle exit, which indicates that changes in the surface temperature occur in the streamwise direction only. This observation

suggests that the experimental setup successfully provides a planar flow inside the flow domain of each jet configuration investigated.

To further examine this assumption for the dual jet with $OR = 1$ and $OR = 5$, the time-averaged local heated wall Nusselt number (Nu_x) profiles along the jet centreline at $Z = 0$ are compared to those obtained along the streamwise location offset from the jet centreline by ± 1 jet width, *i.e.*, $Z = \pm w$, as shown in Figure 5.4. This is similar to the method used by Wang and Tan [1] to validate the two-dimensionality of the flow within their experimental setup, who compare the streamwise velocity profiles attained along $Z = -w, 0$, and w . For the present investigation, a close alignment is observed between the Nu_x profiles corresponding to each Z location for both jet configurations examined, indicating that the flow and heat transfer behaviour is near-identical at all Z locations near the jet centreline for the visible downstream distance.

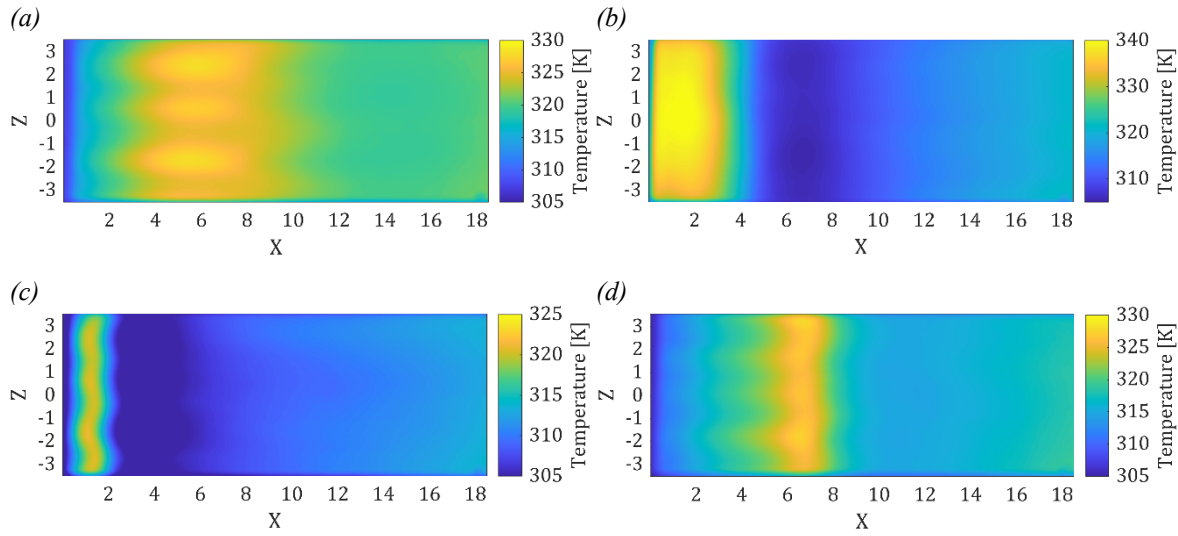


Figure 5.3: Contour plot of the time-averaged surface temperature (T_s) distribution when $Re = 5500$ for (a) a single wall jet, (b) a single offset jet, (c) a dual jet with $OR = 1$ and (d) a dual jet with $OR = 5$.

Having said this, while the overall trend is clear, the Nu_x profiles pertaining to discrete streamwise locations contain a moderate degree of fluctuation, which somewhat hinders the comparison process. The Z –averaged Nu_x profile is therefore introduced to Figure 5.4, where Z –averaged refers to the averaging of all Nu_x values from $-3.5 \leq Z < 3.5$ at each X location. In doing so, a significant reduction in the random noise is observed along the resulting Nu_x profile, therefore providing a clearer indication of the magnitude of the local Nusselt number at a given X location while preserving the overall downstream trend. As the discrete Nu_x profiles at $Z = \pm w$ are positioned within the expected experimental uncertainty of the Z –averaged profile, it can be concluded that the Z –averaging of the Nu_x data is a credible method of removing the excess fluctuations without a loss of information. As a result, all surface variables are averaged with respect to the Z dimension from $-3.5 \leq Z < 3.5$ for the remainder of the investigation. Further to this, as a close alignment is observed between the Z –averaged Nu_x profile and the Nu_x profiles acquired at $Z = \pm w$, it can be deduced that the implemented measures in the experimental setup successfully create the desired two-dimensional flow field inside the fluid domain and the effect of potential 3D flow structures are adequately minimised.

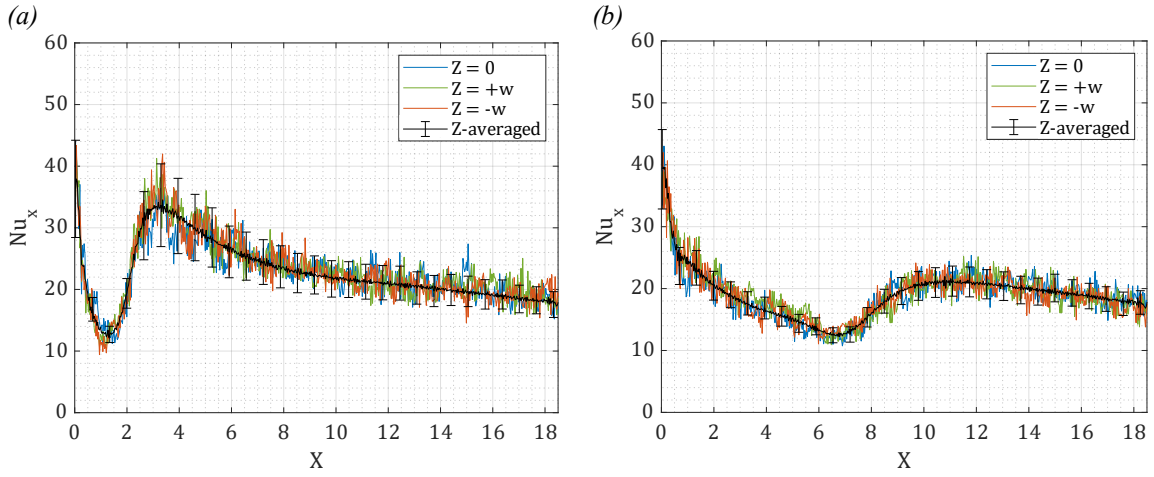


Figure 5.4: Comparison of the Nu_x profiles corresponding to $Z = 0$ and $Z = \pm w$, along with the Z -averaged Nu_x profile, for a dual jet with $Re = 5500$ and $OR = (a) 1$ and $(b) 5$, where the error bars pertain to the Z -averaged dataset.

5.2.Repeatability of Measured Data

To ensure that no time-varying external factor affected the output from the experimental test facility and all instrumentation operated in a consistent manner, it is necessary to assess the repeatability of the measured results. This is achieved through performing a minimum of two test iterations for each set of jet parameters and ensuring the results from each test iteration are aligned with respect to the expected uncertainty interval. Due to the increased time requirement, repeat tests were conducted only during the heat transfer study, as differences in the flow behaviour were considered unlikely if sufficient repeatability was observed in the heat transfer results. A sample of the results acquired from the heat transfer study are shown in Figure 5.5, where the time-averaged Nu_x profiles for multiple test iterations are presented for $Re = 12000$ at $OR = 1$ and $OR = 5$, alongside the experimental uncertainty. A close alignment between the Nu_x profiles is observed for both jet configurations, where the Nu_x profiles relating to succeeding test iterations are within the range of uncertainty values associated with the first test case. The output from the experimental rig can therefore be considered adequately repeatable for testing purposes.

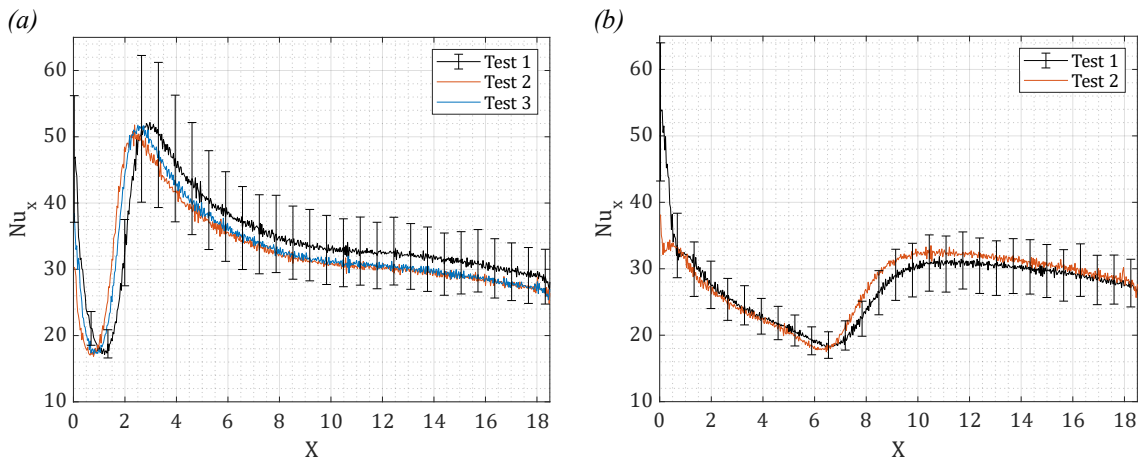


Figure 5.5: Repeatability of time-averaged dual jet Nu_x profiles for $Re = 12000$, where $OR = (a) 1$ and $(b) 5$.

5.3. Comparison with Published Literature

In the absence of published heat transfer data relating to dual jet flow, it is necessary to ensure the credibility of the experimental findings acquired in the present investigation for a dual jet based on the results obtained for a single wall jet (SWJ) and a single offset jet (SOJ). As experimental data for a SWJ and SOJ exist in the published literature to a much greater extent, it is possible to ensure the behaviour of both the SWJ and SOJ from the present experimental facility are consistent with previous experimental studies. Through validating experimental findings for both the SWJ and SOJ, it is possible to ascertain whether the experimental setup is operating correctly and that no abnormalities are introduced to the flow by the test facility that could distort the findings for the subsequent dual jet study.

5.3.1. Single Wall Jet

The following sections discuss the validation of the SWJ experimental data with respect to, first, the flow behaviour and, second, the heat transfer characteristics through comparison with the available published literature.

5.3.1.1. Flow Characteristics

The time-averaged contour plots of velocity magnitude $|U|$ for a single wall jet with $Re = 6000$ and $Re = 12000$ are shown in Figure 5.6(a) and Figure 5.6(b), respectively, onto which the corresponding streamwise velocity vector plots are superimposed. Qualitatively, both plots exhibit typical behaviour for a SWJ, where the downstream flow development is near-identical for both Re cases, only differing in velocity magnitude. For both Re , a distinct potential core region is evident near the jet exit, which gradually diminishes with downstream distance as fluid entrainment with the quiescent ambient fluid acts to slow the average jet velocity. The rate of fluid entrainment increases with streamwise distance, as suggested by the growth of the outer shear layer, and the existence of slower-moving fluid near the solid wall is indicative of boundary layer development.

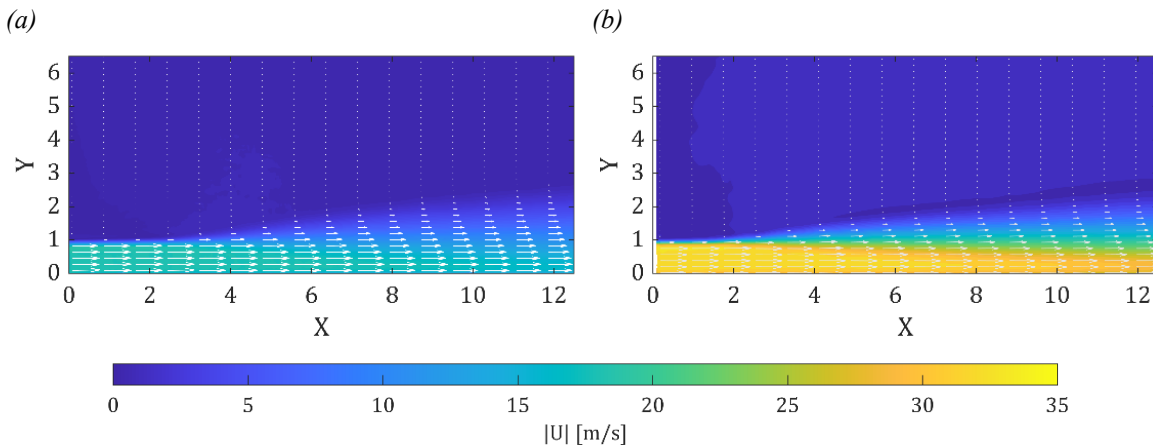


Figure 5.6: Velocity contour and vector plot for single wall jet for $Re = (a) 6000$ and $(b) 12000$.

The decay of the local maximum streamwise velocity u_m with respect to downstream distance is presented in Figure 5.7(a) for a series of Re values, alongside published data from Wang and Tan [19] for $Re = 10000$. The local maximum streamwise velocity in each case is non-dimensionalised with respect to the overall maximum velocity across the entire streamwise domain, $u_{m_{max}}$, which generally occurs at the nozzle exit. The trend acquired for each Re is very similar, where the value of u_m remains relatively constant for some distance downstream, *i.e.*, the potential core, before steadily declining over the remaining downstream distance. The

close alignment between all the profiles indicates that the rate of u_m decay inside the potential core length of a SWJ is relatively unaffected by the jet Reynolds number, however, a slightly greater rate of u_m decay is observed beyond this point for lower Re flows. A good agreement is also observed with that reported by Wang and Tan [19], who obtained a much steadier profile in general but the overall trend is consistent. The minor discrepancies can most likely be attributed to differences in the experimental setup, such as the use of a different working fluid, a shorter nozzle length in advance of the jet exit, differences in the flow conditioning and potential differences in the turbulence intensity at the nozzle outlet, however this parameter was not reported by Wang and Tan [19]. In addition, Wang and Tan [19] noted a larger uncertainty in the time-averaged streamwise velocity of 5% compared to the present study.

To further validate the SWJ flow behaviour, the streamwise velocity profile u at $Re = 12000$ is shown in Figure 5.7(b) for $X = 8, 10$, and 12 , where the inclusion of streamwise velocity data relating to other Re values was deemed unnecessary due to the minor influence of Re on the downstream flow development of the single wall jet. For each X location shown, the streamwise velocity is normalised with respect to the local maximum streamwise velocity u_m and the vertical dimension is scaled with respect to the jet half width associated with the free shear layer. The results from the present investigation indicate that the flow is still developing, as the streamwise velocity profiles at each X location differ somewhat, however, the progressive alignment of the plots with downstream distance signifies the flow is approaching self-similarity. The results are presented alongside that acquired by AbdulNour *et al.* [68] ($X = 10.16, Re = 7700$) and Kim *et al.* [79] ($X = 10, Re = 39000$), where it is clear that the results from the present investigation are approaching that of the published studies upon reaching self-similarity. While minor differences are evident, they can readily be attributed again to differences in the experimental setup, such as the use of a knife-edged nozzle with an aspect ratio of 17.8 in AbdulNour *et al.* [68] or the much lower turbulence intensity of 0.1% at the nozzle exit in the case of Kim *et al.* [79]. Overall, as the SWJ flow data from the current study is generally in line with that presented in the published literature, the SWJ flow behaviour as produced by the experimental test rig can be considered sufficiently validated.

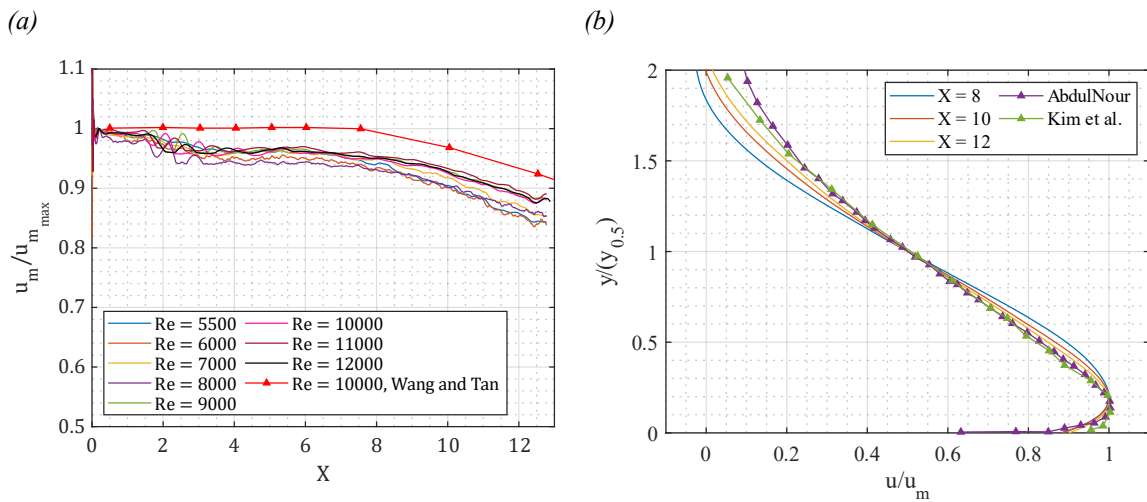


Figure 5.7: (a) Decay of local maximum streamwise velocity with downstream distance for a SWJ compared with that of Wang and Tan [19]. (b) Local streamwise velocity profile for a SWJ for $Re = 12000$ at $X = 8, 10$, and 12 compared to that of AbdulNour *et al.* [68] ($X = 10.16, Re = 7700$) and Kim *et al.* [79] ($X = 10, Re = 39000$).

5.3.1.2. Heat transfer

To validate the measured heat transfer characteristics for the single wall jet, the results are compared with published experimental findings of AbdulNour *et al.* [68] ($Re = 7700$), Mabuchi and Kumada [74] ($Re = 9140$), and Akfirat [73] ($Re = 5540$ and $Re = 8310$). The streamwise Nu_x profiles acquired using the SWJ at various Re is presented in Figure 5.8 alongside the published data, where the resulting trends are generally consistent. For each case, Nu_x initially decreases with streamwise distance from the nozzle exit towards a minimum value, which is synonymous with the thickening of the boundary layer and simultaneous degradation of the potential core region. The location of the local minimum is consistent with the potential core length as provided by Figure 5.7(a), beyond which point the value of Nu_x begins to increase due to the enhanced near-wall mixing following the depletion of the potential core, combined with the laminar-to-turbulent transition of the wall boundary layer. The subsequent decline in Nu_x is indicative of the thickening of the wall boundary layer at a much faster rate and the steady reduction in the mean flow velocity as entrainment effects began to dominate [68, 73]. Increasing the jet Reynolds number does not affect the overall trend, however, Nu_x is enhanced in a uniform manner across all X locations, therefore shifting the Nu_x profile upwards while preserving the locations of the local minimum and maximum, which is similar to the findings of Mabuchi and Kumada [74] and Akfirat [73] for $Re > 1000$.

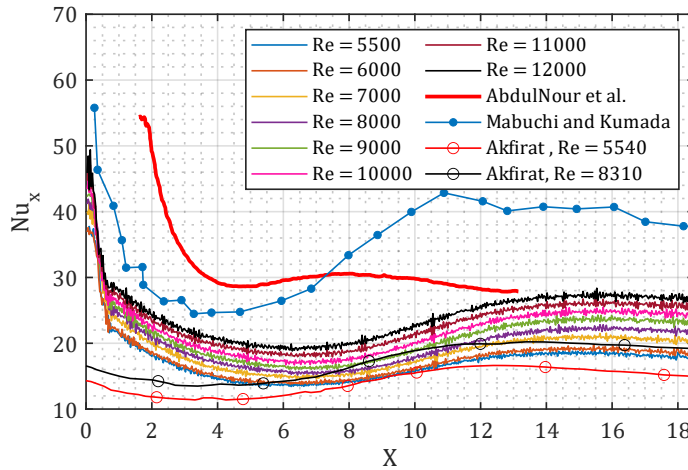


Figure 5.8: Time-averaged Nu_x profiles for a SWJ at various Re compared to the findings of AbdulNour *et al.* [68] for $Re = 7700$, Mabuchi and Kumada [74] for $Re = 9140$ and Akfirat [73] for both $Re = 5540$ and $Re = 8310$.

Despite the similarity in the observed trend, discrepancies are found to exist between the different investigations regarding the magnitude of Nu_x and the streamwise locations of the local minimum and maximum. AbdulNour *et al.* [68] previously noted this inconsistency in the published data and attributed these differences to the use of different nozzle shapes, where a knife-edged nozzle was adopted by AbdulNour *et al.* [68], whereas the remaining investigations used a contoured nozzle. In addition to this, varying aspect ratios were used across all investigations, meaning sidewall effects may be more prominent in certain studies compared to others, as well as the use of an unheated starting length in the case of AbdulNour *et al.* [68] or the preheating of the jet inside the jet nozzle in the case of Akfirat [73]. In each case, details of the turbulence intensity at their respective nozzle exits were not provided, which is again a likely a contributing factor to the differences between the resulting Nu_x profiles. Having said this, the consistency in the overall observed trend compared to the published findings implies the heat transfer characteristics for the SWJ under the flow

conditions dictated by the experimental rig are most likely correct, therefore enabling the completion of the validation process for the single wall jet.

5.3.2. Single Offset Jet

The results pertaining to a single offset jet (SOJ) are validated in similar manner to the single wall jet with respect to both the flow behaviour and the heat transfer characteristics, however, an added complexity is introduced due to the varying offset ratio. The validation procedure for the SOJ is therefore discussed in the following sections, where consideration is given to varying both Re and OR . It should be noted, however, that an equivalent offset ratio value is used throughout the discussion when referring to SOJ flows to remain consistent with the dual jet OR definition and simplify the comparisons. As such, the equivalent OR definition still assumes the presence of the wall jet nozzle, although not in use, where $OR = d/w$, meaning the physical height of the offset jet with respect to the solid wall is identical for a dual jet and a single offset jet with the same offset ratio value.

5.3.2.1. Flow Characteristics

The time-averaged velocity contour plots with the corresponding streamwise velocity vector plots overlaid are shown in Figure 5.9 for a SOJ. Four test cases are chosen for validation, which correspond to $Re = 6000$ and $Re = 12000$ for $OR = 1$ and $OR = 3$.

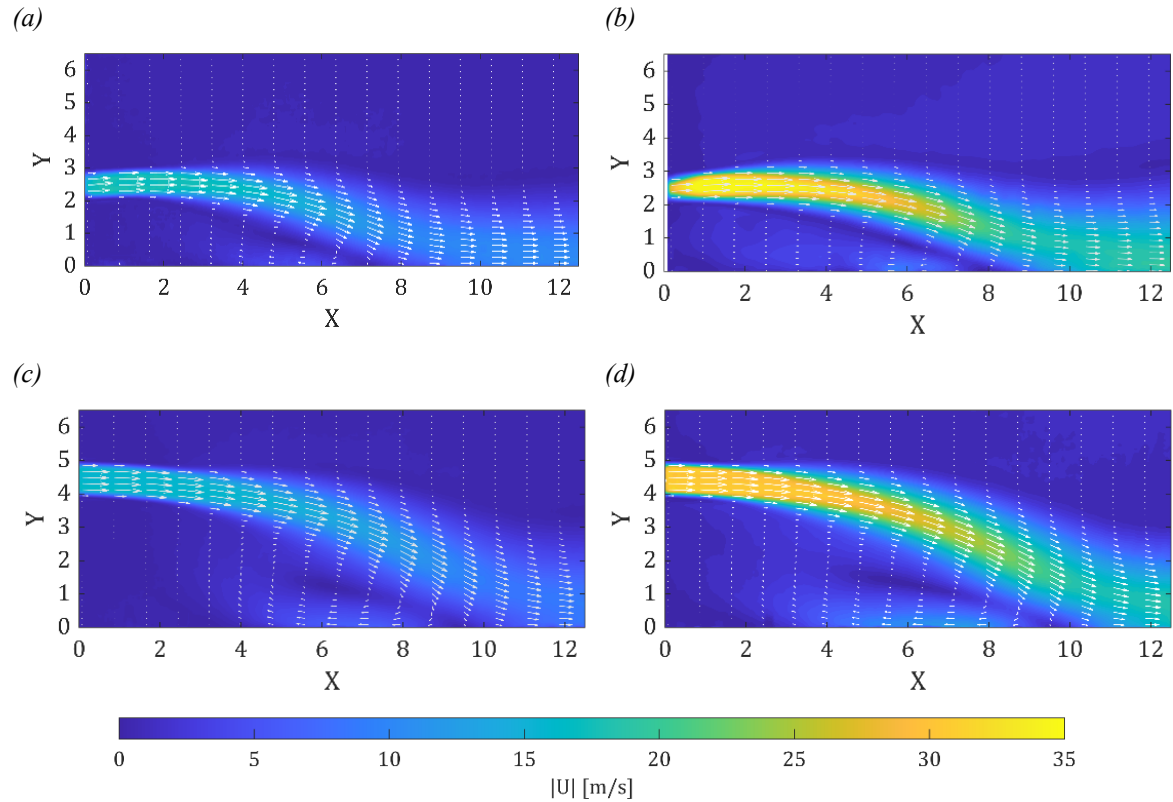


Figure 5.9: Velocity contour and vector plot for a SOJ for (a) $Re = 5500$, $OR = 1$, (b) $Re = 12000$, $OR = 1$, (c) $Re = 5500$, $OR = 3$, and (d) $Re = 12000$, $OR = 3$.

Similar to the SWJ, all plots in Figure 5.9 display typical single offset jet behaviour, where the overall downstream flow development remains consistent for a constant OR values. For all test cases shown in Figure 5.9, a potential core region is created near the jet exit with an initial flow direction parallel to the streamwise dimension. The jet flow subsequently deflects towards the solid boundary due to the creation of a low-pressure

region below the jet exit, *i.e.*, the Coanda effect, and eventually reattaches to the wall at some distance downstream, *i.e.*, X_{rp} . Beyond the reattachment point, the flow develops in a similar manner to a SWJ, where evidence of fluid entrainment and the consequential reduction in the average jet velocity is present for all test cases. While the jet Reynolds number does not affect the downstream jet development, increasing the offset ratio moves X_{rp} further downstream, hence increasing the size of the recirculation zone, which is also evident in results reported by Assouidi *et al.* [134], Gao and Ewing [161], and Kim *et al.* [79].

The streamline plot for the SOJ for $Re = 12000$ is displayed in Figure 5.10 for $OR = 1$ and $OR = 3$. In both cases, a large, slow-moving vortex is present inside the recirculation region, the centre of which is located at (5.87, 0.86) and (5.67, 1.38) for $OR = 1$ and $OR = 3$, respectively. Evidence of a much smaller secondary vortex is also noted where the solid wall meets the jet exit plane, which has been previously noted by many published studies, such as Kim *et al.* [79]. The reattachment point corresponding to $OR = 1$ and $OR = 3$ are identified as the locations at which the $u = 0$ m/s contour intersects the solid wall boundary, thus providing $X_{rp} = 7.91$ and $X_{rp} = 9.56$. Both values obtained are comparable with previously published data relating to SOJ flows, where Wang and Tan [19] obtained $X_{rp} = 6.7$ for $OR = 1$ and Kim *et al.* [79] acquired $X_{rp} \approx 7.2$ and $X_{rp} \approx 10.19$ for equivalent OR values of 1 and 3, respectively.

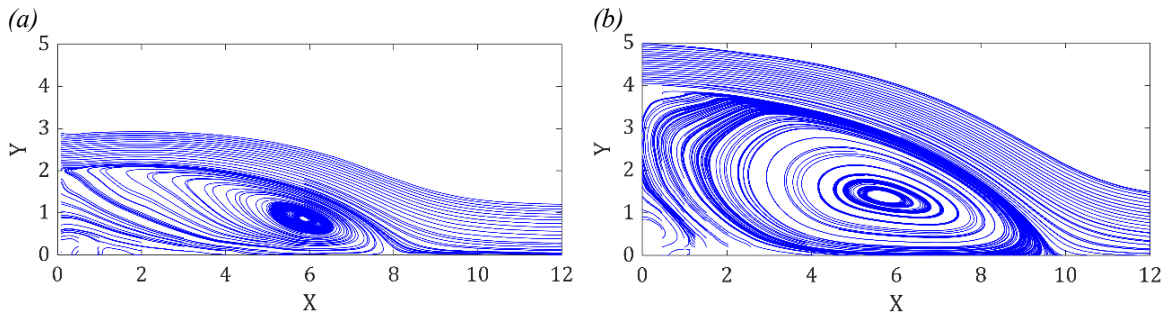


Figure 5.10: Streamline plot showing recirculation region between SOJ and the solid bounding wall for $Re = 12000$ and $OR = (a) 1$ and $(b) 3$.

The decay of the non-dimensionalised maximum streamwise velocity for a series of Re values is presented in Figure 5.11 for $OR = 1$ and $OR = 3$, alongside relevant published data, where the displayed offset ratio values are modified to reflect the definition as per the present study. Similar to the SWJ, the decay of the maximum velocity, and hence the reattachment point, is generally unaffected by the jet Reynolds number, indicated by the close alignment between the respective plots, which was previously noted by Nasr and Lai [76] for $Re \geq 10000$. For $OR = 1$, *i.e.*, Figure 5.11(a), a good agreement between the SOJ data and the published data is observed close to the jet exit, however, a greater disparity exists between the datasets further downstream. The general trend is consistent, with the greatest value of u_m existing near the jet nozzle, followed by a steep reduction in u_m as the jet is deflected towards the solid wall. In the case of Wang and Tan [19], this rate of reduction appears to level off with downstream distance, however, the results from the present investigation suggest a local minimum u_m value near the reattachment point, beyond which u_m begins to increase. This increase in u_m is somewhat evident in the data acquired from Nasr and Lai [76], however considerably less pronounced, who used the same working fluid as the present investigation, *i.e.*, air, and attributed this increase to conversion of static pressure to momentum. The same study noted that this increase in u_m became more exaggerated for higher offset ratios due to a stronger collision with the solid wall, which is evident in the $OR = 3$ data from the current study, as well as that published by Pelfrey and Liburdy [93] for $OR = 6$, both of which

are presented in Figure 5.11(b). As Nasr and Lai [76] had a slightly smaller equivalent OR value of 0.6, the sharper decline, and subsequent rise, in u_m acquired by the present investigation for $OR = 1$ seems reasonable.

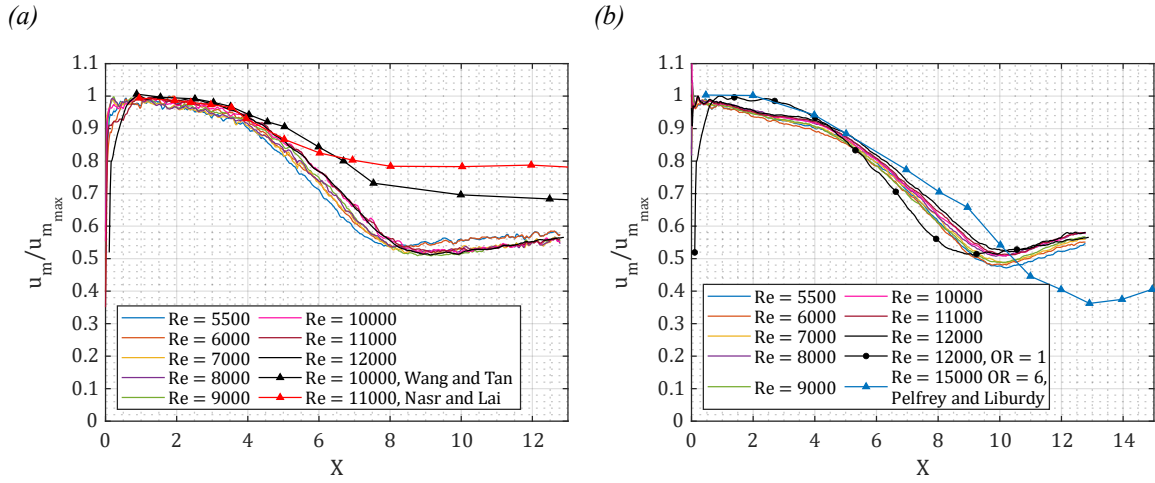


Figure 5.11: Decay of local maximum streamwise velocity with downstream distance for a SOJ with varying Re for (a) $OR = 1$ and (b) $OR = 3$, compared with that of Wang and Tan [19] ($Re = 10000$ $OR = 1$), Nasr and Lai [76] ($Re = 11000$ $OR = 0.6$) and Pelfrey and Liburdy [93] ($Re = 15000$ $OR = 6$).

With regard to the $OR = 3$ data, due to the absence of published experimental data to describe the decay of u_m for a SOJ at $OR = 3$, the validity of the data is assessed based on the shift of the $u_m/u_{m_{max}}$ curve compared to the $OR = 1$ data and that of Pelfrey and Liburdy [93] for $OR = 6$, all of which is presented in Figure 5.11(b). In doing so, it is noted that an increase in the offset ratio moves the location of minimum u_m value further downstream, which is consistent with the increased streamwise position of X_{rp} , while potentially decreasing the value of the minimum u_m value for the lower Re flows. While the magnitude of the minimum u_m value acquired by Pelfrey and Liburdy [93] appears quite low with respect to the current experimental data, the downstream shift in its respective position for an increased OR of 6 is consistent. As the streamwise position of the minimum u_m value for $OR = 3$ appears somewhat mid-way between that of the $OR = 1$ and $OR = 6$ datasets, the shift in the $u_m/u_{m_{max}}$ curve acquired through varying OR for the present experimental setup seems reasonable, which enables the validation of the SOJ flow data for $OR = 3$. The SOJ flow behaviour is therefore regarded as adequately validated with respect to the published literature for all OR considered, meaning subsequent flow data acquired for the dual jet is assumed to have an acceptable level of accuracy.

5.3.2.2. Heat transfer

The time-averaged Nu_x profiles acquired for a SOJ at varying Re are displayed in Figure 5.12 for $OR = 1, 3, 5$ and 7 . The overall Nu_x trend is consistent in each case, where the value of Nu_x sharply decreases at the nozzle exit due to the presence of the secondary recirculation vortex, beyond which the dominating recirculation vortex induces a steady rise in Nu_x towards a local maximum value coinciding with X_{rp} [79]. Following this, as the flow continues to develop with further downstream distance in a similar manner to a SWJ, the value of Nu_x decreases accordingly. This observed trend is constant for all Re investigated, where increasing Re uniformly increases the value of Nu_x at all values of X while maintaining the locations of the local minimum and maximum. Conversely, increasing OR decreases the local maximum Nu_x value, while moving its location further downstream, which is consistent with the growth of the recirculation region as previously discussed.

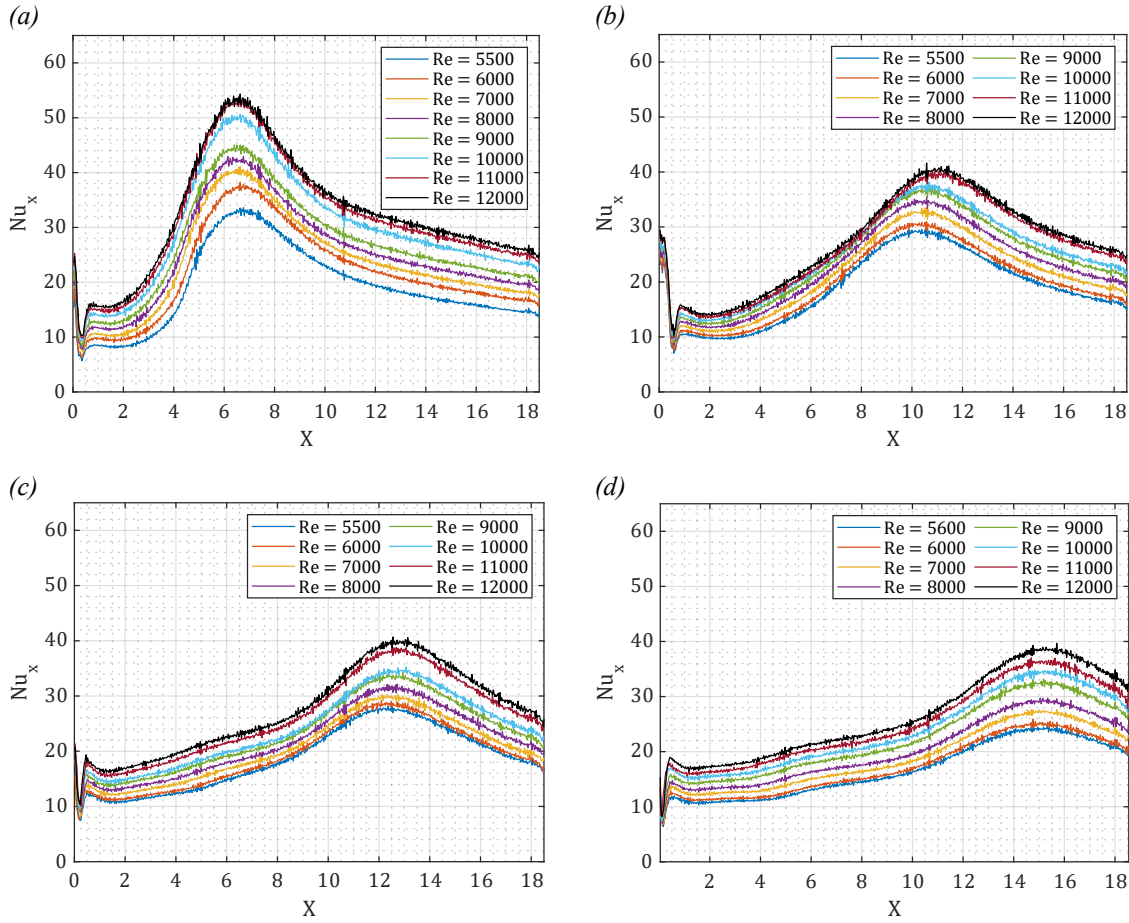


Figure 5.12: Time-averaged Nu_x profiles for a SOJ at various Re for $OR = (a) 1, (b) 3, (c) 5$ and $(d) 7$.

The time-averaged Nu_x profiles acquired using the SOJ for $Re = 12000$ are compared to the published literature for various OR in Figure 5.13. Due to the limited availability of experimental heat transfer data relating to single offset jets at lower Re values, the results are compared to that Kim *et al.* [79] for $Re = 39000$ and Song *et al.* [81] for $Re = 53200$ at similar OR values, both of which adopted the liquid crystal technique. As the jet Reynolds number should affect only the magnitude of Nu_x at each downstream location while preserving the overall trend, the use of SOJ heat transfer data pertaining to a higher Re still provides a viable means of comparison. In doing so, it is noted that the trends associated with Nu_x profiles acquired using the SOJ are consistent with the published literature, where the effect of increasing OR is in agreement with that of Kim *et al.* [79]. The locations of the local minimum and maximum Nu_x values from the present investigation appear to occur slightly upstream compared to the published data, however, discrepancies also exist between that reported by Kim *et al.* [79] and Song *et al.* [81]. As flow instability is an inherent feature of single offset jets, the locations of the local minimum and local maximum Nu_x values are considered sufficiently aligned with the published data to enable the validation of the experimental SOJ data from the present study. In doing so, the acquisition of subsequent heat transfer data for the dual jet flow can be considered both accurate and credible.

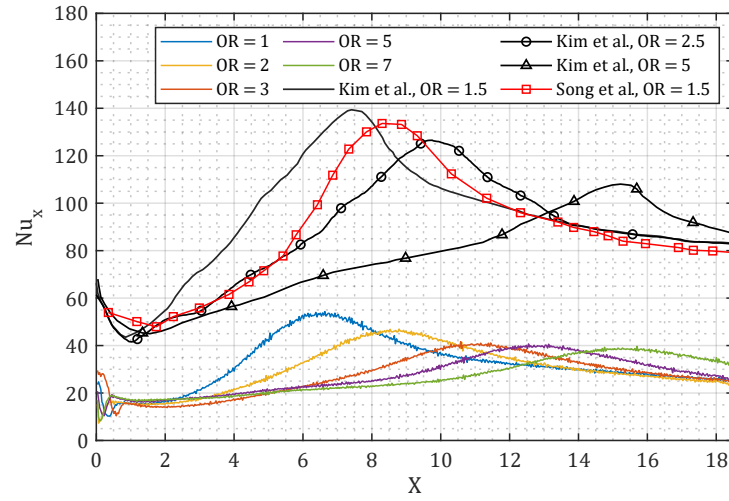


Figure 5.13 Time-averaged Nu_x profiles for a single offset jet at various Re compared to the findings of Kim et al. [45] for $Re = 39000$ and equivalent offset ratio of 1.5.

Chapter 6

Dual Jet Experimental Results

The following sections present the experimental flow and heat transfer results obtained for the dual jet. The dual jet flow behaviour is first discussed in a general sense, where both the time-averaged and time-resolved characteristics are considered, and subsequently related to the corresponding heat transfer findings. The effect of confinement is hence discussed, followed by the effect of varying the jet Reynolds number, offset ratio and velocity ratio.

6.1.Flow Behaviour

Throughout the flow visualisation study, a distinct dual jet flow structure is observed across all jet configurations considered. While adjusting the Re , OR , and V_r influence various aspects of the dual jet flow field, the overall flow behaviour remains relatively consistent. The following sections therefore discuss the flow field results pertaining to a single test case, *i.e.*, $Re = 10000$ and $OR = 1$ for $V_r = 1$, which corresponds to that adopted by Wang and Tan [19] for their PIV study of dual jet flow. First, the time-averaged data is presented, where ‘time-averaged’ refers to the averaging of all 2000 velocity fields obtained within the 1 second testing period used during the PIV study, *i.e.*, 2000 images captured at 2000 Hz . Following this, the time-resolved flow features are discussed.

6.1.1. Time-Averaged Analysis

6.1.1.1. Flow Structure

The time-averaged velocity field $|U|$ obtained for the dual jet with $Re = 10000$ and $OR = 1$ is presented in Figure 6.1, onto which the corresponding velocity vector field is superimposed, where equidistant streamwise intervals of 0.2 in X are used for the velocity vector profiles. In doing so, the typical dual jet behaviour is observed, as previously noted by Wang and Tan [19]. As the jets emerge from their respective nozzle exits, an immediate deflection toward the other jet is noted in Figure 6.1, which creates a distinct region of slow-moving fluid between the jets close to the jet exit plane, *i.e.*, the recirculation region. Further downstream, the end of the recirculation region is identified at a stagnation point in the flow, *i.e.*, the merge point, beyond which the velocity vector profiles exhibit a ‘double-peak’ characteristic, indicating the jets have yet to fully combine. The extent of this deflection in the vector velocity profile is found to gradually diminish with downstream

distance, signifying the interaction between the two jets as they combine into one, before eventually disappearing at the combined point (cp). Once fully combined, the dual jet flow continues to develop in a manner similar to a typical wall jet, where Figure 6.1 provides evidence of jet spreading and increased fluid entrainment with downstream distance, therefore slowing the overall mean jet velocity.

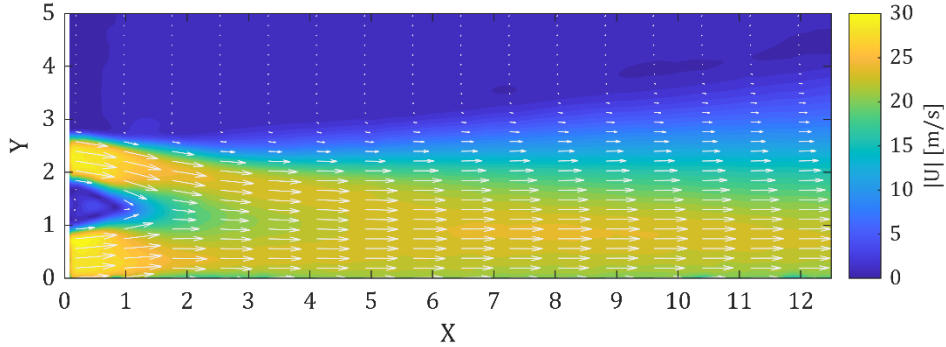


Figure 6.1: Contour and vector plot of time-averaged velocity field for a dual jet with $Re = 10000$ and $OR = V_r = 1$.

To further investigate the dual jet flow structure, the corresponding streamlines are plotted in Figure 6.2(a). The streamlines suggest that a very steady, predictable dual jet flow structure can be obtained through time-averaging the flow data, where the streamlines associated with each jet appear to deflect towards each other in quite a monotonic fashion, join after some downstream distance and continue to flow together in parallel in the streamwise direction. It is interesting to note the near wall flow behaviour in the vicinity of the merge point (mp), where the upward deflection of the wall jet appears to create a region of slow-moving fluid or separation region at the solid wall. The Coanda effect ensures the wall jet remains attached, however, and diverts all streamlines back towards the solid wall. The extent of this streamline deflection from the solid wall appears to be greatest just downstream of mp .

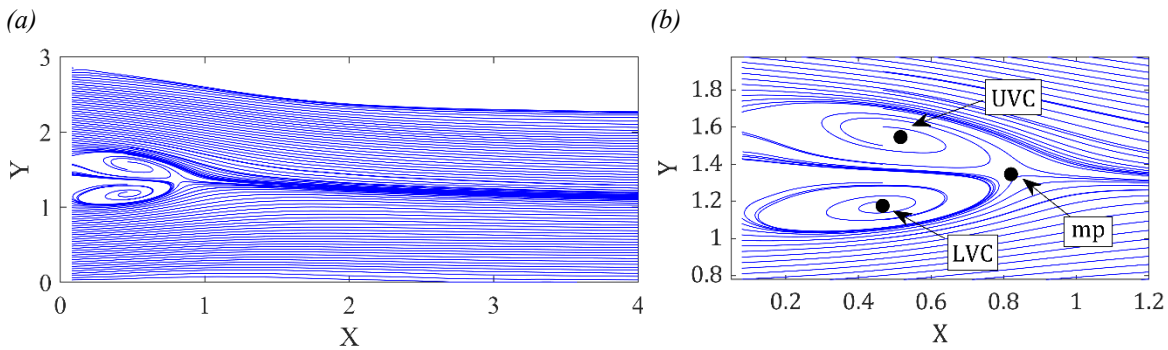


Figure 6.2: (a) Streamline plot and (b) Close-up of recirculation region for dual jet with $OR = V_r = 1$ and $Re = 10000$.

The streamlines also reveal more information regarding the flow structure inside the recirculation zone, where two distinct, steady counter-rotating vortices are observed in this region. A close-up of the recirculation zone is shown in Figure 6.2(b), where the locations of the LVC , UVC and mp can subsequently be identified. In doing so, the upper and lower vortices appear to be of a similar size with their respective centres somewhat aligned in the same vertical plane, albeit the UVC is located slightly further downstream. Although previous researchers have noted unsteady, von Kármán-like vortex shedding for a dual jet with jet with $Re = 10000$ and $OR = 1$, such as Wang and Tan [19], Li *et al.* [2] and Mondal *et al.* [20], the present research suggests the

steady, counter rotating flow regime is applicable when the flow data is time-averaged. Conversely, previous numerical studies such as Mondal *et al.* [20] suggest that the steady and unsteady dual flow regimes cannot occur concurrently, and are instead two separate flow phenomena. The current results therefore bring this conclusion of past researchers into question, suggesting that despite the possible occurrence of vortex shedding in the time-resolved behaviour, the time-averaged results may still present as two counter-rotating vortices. This implies that the two counter-rotating vortices presented by numerous past studies may be just a time-averaged feature and not a physical reality inside the range of Re examined.

6.1.1.2. Identification of mp, cp, UVC and LVC

The procedure by which the merge point, the combined point, the upper vortex centre, and the lower vortex centre are identified follows that of Kumar and Das [1]. As presented in Figure 6.3, the isolines corresponding to the positions at which either the local streamwise or transversal flow velocity component equated to 0 m/s are plotted, *i.e.*, $u, v = 0$. The crossover locations indicate the positions of mp (X_{mp}, Y_{mp}) = (0.82, 1.35), UVC (X_{UVC}, Y_{UVC}) = (0.52, 1.55) and LVC (X_{LVC}, Y_{LVC}) = (0.47, 1.18).

In addition to this, the isolines corresponding to $du/dy = 0$ are plotted in Figure 6.3, which indicates the positions of the local streamwise velocity minima and maxima at each downstream location inside the flow field. The location of cp coincides with the furthest downstream extent of the local deflection in the streamwise velocity profile towards a minimum value, beyond which only one instance of $du/dy = 0$ exists at each downstream location for the fully combined jet. The combined point location is hence identified as (X_{cp}, Y_{cp}) = (4.96, 1.27). Based on the positions of mp and cp , the limits of the converging region, merging region, and combined region can hence be defined as $0 \leq X \leq 0.82$, $0.82 < X \leq 4.96$ and $X > 4.96$, respectively.

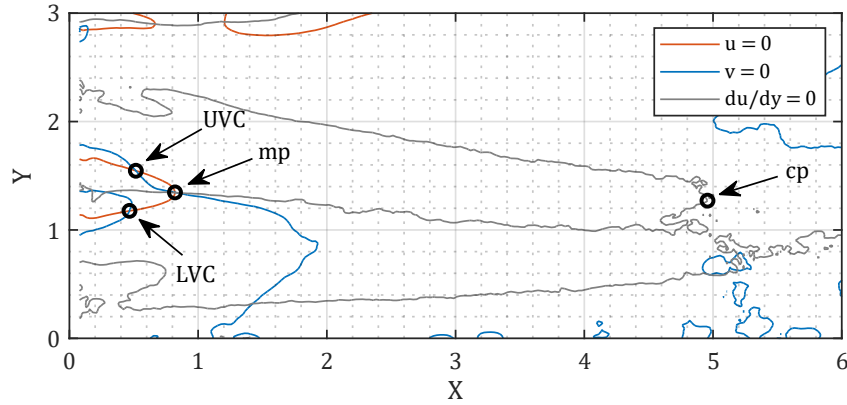


Figure 6.3: Identification of the locations of the merge and combined points, upper and lower vortex centres, for a dual jet with $OR = V_r = 1$ and $Re = 10000$.

The streamwise positions of mp and cp are comparable to that of Wang and Tan [19], who noted $X_{mp} = 0.75$ and $X_{cp} = 6.4$ but did not report the transverse positions of any notable flow features. While the value of X_{cp} is slightly less than that of Wang and Tan [19], it is significantly closer in value than that reported by any of the previously published numerical studies for the same dual jet parameters, such as $X_{cp} = 16.72$ as per Mondal *et al.* [21]. In general, the published numerical studies provide relatively similar values for X_{mp} as Wang and Tan [19], however, the values for X_{cp} appear to be considerably overpredicted in comparison. Regarding the locations of UVC and LVC , Wang and Tan [19] do not report on the occurrence of two counter-

rotating vortices in their time-averaged flow field, therefore a comparison cannot be made with the present investigation.

6.1.1.3. Shear Layers and Boundary Layer Development

The development of the shear layers, based on the jet half-width locations $(y_{0.5})_1$, $(y_{0.5})_2$, and $(y_{0.5})_3$, and the wall boundary layer δ_{bl} is shown in Figure 6.4, alongside that of Wang and Tan [19]. Beginning with the development of the outer shear layer of the offset jet (Layer 1), the value of $(y_{0.5})_1$ initially decreases with downstream distance, following the deflection of the offset jet, and reaches a minimum value near $X = 4$. Beyond this point, the increasing rate of entrainment and mixing with the ambient induces a subsequent growth in $(y_{0.5})_1$, which continues over the remaining downstream distance. The development of the inner shear layers of the offset and wall jets (Layer 2 and Layer 3) follows the shape of the recirculation region, where $(y_{0.5})_2$ and $(y_{0.5})_3$, respectively, converge towards each other with streamwise distance until eventually merging at approximately $X = 1.4$. Finally, the boundary layer thickness δ_{bl} generally increases with downstream distance, as expected. This rate of growth is initially quite steady, however, a sudden increase is observed near $X = 4.6$, which may be an indication of the transition to turbulence in the boundary layer or a dual jet flow feature upon entering the combined region. Further flow data near the solid wall in this region would be required to fully ascertain the cause of this phenomenon.

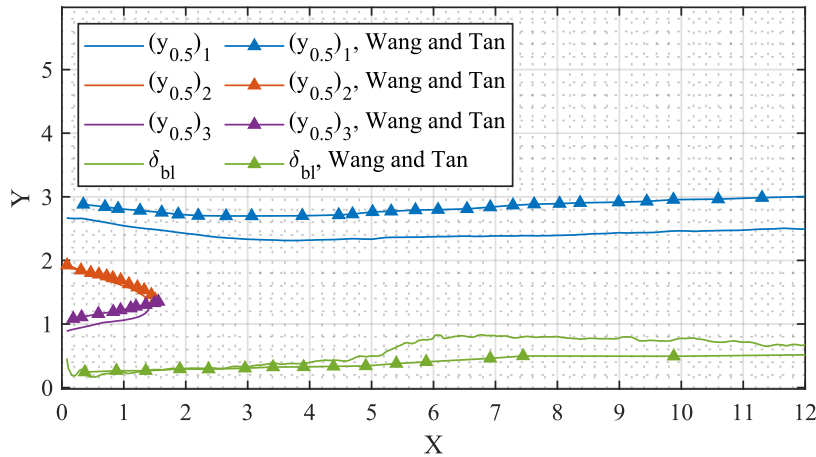


Figure 6.4: Shear layer and boundary layer development for a dual jet with $Re = 10000$ and $OR = V_r = 1$ compared to that of Wang and Tan [19].

The present data is in relatively good agreement with that of Wang and Tan [19] with respect to the overall trends, however, discrepancies exist regarding the exact placement of $(y_{0.5})_1$. This discrepancy is attributed to the presence of the boundary layer on the upper side of the offset jet inlet profile, as previously discussed in relation to Figure 5.1, thus lowering the starting position of $(y_{0.5})_1$ along the jet inlet plane and inducing subsequent downstream effects. This discrepancy is relatively minor, however, and is not considered to have significantly impacted other flow parameters of interest.

Shear layers are also characterised as thin regions of concentrated vorticity, across which the tangential velocity varies significantly [162]. In Figure 6.5, the jet half-width locations and boundary layer thickness are transposed onto the vorticity contour plot for the flow field. Indeed, regions of highly concentrated vorticity are evident along the shear layer locations, where the distribution of the highly sheared regions are centred about the jet-half width locations or distinctly contained within the wall boundary layer. For $(y_{0.5})_1$, the

vorticity magnitude inside Layer 1 appears to be greatest towards the jet exit, where velocity discontinuity between the jet flow and the quiescent ambient is most extreme, and the growing rate of fluid entrainment with streamwise distance decreases the vorticity magnitude while simultaneously increasing the shear layer thickness. The vorticity distributions along $(y_{0.5})_2$ and $(y_{0.5})_3$ are quite similar in magnitude but oppose in sign, with the negative vorticity associated with Layer 2 and the positive vorticity inside Layer 3. As per the nature of the inner shear layers, the vorticity magnitude associated with Layer 2 and Layer 3 decays rapidly in the region where the two shear layers meet, thus indicating their distinct ending point. High negative vorticity is also evident inside the wall boundary layer, as expected, the distribution of which again suggests a wall jet separation region from the solid boundary in the vicinity of mp .

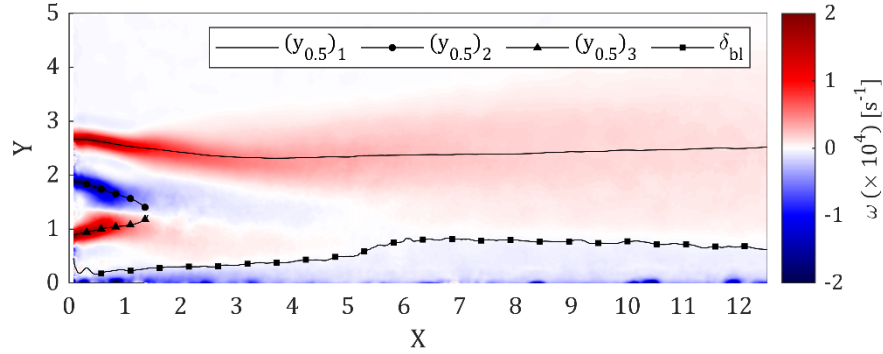


Figure 6.5: Comparison of the dual jet vorticity contour plot with jet half-width locations and boundary layer thickness for $Re = 10000$ and $OR = V_r = 1$.

6.1.1.4. Streamwise Velocity Characteristics

The normalised streamwise velocity plots at various downstream locations are plotted in Figure 6.6(a) and compared with that acquired by Wang and Tan [19] at $X = 10$. To remain consistent with Wang and Tan [19], the y-coordinate is normalised with respect to the outermost jet half width location, *i.e.*, $(y_{0.5})_1$, while the velocity is non-dimensionalised with respect to the local maximum streamwise velocity, u_m . Within the merging region, *i.e.*, $X = 2$ and $X = 4$, the velocity profile is characterised by two distinct peaks, signifying the positions of the maximum streamwise velocity contribution from the offset jet and wall jet, *i.e.*, $u_{m,o}$ and $u_{m,w}$ respectively. A local deflection in the velocity profile is thus created, which gives rise to a local minimum value u_{min} , the value of which within the merging region is always positive. As X increases, the momentum exchange between the two jets induces an increase in u_{min} relative to u_m , which leads to the eventual suppression of the deflection point. Beyond this location, *i.e.*, within the combined region, only a single peak in the streamwise velocity profile exists, similar to that of a single wall jet, which is acquired for $X \geq 6$. At $X = 10$, a good agreement with Wang and Tan [19] is observed, which indicates a very similar flow development between both investigations. As a continual change in the streamwise velocity profiles is evident from $X = 8$ to $X = 12$, it is clear that self-similarity has not yet been reached within the visible flow domain. This finding agrees with that reported by Wang and Tan [19], who achieved self-similarity after $X = 15$. Based on the minor incremental change between successive velocity profiles from $X = 8$ to $X = 12$, it is reasonable to assume that self-similarity will be attained after a similar downstream distance in the present investigation.

The decay of the local maximum streamwise velocity u_m with respect to downstream distance is presented in Figure 6.6(b), where the local maximum streamwise velocity is non-dimensionalised with respect to the overall

streamwise maximum velocity, $u_{m_{max}}$, that occurs immediately after the nozzle exit. In doing so, a distinct trend is observed, which is generally in reasonable agreement with that reported by Wang and Tan [19]. Within the converging region, u_m appears to decrease almost linearly with downstream distance following the initial rise to $u_{m_{max}}$ immediately downstream of the nozzle exit. This rise in u_m is likely due to residual boundary layer growths effects from inside the jet nozzle, where the conservation of momentum induces a small velocity increase. The subsequent linear decline in u_m is consistent with jet entrainment with slower-moving fluid inside the recirculation zone, as well as the deflection of both jets away from the streamwise flow direction.

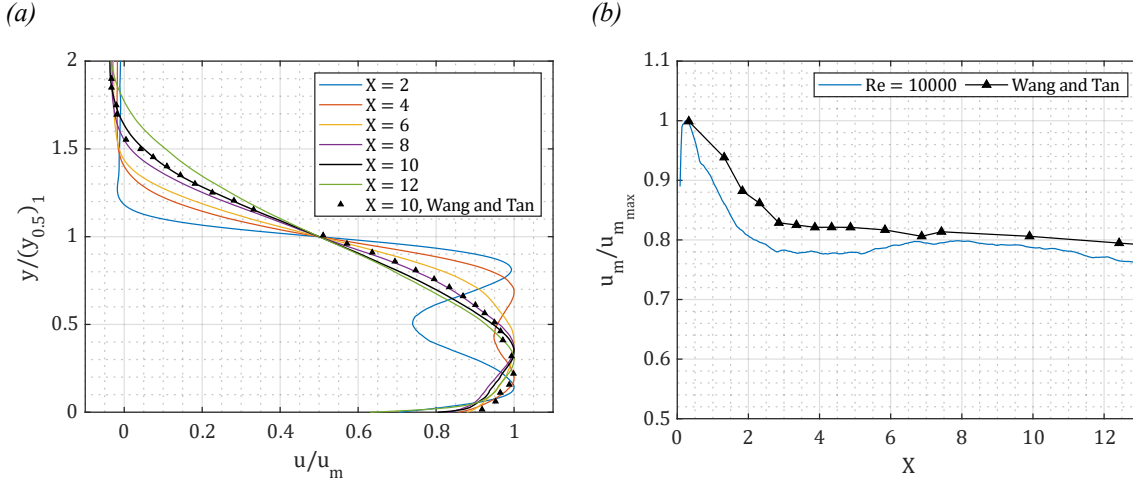


Figure 6.6: Normalised (a) velocity profiles at various downstream locations and (b) streamwise decay of the maximum velocity for a dual jet with $Re = 10000$ and $OR = V_r = 1$ alongside that of Wang and Tan [19].

Inside the merging region, the value of u_m continues to decrease in a similar fashion before starting to level off after $X = 2$. This flattening of the u_m profile is likely due to the combined effect of the jet interaction, where the momentum contribution from the wall and offset jets to the central slower-moving fluid acts to decrease the overall velocity, as well the realignment of the flow streamlines with the streamwise dimension, therefore inducing an apparent increase in the streamwise velocity of the jet. As both effects act in opposition, it seems reasonable that a flattening effect would be observed. In the combined region, the value of u_m subsequently increases towards a maximum value near $X = 8$, which is consistent with the dual jet reaching a fully combined state within which the total momentum is greater than that of the wall jet or offset jet individually. Beyond this point, the maximum velocity decreases in a monotonic fashion as entrainment effects from the ambient fluid begin to dominate and slow the overall mean jet velocity. Across all regions, a reasonable agreement is obtained with Wang and Tan [19], certainly in terms of the overall trend, although the present study measures consistently lower values, especially in the converging and merging regions. These discrepancies are likely due to differences in the experimental setup, which include the use of a different working fluid, however, both setups successfully emulate the same overall dual jet behaviour.

Finally, the turbulence intensity distribution inside the dual jet flow field is described by the contour plot in Figure 6.7, where the standard deviation of the time-resolved velocity magnitude at each location is scaled with respect to the average streamwise velocity at the jet outlet. While high turbulence intensity values are identified in the shear layer regions, it is interesting to note the existence of a distinct region of very high turbulence intensity that primarily exists inside the merging region. This highly fluctuating region corresponds to the wall

jet and offset jet interaction and mixing between the two jets, where the intensity of the interaction is greatest in the vicinity of the merge point and decreases with increasing downstream distance.

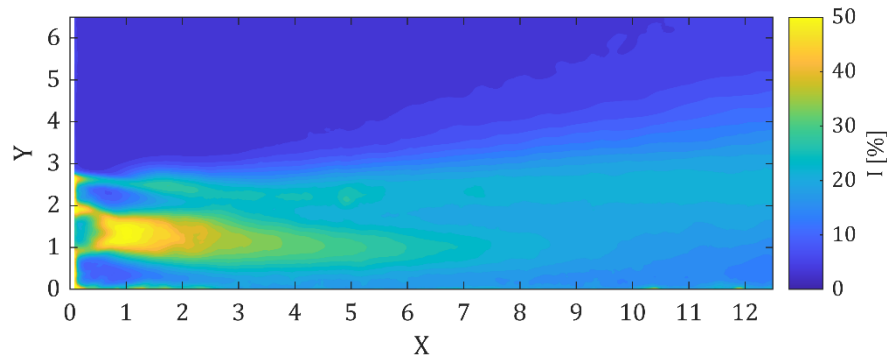


Figure 6.7: Turbulence intensity distribution inside a dual jet flow field for $Re = 10000$ and $OR = V_r = 1$.

6.1.2. Time-Resolved Analysis

The time-resolved analysis of the dual jet for $Re = 10000$ and $OR = 1$ reveals a highly fluctuating flow field, where it is evident that an unsteady interaction takes place between the wall jet and the offset jet. The instantaneous velocity contour plots for a sample set of 4 successive instances in time are presented in Figure 6.8, where it was clear to see that the velocity field changes continuously over the total testing time of 1 second when captured at the fixed frame rate of 2000 Hz. A periodic series of high velocity regions are observed in the streamwise direction of the dual jet flow field for all time instances considered, which is indicative of a von Kármán-like shedding of vortices from the jet inlet plane. The successive flow fields in Figure 6.8 imply that the vortices are shed from the wall jet and the offset jet in an alternating manner, as indicated by the periodic elongation and contraction of the respective jet potential core region, where the wall jet and offset jet are consistently in opposition. With each elongation and contraction of a jet potential core, a new discrete high velocity region is introduced to the dual jet flow field, which is subsequently advected downstream and eventually dispersed after a certain streamwise distance due to the viscosity of the flow field.

To further examine this periodic dual jet flow field, the streamlines corresponding to the instantaneous flow fields shown in Figure 6.8 are plotted in Figure 6.9, where the unsteady interaction is observed between Layer 2 and Layer 3. The streamlines plots demonstrate that Layer 2 and Layer 3 are pulled into the low-pressure recirculation region in an alternating fashion, which thus induces the formation of the lower and upper vortex, respectively. In each case, the corresponding vortex grows in size and strength until reaching the point at which it becomes unstable, enabling it to break free and convect downstream. This process is then repeated by the opposing vortex, meaning at no point in time do the upper and lower vortex coexist with equal size and strength. The periodic instability of the upper and lower vortex gives rise to the characteristic von Kármán vortex street generally associated with flow past a bluff body, the occurrence of which is also evident in the experimental results produced by Wang and Tan [19]. The periodic shedding of vortices acts to enhance the interaction and mixing between the wall and offset jet and enables them to reach a fully combined state over a relatively short distance.

A vortex, however, is defined as a region of concentrated vorticity. Therefore the data presented in Figure 6.8 and Figure 6.9 cannot be used standalone to locate the positions of all vortices in the streamwise direction,

along with their respective direction of rotation, or definitively identify the presence of a von Kármán street. The instantaneous vorticity contour plot at $t = 0.2045$ s is hence plotted in Figure 6.10(a), where distinct regions of high vorticity are clear. Layer 1 is primarily characterised by discrete regions of positive vorticity, which is indicative of the formation of Kelvin-Helmholtz roll-up vortices with an anti-clockwise rotation due to entrainment with the quiescent ambient along this region. The respective sign of the vorticity associated with Layer 2 and Layer 3 are in distinct opposition, where Layer 2 is largely characterised by negative vorticity, and Layer 3 positive. For the specific instance in time shown, the lower vortex presents as much larger than the upper vortex, which is indicative of the phase of the vortex shedding cycle. The vorticity plot enforces the consensus that the upper vortex and lower vortex in a dual jet are counter-rotating in nature, where the negative vorticity which characterises the upper vortex implies a clockwise rotation, whereas the positive vorticity of the lower vortex indicates an anti-clockwise direction of rotation. Downstream of the recirculation zone, the observed regions of alternating positive and negative vorticity along the streamwise dimension correspond to the pairs of vortices previously shed from jet inlet plane and subsequently convected downstream, the combination of which form the von Kármán vortex street.

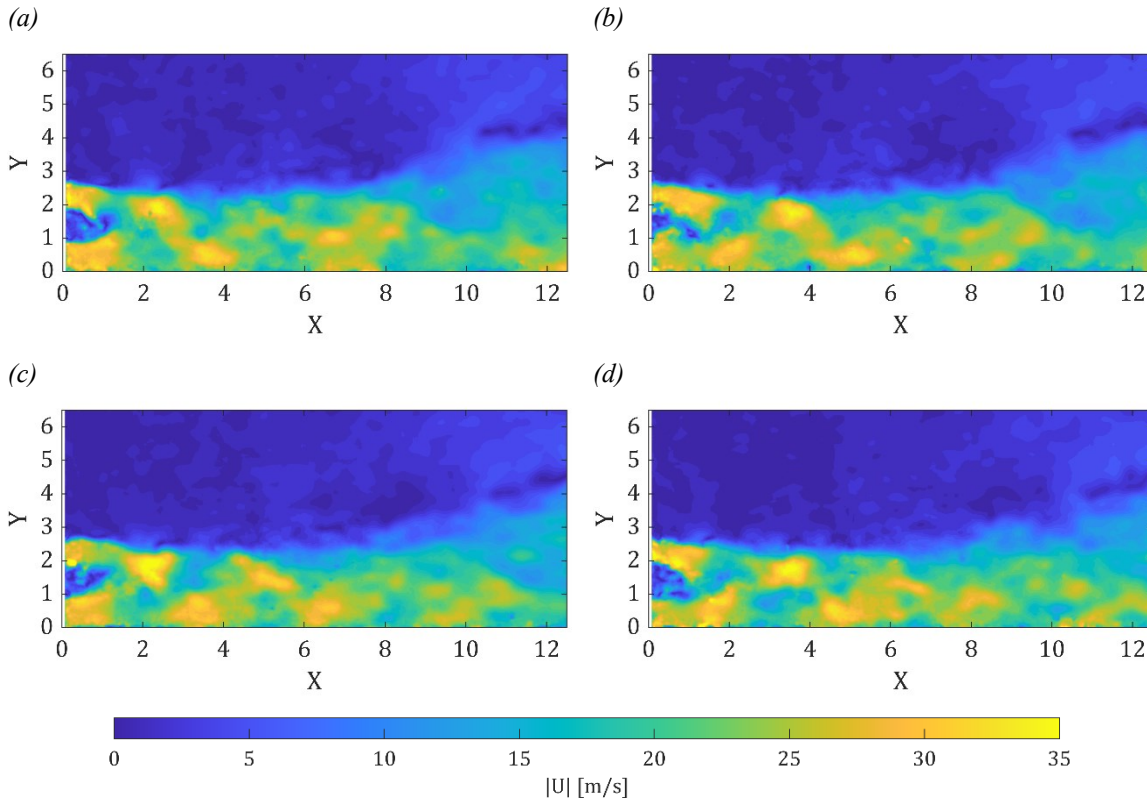


Figure 6.8: Instantaneous velocity contour plots for a dual jet with $Re = 10000$ and $OR = V_r = 1$, where (a) $t = 0.0035$ s, (b) $t = 0.004$ s, (c) $t = 0.0045$ s and (d) $t = 0.005$ s.

The use of vorticity as a means of vortex identification, however, may fail to identify all vortical structures in a given flow field due to its inability to discriminate between swirling and shearing flows, as previously noted by Wang and Tan [19]. Wang and Tan [19] thus adopted the λ_2 criterion originally proposed by Hussain and Jeong [163], where λ_2 refers to the median of the three eigenvalues associated with the $S^2 + \Omega^2$ matrix, and S and Ω are the symmetric and antisymmetric components of the velocity gradient matrix $\Delta \mathbf{U}$ such that $S_{ij} = \frac{1}{2}(\Delta \mathbf{U}_{i,j} + \Delta \mathbf{U}_{j,i})$ and $\Omega_{ij} = \frac{1}{2}(\Delta \mathbf{U}_{i,j} - \Delta \mathbf{U}_{j,i})$. The λ_2 value hence provides the centre locations and associated strength of discrete vortices present in the flow field, where $\lambda_2 < 0$ is indicative of a swirling flow, *i.e.*, within

the vortex core, and $\lambda_2 > 0$ implies a shearing flow [19, 164]. The λ_2 value therefore cannot determine the direction of rotation for the vortex in question.

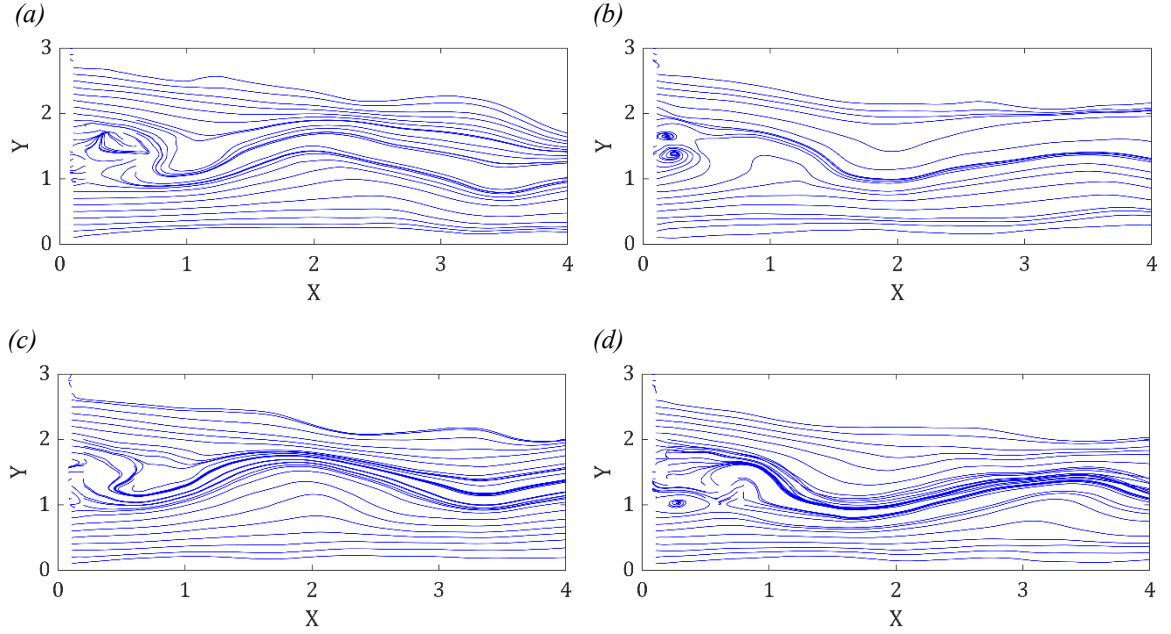


Figure 6.9 Instantaneous streamline plots for a dual jet with $Re = 10000$ and $OR = V_r = 1$, where (a) $t = 0.0035$ s, (b) $t = 0.004$ s, (c) $t = 0.0045$ s and (d) $t = 0.005$ s.

For the present investigation, a similar parameter is available through the DaVis 10 software, termed the swirling strength and denoted by λ_{ci} [150]. The value of λ_{ci} is again determined based on the velocity gradient matrix $\Delta \mathbf{U}$, where it is understood that $\Delta \mathbf{U}$ possesses a pair of eigenvalues in the form $a \pm \sqrt{b}$, where the discriminant b is determined using equation 6., such that

$$b = \frac{\partial u}{\partial y} \frac{\partial v}{\partial x} - \frac{1}{2} \left(\frac{\partial u}{\partial x} \frac{\partial v}{\partial y} \right) + \frac{1}{4} \left(\left(\frac{\partial u}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial y} \right)^2 \right) \quad (6.1)$$

Depending on the sign of b , the eigenvalues of $\Delta \mathbf{U}$ can either exist as a set of two real values, *i.e.*, $b > 0$, or a complex conjugate pair, *i.e.*, $b < 0$. A swirling flow is indicated by $b < 0$, while a positive value of b implies a shearing flow. In the DaVis 10 software, λ_{ci} is hence calculated using $\lambda_{ci} = \max(0, -b)$, which eliminates all regions of shearing flow to display the discrete locations of swirl only, *i.e.*, the vortex positions.

The results acquired following the application of this process to the time instance presented in Figure 6.10(a) is shown in Figure 6.10(b), where the positions of all notable vortices within the flow field are more clearly defined. When compared to Figure 6.10(a), the positions of the identified vortices as per Figure 6.10(b) are found to directly correspond to the regions of concentrated vorticity, while the regions of sheared flow are omitted, thus improving the clarity of the vortex pattern within the flow field. In doing so, the existence of the von Kármán vortex street along the inner shear layers of the wall and offset jet is highlighted, as well as the existence of Kelvin-Helmholtz roll-up vortices along the outer shear layer of the offset jet, which is consistent with the dual jet vortex pattern reported by Wang and Tan [19]. When applied across the total dataset, a translational velocity of $u_{trans} = 0.76u_{jet}$ is identified for the vortices, calculated based on the average streamwise distance a single vortex is convected through during a single timestep, and is comparable to that

obtained by Wang and Tan [19] of $0.84u_{jet}$. To remain consistent with Wang and Tan [19], a vector plot corresponding to $U - u_{trans}$ is overlayed onto Figure 6.10(b), which further demonstrates the nature of the swirling flow inside each vortex core.

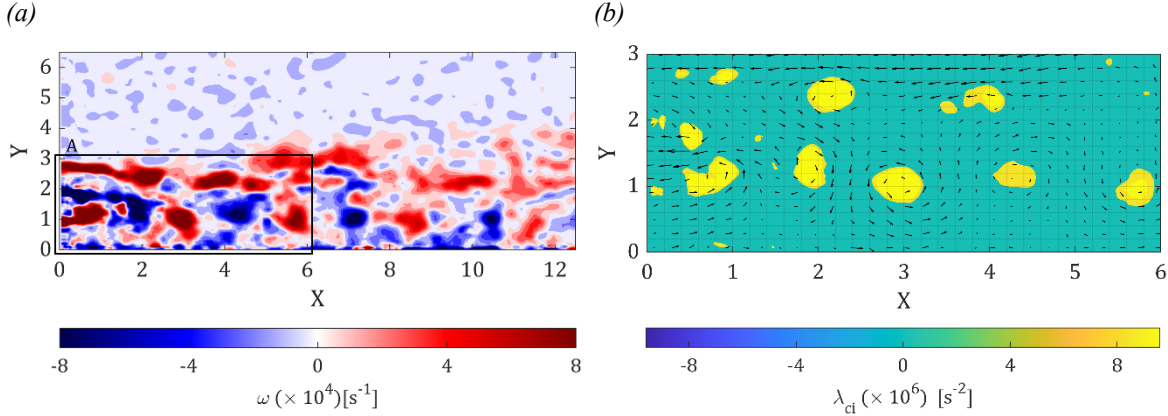


Figure 6.10: (a) Vorticity and (b) swirling strength contour plot (with $U - u_{trans}$ vector plot overlayed) for a dual jet with $Re = 10000$ and $OR = V_r = 1$ at $t = 0.2045$ s, where (b) focusses only on the region marked as 'A' in (a).

Finally, to determine the vortex shedding frequency, the transverse velocity v at the merge point is plotted with respect to time in Figure 6.11(a). The transverse velocity at this point is observed to fluctuate considerably about 0 m/s between the approximate limits of ± 25 m/s, which is consistent with the periodic shedding of vortices in this region. To extract the vortex shedding frequency, the power spectral density (PSD) of the velocity signal is acquired using the DaVis 10 software. The PSD is plotted in Figure 6.11(b), which produces a singular dominant peak at $f = 914$ Hz, i.e., the shedding frequency f_s . Using the measured average jet velocity at the nozzle outlet of $u_{jet} = 25.46$ m/s, the final Strouhal number associated with the vortex shedding frequency equates to $St = fw/u_{jet} = 0.251$, which is in excellent agreement with that acquired by Wang and Tan [19] for the same configuration of $St = 0.25$. Due to the overall close alignment between both experimental datasets, the time-resolved dual jet flow characteristics from the present experimental facility can be considered adequately validated with respect to the published literature, which thus ensures the credibility of the time-resolved dual jet flow behaviour acquired over the wider range of experimental jet parameters.

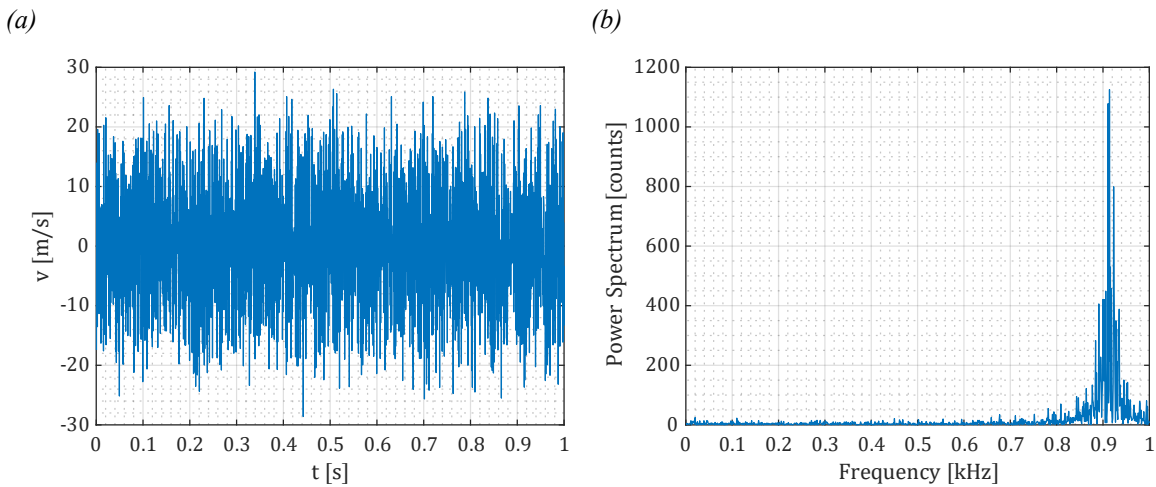


Figure 6.11: Analysis of the transverse velocity component v at mp for a dual jet with $Re = 10000$ and $OR = V_r = 1$. (a) Change in v with respect to time t , (b) Power spectral density (PSD) of v time signal.

6.2. Heat Transfer Characteristics

The results of the heat transfer study reveal distinct dual jet heat transfer behaviour, where the overall Nu_x trend remains relatively consistent across the range of jet parameters considered. The general dual jet heat transfer characteristics are therefore discussed in the following sections for a single test case, *i.e.*, $Re = 10000$ and $OR = 1$ for $V_r = 1$, the measured flow behaviour for which was previously validated with respect to Wang and Tan [19] in section 6.1. For the heat transfer study, the results are only discussed from a time-averaged perspective, where ‘time-averaged’ refers to the averaging of all 500 thermal datasets obtained within the 20 second testing period. While time-resolved data was obtained, no obvious time dependent features were observed within the heat transfer data as the IR camera imaging frequency was not sufficiently high compared to the vortex shedding frequency. As a result, a time-resolved analysis is omitted from the succeeding discussion regarding the dual jet heat transfer behaviour.

6.2.1. Time-Averaged Analysis

The transfer of heat to the dual jet from the test surface is analysed in terms of the local surface Nusselt number Nu_x , which is plotted with respect to the non-dimensionalised streamwise distance from the nozzle exit in Figure 6.12 for $Re = 10000$ and $OR = 1$. As shown in Figure 6.12, an initial decline Nu_x is observed close to the nozzle exit towards a local minimum value at approximately $X = 1$. A sharp increase in Nu_x subsequently occurs, leading to an eventual Nu_x maximum peak near $X = 2.8$. The value of Nu_x decreases over the remaining downstream distance, with a rate of decline that is initially quite sharp but reduces for higher values of X . This distinct Nu_x trend obtained varies greatly from that of the SWJ or the SOJ previously discussed in relation to Figure 5.8 and Figure 5.12, respectively, thus highlighting the requirement for accurate published data relating to the heat transfer characteristics of dual jet flows. To understand the underlying flow phenomena driving the Nu_x trend, the heat transfer data is plotted alongside the corresponding streamline plot and the streamwise decay of the maximum velocity, where the X axis is common to all three plots to allow for a direct comparison.

In doing so, the initial decline in Nu_x close to the nozzle exit can clearly be attributed to the deflection of the wall jet away from the solid wall due to the low-pressure recirculation zone. This creates a region of slower-moving fluid or separation region close to the solid wall, thus reducing the local rate of heat transfer. The extent of the wall jet deflection appears to be greatest at approximately $X = 1$, *i.e.*, slightly downstream of the merge point at $X_{mp} = 0.82$, which coincides with the occurrence of the local minimum value in the Nu_x profile, $Nu_{x_{min}}$. Beyond this point, the interaction between the jets is initiated and the dual jet flow is redirected back towards the solid wall, which contributes towards the sharp rise in Nu_x in this region. The local maximum value in the Nu_x profile at $X \approx 2.8$, $Nu_{x_{max}}$, coincides with the impingement location of the dual jet with the solid wall, downstream of which the flow remains in contact with the solid boundary. The subsequent decline in Nu_x over the remaining downstream distance can hence be attributed to the continual growth of the boundary layer at the solid boundary, as well as a reduction in the intensity of the jet interaction as the effects of entrainment with the quiescent air begin to dominate and slow the overall mean jet velocity. As there is no secondary increase in Nu_x along this region, it indicates that a natural transition to turbulence does not take place, rather the boundary layer is already fully turbulent by this point. As such, it can be inferred that the

separation of the bulk flow from the solid wall boundary induces a sudden transition of the boundary layer to turbulent.

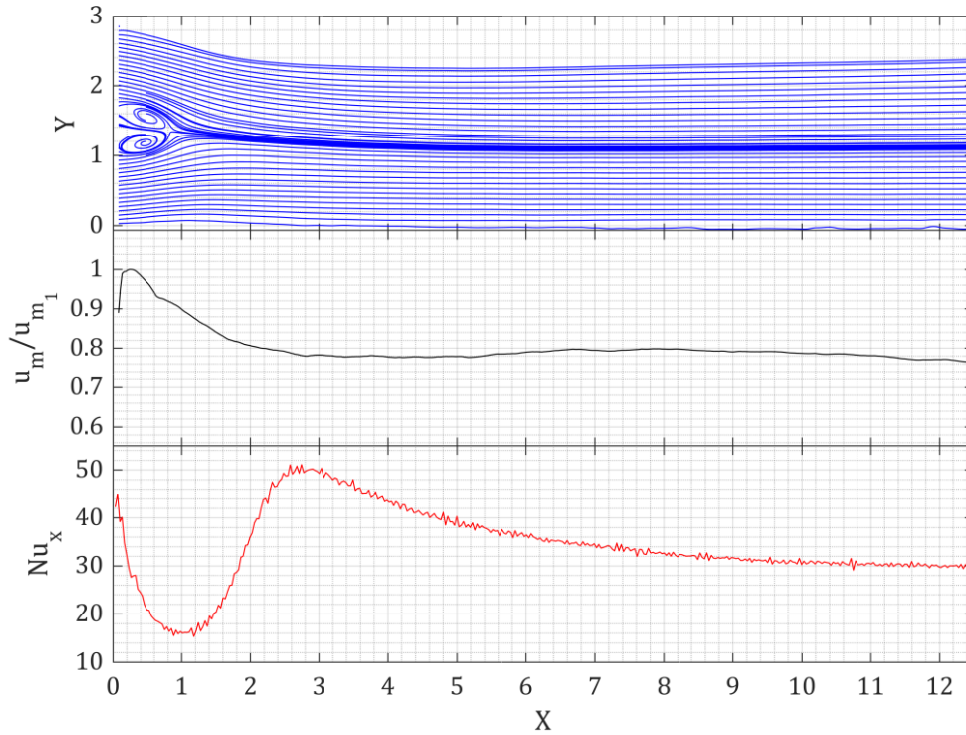


Figure 6.12: Time-averaged Nu_x profile for a dual jet with $Re = 10000$ and $OR = V_r = 1$ alongside the corresponding streamline plot and decay of maximum streamwise velocity.

Despite the logical reasoning behind the resulting Nu_x profile, it is noted that the downstream Nu_x trends for a dual jet vary considerably throughout the published literature. While most studies note the existence of a peak in the Nu_x profile, such as Mondal *et al.* [22], Hnaïen *et al.* [10], and Kumar [83], the occurrence of a local minimum in advance of this is reported to a much lesser extent. A comparison between the experimental results from the present study and the published literature for $Re = 10000$, $OR = 1$ and $V_r = 1$ is shown in Figure 6.13, where it is clear to see that the Nu_x trends acquired by Vishnuvardhanarao and Das [28], Mondal *et al.* [22], and Singh and Dewan [24] do not align with the present investigation, but also do not agree with each other. Each of the published studies used for comparison in Figure 6.13 were numerical investigations, where Vishnuvardhanarao and Das [28] and Mondal *et al.* [22] both adopted the $k - \varepsilon$ turbulence model and Singh and Dewan [24] used the SST $k - \omega$ turbulence model. Each of the numerical studies adopted a constant temperature boundary condition in the solid wall, where Vishnuvardhanarao and Das [28] compared the results acquired for both a uniform wall temperature and uniform wall flux boundary condition. A uniform velocity profile was applied at the respective jet inlets for each study, where a turbulent length scale of $0.07w$ was specified across all three studies and a turbulence intensity of 5% was adopted by both Mondal *et al.* [22] and Singh and Dewan [24] while Vishnuvardhanarao and Das [28] used 2%.

While all datasets indicate a general decline in Nu_x in the far downstream region, the observed heat transfer behaviour close to the nozzle exit varies greatly. Firstly, Vishnuvardhanarao and Das [28], presents quite an unsteady profile, despite the use of a steady state model, where Nu_x appears to initially increase at the nozzle exit but decline in an oscillatory manner with further downstream distance. The values associated with the far downstream region provide the best agreement with the current study, however, the occurrence of a local

maximum followed by a local minimum close to the nozzle exit directly contradicts the current study. Conversely, Mondal *et al.* [22] does not indicate the presence of a local Nu_x minimum whatsoever, where Nu_x increases towards a maximum value upon exiting the nozzle before steadily decreasing with further streamwise distance. Finally, the results of Singh and Dewan [24] differ most when compared to the present study, where the greatest Nu_x value is reported at the nozzle exit and diminishes with X in an almost logarithmic fashion.

Despite these differences, it should be noted that the discussed published studies relating to transfer of heat to a dual jet flow were entirely numerical in nature and had only the flow velocity data provided by Wang and Tan [19] to validate their respective numerical models. In the case of Mondal *et al.* [22] and Singh and Dewan [24], the numerical models were validated using the experimental results for a SOJ published by Kim *et al.* [79] and Song *et al.* [81], respectively, which is an approach used by many of the other numerical dual jet studies found in the literature. However, the flow field associated with a dual jet varies greatly from that of a SOJ, as previously discussed, meaning the credibility of this method of validation is contestable. In the case of both Mondal *et al.* [22] and Singh and Dewan [24], both studies obtained a reasonable agreement with their chosen experimental dataset for the SOJ data in the vicinity of the local maximum, however, neither model captured the initial decline in Nu_x at the nozzle exit resulting from the secondary recirculation region. This implies that both models do not necessarily capture accurate Nu_x data close to the nozzle exit. In addition to this, neither study presented the results for a SWJ, the Nu_x data from which is heavily influenced by the laminar-to-turbulent transition in the boundary layer. The inconsistencies between the Nu_x datasets shown in Figure 6.13 therefore demonstrate the requirement for published experimental heat transfer data specific to dual jet flows, as a credible means of numerical model validation would introduce consistency across all published dual jet studies. In addition, as all the published studies presented in Figure 6.13 adopted fully turbulent numerical models, it is reasonable to assume that the use of a transitional model may have provided a more accurate solution.

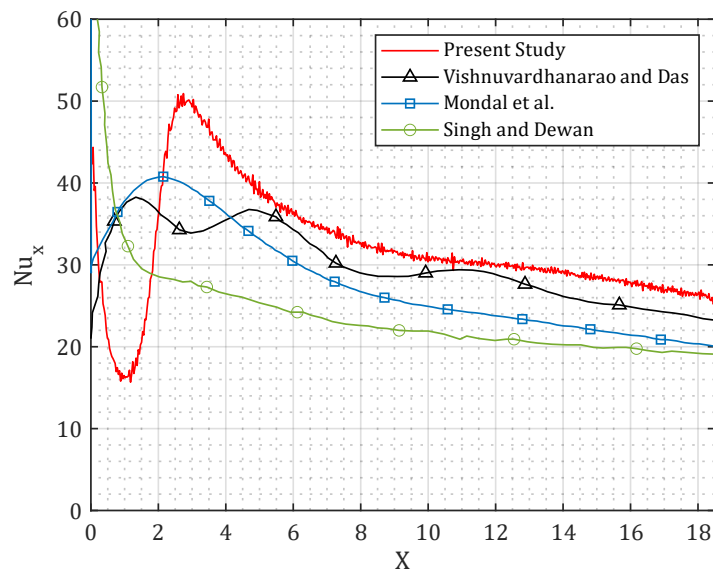


Figure 6.13: Comparison of the dual jet Nu_x profile for $Re = 10000$ and $OR = V_r = 1$ with that obtained by Vishnuvardhanarao and Das [28], Mondal *et al.* [22] and Singh and Dewan [24] for identical dual jet flow parameters.

6.3.Effect of Confinement

As previously discussed, the purpose of the enclosure in the experimental setup is to mitigate jet spreading effects and encourage a planar flow field along the mid-plane of the fluid domain. To investigate the effect of confinement of the dual jet and ensure the dual jet flow field was not affected by the presence of the enclosure in a negative manner, additional experiments without the transparent enclosure were conducted. The flow data obtained with and without the enclosure for $OR = 1$ is presented as velocity contour plots in Figure 6.14 for a low and high jet Reynolds number value, *i.e.*, $Re = 5500$ and $Re = 12000$, onto which the corresponding velocity vector plots are superimposed. For simplicity, the ‘with’ and ‘without’ the enclosure test cases are termed confined and unconfined, respectively, and the positions of the merge point, the combined point, the upper vortex centre, and the lower vortex centre are marked for each flow field.

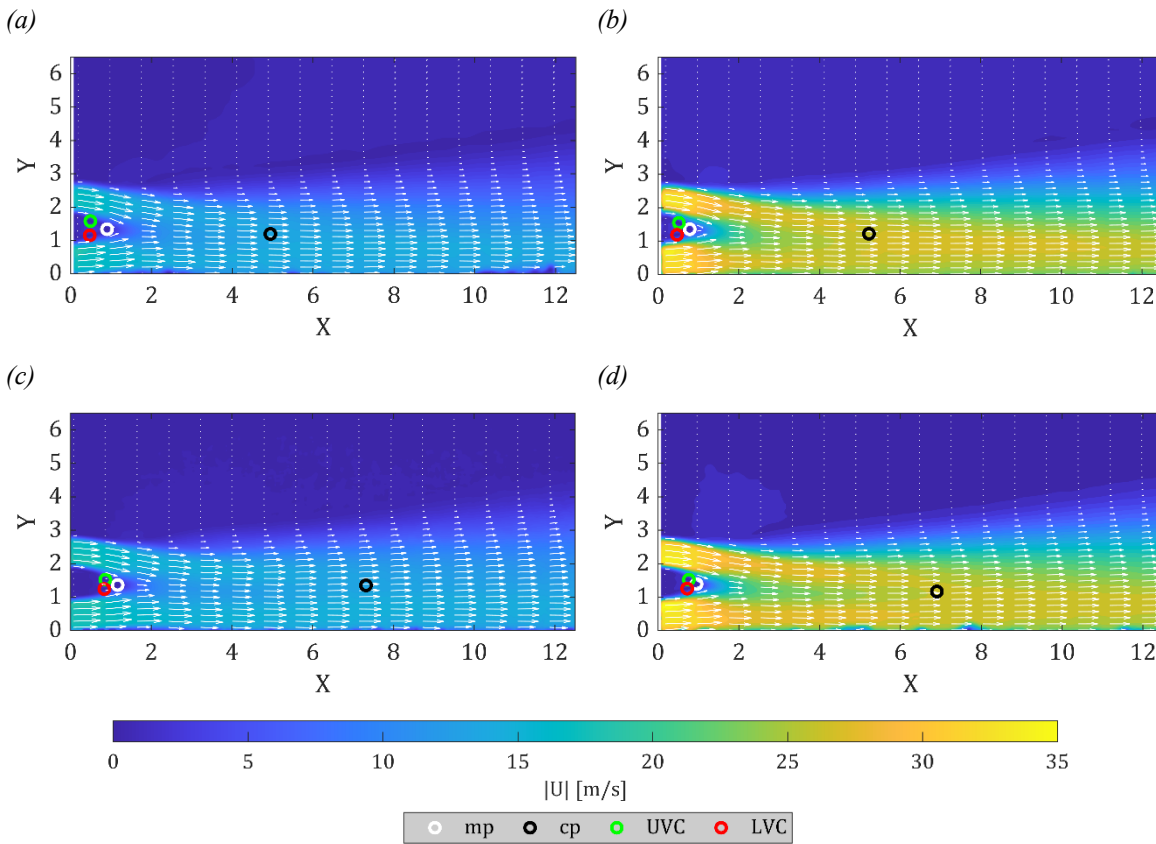


Figure 6.14: Velocity contour and vector plots comparing the effect of the confinement for $OR = 1$ and $V_r = 1$, where (a, c) $Re = 5500$ and (b, d) $Re = 12000$. Top/bottom row shows confined/unconfined cases, respectively.

In each case, the removal of the enclosure results in a larger recirculation region, where the positions of *mp*, *UVC* and *LVC* move noticeably downstream, which is more obvious for the lower Re case. This finding is similar to that reported by Nasr and Lai [76], who noted that the rate of deflection of a single offset jet towards the solid wall is reduced without side walls, hence moving the reattachment point further downstream and increasing the extent of the recirculation zone. This is attributed to the lower, sub-atmospheric pressures that can be achieved inside the recirculation zone through the use of side walls, where the side walls act to bound the recirculation region. Without side walls, however, entrainment between the recirculation zone and surrounding ambient can readily take place, thus raising the achievable pressure in this region and reducing the deflection of the SOJ. Nasr and Lai [76] therefore strongly recommended the use of side walls when conducting

2-dimensional flow studies involving single offset jets in order to reduce the impact of this 3-dimensional ‘pressure-leakage’ effect. As the increased recirculation region size is consistent with the current study, the pressure-leakage effect and subsequent increase in pressure inside the recirculation region are considered adequate reasoning for the results produced by the unconfined tests.

In the case of the dual jet, however, the lessened deflection of the offset jet also affects flow development downstream of the recirculation zone, as the intensity of the interaction between the jets is subsequently reduced. This results in a slower downstream jet development and lengthening of the merging region, as indicated by the increased streamwise position of the combined point for both Re cases presented. This effect is more pronounced for higher OR cases, such as that presented in Figure 6.15 for $OR = 3$ for the same Re values as Figure 6.14, where the location of cp moves beyond the scope of the experimental field-of-view. Further to this, a higher pressure inside the recirculation zone means the deflection of the wall jet away from the solid boundary is less noticeable, where the wall jet development in Figure 6.15 appears relatively unaffected by the presence of the offset jet in the unconfined case compared to the confined case. In addition to the reasons discussed, the observed differences between the confined and unconfined test cases may have been further exacerbated by jet spreading effects, as well as the effect of other unknown 3D flow structures.

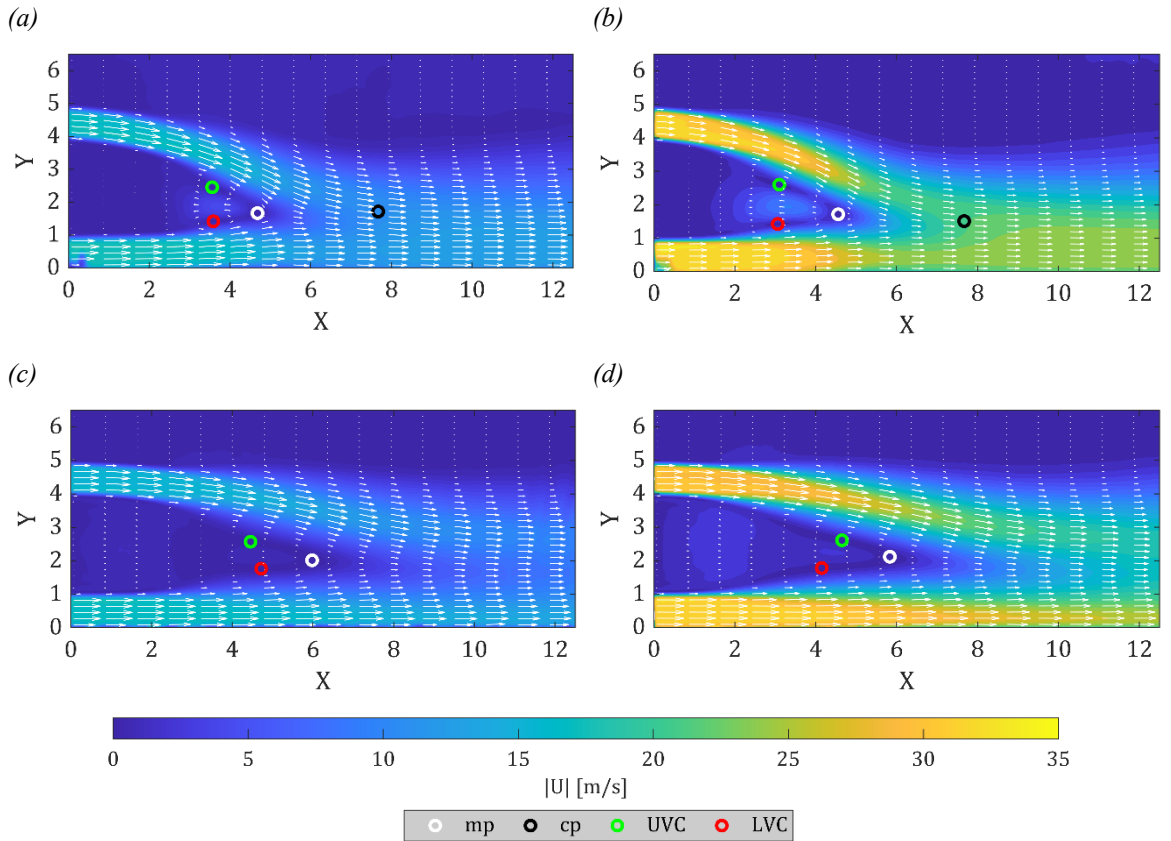


Figure 6.15: Velocity contour and vector plots comparing the effect of the confinement for $OR = 3$ and $V_r = 1$, where (a, c) $Re = 5500$ and (b, d) $Re = 12000$. Top/bottom row shows confined/unconfined cases, respectively.

To investigate the effect of confinement on the heat transfer characteristics of the dual jet flow, the time-averaged Nu_x profiles pertaining to the confined and unconfined flow data presented in Figure 6.14 and Figure 6.15 are displayed in Figure 6.16, *i.e.*, $Re = 5500$ and $Re = 12000$ for $OR = 1$ and $OR = 3$. The Nu_x profiles presented in Figure 6.16(a) indicate that the effect of the jet confinement on the time-averaged wall heat transfer

for $OR = 1$ is relatively minimal, where only slight differences are visible close to the nozzle exit. As previously mentioned with regard to the corresponding flow data presented in Figure 6.14, the lower Re case appears to be more obviously affected by the presence of the enclosure, where the streamwise position of both the local Nu_x minimum and Nu_x maximum increase for the unconfined case as the merge point moves further downstream. Regarding the respective change in magnitude of both $Nu_{x_{min}}$ and $Nu_{x_{max}}$, the results obtained are consistent with a less significant deflection of the wall jet from the solid wall. In the case of less a pronounced wall jet deflection, the intensity of the subsequent re-impingement with the solid boundary is greatly reduced, which would logically increase and decrease the values of $Nu_{x_{min}}$ and $Nu_{x_{max}}$, respectively.

For higher offset ratios, however, the effect of removing the enclosure on the heat transfer characteristics is considerably more apparent, as demonstrated in Figure 6.16(b) for $OR = 3$, which is consistent with that observed for the respective flow data. Similar to that noted for $Re = 5500$ with $OR = 1$, the growth of the recirculation region for both Re examined moves the positions of $Nu_{x_{min}}$ and $Nu_{x_{max}}$ further downstream, while respectively increasing and decreasing their associated magnitudes. In the far downstream region, however, a greater alignment between the confined and unconfined test cases is acquired. It is interesting to note that the removal of the enclosure results in an overall smoothing of the Nu_x profile towards that typical of a SWJ, which indicates that the full benefit of the additional offset jet in the transfer of heat is not attainable without the addition of the enclosure for higher OR values. Due to the vast difference in the results acquired for the confined and unconfined test cases, it can be concluded that the use of the enclosure is vital for enhancing the planarity of the flow field, as well as producing results that can be considered both credible and accurate with respect to the published literature, as previously discussed for a single wall jet, a single offset jet and a dual jet.

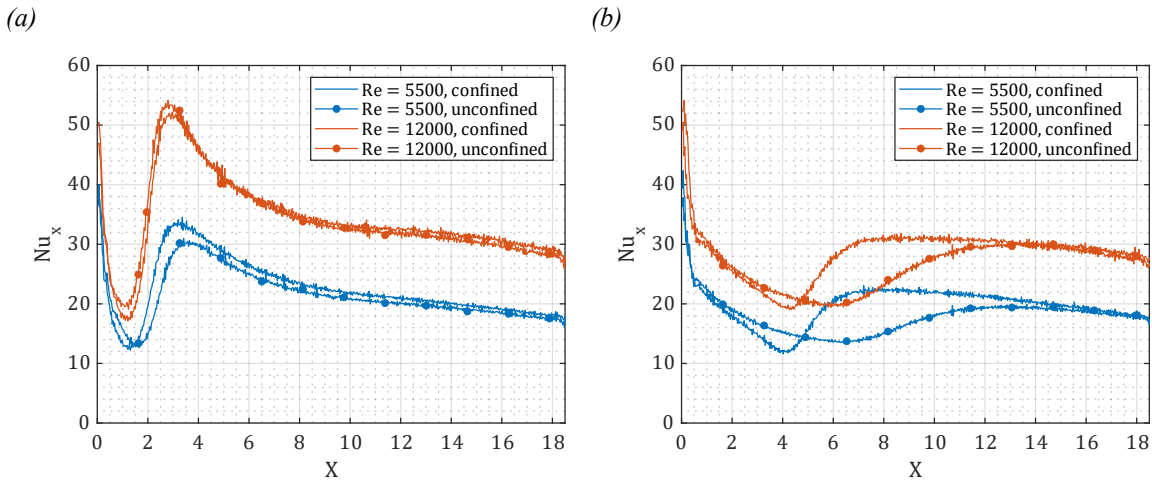


Figure 6.16: Comparison of the dual jet Nu_x profiles with and without the smooth enclosure, i.e., confined vs unconfined, for $Re = 5500$ and 12000 with $OR = (a) 1$ and $(b) 3$, where $V_r = 1$.

6.4. Effect of Reynolds Number

The following sections discuss the effect of the Reynolds number on the dual jet flow structure and the subsequent heat transfer implications. As no other experimental studies exist in the published literature relating to dual jet flow over an expanded range of Re , it is not possible to obtain a direct validation for the observed trends, however comparisons are drawn between the present investigation and that reported by published numerical investigations which analysed the effect of Re [10, 22-24, 28, 84, 108]. The findings of these numerical studies are noted as quite being contradictory in nature, however, such as that of Mondal *et al.* [22] and Singh and Dewan [24], who, for the same set of dual jet parameters, reported the value of X_{mp} to increase and remain constant, respectively, with increasing Re . Conversely, Kumar *et al.* [23] noted a decreasing X_{mp} value with increasing Re for a higher offset ratio range of $3 \leq OR \leq 9$. These discrepancies further highlight the requirement for accurate experimental data pertaining to dual jet flow, and a degree of caution is therefore exercised when drawing comparisons with the findings from the present investigation.

For the following sections, flow data is first presented for $5500 \leq Re \leq 12000$ and $V_r = 1$ at the extreme offset ratio cases, *i.e.*, $OR = 1$ and $OR = 3$, where both the time-averaged and time-resolved Re effects are considered. The heat transfer findings are subsequently shown for a wider OR range, *i.e.*, $1 \leq OR \leq 7$, the understanding for which is derived from the previously established flow behaviour.

6.4.1. Flow Field Characteristics

To examine the effect of the Re on the time-averaged dual jet flow structure, the time-averaged velocity $|U|$ fields obtained for a dual jet with $OR = 1$ are presented in Figure 6.17 for $Re = 6000, 8000, 10000$ and 12000 , where the corresponding velocity vector field is superimposed onto each contour plot. In each case, both jets are observed to deflect towards one another upon exiting the jet nozzle, which leads to the creation of a distinct recirculation zone in this region, downstream of which the flow continues to develop as the jets progress towards a fully combined state. Upon initial inspection, the contour plots indicate that as Re increases, the velocity magnitude of the flow increases accordingly, however the general behaviour of the dual jet remains relatively consistent. This is inferred from Figure 6.17 based on the similarity in the shape of the high velocity regions in the contour plots for each Re shown, as well as the respective streamwise development of velocity profiles, where the apparent size and shape of the recirculation region seems somewhat predictable. This finding is repeated in Figure 6.17 for $OR = 3$, where, aside from the increasing magnitude of the flow velocity, the time-averaged dual jet flow structure remains relatively constant with increasing Re . To further examine the effect of Re on the time-averaged flow characteristics of a dual jet, the streamlines corresponding to a high and a low Re case are plotted in Figure 6.19 for $OR = 1$ and $OR = 3$. In doing so, the flow structure inside the recirculation region is revealed as two-counter rotating vortices regardless of the chosen jet Reynolds number, as well as a slight separation of the wall jet from the solid wall boundary, which again asserts the assumption that Re does not affect the overall jet structure. This point is further reiterated in Figure 6.20 regarding the shear layer and boundary layer development of the dual jet for $OR = 1$ and $OR = 3$, where a close alignment is evident between the respective plots pertaining to the growth of $(y_{0.5})_1$, $(y_{0.5})_2$, $(y_{0.5})_3$ and δ_{bl} for the range of Re considered.

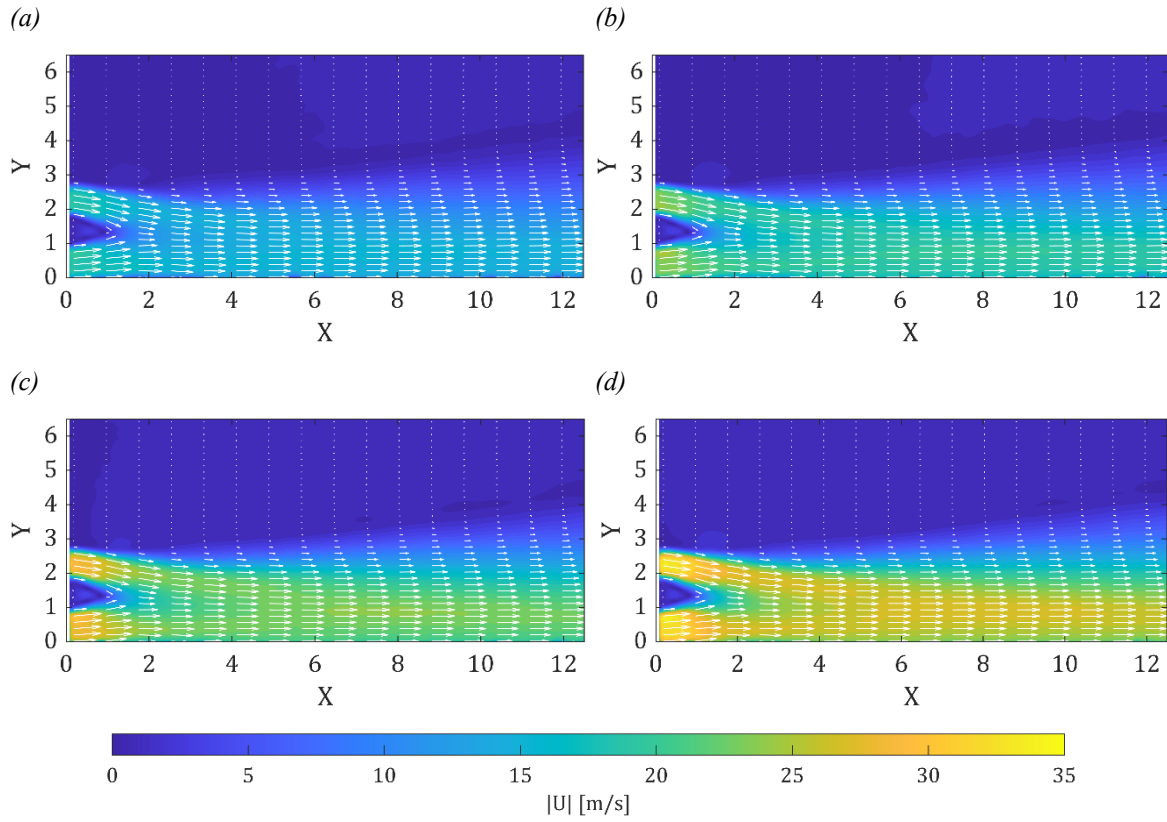


Figure 6.17: Contour and vector plot of the time-averaged velocity field for a dual jet with $OR = 1$ and $V_r = 1$, where $Re =$ (a) 6000, (b) 8000, (c) 10000 and (d) 12000.

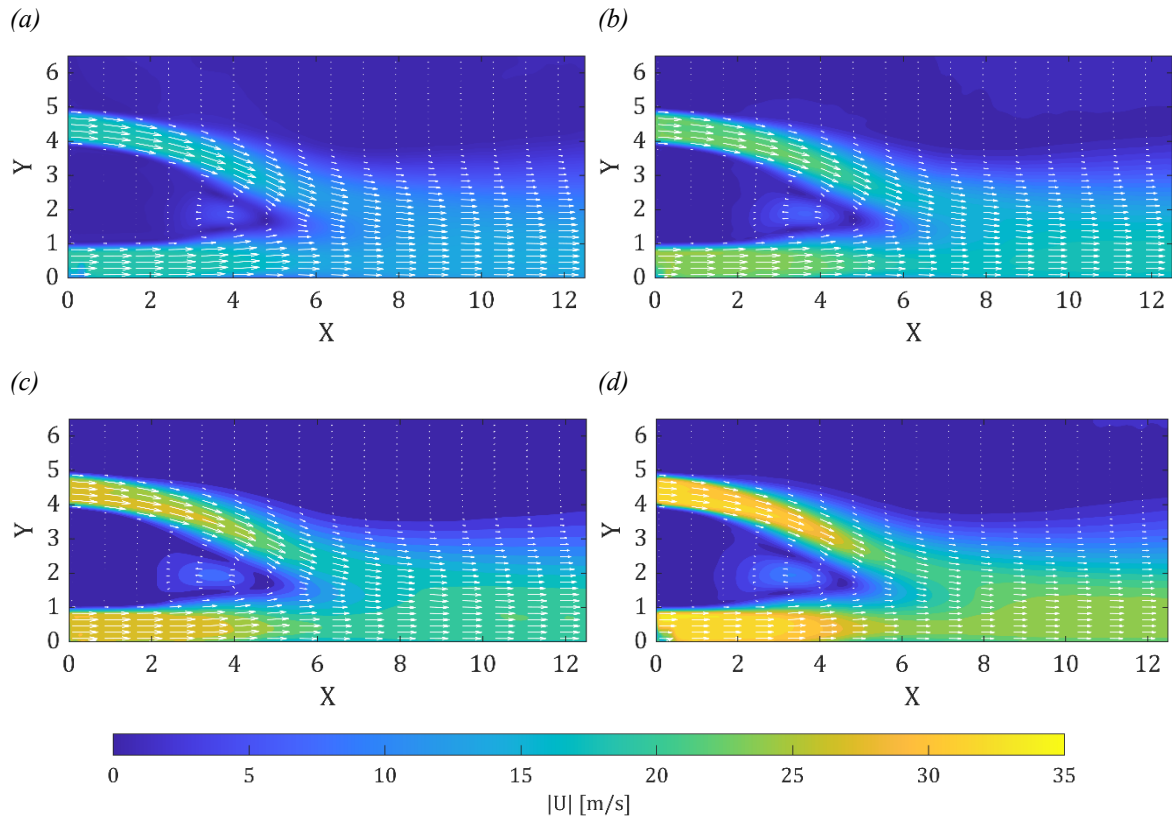


Figure 6.18: Contour and vector plot of the time-averaged velocity field for a dual jet with $OR = 3$ and $V_r = 1$, where $Re =$ (a) 6000, (b) 8000, (c) 10000 and (d) 12000.

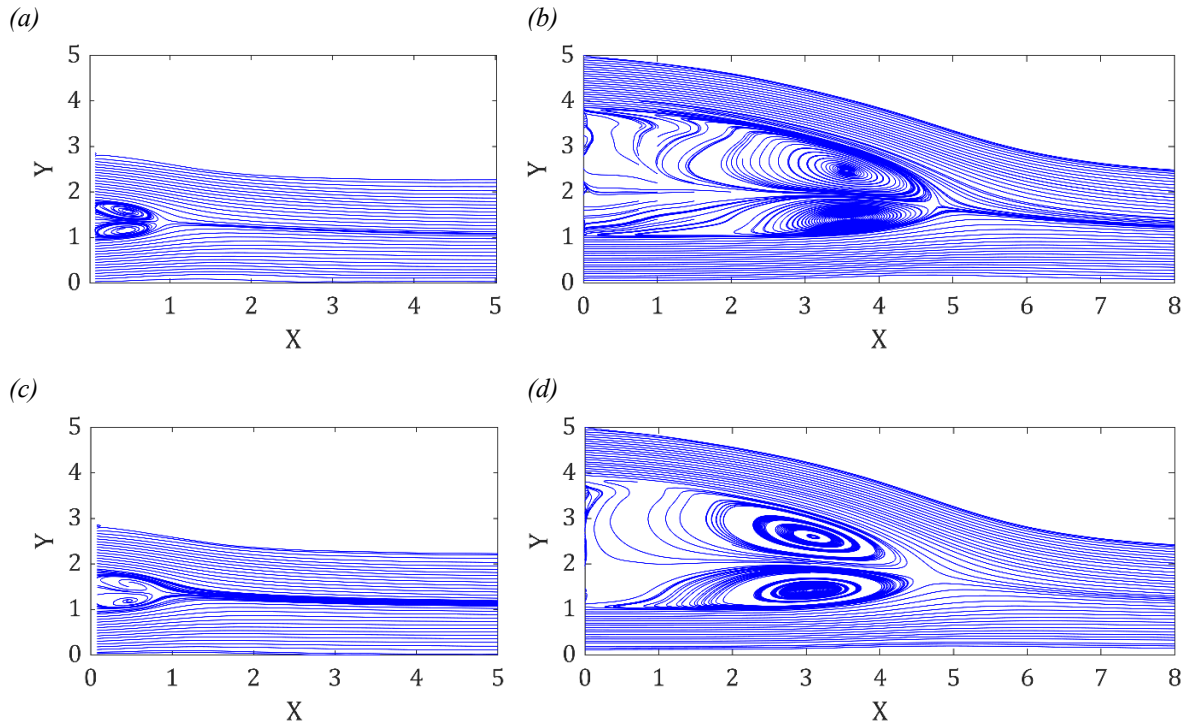


Figure 6.19: Comparison of dual jet streamline plot for changing Re when $V_r = 1$, where (a) $Re = 6000$, $OR = 1$ (b) $Re = 6000$, $OR = 3$ (c) $Re = 12000$, $OR = 1$ and (d) $Re = 12000$, $OR = 3$.

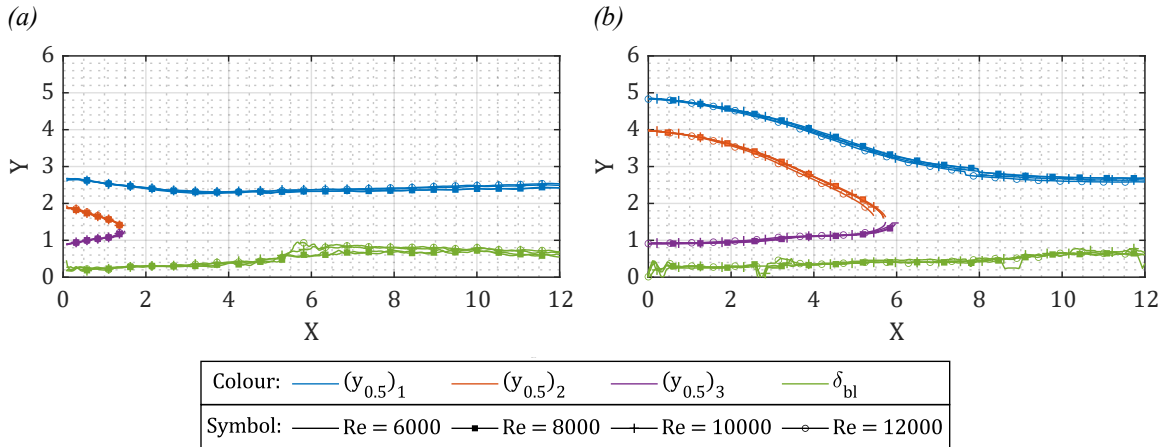


Figure 6.20: Shear layer and boundary layer development for $Re = 6000, 8000, 10000$ and 12000 for (a) $OR = 1$ and (b) $OR = 3$ and $V_r = 1$.

While the general flow structure is relatively unaffected by changing Re , slight variations in the positions of mp , cp , UVC and LVC can be observed from Figure 6.21, which presents their respective X and Y locations for $5500 \leq Re \leq 12000$ and $OR = 1, 2, 3$, the data for which is contained in Table B.1 of Appendix B. Beginning with $OR = 1$, the plots contained in Figure 6.21(a) and Figure 6.21(b) suggest that the merge point moves marginally upstream with increasing Re , while moving further from the solid wall. As this change is less than $0.2w$ for the Re range investigated, it is difficult to draw a definitive conclusion based on the $OR = 1$ data alone. This trend, however, becomes significantly more exaggerated for $OR = 2$, as shown in Figure 6.21(c) and Figure 6.21(d), meaning that despite how seemingly minor the incremental change in X_{mp} and Y_{mp} appear, the resulting trend is absolute. This trend alters somewhat for $OR = 3$, where the data presented in Figure 6.21(e) and Figure 6.21(f) indicates an initial increase followed by a subsequent decrease in the streamwise position of the merge point with increasing Re , while the transverse position appears constant. It

is interesting to note, however, that the observed trend for the merge point at each OR value does not agree with that previously reported by Mondal *et al.* [22], Singh and Dewan [24] and Kumar *et al.* [23]. In addition to this, the values for the merge point reported by previous studies are generally further downstream compared to the present study, as well as further from the solid wall, e.g., $(X_{mp}, Y_{mp}) = (1.49, 1.42)$ or $X_{mp} = 1.54$ for $Re = 10000$ and $OR = 1$, as per Singh and Dewan [24] and Mondal *et al.* [20], respectively.

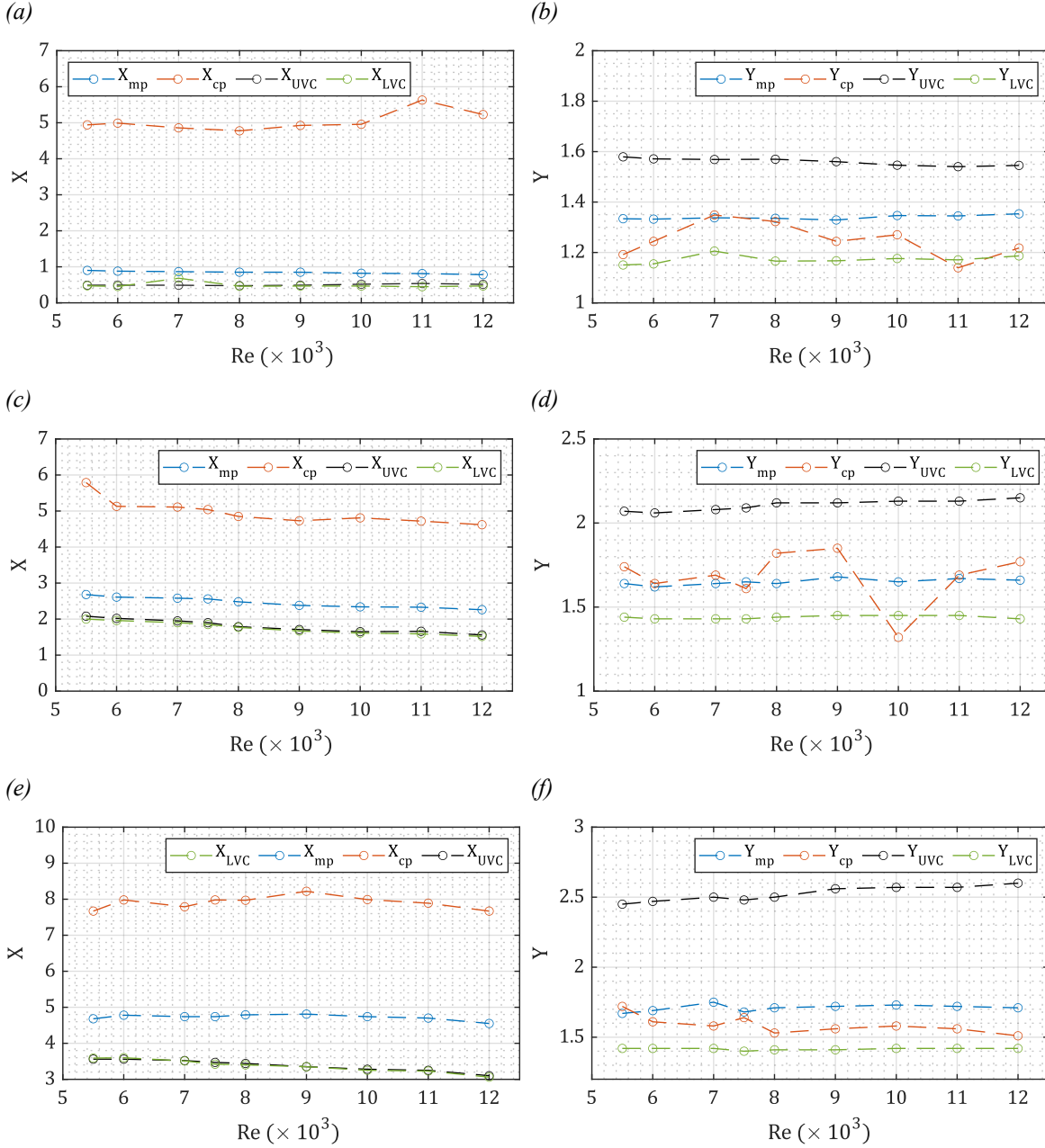


Figure 6.21: Effect of Re on the X-location (left) and Y-location (right) of mp , cp , UVC and LVC , for a dual jet with various OR and $V_r = 1$, where (a, b) $OR = 1$, (c, d) $OR = 2$, (e, f) $OR = 3$.

Conversely, the trends associated with the X and Y positions of the respective centres of the upper and lower vortex are considerably more agreeable for $OR = 2$ and $OR = 3$ compared to $OR = 1$. For the higher OR values, increasing Re moves UVC and LVC closer to the jet exit and upwards, away from the solid wall boundary. While the trend associated with Y_{LVC} appears similar, the values of X_{UVC} and X_{LVC} do not change appreciably with increasing Re for $OR = 1$, while the values of Y_{UVC} indicate the upper vortex moves closer

to the solid wall. For all OR values, UVC and LVC remain mostly aligned in the same vertical plane regardless of the jet Reynolds number or underlying trend. As the positions of UVC and LVC are reported in the literature to a much lesser extent compared to mp , a comparison could only be made to that provided by Kumar [27] for the present set of jet parameters, *i.e.*, $(X_{UVC}, Y_{UVC}) = (1.31, 2.39)$ and $(X_{LVC}, Y_{LVC}) = (1.16, 1.38)$ for $Re = 15000$ and $OR = 2$. Considering the apparent trend, these values seem to agree with the present investigation, however, it is noted that a greater misalignment exists between the vortex centres regarding their streamwise positions.

From these findings, it is reasonable to conclude that increasing Re has a somewhat ‘squeezing’ effect on the recirculation zone in the case of $OR = 1$, as indicated by the discrete positions of mp , UVC and LVC moving closer together. In addition to this, the increasing value of Y_{mp} suggests that the deflection of the wall jet away from the solid boundary becomes more exacerbated at higher Re , all of which is most likely brought about through an increasing rate of entrainment with the recirculation zone and subsequent pressure reduction in this region. These trends are replicated to some degree for $OR = 2$, where the data presented in Figure 6.21(c) and Figure 6.21(d) show a definite shortening of the recirculation zone in the streamwise direction with rising Re , as well as an increased deflection of the wall from the solid wall boundary, although the extent to which this occurs is far less obvious in comparison.

The trends associated with $OR = 2$ primarily differ from that of $OR = 1$ due to the positions of the upper and lower vortex, where the $OR = 2$ data clearly indicates that both UVC and LVC move upstream towards the jet exit and further from the solid wall boundary. The reduction in both X_{UVC} and X_{LVC} is consistent with the reduction in the streamwise length of the recirculation zone, while the increased value of Y_{LVC} could again be attributed to the increasing rate of entrainment in this region. The increase in Y_{UVC} is much greater in comparison, however. This can simply be attributed to the greater vertical dimension of the recirculation zone at each new upstream position, which is similar for the $OR = 3$ case. Conversely, in the case of $OR = 1$, a similar movement in the UVC and LVC positions is not possible due to the significantly smaller dimensions of the recirculation zone and the fact that the upper and lower vortices occupy the total available space in this region, as previously indicated in Figure 6.19. Finally, the data presented for $OR = 3$ shows that the size of the recirculation region remains approximately constant, where, although an increased rate of entrainment occurs, the subsequent reduction in pressure is not enough to affect the deflection of either jet.

Unlike the other flow features, the relationship between the position of the combined point and the jet Reynolds number is considerably less defined. Regarding the X position of cp , Figure 6.21(a) suggests that the downstream location is relatively independent of Re for $OR = 1$, as there is no apparent trend in the plotted data. Conversely, for $OR = 2$, a clear, decreasing trend is observed for X_{cp} from Figure 6.21(c), however, this trend is not replicated in Figure 6.21(e) for $OR = 3$, where an initial increase followed by a subsequent decrease in X_{cp} is evident. Overall, it could be argued that the trend associated with X_{cp} follows that of X_{mp} for the respective OR value, which implies that the length of the combined region remains constant regardless of Re for a given offset ratio. The Y position of cp tends to fluctuate greatly in comparison, particularly for $OR = 1$ and $OR = 2$, whereas the value of Y_{cp} is generally decreasing for $OR = 3$. These fluctuations in the position of cp with respect to Re are to be expected, however, as cp marks the end of the merging region, within which

the majority of mixing and interaction of the two jets take place. It is therefore expected that the flow in this region would be highly chaotic and therefore behave in a less predictable manner.

The decay of local maximum streamwise velocity with downstream distance over the total experimental Re range is presented in Figure 6.22 for both $OR = 1$ and $OR = 3$, where the local maximum streamwise velocity u_m is normalised with respect to the overall maximum velocity $u_{m_{max}}$. While the trend associated with the streamwise decay of u_m is unchanging for each Re value considered, the plots appear more closely aligned near the jet exit and a greater dispersion is evident further downstream. For both OR cases, this dispersion initially takes place in the vicinity of the $u_m/u_{m_{max}}$ minimum, which was previously identified as the point inside the merging region at which the flow direction begins to realign with the streamwise dimension following the initial jet deflection. As the greater values of $u_m/u_{m_{max}}$ occur in conjunction with the higher Re values, it indicates that the rate of decay of maximum streamwise velocity for a dual jet flow decreases with increasing Re by virtue of its increased inertia. In addition to this, due to the location at which the dispersion occurs, the plots suggest that dual jet flows with lower Re experience a greater loss of momentum in the vicinity of the recirculation zone, which becomes apparent only after the flow realigns with the streamwise direction.

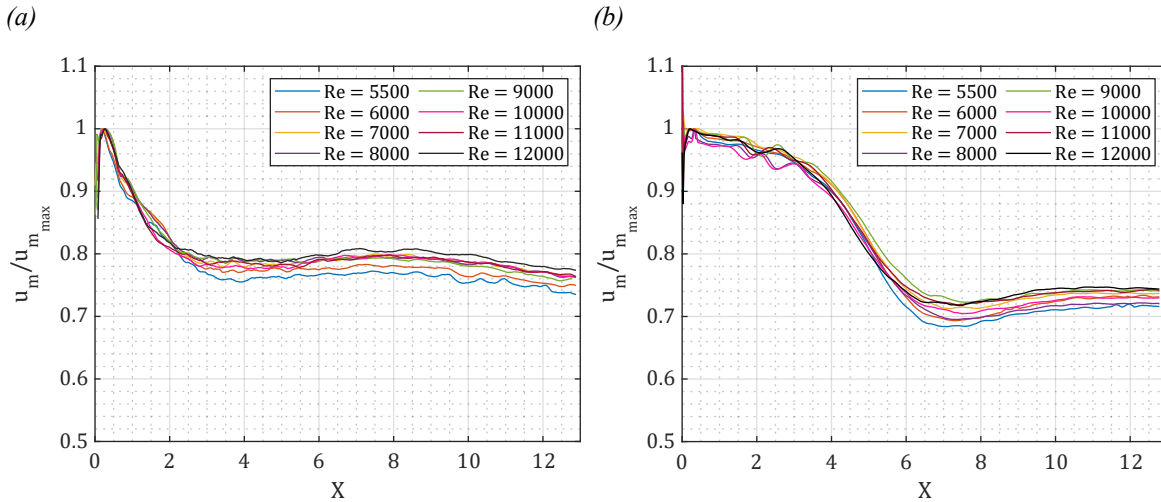


Figure 6.22: Decay of the maximum streamwise velocity, normalised with respect to the overall maximum velocity, for various Re and $V_r = 1$, where (a) $OR = 1$ and (b) $OR = 3$.

The normalised streamwise velocity plots at various downstream locations for $OR = 1$ and $OR = 3$ are displayed in Figure 6.23 and Figure 6.24, respectively, for the range of Re examined. There is generally a close alignment between the relevant datasets at each downstream location, particularly in the combined region, where the velocity plots are characterised by a single local maximum point, *i.e.*, from $X \geq 6$ for $OR = 1$ and $X \geq 10$ for $OR = 3$. The greatest dispersion between the velocity profiles is observed within the merging region for both cases, identifiable by the existence of a deflection in the velocity profile, which can be attributed to the varying streamwise length of the recirculation zone. For $OR = 1$, the value of u/u_m at the local minimum at $X = 2$ is generally less for lower Re , which is consistent with the slower flow development associated with the increased streamwise position of the merge point. For $OR = 3$, the trend associated with the local minimum at $X = 6$ is not as clearly defined, however, it follows the somewhat ambiguous trend associated with X_{mp} identified for $OR = 3$. Based on these observations, it is possible to conclude that the effect of the jet Reynolds number on the streamwise velocity profile of a dual jet flow is more apparent upstream of the combined point,

i.e., within the converging and merging regions. Beyond this point, the differences between the non-dimensionalised velocity profiles becomes almost imperceptible as the flow approaches self-similarity. It should be noted, however, as the field-of-view of the experimental setup is limited, it is not possible to assess the effect of Re on the downstream position at which the dual jet achieves self-similarity.

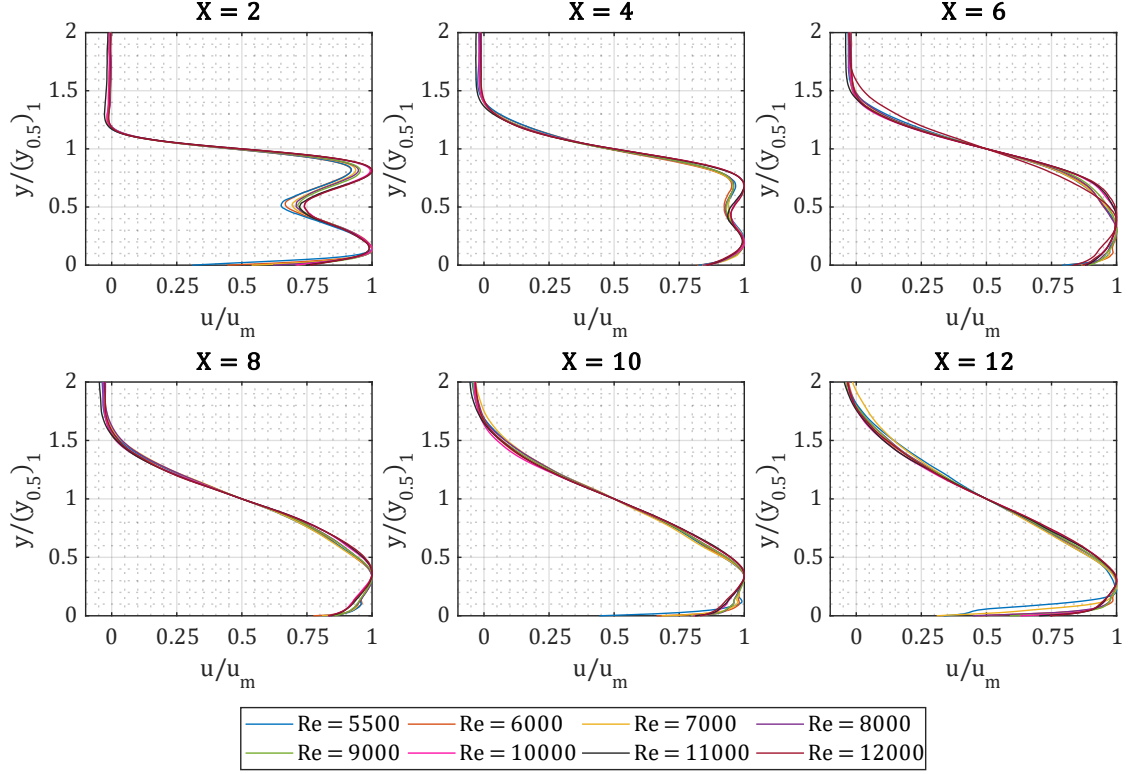


Figure 6.23: Comparison of the normalised velocity profiles for each Re at various downstream locations for a dual jet with $OR = 1$ and $V_r = 1$.

Finally, the effect of the Re on the time-resolved flow behaviour of a dual jet is examined in Figure 6.25, which focuses on the change in the vortex shedding characteristics for $OR = 1$ and $Re = 6000, 8000, 10000$ and 12000 . Figure 6.25(a) presents the power spectral density (PSD) of the time signal associated with the transverse velocity v at a location in the vicinity of the merge point for each of the Re values in question. In doing so, a clear dominant frequency is revealed for each Reynolds number, which corresponds to the vortex shedding frequency. The respective values are $f_s = 603 \text{ Hz}$, 756 Hz , 914 Hz and 953 Hz for $Re = 6000, 8000, 10000$ and 12000 . Based on these values, increasing the dual jet Reynolds number clearly increases the vortex shedding frequency, which is generally the case for flow past a bluff body. Figure 6.25(a) hence presents the Strouhal number associated with the vortex shedding frequency, the value of which remains at an approximate constant value of 0.25 over the total Re range. A minor decrease in St is evident at $Re = 12000$, however, further data would be required at higher Re values to confirm the existence of any additional trends.

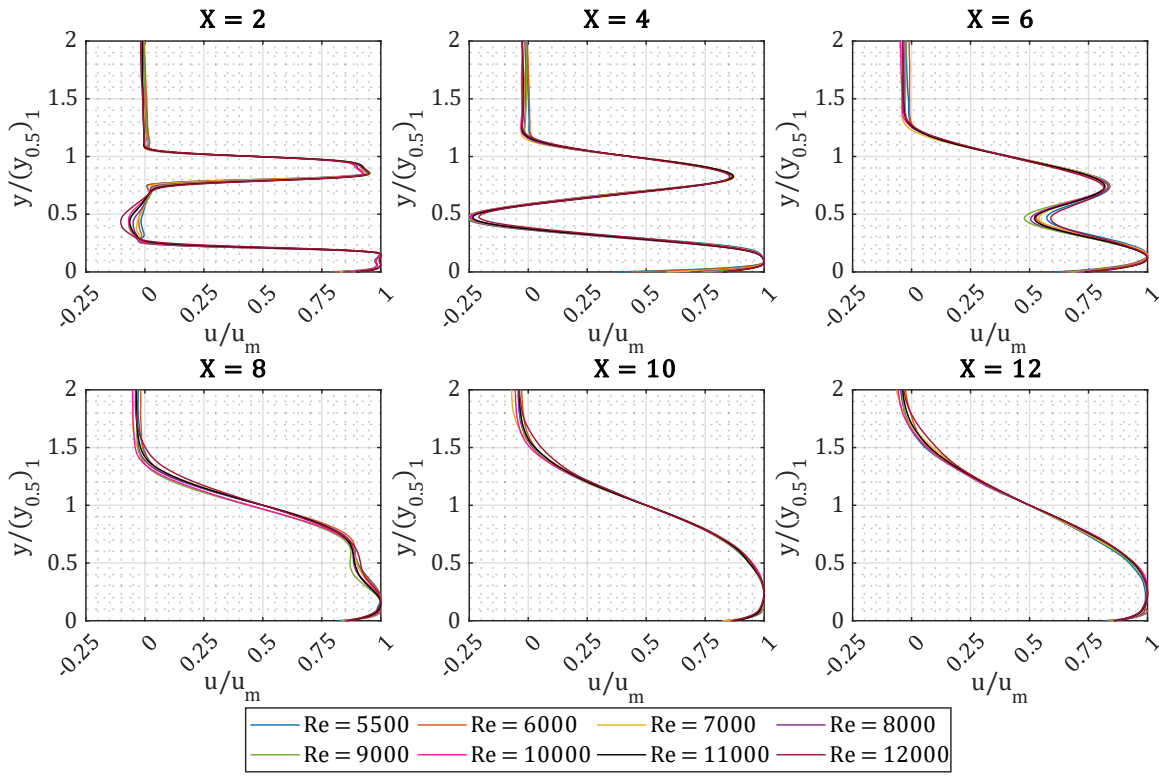


Figure 6.24: Comparison of the normalised velocity profiles for each Re at various downstream locations for a dual jet with $OR = 3$ and $V_r = 1$.

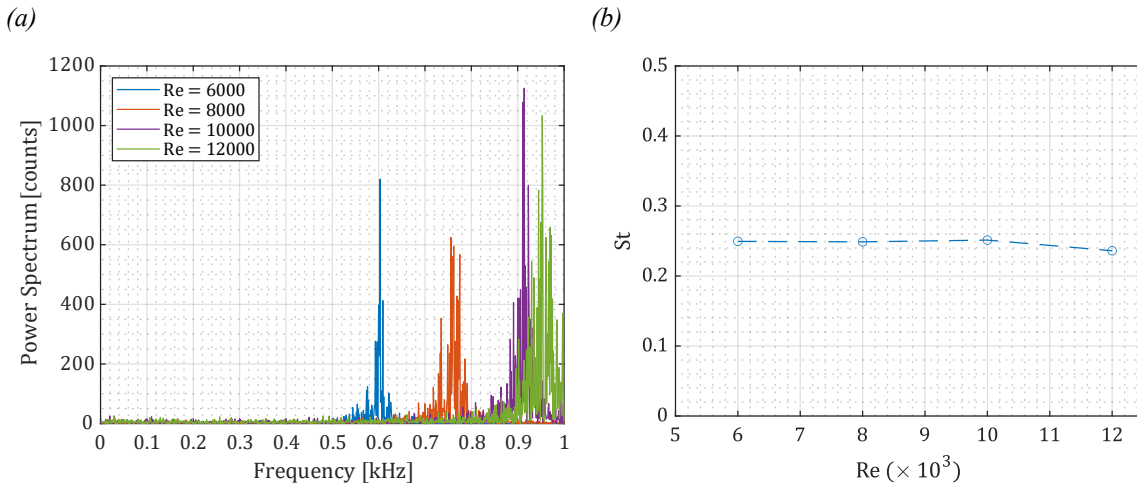


Figure 6.25: (a) Power spectral density (PSD) and (b) Strouhal number (St) associated with the v time signal in the vicinity of mp for a dual jet with $OR = 1$, $V_r = 1$ and $Re = 6000, 8000, 10000$ and 12000 .

6.4.2. Heat Transfer Behaviour

The influence of the jet Reynolds number on the heat transfer characteristics of a dual jet flow is discussed in the succeeding sections in two stages; first, the effect of altering Re on the time-averaged local Nusselt number Nu_x profile is examined, following which the surface-averaged Nu_x is analysed. For the heat transfer analysis, data pertaining to a much wider range of offset ratios is available, making it possible to confirm the existence of trends regarding Re , as well as infer details of the flow behaviour at the higher OR for which the obtainment of PIV data was not possible.

6.4.2.1. Time-Averaged Local Nusselt Number Profiles

The time-averaged Nu_x profiles for $5500 \leq Re \leq 12000$ are displayed in Figure 6.26 for $OR = 1, 3, 5$ and 7 . For all cases examined, the trend remains consistent regardless of the Reynolds number, where Nu_x initially decreases towards a minimum value, which has previously been identified as the location after the merge point at which the wall jet deflection from the solid wall boundary is greatest. Nu_x then increases towards a local maximum value as the jet reattaches to the solid wall, all of which takes place within the merging region. Beyond this point, the value of Nu_x proceeds to decrease in accordance with the slowing of the overall mean jet velocity.

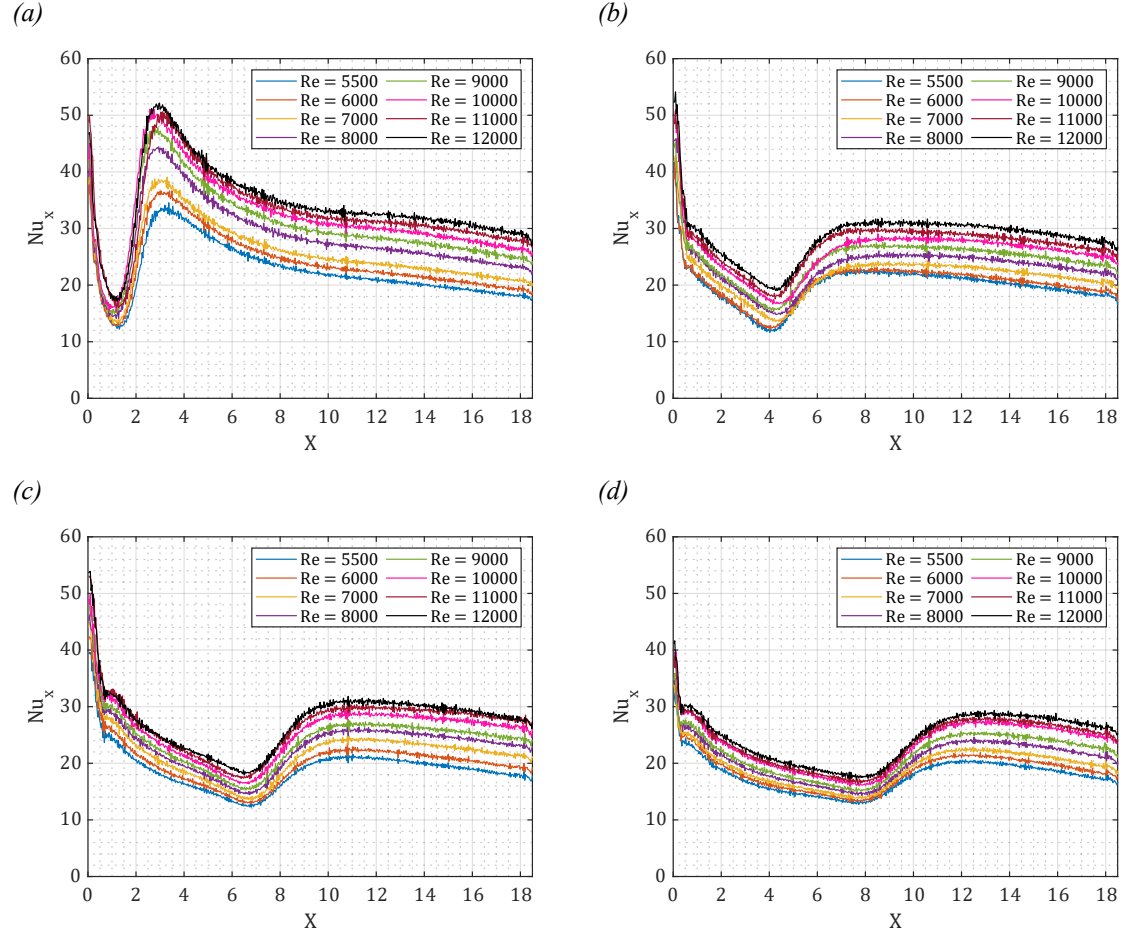


Figure 6.26: Time-averaged Nu_x profiles for a dual jet for different Re values at $V_r = 1$ for $OR =$ (a) 1, (b) 3, (c) 5 and (d) 7.

For each OR presented, increasing the value of Re appears to enhance Nu_x at all downstream locations while maintaining the approximate X locations of all notable downstream features, which is similar to that observed for the single wall jet or single offset jet. As the effect of Re can be viewed as an approximate ‘shift’ of the Nu_x profile upwards along the vertical axis, it can be inferred that the enhancement of Nu_x at each streamwise location is somewhat uniform, which results in the preservation of the overall trend. While this enhancement of Nu_x with increasing Re is to be expected since the increased momentum of the jet should naturally induce a greater rate of heat transfer, upon closer inspection, there are slight variations in the extent of the Nu_x augmentation depending on the downstream region. For example, the plots appear more closely grouped inside the converging region, where the alignment between the plots becomes closer in the vicinity of $Nu_{x_{min}}$. Beyond

this point, as Nu_x increases rapidly towards $Nu_{x_{max}}$, so too does the dispersion between the plots. Downstream of $Nu_{x_{max}}$, which is primarily the combined region, there is a definite increase in the enhancement of Nu_x between successive Re test cases compared to the converging region for all OR shown.

The reasoning for this observation can readily be obtained from the fluid flow behaviour, in particular Figure 6.22, which describes the decay of maximum velocity with downstream distance. As previously discussed, increasing Re decreases the rate of decay of maximum streamwise velocity for a dual jet flow, as the lower Re flows experience as greater loss of momentum in the vicinity of the recirculation zone. As the rate of momentum loss from a dual jet flow with changing Re is not proportional, it is to be expected that the induced heat transfer behaviour would follow suit. As such, the magnitude of the enhancement of Nu_x in the far downstream region is greater than that experienced the near jet region for increasing Re due to the better preservation of jet momentum succeeding the recirculation zone that can result from the increased jet inertia.

Finally, it is interesting to note the positioning of $Nu_{x_{min}}$ and $Nu_{x_{max}}$ with increasing Re . While their respective streamwise positions appear somewhat consistent, slight variations are observed, particularly for higher OR . For $OR = 3$, *i.e.*, Figure 6.22(b), the X location of $Nu_{x_{min}}$ appears to move downstream within increasing Re , an explanation for which could potentially be attributed to an elongation of the separation region with increasing Re . This observation can also be applied to $OR = 5$ and $OR = 7$, shown in Figure 6.22(c) and Figure 6.22(d), respectively. With regard to $OR = 1$, while the positioning $Nu_{x_{min}}$ appears considerably more consistent, it is likely the effect of the elongation of the separation zone on the heat transfer behaviour with increasing Re is offset by the movement of the merge point upstream, which is not the case for $OR = 3$. While other, perhaps more obvious, differences exist between the Nu_x curves for changing OR , they are discussed in greater detail in section 0.

When compared to the published numerical data relating to dual jet flows and their corresponding heat transfer characteristics, some of the observed trends are consistent, while others are in direct opposition. For example, the enhancement of Nu_x at all streamwise locations with increasing Re is reported by Vishnuvardhanarao and Das [28], Mondal *et al.* [22], Hnaïen *et al.* [84], and Singh and Dewan [24], however, the Nu_x profile changes drastically between each study, as previously discussed in section 6.2 for $Re = 10000$. Further to this, while Vishnuvardhanarao and Das [28], Hnaïen *et al.* [84], and Singh and Dewan [24] note no obvious change in behaviour with varying Re , Mondal *et al.* [22] suggests that the location of a Nu_x peak moves downstream with increasing Re for $OR = 1$ due to a delay in the onset of the jet interaction. However, the present study did not corroborate this finding experimentally for the same set of jet parameters.

6.4.2.2. Surface-Averaged Nusselt Number

The variation in the surface-averaged Nusselt number $\overline{Nu_x}$ with respect to Re is presented in Figure 6.27 for $OR = 1, 2, 3, 5$ and 7 , which is acquired through averaging the relevant Nu_x data over the entire downstream domain of $18.5w$. A clear linear relationship is evident in each case, where an increase Re leads to a corresponding, proportionate rise in $\overline{Nu_x}$, where the rate of increase in $\overline{Nu_x}$ appears to rise with decreasing OR , as previously noted by Hnaïen *et al.* [84]. While a linear trend in $\overline{Nu_x}$ with Re is unusual finding for turbulent flow, a linear relationship between $\overline{Nu_x}$ and Re has also been reported in other numerical dual jet studies, such as Vishnuvardhanarao and Das [28] and Mondal *et al.* [22]. This may be due to the streamwise

distance over which the local Nu_x values have been averaged, where increasing the length of the test area may result in an $\overline{Nu_x}$ trend requiring Re to have exponent of less than 1 for proportionality, which would be considered more typical of turbulent flow. Having said this, both Vishnuvardhanarao and Das [28] and Mondal *et al.* [22] averaged their Nu_x data over much longer streamwise distances, meaning a direct numerical comparison with the present investigation cannot be made, however, the linear trend certainly remains consistent regardless.

For the present investigation, the linear fit is applied to Re and $\overline{Nu_x}$ dataset for each OR value, the resulting equations for which are contained within Figure 6.27. In doing so, the coefficient of determination, or r^2 value, varies from 0.9816 to 0.9990, which confirms the suitability of the linear fit. Similar to the previous statement, the slope associated with the line of best fit increases with decreasing OR , which indicates the effect of Re is more apparent at lower OR . The relationship between $\overline{Nu_x}$, Re and OR is further discussed in section 6.5.2.2.

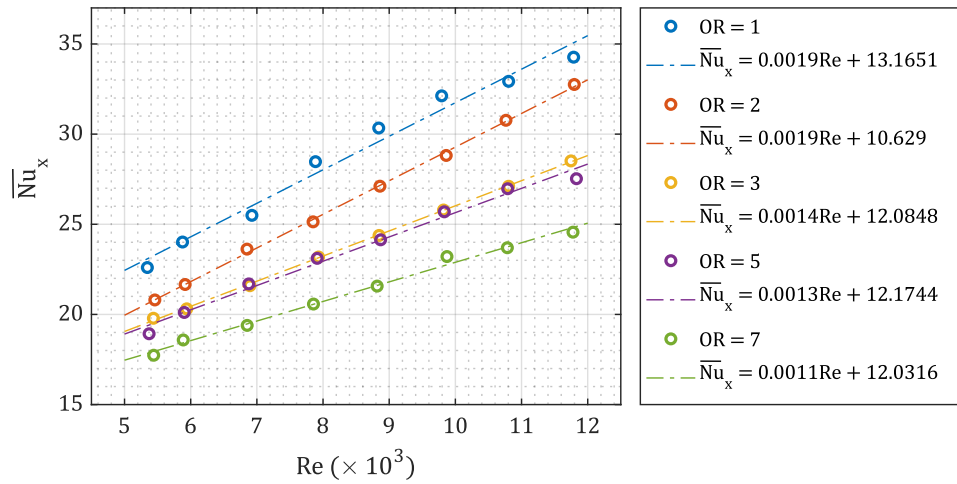


Figure 6.27: Relationship between the surface-averaged, time-averaged Nusselt number $\overline{Nu_x}$ and the dual jet Reynolds number for $V_r = 1$.

6.5. Effect of Offset Ratio

The following sections discuss the effect of varying the offset ratio on the flow behaviour and induced heat transfer characteristics of a dual jet. Similar to the effect of Re , there is no experimental data available in the published literature pertaining to a dual jet with an offset ratio greater than 1. The effect of OR for a dual jet has been explored through numerical means in studies such as Li *et al.* [92], Mondal *et al.* [20], Kumar [27], Kumar [83], Hnaïen *et al.* [84] and Hnaïen *et al.* [25], and, while the observed trends are generally agreeable, no experimental evidence has ever corroborated their findings. Most notably, studies such as Mondal *et al.* [20] suggest that the vortex shedding phenomenon that is observed inside a dual jet flow field is a feature of certain offset ratio values only, and a more stable flow field can be observed for $OR < 2.1$. The following sections therefore analyse the time-averaged and time-resolved flow data of a dual jet for $OR = 1, 2$ and 3 at the extreme Re values to verify whether or not this flow feature can be replicated in an experimental scenario. Subsequent to this, the changes induced to the heat transfer behaviour are examined with respect to the observed changes in the flow field.

6.5.1. Flow Field Characteristics

The time-averaged flow field obtained for a dual jet is presented in Figure 6.28 for $OR = 1, 2$ and 3 by means of the time-averaged velocity $|U|$ contour plot, onto which the corresponding streamlines are overlayed. As section 6.4.1 indicated that the overall flow structure is generally unaffected by Reynolds number, only a single jet Reynolds number of $Re = 12000$ is shown in Figure 6.28. For all OR shown, the general dual jet flow behaviour is relatively consistent, where both jets initially deflect towards each other upon exiting the jet nozzle and eventually merge at some downstream distance, which leads to the creation of the recirculation zone immediately downstream of the nozzle exit. From the plots shown in Figure 6.28, it is clear that increasing OR increases both the streamwise and transverse dimensions of the recirculation zone, which thus moves the location of the merge point downstream. Inside the recirculation zone, two counter-rotating vortices are observed for each OR , the size of which appears to grow with the increasing size of recirculation zone. In the case of $OR = 1$, the streamwise location of the respective vortex centres appears somewhat midway between X_{mp} and the nozzle exit. This is not the case for the higher OR values, however, where the location of both UVC and LVC are generally biased towards the merge point.

To further examine the changes in the flow structure, the development of the shear layers and the boundary layer for $OR = 1$ and $OR = 3$ are plotted in Figure 6.29 for $Re = 5500$. By virtue of the change in the offset jet height, the starting positions of $(y_{0.5})_1$ and $(y_{0.5})_2$ for $OR = 3$ are naturally higher to correspond with the upper and lower edge of the offset jet nozzle exit. Similar to that observed for $OR = 1$, the initial decrease in $(y_{0.5})_1$ and $(y_{0.5})_2$ for $OR = 3$ follows the deflection of the offset jet, where the total downstream existence of $(y_{0.5})_2$ is constrained to the recirculation zone. Layer 1 continues to develop beyond the recirculation zone, however, unlike $OR = 1$, the value of $(y_{0.5})_1$ for $OR = 3$ does not start to increase at any point, but instead appears to approach that of $OR = 1$ in the far downstream region. Inside the wall jet region, the boundary layer growth appears unaffected by the increased offset ratio, however, the difference in the downstream development of $(y_{0.5})_3$ for $OR = 1$ and $OR = 3$ is consistent with a delay in the deflection of the wall jet due to the elongation of the recirculation region.

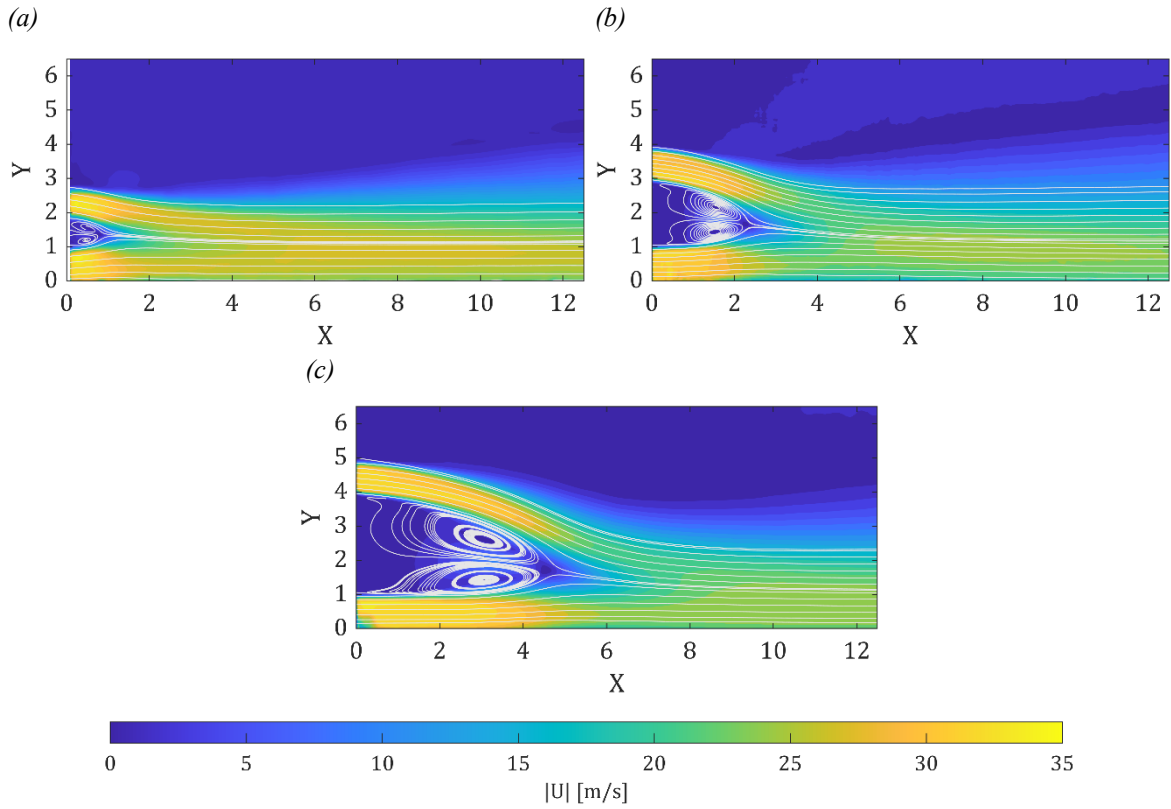


Figure 6.28: Contour plot and streamlines for the time-averaged velocity field of a dual jet with $Re = 12000$ and $V_r = 1$, where $OR = (a) 1, (b) 2$ and $(c) 3$.

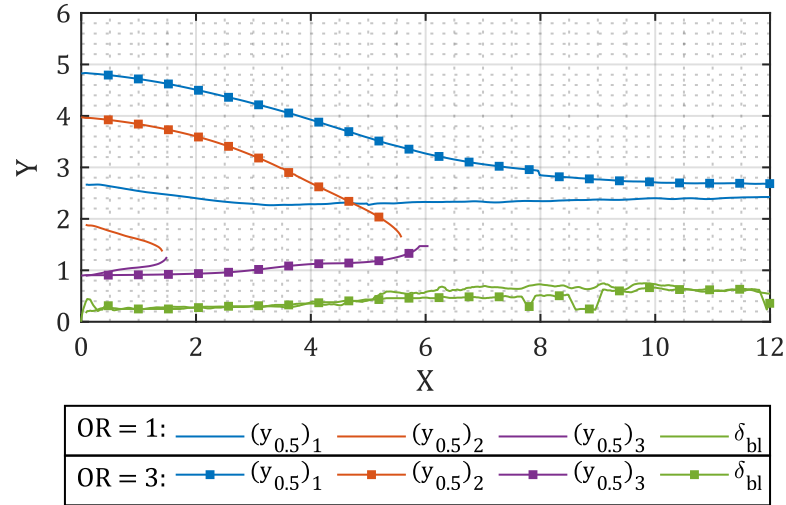


Figure 6.29: Comparison of shear layer and boundary layer development for $OR = 1$ and $OR = 3$ in a dual jet with $Re = 5500$ and $V_r = 1$.

The effect of the offset ratio on the positioning of mp , cp , UVC and LVC is presented in Figure 6.30 for $Re = 5500$ and $Re = 12000$, where the observed trends are generally consistent for both Re considered. Beginning with the merge point, Figure 6.30(a) shows that increasing OR pushes the position of the merge point further downstream, which is obvious considering the growth in the recirculation zone. This change in X_{mp} is linear for both Re cases, which should continue for higher OR values. While an increasing trend is also evident for Y_{mp} in Figure 6.30(b), the change in Y_{mp} between $OR = 1$ and $OR = 2$ is greater than that observed between $OR = 2$ and $OR = 3$, indicating a potential limiting value. This is consistent with that previously observed for

the development of $(y_{0.5})_1$ for $OR = 3$, meaning that despite an increase in OR , the height of the dual jet cannot grow unboundedly in the far downstream region, but instead approaches that of lower OR values.

The trends associated with the positioning of UVC and LVC are consistent with previous studies, such as Kumar [27], whereby an increase in OR increases the size of both the upper and lower vortex, where the growth of the lower vortex consequentially pushes the upper vortex upwards. This can be interpreted from Figure 6.30(b) as a minor increase in Y_{LVC} , but a much greater increase in Y_{UVC} . The trend associated with the increase in Y_{LVC} follows that of Y_{mp} , which again indicates the existence of a limiting Y_{LVC} value as OR continues to increase. Conversely, the trend associated with Y_{UVC} implies that Y_{UVC} is free to grow unbounded with increasing OR , as its position depends on the extent of the offset jet deflection. Finally, the positions of both vortex centres move downstream with increasing OR , while remaining in the same vertical plane.

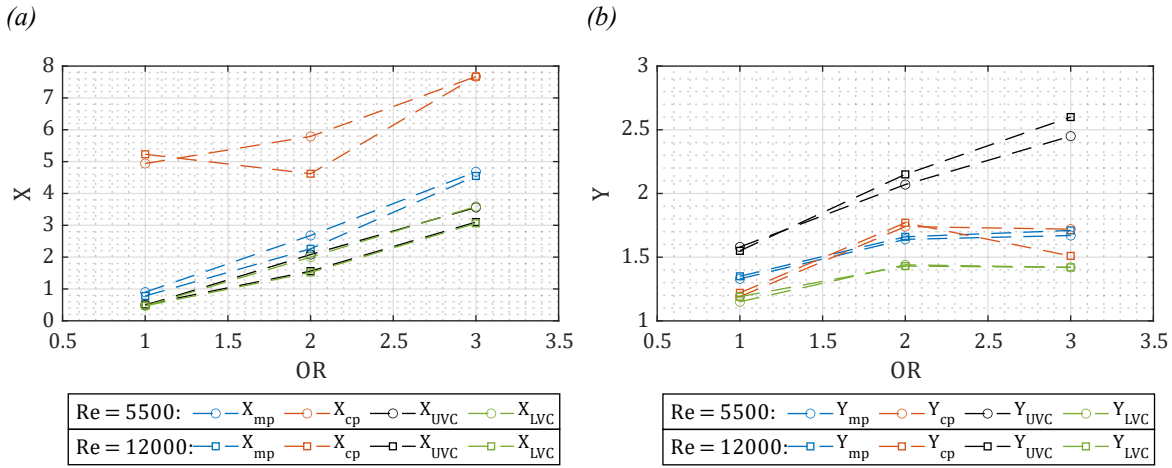


Figure 6.30: Effect of OR on the (a) X -location and (b) Y -location of mp , cp , UVC and LVC , for a dual jet with $Re = 5500$ and $Re = 12000$ and $V_r = 1$.

The position of the combined point appears to follow a similar trend to the merge point, where the data suggests it moves downstream and further from the solid wall with increasing OR . Regarding the streamwise position of cp , the rate of increase of X_{cp} with rising OR is generally less than that of X_{mp} , which indicates a shortening of the merging region with higher OR values. In addition to this, the value of Y_{cp} appears to be approaching a limiting value, similar to Y_{mp} and Y_{LVC} , which again indicates that the spreading rate of the dual jet in the downstream region is relatively consistent regardless of OR . As previously discussed in relation to the combined point for $Re = 10000$ and $OR = 1$, the streamwise position of cp for the higher OR values continues to be considerably less than that reported in the published numerical studies relating to dual jet flows, such as $X_{cp} = 16.25$ ($OR = 2$ and $Re = 15000$) according to Kumar [27] or $X_{cp} = 36.27$ ($OR = 2.5$ and $Re = 10000$) as per Mondal *et al.* [20].

The streamwise decay of the non-dimensionalised maximum velocity with respect to changing OR is presented in Figure 6.31 for $Re = 5500$ and $Re = 12000$. For each test case shown, the overall trend regarding $u_m/u_{m_{max}}$ is constant, where an initial decrease occurs upon ejection from the nozzle exit, followed by an increase as the jet deflection realigns with the streamwise direction, beyond which $u_m/u_{m_{max}}$ continues to decline. Through increasing OR , it is clear that the streamwise position of the local $u_m/u_{m_{max}}$ minimum moves further downstream, which is consistent with the increased value of X_{mp} and, hence, the extended region over

which the jets remain deflected. In addition to this, the initial drop in $u_m/u_{m_{max}}$ near the jet exit is delayed for higher OR values, which again insinuates the delayed onset of the jet deflection in this region. From the data presented, it appears that the curves become more aligned in the far downstream region, however, it is clear that $OR = 1$ maintains the greatest value of u_m in this region. Increasing OR therefore results in a lower u_m value at the local minimum position, beyond which the recovery of u_m is generally less. The results of the present investigation are also compared to numerical results of Kumar [27] for $Re = 15000$ and $OR = 2$ in Figure 6.31(b), which were obtained using the $k - \varepsilon$ turbulence model. In doing so, a similar trend is observed, where the location of the local minimum is consistent with that obtained by the present study for $OR = 2$, however, the overall rate of decay is considerably less. The results suggest that published dual jet numerical models generally underestimate the rate of the flow development, which is consistent with the greater values obtained for X_{cp} .

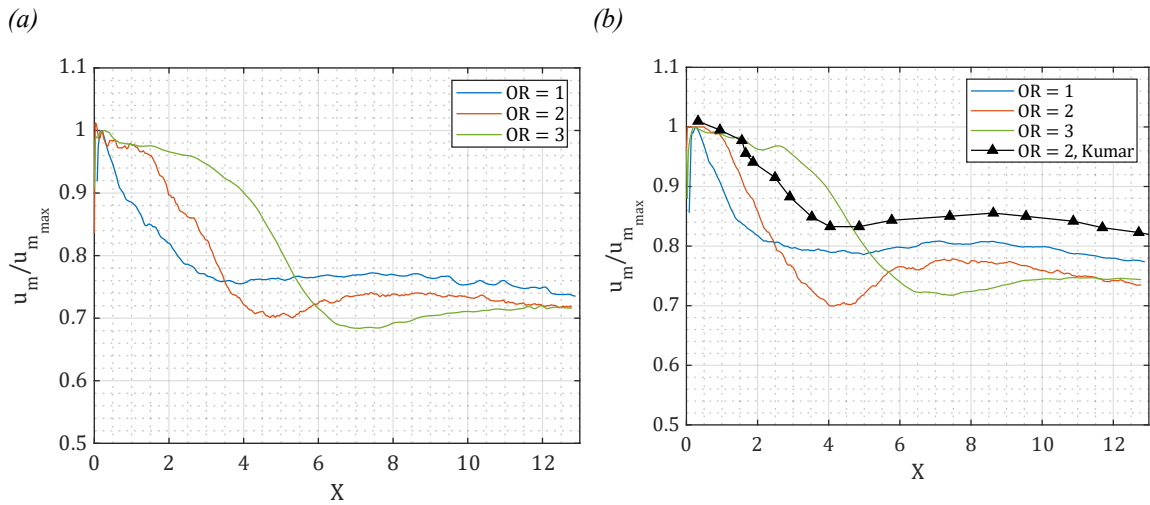


Figure 6.31: Decay of the local maximum streamwise velocity, normalised with respect to the overall maximum velocity, for a dual jet with $V_r = 1$ and varying OR , where (a) $Re = 5500$ and (b) $Re = 12000$, compared to that of Kumar [27] for $Re = 15000$ and $OR = 2$.

The velocity profiles at various downstream locations are shown in Figure 6.32 for $Re = 12000$ and $OR = 1, 2$ and 3 . As expected, the profiles differ greatly from $2 \leq X \leq 8$, as not all of the dual jet flows in question have reached their respective combined regions by this stage. As the size of the recirculation zone, the merge point location and the combined point location change appreciably as OR is varied from 1 to 3 , it seems reasonable that the greatest difference in the velocity profiles would be experienced closer to the jet exit. As such, the velocity profile development for $OR = 3$ is slower than that of $OR = 2$, which is slower than that of $OR = 1$, as the notable flow features are located progressively further downstream with increasing OR . Conversely, a much closer alignment is observed between the velocity profiles for $X \geq 10$, at which point all jet configurations have reached the combined region. This alignment appears to improve with downstream distance, which implies that the effect of OR becomes negligible at further downstream distances.

To examine the effect of the offset ratio on the time-resolved flow behaviour of a dual jet, the instantaneous flow field acquired for a dual jet with $Re = 10000$ is displayed in Figure 6.33 for $OR = 2$ and $OR = 3$. As aforementioned, studies such as Mondal *et al.* [20] postulated that the vortex-shedding phenomenon previously observed inside the dual jet flow fields is a feature of a certain range of OR , outside of which the stable, counter-rotating vortex regime is observed within the recirculation zone of the instantaneous flow fields. In the case of

Mondal *et al.* [20], when investigated numerically using the $k - \varepsilon$ turbulence model, vortex shedding was only observed for a dual jet with $Re = 10000$ if $0.7 \leq OR \leq 2.1$. However, the results of the present investigation suggest that vortex shedding may also be observed for $OR > 2.1$. The instantaneous velocity contour plots and corresponding streamlines shown in Figure 6.33 indicate instabilities in the flow behaviour for both $OR = 2$ and $OR = 3$, where neither case presents two stable vortices in the recirculation zone. Instead, the flow behaviour inside this region appears more unpredictable. There is clear evidence of vortex growth inside the recirculation zone, which appear to be subsequently shed and convected downstream. In the case of $OR = 2$, the presence of alternating, high velocity regions is a clear indication of a von Kármán-like shedding of vortices, however, this is far less obvious in the case of $OR = 3$. As the shedding occurs much further downstream in this case, it is likely that the vortices much dissipate at a faster rate, as the dual jet must still exhibit consistent behaviour in the far downstream region regardless of OR , as previously discussed.

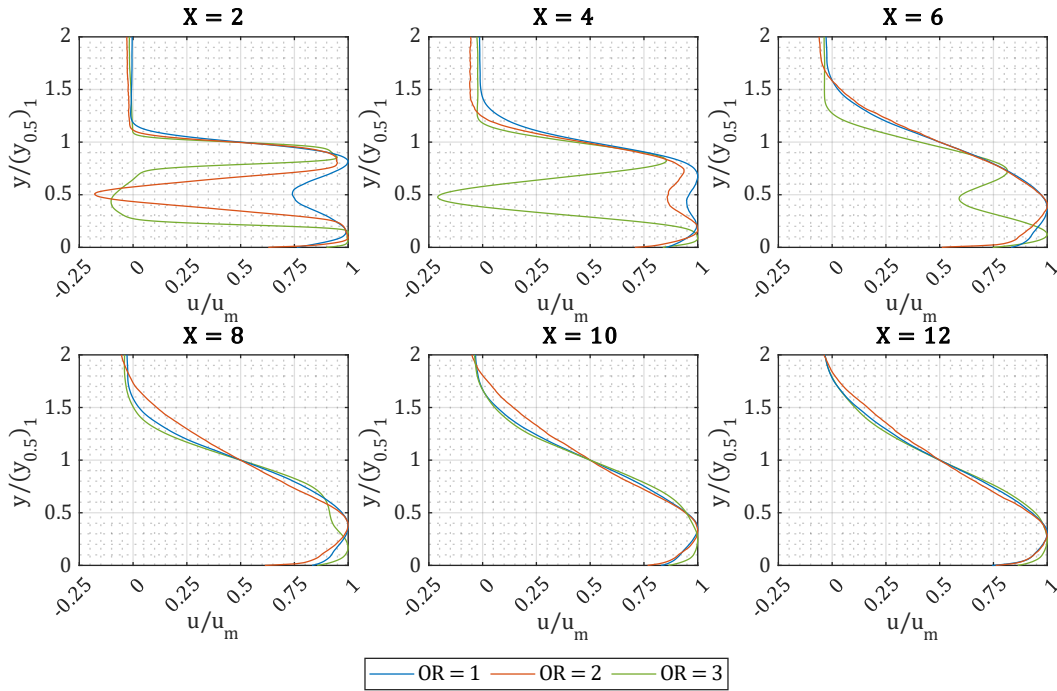


Figure 6.32: Comparison of the normalised velocity profiles for $OR = 1, 2$ and 3 at various downstream locations for a dual jet with $Re = 12000$ and $V_r = 1$.

To further examine the vortex shedding characteristics of the higher offset ratio values, the instantaneous vorticity contour plots corresponding to $OR = 2$ and $OR = 3$ are presented in Figure 6.34, onto which the isolines pertaining to constant swirling strength λ_{ci} are overlayed. In doing so, the high vorticity regions which relate to swirling flow only are identified within the flow field, which include the Kelvin Helmholtz roll-up vortices along the outer shear layer of the wall jet, as well as the swirling regions of flow along Layer 2 and Layer 3 inside the recirculation zone. In both cases, vortices are present inside the dual jet flow field downstream of the recirculation zone, where the wall and offset jets primarily interact, the vorticity of which alternate from positive to negative in a regular fashion. This is clear evidence of periodic vortex shedding from the recirculation zone, the apparent translational velocity of which is $u_{trans} = 0.68u_{jet}$ and $u_{trans} = 0.6u_{jet}$ for $OR = 2$ and $OR = 3$, respectively. These values can be compared to that previously acquired for $OR = 1$ of $u_{trans} = 0.76u_{jet}$.

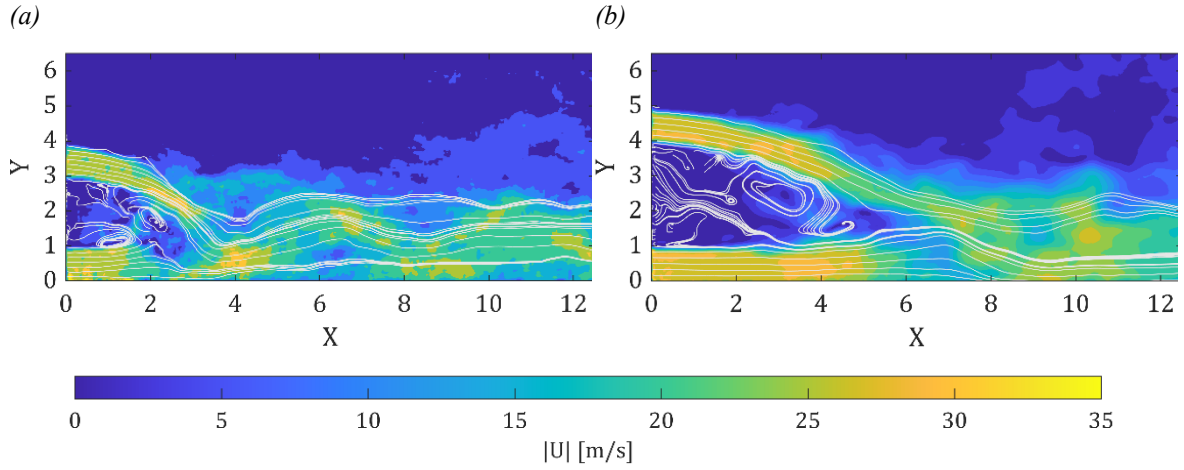


Figure 6.33: Instantaneous contour plot and streamlines for the velocity field of a dual jet with $Re = 10000$ and $V_r = 1$ at a sample timestep t , where $OR = (a) 2$ and $(b) 3$.

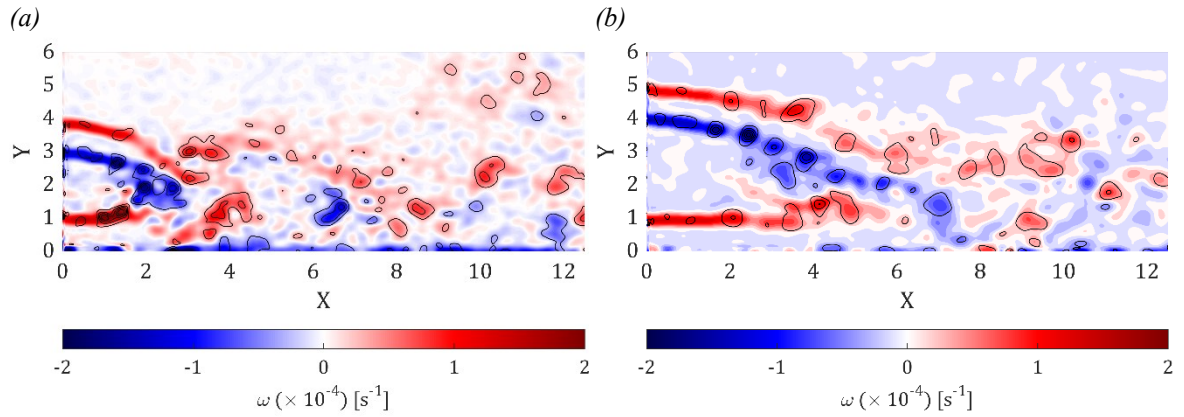


Figure 6.34: Instantaneous vorticity contour plot of a dual jet with $Re = 10000$ and $V_r = 1$, where $OR = (a) 2$ and $(b) 3$, alongside isolines of constant swirling strength λ_{ci} .

The power spectral density (PSD) of the time signal associated with the transverse velocity v at a location in the vicinity of the merge point is presented in Figure 6.35(a) for $Re = 10000$ and $OR = 1, 2$ and 3 . A clear dominant frequency is revealed in each case, which corresponds to the vortex shedding frequency. When scaled with respect to the lower OR values, the dominant frequency for $OR = 3$ appears significantly less prominent, however, the dominant frequency for $OR = 3$ is still very much present and enables the identification of the shedding frequency. In doing so, shedding frequency values of $f_s = 914 \text{ Hz}$, $f_s = 505 \text{ Hz}$ and $f_s = 377 \text{ Hz}$ can be identified for $OR = 1, 2$ and 3 , respectively, which indicates that the shedding frequency decreases with increasing OR . The corresponding Strouhal numbers are plotted in Figure 6.35(b), where the decreasing trend with increasing OR is consistent with Mondal *et al.* [20]. Based on this finding, it stands to reason that there should exist an OR value at which vortex shedding ceases to occur, although its identification is beyond the limitations of the present experimental setup.

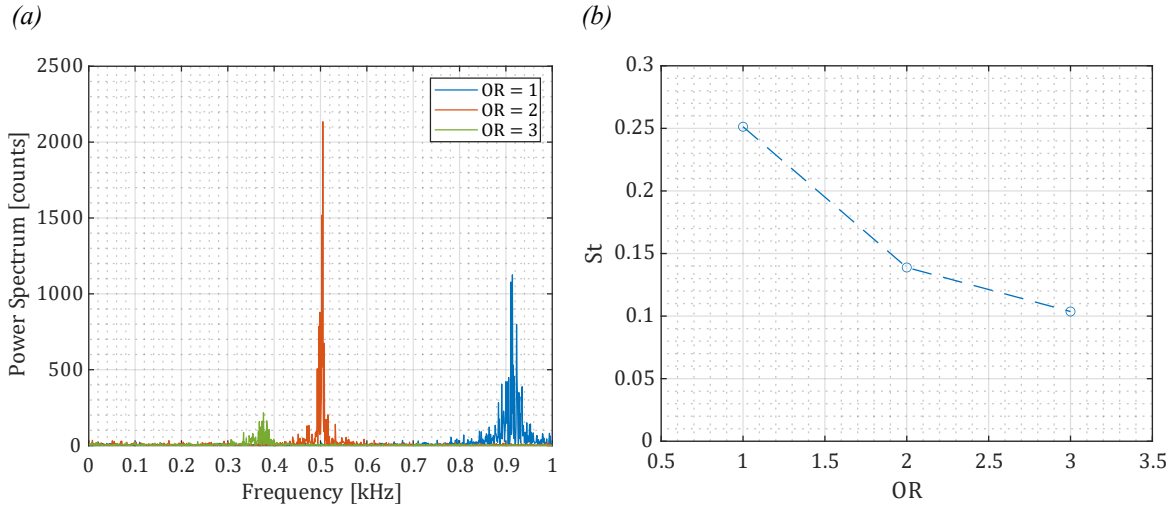


Figure 6.35: (a) Power spectral density (PSD) and (b) Strouhal number (St) associated with the v time signal in the vicinity of mp for a dual jet with $Re = 10000$ and $V_r = 1$ for $OR = 1, 2$ and 3 .

6.5.2. Heat Transfer Behaviour

The influence of the offset ratio on the heat transfer characteristics of a dual jet flow is discussed in the following sections, with respect to both the time-averaged local Nusselt number Nu_x profile and the surface-averaged Nu_x . The influence of OR on the induced heat transfer in the solid wall has been investigated in studies such as Kumar [83], Farooq and Das [29], and Hnaïen *et al.* [84] though numerical means, and, while a general decline in $\overline{Nu}_x$ has been observed with increasing OR , very different trends have been reported regarding the downstream local Nusselt number profile. For example, Hnaïen *et al.* [84] reported the occurrence of a single Nu_x peak near the nozzle exit, the position of which is constant with increasing OR , while Kumar [83] noted a secondary Nu_x peak for $OR > 2$ that moved downstream with higher OR values. In addition, the change in value of all streamwise Nu_x values appear to be considerably greater in the case of Hnaïen *et al.* [84], while Kumar [83] reported more subtle changes overall.

6.5.2.1. Time-Averaged Local Nusselt Number Profiles

The time-averaged Nu_x profiles for $OR = 1, 2, 3, 5$ and 7 are displayed in Figure 6.36 for $Re = 6000, 8000, 10000$ and 12000 . As previously discussed, all Nu_x profiles contain a single minimum point $Nu_{x_{min}}$ where the wall jet separates from the solid wall boundary, followed by a local maximum point $Nu_{x_{max}}$ where the flow reattaches, beyond which Nu_x continues to decline. In the case of $Re = 6000$, Figure 6.36(a), it is clear that an increase in OR moves the streamwise position of both $Nu_{x_{min}}$ and $Nu_{x_{max}}$ further downstream, which is consistent with the delayed separation of the wall jet and elongation of the separation region. In doing so, the value of $Nu_{x_{min}}$ remains relatively consistent regardless of OR , while $Nu_{x_{max}}$ clearly declines within increasing OR . For $OR \leq 2$ a ‘peaking’ effect is observed in the vicinity of $Nu_{x_{max}}$, which is likely due the higher velocity at which the flow re-impinges on the solid wall at this further upstream position. For $OR \geq 3$, this peaking is not observed, however, $Nu_{x_{max}}$ continues to decline as the flow reattaches further downstream with a lower flow velocity. In the far downstream region, a close alignment is observed between the Nu_x profiles, which implies that the effect of OR is negligible in this region, which is consistent with that previously noted in the analysis of the flow data.

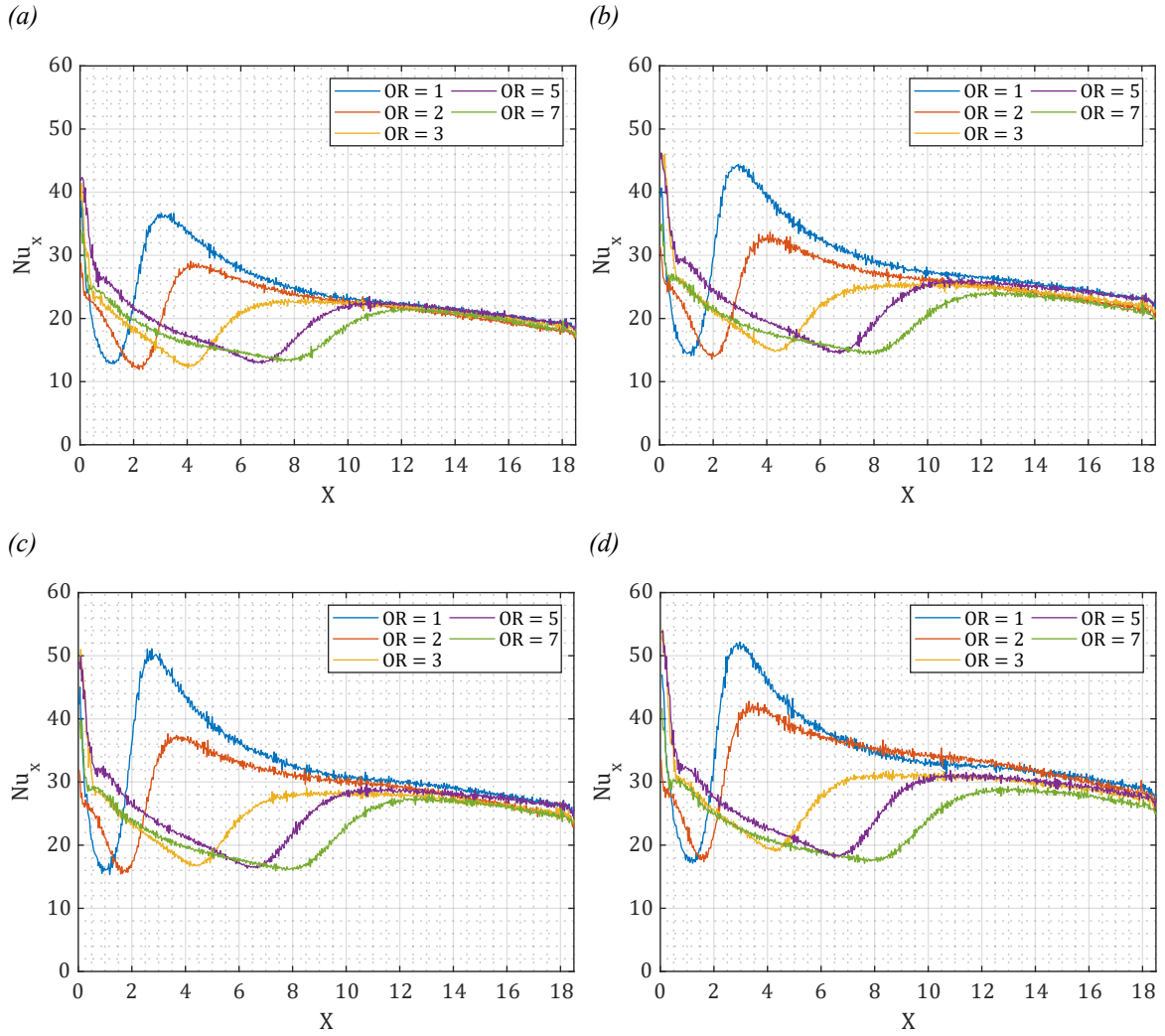


Figure 6.36: Time-averaged Nu_x profiles for a dual jet for different OR values and $V_r = 1$ for $Re =$ (a) 6000, (b) 8000, (c) 10000 and (d) 12000.

In the case of $Re = 6000$, the transition in the Nu_x profile from $OR = 1$ to $OR = 7$ is quite smooth, where the profiles appear quite evenly spaced in the streamwise dimension. Upon increasing Re , this spacing becomes distinctly uneven, whereby the positions of $Nu_{x_{min}}$ and $Nu_{x_{max}}$ for $OR = 1$ and $OR = 2$ remain biased towards the nozzle exit while that of the remaining higher OR values move slightly further downstream. In addition, the peaking at $Nu_{x_{max}}$ for $OR = 1$ and $OR = 2$ becomes significantly more exaggerated at higher Re , which further enhances the disparity in the Nu_x profiles compared to that of $OR \geq 3$. In the case of $OR = 1$, it was previously noted that the streamwise position of the merge point moves slightly upstream with increasing Re and, when combined with the delayed onset that can be induced in the wall jet separation due to the higher flow inertia, the position of the $Nu_{x_{min}}$ ultimately remains quite constant. Conversely, the flow data presented for $OR = 2$ indicated a much greater reduction in X_{mp} for increasing Re , meaning the upstream shift in $Nu_{x_{min}}$ is significantly more obvious. As the reattachment point thus experiences a higher flow velocity, not only as a result of the increased Re , but also by virtue of this new upstream position, the value of $Nu_{x_{max}}$ is enhanced significantly.

As the Nu_x profiles associated with $OR = 5$ and $OR = 7$ exhibit a similar behaviour to that of $OR = 3$ for changing Re , it can reasonably be assumed that the flow characteristics must also be consistent. Therefore, as

the flow data for $OR = 3$ presented minimal change in the length of the recirculation zone with respect to changing Re , in addition to an elongation of the separation region, it is assumed that similar effects occur in the cases of $OR = 5$ and $OR = 7$. In doing so, the position of $Nu_{x_{min}}$ and $Nu_{x_{max}}$ are moved slightly further downstream with increasing Re , which prevents the occurrence of ‘peaking’. Therefore, for a constant Re , the effect of altering OR for $OR \geq 3$ is the movement of both $Nu_{x_{min}}$ and $Nu_{x_{max}}$ further downstream in a consistent fashion, while maintaining the close alignment between the Nu_x profiles further downstream.

When compared to the published literature in Figure 6.37, the Nu_x profiles from the present investigation are distinctly different from those of previous numerical studies for a similar set of jet parameters. The data presented in Figure 6.37(a) pertains to data for $OR = 2$ obtained for similar Re values. While the trends obtained by Kumar [83] and Hnaien *et al.* [84] using the $k - \varepsilon$ and $k - \omega$ turbulence models, respectively, are quite similar to each other, the published numerical studies do not align with the present study and fail to simulate the local minimum in the Nu_x profile. Figure 6.37(b) hence demonstrates the secondary peak reported by Kumar [83] for $OR = 4$, the existence of which is also inconsistent with the present experimental data when compared to that of $OR = 3$ and $OR = 5$.

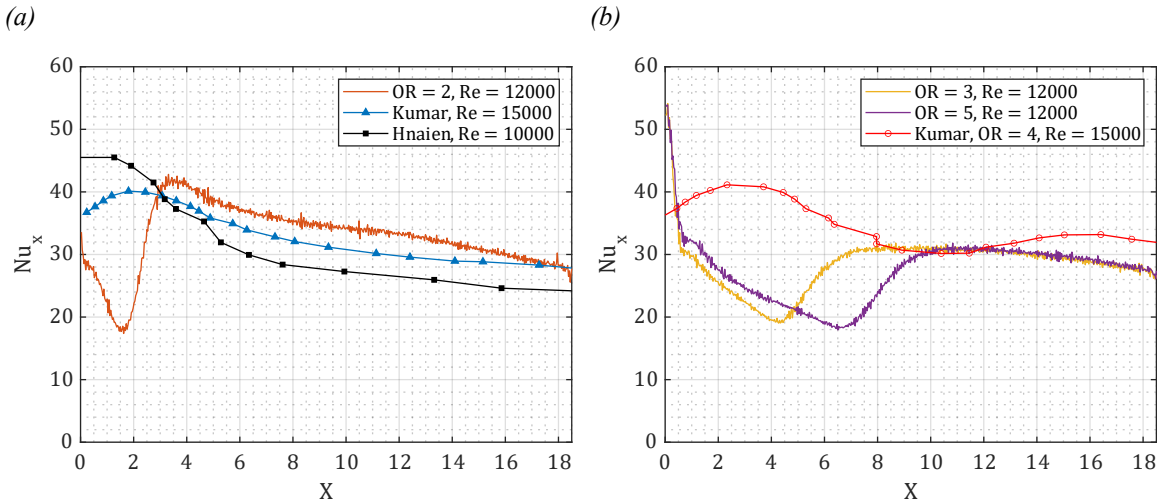


Figure 6.37: Comparison of dual jet Nu_x profile obtained for $Re = 12000$ and $V_r = 1$ with that of (a) Kumar [83] ($Re = 15000, OR = 2$) and Hnaien *et al.* [84] ($Re = 10000, OR = 2$) for $OR = 2$ and (b) Kumar [83] ($Re = 15000, OR = 4$) for $OR = 3$ and $OR = 5$.

6.5.2.2. Surface-Averaged Nusselt Number

The variation in the surface-averaged Nusselt number $\overline{Nu}_x$ with respect to OR is presented in Figure 6.38 for $Re = 6000, 8000, 10000$, and 12000 . For the data presented, it is clear that an increase in OR causes a general decline in $\overline{Nu}_x$, however the rate of decline is greater at lower OR and decreases with increasing OR . Using the MATLAB Curve Fitter tool, an exponential fit was identified as the most suitable for describing the resulting trend between OR and $\overline{Nu}_x$ based on the r^2 value which varied from 0.9519 to 0.9768, the corresponding equations for which are stated in Figure 6.38 for each Re . Based on the line of best fit, it is clear that the effect of OR is more obvious in the low OR range, where the rate of decline is much greater. This effect is more apparent for lower Re values, where the trendline appears to approach straight-line linear relationship with increasing Re .

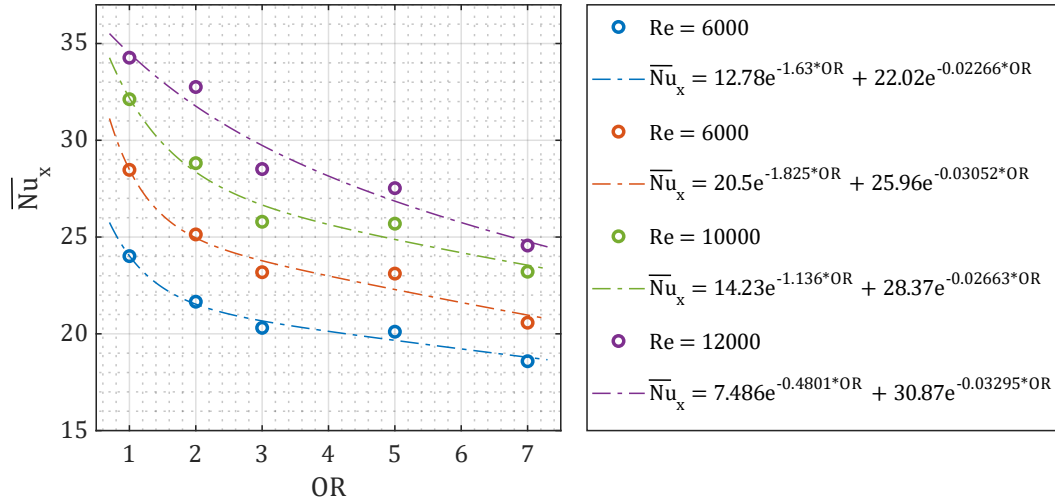


Figure 6.38: Relationship between the surface-averaged, time-averaged Nusselt number $\overline{Nu}_x$ and the dual jet offset ratio for $V_r = 1$.

The data presented in Figure 6.39 plots $\overline{Nu}_x/Re^{0.5}$ against OR as a means to obtain a singular correlation relating all three variables in the form $\overline{Nu}_x/Re^m = f(OR)$. The Reynolds number exponent, m , is first obtained through fitting a power law to the data previously presented in Figure 6.27, where $m \approx 0.5$ provides the best relative fit across all OR cases based on the least squares approach. Through the division of $\overline{Nu}_x$ by Re^m , the Reynolds number dependence is removed from the dataset and the effect of OR on $\overline{Nu}_x$ is hence isolated, as shown in Figure 6.39. The resulting trend indicates an almost exponential decline in $\overline{Nu}_x/Re^{0.5}$ with increasing OR , where fitting a power law to the data enables the derivation of the corresponding correlation. The relationship between $\overline{Nu}_x$, Re and OR for $V_r = 1$ is hence summarised by equation 6.2.

$$\overline{Nu}_x/Re^{0.5} = 0.4413 \times OR^{-0.0997} - 0.1281 \quad (6.2)$$

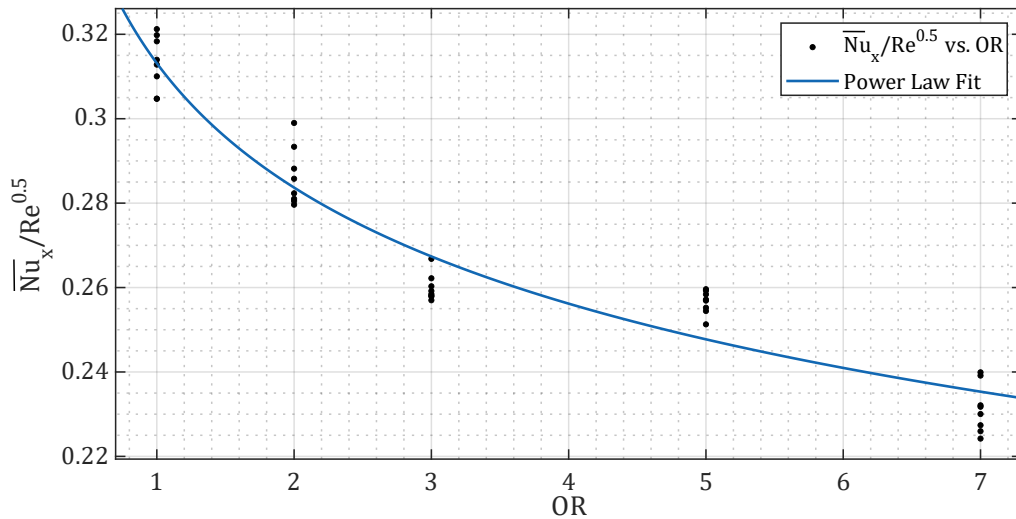


Figure 6.39: Power law fit describing the relationship between $\overline{Nu}_x/Re^{0.5}$ and OR for $V_r = 1$.

6.6. Effect of Velocity Ratio

The following sections discuss the effect of varying the velocity ratio on the flow characteristics of a dual jet and the subsequent impact on the heat transfer behaviour. Comparisons are made at each offset ratio, and, in all cases presented from this study, the total mass flowrate is maintained at a constant value. The effect of V_r has been examined to a much lesser extent in the published literature, with the majority of studies focussing specifically on the flow aspects [25, 92, 96]. Only a single numerical study exists in the published literature which analyses the influence of V_r on the Nu_x profile in the solid wall, *i.e.*, Vishnuvardhanarao and Das [28]. As such, the findings of Vishnuvardhanarao and Das [28] are yet to be corroborated, while all remaining V_r studies require validation through experimental means.

The analysis of the effect of V_r for the present study requires the introduction of the total Reynolds number term, Re_{tot} , which equates to the sum of the individual jet Reynolds numbers pertaining to the wall and offset jet, such that $Re_{tot} = Re_w + Re_o = (u_w + u_o)w/\nu$. In doing so, it can be assumed for a specified Re_{tot} value, the total mass flow rate of the dual jet is constant, regardless of the ratio of u_w to u_o . This aids in the analysis of the velocity ratio as test cases of the same constant mass flow rate can easily be identified. The subsequent sections therefore adopt this term for the analysis of the effect of V_r , where the impact on the overall flow structure is initially considered, followed by the induced heat transfer characteristics.

6.6.1. Flow Structure

The time-averaged flow fields obtained for a dual jet with $V_r = 0.5, 0.8, 1.5$ and 2 are presented in Figure 6.40, Figure 6.41, and Figure 6.42 for $OR = 1, 2$ and 3 , respectively, where two Re_{tot} cases are shown for each V_r and OR combination. In each case, the flow fields are represented by means of the time-averaged velocity $|U|$ contour plot, onto which the corresponding streamlines are overlayed. In doing so, a distinct change in the flow behaviour is observed as it transitions from that an offset jet dominated flow to a wall jet dominated flow, which appears consistent regardless of OR .

For $V_r < 1$, it is clear that in increasing u_o relative to u_w , the degree to which the offset jet is deflected reduces and the weaker wall jet is pulled more from the solid wall. The thickness of the separation region at the solid wall boundary is hence increased with decreasing V_r , to the extent that flow reversal can occur in this region at $V_r = 0.5$. Conversely, for $V_r > 1$, the inertia of the wall jet is much greater than that of the offset jet, meaning the offset jet experiences a much greater deflection in comparison and the separation of the wall jet from the solid boundary is reduced significantly. Inside the recirculation region, the lower vortex appears to grow in size with decreasing $V_r < 1$ in an asymmetric manner to occupy more space inside the recirculation region, which acts to suppress the upper vortex. Increasing V_r above 1 therefore has the opposite effect, whereby the upper vortex increases in size in a similar manner and suppresses the lower vortex. This effect is much clearer in the higher OR cases by virtue of the increased size of the recirculation region, *i.e.*, Figure 6.41 and Figure 6.42, while the presence of the second vortex at the extreme V_r values is much more difficult to detect in the case of $OR = 1$.

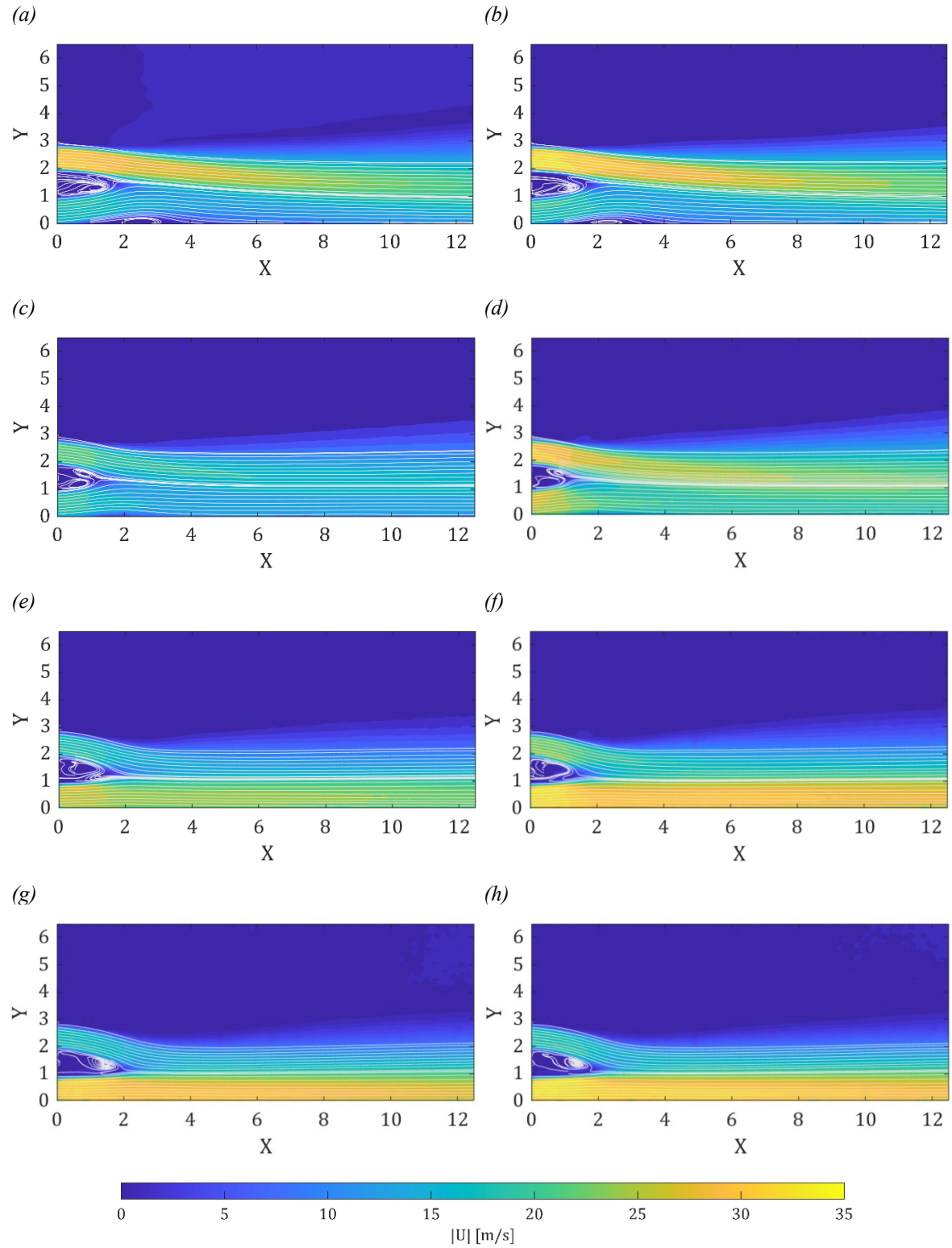


Figure 6.40: Effect of V_r on the dual jet flow structure for $OR = 1$, where 2 mass flow rates are displayed horizontally for each V_r case. (a) $V_r = 0.5$, $Re_{tot} = 16000$, (b) $V_r = 0.5$, $Re_{tot} = 18000$, (c) $V_r = 0.8$, $Re_{tot} = 13500$, (d) $V_r = 0.8$, $Re_{tot} = 20000$, (e) $V_r = 1.5$, $Re_{tot} = 15000$, (f) $V_r = 1.5$, $Re_{tot} = 20000$, (g) $V_r = 2$, $Re_{tot} = 16000$, (h) $V_r = 2$, $Re_{tot} = 18000$.

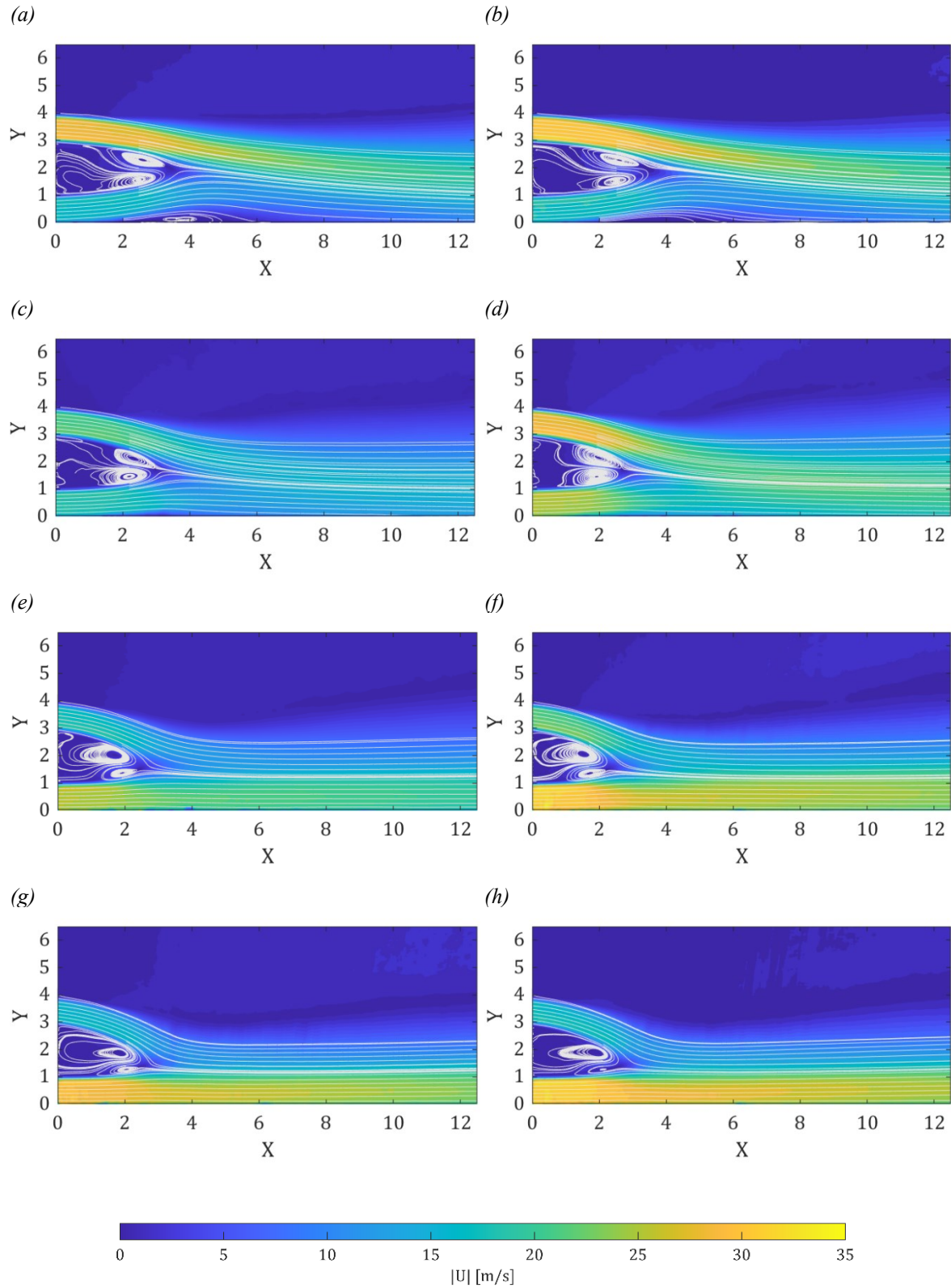


Figure 6.41: Effect of V_r on the dual jet flow structure for $OR = 2$, where 2 mass flow rates are displayed horizontally for each V_r case. (a) $V_r = 0.5$, $Re_{tot} = 16000$, (b) $V_r = 0.5$, $Re_{tot} = 18000$, (c) $V_r = 0.8$, $Re_{tot} = 13500$, (d) $V_r = 0.8$, $Re_{tot} = 20000$, (e) $V_r = 1.5$, $Re_{tot} = 15000$, (f) $V_r = 1.5$, $Re_{tot} = 20000$, (g) $V_r = 2$, $Re_{tot} = 16000$, (h) $V_r = 2$, $Re_{tot} = 18000$.

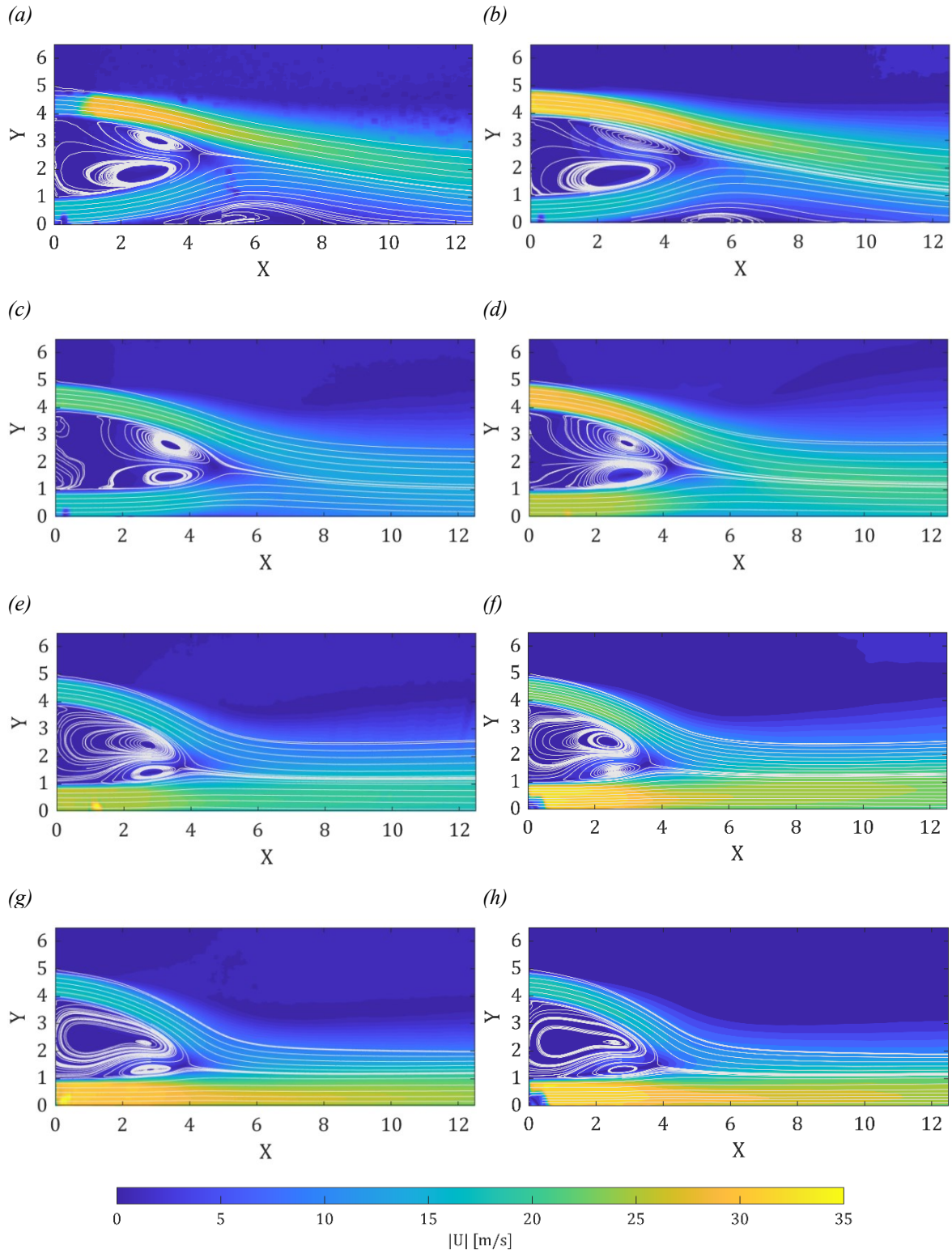


Figure 6.42: Effect of V_r on the dual jet flow structure for $OR = 3$, where 2 mass flow rates are displayed horizontally for each V_r case. (a) $V_r = 0.5$, $Re_{tot} = 16000$, (b) $V_r = 0.5$, $Re_{tot} = 18000$, (c) $V_r = 0.8$, $Re_{tot} = 13500$, (d) $V_r = 0.8$, $Re_{tot} = 20000$, (e) $V_r = 1.5$, $Re_{tot} = 15000$, (f) $V_r = 1.5$, $Re_{tot} = 20000$, (g) $V_r = 2$, $Re_{tot} = 16000$, (h) $V_r = 2$, $Re_{tot} = 18000$.

At the V_r extremities, *i.e.*, $V_r = 0.5$ and $V_r = 2$, the increment in Re_{tot} for a constant V_r in Figure 6.40, Figure 6.41, and Figure 6.42 is less than that presented for the mid-range V_r values due to the flow delivery limitations of the experimental setup, which hence limits the number of possible V_r and Re_{tot} combinations. Despite this, it is clear that for the V_r and Re_{tot} combinations shown that, aside from an increase in the velocity magnitude, the overall flow structure of a dual jet at a specified V_r does not change with increasing Re_{tot} . The observed changes in the flow behaviour with Re_{tot} are similar to that previously discussed in section 6.4 regarding the effect of Re , which, when compared to the induced flow behaviour due to varying V_r , are relatively minimal. As the effect of V_r appears reasonably consistent regardless of the value of Re_{tot} , the remaining analysis of V_r focusses primarily on that attained for a single Re_{tot} value of 16000, as this mid-range Re_{tot} values allows for a greater range of V_r combinations.

The vorticity contour plots relating to $V_r = 0.5$ and $V_r = 2$ for $Re_{tot} = 16000$ and $OR = 2$ are displayed in Figure 6.43, which represents the change in the vorticity field through interchanging the wall and offset jet velocities. Compared to that discussed previously for $V_r = 1$, the vorticity field changes considerably through adjusting V_r . In both cases, the positive vorticity along the outer region is consistent with the development of Layer 1 and the associated entrainment with the ambient. For $V_r = 0.5$, however, the remainder of the dual jet flow field is dominated by negative vorticity, as the inner shear layer of the offset jet, *i.e.*, Layer 2, and the new shear layer associated with the wall jet separation appear to combine and act to reattach the jet flow to the solid wall. Conversely, the vorticity inside the flow field for $V_r > 1$ is primarily positive, which is consistent with the rapid entrainment of the offset jet into the wall jet flow, leading to an elongation of Layer 3 and its eventual merger with Layer 1.

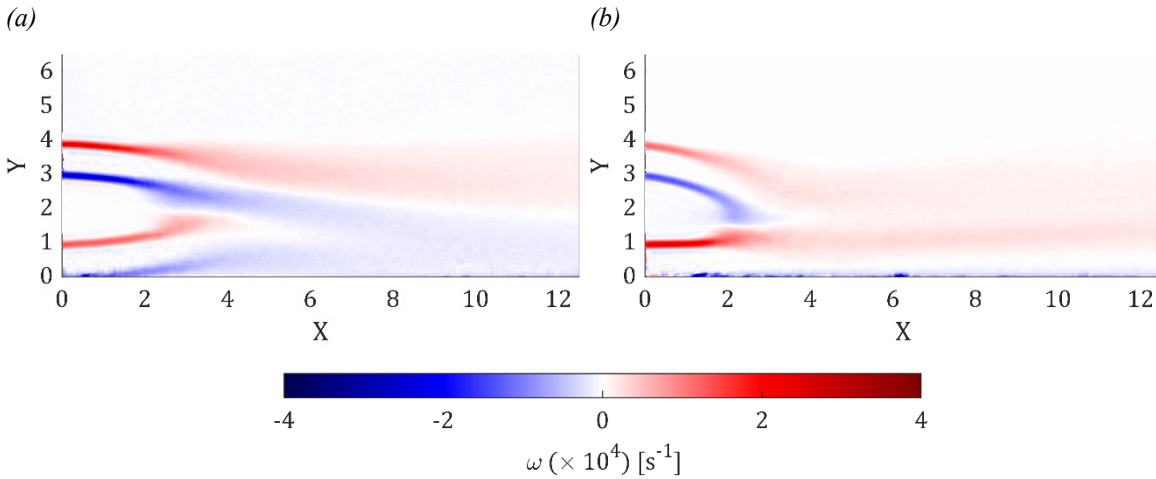


Figure 6.43: Effect of V_r on the vorticity contour plot for a dual jet with $Re_{tot} = 16000$ and $OR = 2$, where $V_r = (a) 0.5$ and $(b) 2$.

The effect of the velocity ratio on the positioning of *mp*, *cp*, *UVC* and *LVC* is presented in Figure 6.44 for $OR = 1, 2$ and 3 at $Re_{tot} = 16000$, the data for which is contained in Table B.2 in Appendix B. As noted previously regarding the overall flow structure, the observed trends are generally consistent for each OR value considered, which aids in the simplification of the analysis of V_r . Beginning with the merge point, it is interesting to note that in increasing the velocity ratio from 0.5 to 2, the streamwise position of *mp* generally decreases with increasing V_r towards a minimum value, following which X_{mp} appears to increase. This trend seems logical, as the size of the recirculation zone in the case of a single offset jet is distinctly greater than that

of a dual jet for a similar offset ratio, as previously demonstrated by Wang and Tan [19], which pushes the reattachment point further downstream compared to the merge point. As this occurs as a result of the lower pressure inside the recirculation zone in the case of a dual jet flow, it seems plausible that the minimum value of X_{mp} occurs in conjunction with the V_r value that induces the lowest pressure inside the recirculation zone. For $OR = 1$, the minimum streamwise position of the merge point exists at $V_r = 1$, however, the data suggests that this position moves downstream with increasing OR . For $OR = 3$, however, X_{mp} appears to initially stay somewhat consistent with increasing V_r for $V_r \leq 1$, beyond which X_{mp} exhibits a similar behaviour to that observed for the other OR values. Conversely, a consistent decreasing trend is observed in the vertical position of mp for all OR examined, whereby the merge point moves closer to the solid wall with increasing V_r , which was also reported by Mondal *et al.* [96].

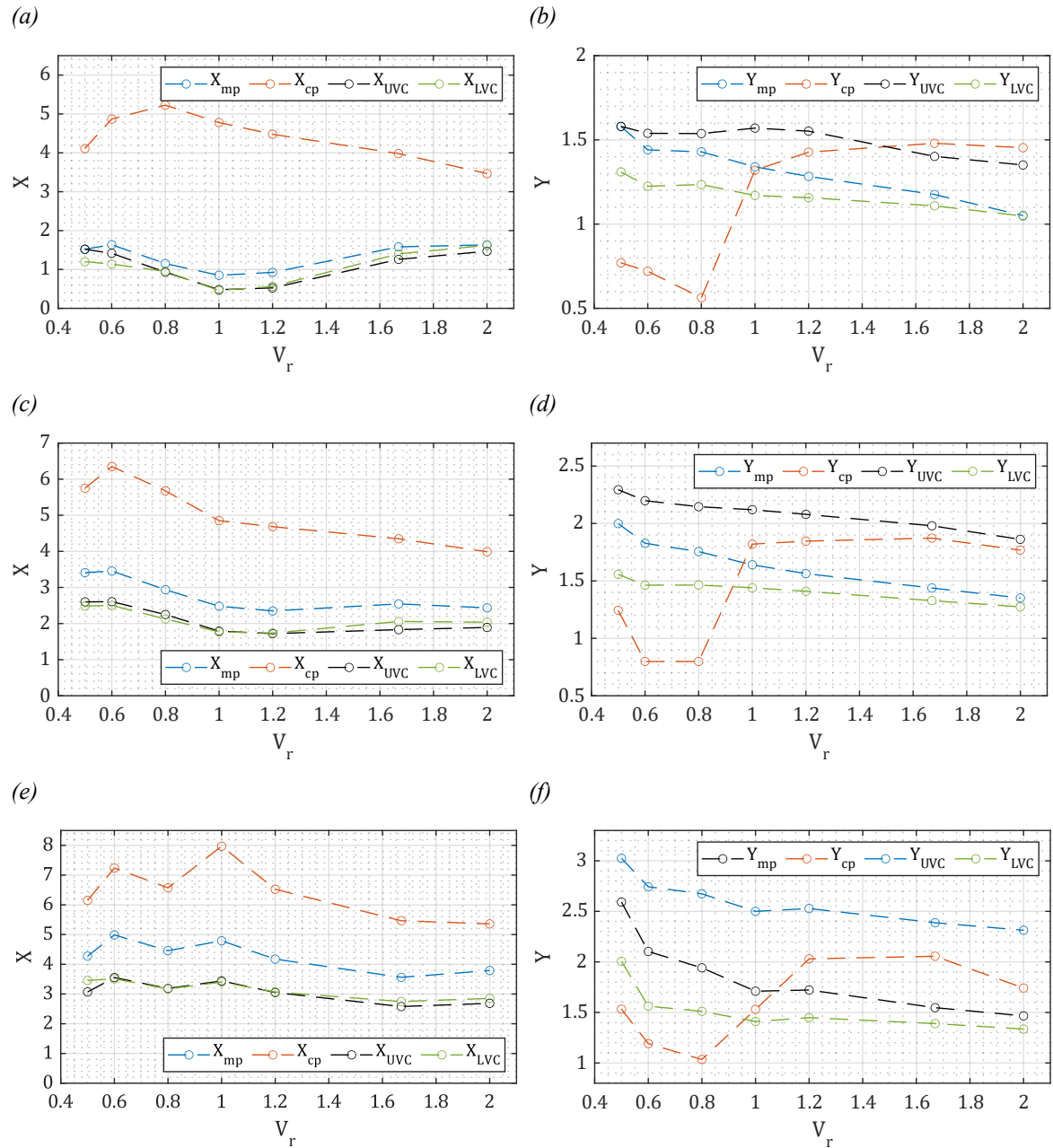


Figure 6.44: Effect of V_r on the X-location (left) and Y-location (right) of mp , cp , UVC and LVC , for a dual jet with various OR and $Re_{tot} = 16000$, where (a, b) $OR = 1$, (c, d) $OR = 2$ and (e, f) $OR = 3$.

The trends exhibited by the merge point are generally reflected in that of the upper and lower vortex centres, whereby a minimum value of X_{UVC} and X_{LVC} exist when V_r is varied in the range $0.5 \leq V_r \leq 2$, while Y_{UVC} and Y_{LVC} decrease in a consistent manner. Despite the relative change in size of the upper and lower vortex through varying V_r , it is interesting to note that both vortices still remain generally aligned in the same vertical plane, with only a slight deviation at the V_r extremities, with X_{UVC} further downstream for $V_r < 1$ and vice versa for $V_r > 1$. Regarding the vertical position of the vortex centres, while the rate of decline with increasing V_r is similar for both Y_{UVC} and Y_{LVC} , it is significantly less when compared to that of Y_{mp} . As a result, the vertical height of mp appears to approach that of UVC or LVC when V_r is decreased below or increased above 1, respectively. It is interesting to note, however, that the discussed trends do not quite align with that of Hnaïen *et al.* [25], who reported that X_{mp} , X_{UVC} and X_{LVC} all decreased with increasing V_r . While the decline in Y_{mp} and Y_{LVC} is consistent, Hnaïen *et al.* [25] found Y_{UVC} was independent of V_r .

With regard to the combined point, the streamwise position of cp initially increases with increasing V_r towards a maximum value, beyond which it starts to decrease. The maximum value for X_{cp} generally occurs when $V_r \leq 1$, therefore a more offset jet dominated flow, which seems reasonable due to the fact that for $V_r \approx 1$, the rate of decay associated with u_{m_o} is generally more than that of u_{m_w} due to the greater deflection of the offset jet. Therefore, as u_{m_o} is increased relative to u_{m_w} , thus lowering V_r , there must exist a value of V_r at which their respective rates of decay become equal, at which point X_{cp} is maximised. The trend associated with Y_{cp} also reflects this narrative, whereby the minimum value for X_{cp} generally occurs in tandem with a value for Y_{cp} that is closest to the solid wall, therefore indicating that it was the dissolution of u_{m_w} that dictated the position of the combined point. When V_r is further decreased, Y_{cp} starts to increase in a manner that is consistent with the increasing deflection of the wall jet, thereby indicating that the position of the combined point is still defined by the termination of u_{w_m} . When V_r is increased above the value that produces the minimum X_{cp} , a significant jump in Y_{cp} is noted initially, which indicates the position of the combined point is now defined by the termination of u_{w_o} , that eventually starts to decline as the deflection of the offset jet increases.

The streamwise decay of the non-dimensionalised maximum velocity with respect to changing V_r is presented in Figure 6.45 for $Re_{tot} = 16000$ and $OR = 1, 2$ and 3 . For each OR , the effect of both decreasing and increasing V_r are separately compared to that of $V_r = 1$, where the different trends observed for $0.5 \leq V_r \leq 1$ and $1 \leq V_r \leq 2$ remain relatively consistent across all OR values. For $V_r < 1$, the data suggests that decreasing V_r leads to a linearisation of the rate of decay of u_m with downstream distance, which induces a slower rate of decay near the nozzle exit but a much faster rate of decay further downstream. As the position at which the local minimum in $u_m/u_{m_{max}}$ occurs has previously been attributed to the realignment of the flow streamlines with the streamwise direction following the initial deflection, it seems reasonable that the rate of decay near the nozzle exit would be slowed as the offset jet is deflected to a lesser extent. In addition, as the jet remains in this deflected trajectory over a further downstream distance, this would induce a more constant rate of decay, which would gradually remove the occurrence of a local minimum maintain a faster rate of decay further downstream. For $OR = 3$, decreasing V_r appears to increase the rate of decay at all downstream positions, including the region close to the jet exit, which may be induced as a result of the much larger separation region created at the solid wall boundary and subsequent loss of momentum due to flow reversal effects.

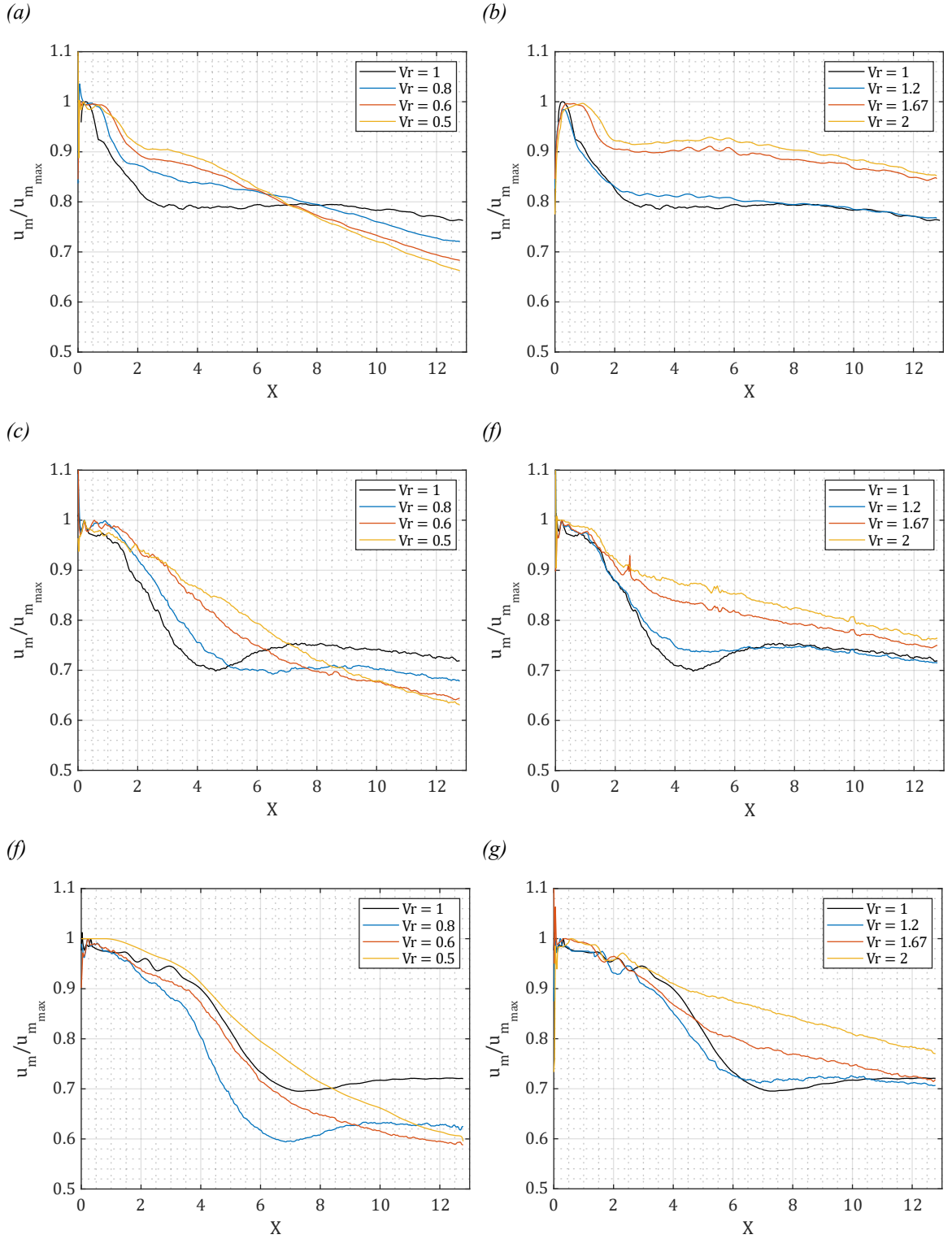


Figure 6.45: Effect of decreasing (left) and increasing (right) V_r with respect to $V_r = 1$ on the decay of the maximum streamwise velocity, normalised with respect to the overall maximum velocity, for a dual jet with $Re_{tot} = 16000$ and $OR = (a, b) 1$, $(c, d) 2$ and $(e, f) 3$.

A similar linearisation of the rate of decay of u_m is observed for increasing V_r above 1 for $OR = 2$ and $OR = 3$, however, this acts to reduce the rate of decay across the entire downstream domain in this instance, which is similar to that noted by Li *et al.* [92]. As V_r is increased, it is clear that the prominence of the local minimum point gradually deteriorates, which appears to induce a constant rate of decay along the streamwise direction

by $V_r = 2$. This suggests a reduced deflection in the wall jet with increasing V_r , however, the presence of the offset jet still slows the overall rate of decay when compared to a single wall jet. For $OR = 1$, increasing V_r does not appear to affect the overall shape of the $u_m/u_{m_{max}}$ profile, but instead shifts the profile upwards to suggest a slower rate of decay but an otherwise consistent trend. The local minimum value does appear to somewhat flatten and move upstream with increasing V_r , beyond which the behaviour is comparable to a single wall jet. From the data presented, it appears that for $OR = 1$, the effect of increasing V_r enables the downstream decay of u_m to resemble that of a SWJ much sooner, where the presence of the offset jet causes only a sharp decrease in u_m close to the nozzle exit, following which the trend recovers rapidly. For greater OR values, as mp exists further downstream, the effect of the offset jet is apparent over a much longer downstream distance.

The velocity profiles at $X = 8, 10$, and 12 are shown in Figure 6.46 for $V_r = 0.5$ and $V_r = 2$ for both $OR = 1$ and $OR = 3$, where Re_{tot} is maintained at as constant value of 16000 . In each case, the profiles are compared with that previously presented for a dual jet of the same OR with $V_r = 1$ at $X = 12$, where $Re = 12000$. For $V_r = 0.5$, the velocity profile changes considerably with X for both OR shown. These profiles differ greatly from the $V_r = 1$ test case, which indicates that the flow has not yet reached self-similarity inside the visible flow domain. Conversely, the velocity profiles presented for $V_r = 2$ do not appear to change with X for both OR shown, which implies that the flow has in fact reached self-similarity. When compared to $V_r = 1$, it is interesting to note that the profile obtained furthest downstream for $OR = 3$, *i.e.*, $X = 12$, is much more closely aligned with its respective $V_r = 1$ test case compared to $OR = 1$. The velocity profile for $OR = 1$ at $X = 12$ instead resembles that of a single wall jet, as previously mentioned in relation to the decay of u_m . This is further highlighted through the comparison with the single wall jet data of Kim *et al.* [79]. This thus indicates that for $OR = 1$, the effect of the offset jet is restricted to a region close to the nozzle exit for $V_r > 1$, and little to no effects are observed further downstream. As OR is increased, the effect of the offset jet extends further downstream as a result of the lengthened recirculation zone. Despite this, it can reasonably be concluded that increasing V_r above 1 increases the rate of flow development with downstream distance, enabling the dual jet to reach self-similarity much sooner, while reducing V_r below 1 acts to slow the streamwise jet development.

Finally, the effect of the V_r on the time-resolved flow behaviour of a dual jet is examined in Figure 6.47, where the PSD of the time signal associated with transverse velocity v is displayed for a dual jet with $Re_{tot} = 16000$ and varying V_r for $OR = 1, 2$, and 3 in Figure 6.47(a), Figure 6.47(b) and Figure 6.47(c), respectively. A clear dominant frequency (f_s) is evident across the majority of cases. However, a dominant frequency could not be identified in many of the PSD associated with the extreme V_r cases, which indicates that the periodic shedding of vortices does not occur at these V_r values. For $OR = 1$, the effect of decreasing V_r below 1 appears to increase f_s , however, a decrease is noted in the case of $V_r = 0.5$. Conversely, increasing V_r above 1 causes a sharp rise in f_s until $V_r = 1.2$, beyond which a dominant frequency is no longer present. This finding seems reasonable considering the previous analysis of the time-averaged flow revealed a closer similarity to that of a single wall jet in this V_r range as opposed to a traditional dual jet, meaning the periodicity of the flow is likely dampened as the SWJ flow behaviour begins to dominate.

This does not appear to be the case or $OR = 2$, where increasing V_r above 1 generally increases f_s , and a dominant frequency is still present by $V_r = 2$. In addition to this, decreasing V_r below 1 causes a decrease in f_s and a dominant frequency is no longer present by $V_r = 0.5$. A similar trend is noted for $OR = 3$, however,

the range within which a detectable shedding frequency exists is reduced to $0.8 \leq V_r \leq 1.67$. At the V_r extremities, it is clear that the dominant vorticity within the dual jet flow field prevents a vortex of opposing sign to be shed from the recirculation region, which removes the periodicity of the flow field. The range of V_r values within which periodicity is apparent therefore appears to narrow with increasing OR , as the vorticity generated by the weaker jet reduces compared to the more dominant jet flow. It should be noted, however, that the flow regime inside the recirculation zone outside the periodic range of V_r values is not necessarily steady, as published studies such as Mondal *et al.* [96] have suggested. Instead, vortices which complement the sign of the dominant vorticity inside the flow field are still shed and convected downstream, meaning the respective sizes of the two vortices inside the recirculation zone fluctuate considerably on a continuous basis.

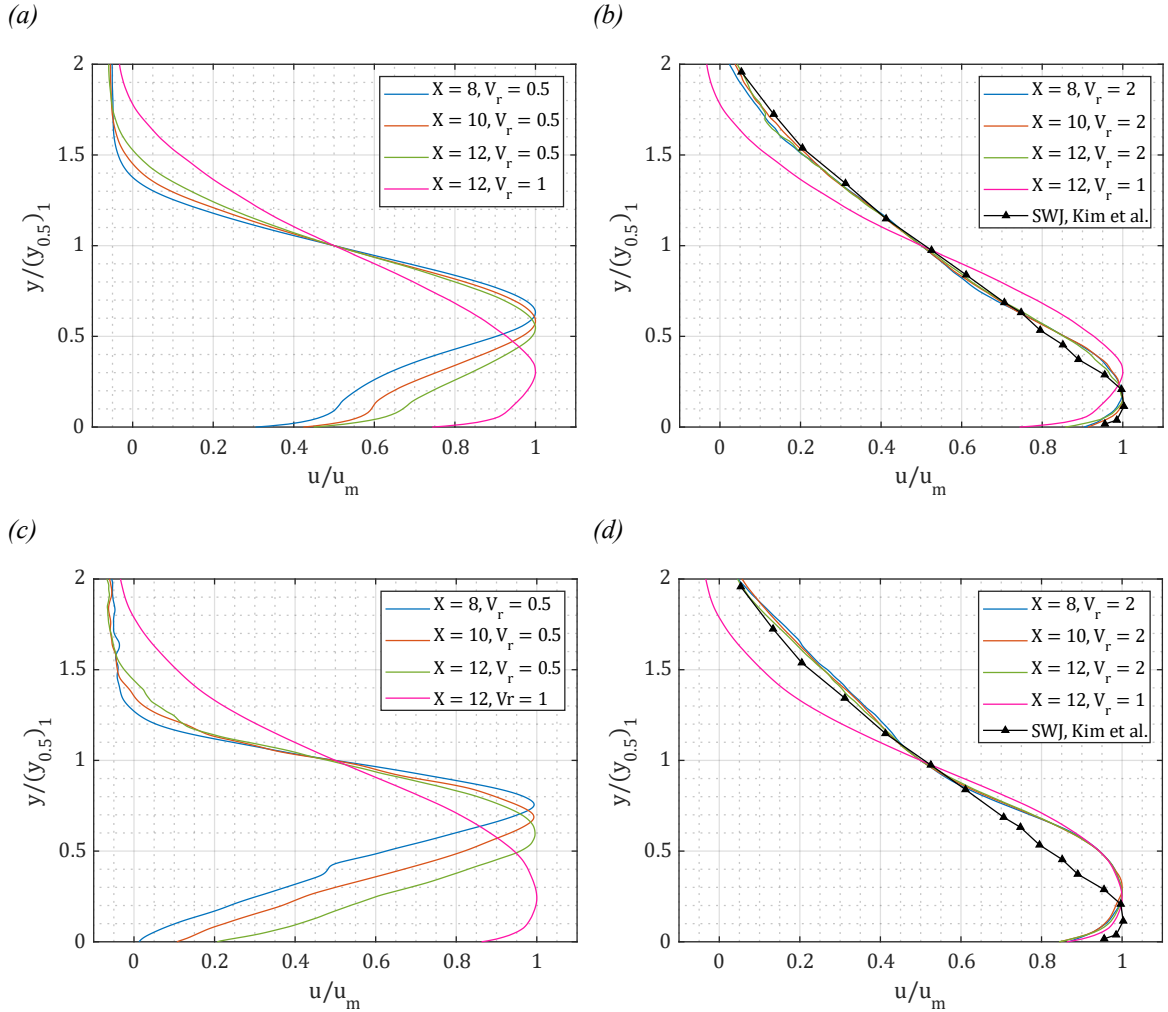


Figure 6.46: Comparison of the normalised velocity profiles for $V_r = 0.5$ (left) and $V_r = 2$ (right) at $X = 8, 10$, and 12 for a dual jet with $Re_{tot} = 16000$ to that of dual jet with $Re = 12000$ and $V_r = 1$ for the same OR , where (a, b) $OR = 1$ (c, d) $OR = 3$. The $V_r = 2$ cases are also compared to Kim *et al.* [79] for a SWJ ($X = 10, Re = 39000$).

The Strouhal numbers corresponding to the relevant shedding frequencies are presented in Figure 6.47(d) for each OR , from which the induced trends can easily be identified. As the velocities associated with each jet are generally unequal for each V_r value, the Strouhal number is calculated based on the average inlet jet velocity, which is constant for a given Re_{tot} . For $OR = 1$, it is clear that following a small increase, St generally decreases with increasing V_r until $V_r = 1$, beyond which St increases until the limiting V_r value is reached. This trend does not agree with that proposed by Mondal *et al.* [96], who reported a continuously increasing St

value with increasing V_r inside the range $0.78 \leq V_r \leq 1.34$ for $Re = 10000$, outside of which a stable flow regime was observed. It should be noted, however, Mondal *et al.* [96] varied V_r through incrementing u_w and maintaining u_o at a constant reference value, therefore Re_{tot} was altered consistently throughout. As it has been previously discussed that increasing Re can increase the shedding frequency, this may account for the difference in the trends observed. The trend does, however, agree with that observed for $OR = 2$ and $OR = 3$, where St continuously increases with increasing V_r inside their respective periodic V_r ranges.

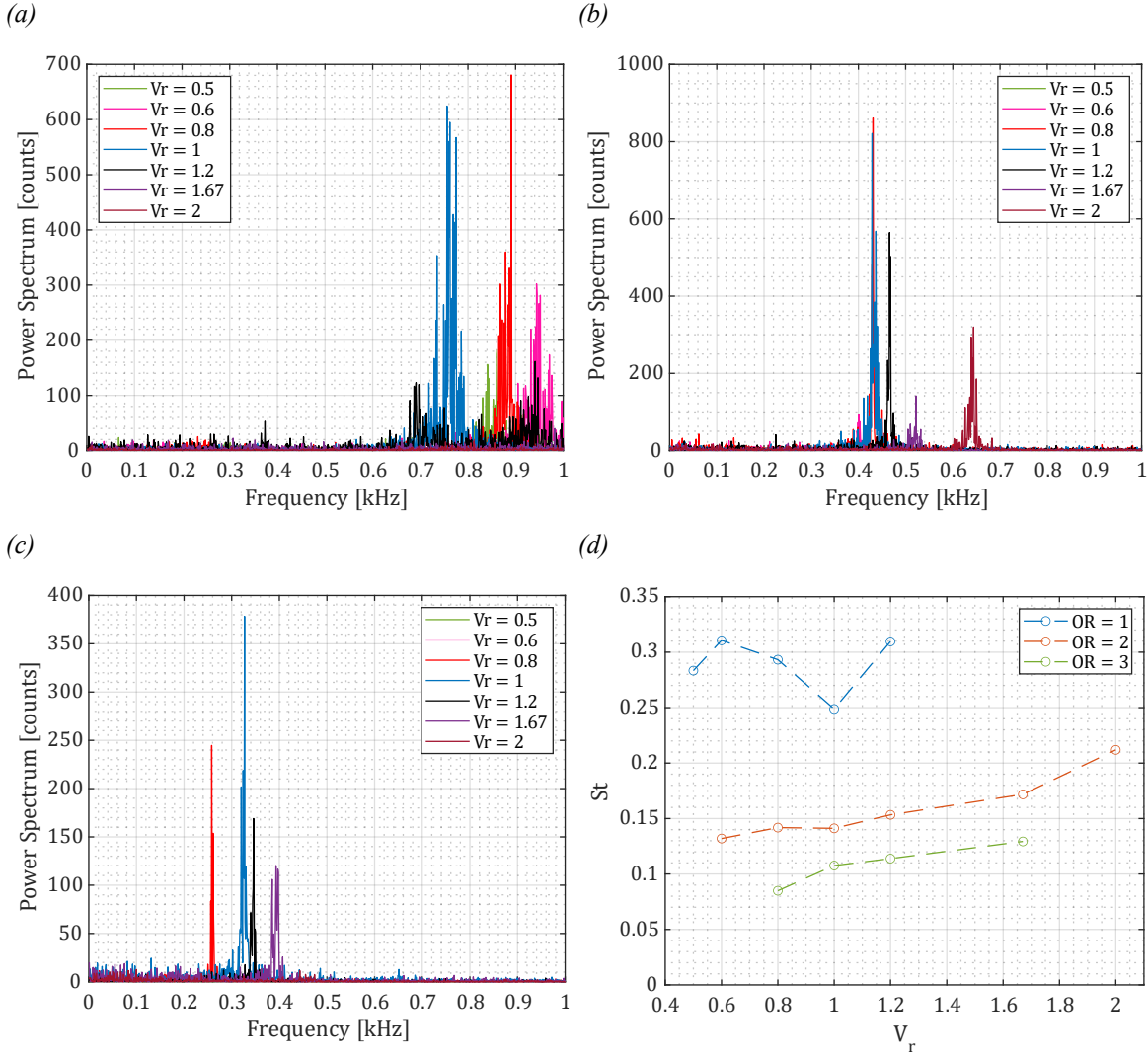


Figure 6.47: Effect of V_r on the power spectral density (PSD) associated with the v time signal for a dual jet with $Re_{tot} = 16000$ for $OR = (a) 1, (b) 2$ and $(c) 3$, and (d) the corresponding Strouhal number St trend.

6.6.2. Heat Transfer Characteristics

The influence of the velocity ratio on the heat transfer behaviour from the solid boundary is discussed in this section, where the analysis is primarily focused on the effect of varying V_r for a constant mass flow rate or Re_{tot} . For completeness, the effect of varying the total mass flow rate for a constant V_r is first discussed to ensure that the behaviour is relatively consistent for a given V_r value. This enables the simplification of the discussion regarding the effect of V_r , where the data is discussed for a single Re_{tot} test case, *i.e.*, $Re_{tot} = 16000$, and the trends are hence assumed to be applicable for other Re_{tot} values.

6.6.2.1. Effect of Mass Flow Rate

To investigate the effect of the mass flow rate on the heat transfer characteristics for a constant V_r value, the time-averaged Nu_x profiles for varying Re_{tot} values are presented in Figure 6.48 and Figure 6.49, which pertain to $V_r = 0.8$ and $V_r = 1.5$, respectively, *i.e.*, a sample case for $V_r < 1$ and $V_r > 1$. In each case, data is presented for $OR = 1, 3, 5$ and 7 , where the observed trend with respect to Re_{tot} appears to remain consistent regardless of OR . For $V_r = 0.8$, *i.e.*, Figure 6.48, the effect of increasing Re_{tot} is a proportional enhancement of Nu_x at each downstream location, which shifts the overall profile upwards along the y -axis without significantly affecting the overall shape. This is consistent with that previously discussed regarding the overall flow structure, where, aside from the increased flow momentum, increasing Re_{tot} does not affect the overall flow structure for a given V_r .

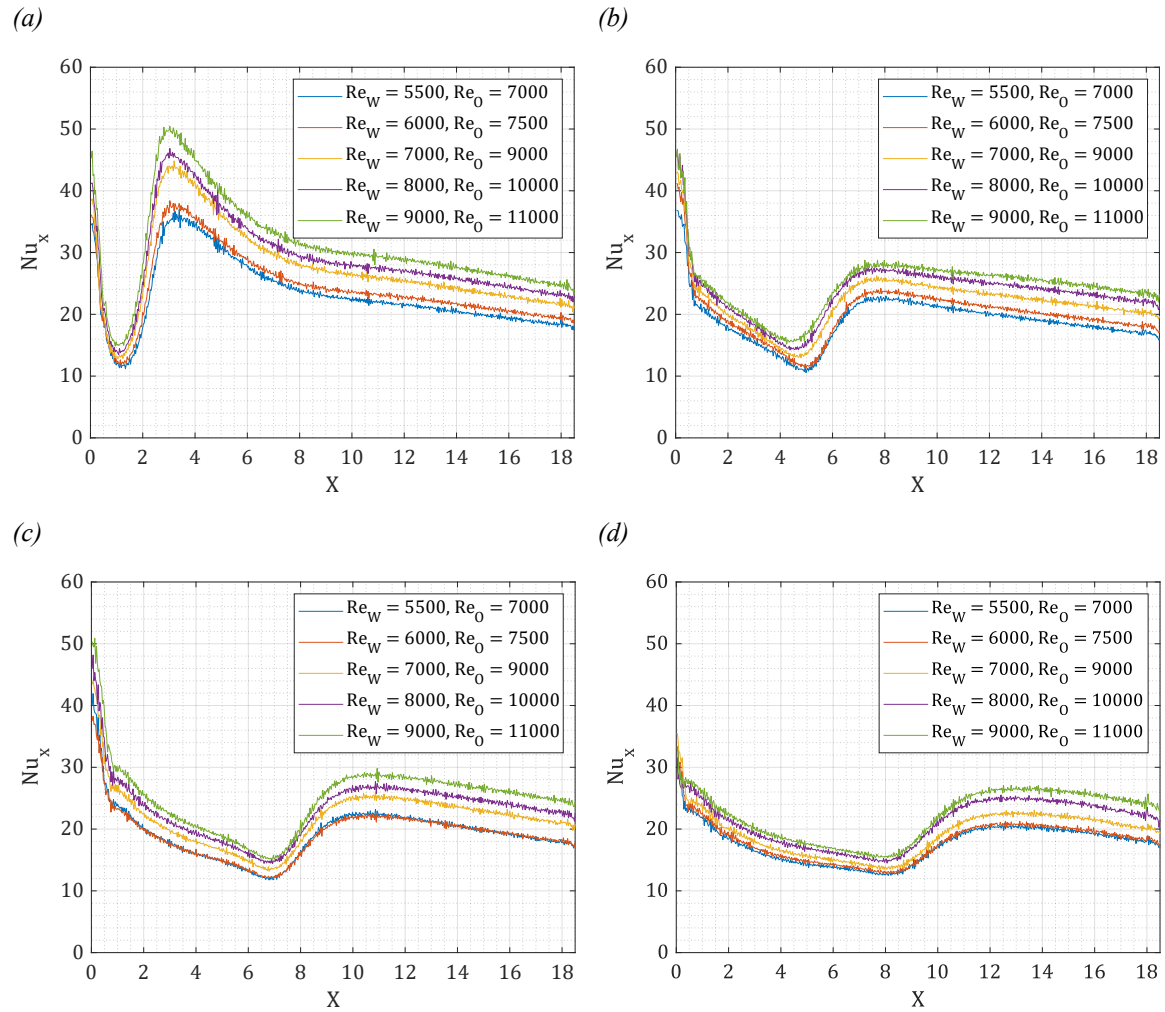


Figure 6.48: Effect of changing total mass flow rate $\dot{m}$, and hence the total Reynolds number Re_{tot} , for a constant velocity ratio of $V_r = 0.8$, where $OR = (a) 1, (b) 3, (c) 5$ and $(d) 7$.

There is a slight discrepancy, however, regarding the streamwise position of $Nu_{x_{min}}$ for $OR = 1$ and $OR = 3$, which appears to move upstream with increasing Re_{tot} , while the location of $Nu_{x_{max}}$ stays constant. For $OR = 1$, a similar reasoning can be applied to that previously discussed in relation to the effect of Re in section 6.4, whereby increasing Re decreases X_{mp} which causes $Nu_{x_{min}}$ to move further upstream. This was not originally noted in section 6.4 regarding $OR = 3$, however, for $V_r = 0.8$, a consistent decreasing trend is observed, where $X_{mp} = 4.72, 4.46$, and 4.19 for $Re_{tot} = 13500, 16000$ and 20000 , respectively. This suggests that a lower

pressure is induced in the recirculation region for increasing Re_{tot} when $V_r < 1$, which can increase the deflection of the wall jet which subsequently moves mp and $Nu_{x_{min}}$ upstream. For the greater OR values, however, this effect is not observed, meaning the offset jet is located far enough from the wall jet that the pressure inside the recirculation zone cannot increase with increasing Re_{tot} , meaning the extent of the deflection of either jet can remain relatively unaffected.

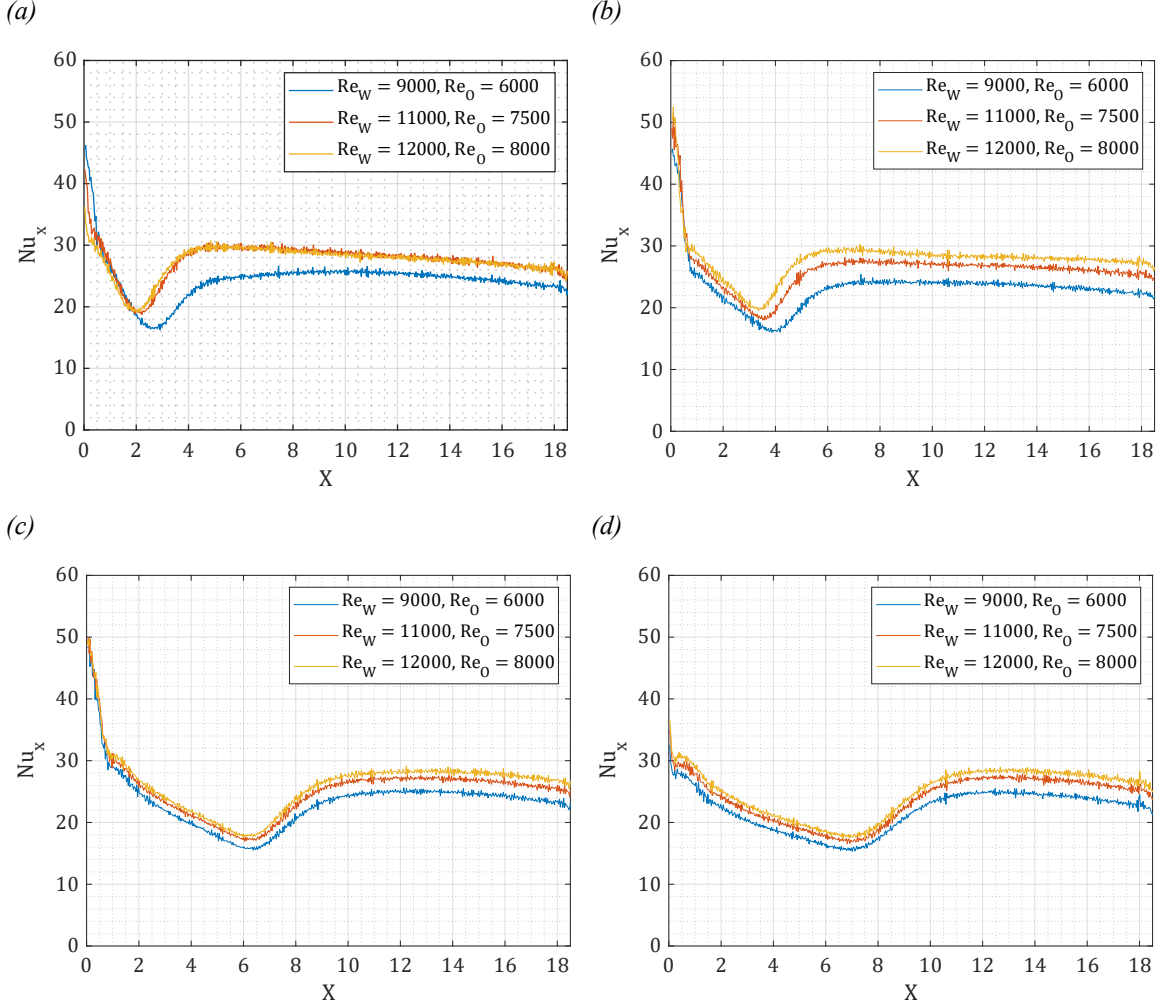


Figure 6.49: Effect of changing total mass flow rate $\dot{m}$, and hence the total Reynolds number Re_{tot} , for a constant velocity ratio of $V_r = 1.5$, where $OR =$ (a) 1, (b) 3, (c) 5 and (d) 7.

In a similar manner, the effect of increasing Re_{tot} for $V_r = 2$ is an increase in Nu_x at each streamwise location while preserving the overall trend, as shown in Figure 6.49. Similar to $V_r = 0.8$, the position of $Nu_{x_{min}}$ appears to move upstream with increasing Re_{tot} for $OR = 1$ and $OR = 3$, which again suggests that the proximity of the jets with respect to each other enables the wall jet to increase the extent of the offset jet deflection with increasing Re_{tot} , which thus moves the merge point upstream, along with the streamwise position of $Nu_{x_{min}}$. In this case, however, it appears that $Nu_{x_{max}}$ also moves upstream with increasing Re_{tot} , which can be attributed to the higher velocity of the wall jet enabling it to reattach to the solid wall much sooner, where the entrainment from the offset jet is too weak to sustain the separation region. The movement of $Nu_{x_{min}}$ and $Nu_{x_{max}}$ with changing Re_{tot} is again not observed at higher OR values, which implies the separation of the jets is sufficiently great as to ensure the deflection of neither jet is affected by increasing mass flow rate, even when the relative strength of each jet is varied. It is interesting to note, however, that while Vishnuvardhanarao

and Das [28] reported a similar enhancement of Nu_x at each downstream location by increasing Re for a constant V_r , the locations of $Nu_{x_{min}}$ and $Nu_{x_{max}}$ were generally constant, despite their use of $OR = 1$. While an instance of $Nu_{x_{min}}$ and $Nu_{x_{max}}$ was not reported for every V_r case, the resulting trend in the cases where they were present is more comparable to that obtained by the current investigation for $OR \geq 5$.

Despite the differences discussed, it is clear that the overall flow behaviour remains very consistent with changing Re_{tot} , which thus induces a characteristic Nu_x profile in the solid wall boundary for a given V_r value. It is therefore expected that the trends observed regarding the change in the Nu_x profile with varying V_r should be relatively consistent across all mass flow rates, with only minor deviations with altering Re_{tot} . This allows for the simplification of the remaining analysis for the effect of V_r to a single mass flow rate, *i.e.*, $Re_{tot} = 16000$, where a mid-range Re_{tot} value is chosen to maximise the available V_r combinations within the limits of the experimental setup.

6.6.2.2. Effect of Velocity Ratio

The effect of varying V_r for a constant Re_{tot} value of 16000 is analysed in two stages; first, the effect of reducing V_r below 1, *i.e.*, the heat transfer characteristics of an offset jet dominated flow, and, second, the effect of increasing V_r above 1, *i.e.*, a wall jet dominated flow. The effect of reducing V_r below 1 for $Re_{tot} = 16000$ is examined in Figure 6.50 for $OR = 1, 3, 5$ and 7 , which reveals a different trend depending on the offset ratio. Beginning with $OR = 1$, the time-averaged Nu_x profiles shown in Figure 6.50(a) indicate that both $Nu_{x_{min}}$ and $Nu_{x_{max}}$ move downstream with decreasing V_r , while also decreasing in value. This can be attributed to the growth of the separation region at the solid wall with decreasing V_r , where the elongation of the separation region means the point at which the thickness of the separation region is greatest occurs further downstream, *i.e.*, the location of $Nu_{x_{min}}$. As the flow subsequently reattaches to the solid wall further downstream, at which point the mean jet velocity is lower, the value of $Nu_{x_{max}}$ that occurs at this point is hence reduced. Further downstream, the effect of the increased rate decay of maximum velocity associated with lower V_r is clear, where the decline of Nu_x in this region appears much sharper with decreasing V_r , and a closer alignment between the profiles is observed in the far downstream region, where the respective mean jet velocities are comparable.

For $OR = 3$, *i.e.*, Figure 6.50(b), the resulting trend differs quite considerably, whereby a decrease in V_r moves $Nu_{x_{min}}$ and $Nu_{x_{max}}$ upstream. As the rate of decline in Nu_x appears to increase with lower V_r values near the jet exit, it implies that the extent of the wall rate deflection is increased, meaning the thickness of the separation region is much greater. As a result, the point at which the separation region reaches its maximum thickness occurs sooner, hence $Nu_{x_{min}}$ occurs further upstream with a slightly attenuated value. Similar to $OR = 1$, as the reattachment point is now located further upstream, $Nu_{x_{max}}$ is moved upstream with a consequential increase in value. A similar trend is observed for $OR = 5$ and $OR = 7$, *i.e.*, Figure 6.50(c) and Figure 6.50(d), respectively. An unusual effect becomes apparent for lower V_r at the higher OR values, by which a secondary peak in the Nu_x profile appears further downstream. This can be attributed to the development of the local velocity profile for higher OR flows, where the location of the local maximum u_m is initially located far from the solid wall but gradually approaches it with downstream distance. There exists a point, therefore, that the proximity of u_m from the solid wall boundary is sufficiently low as to induce an enhancement in the local Nu_x

value, which, in the case of higher OR flows, occurs downstream of the reattachment point. This hence appears as an additional peak in the Nu_x profile and occurs further downstream with increasing OR .

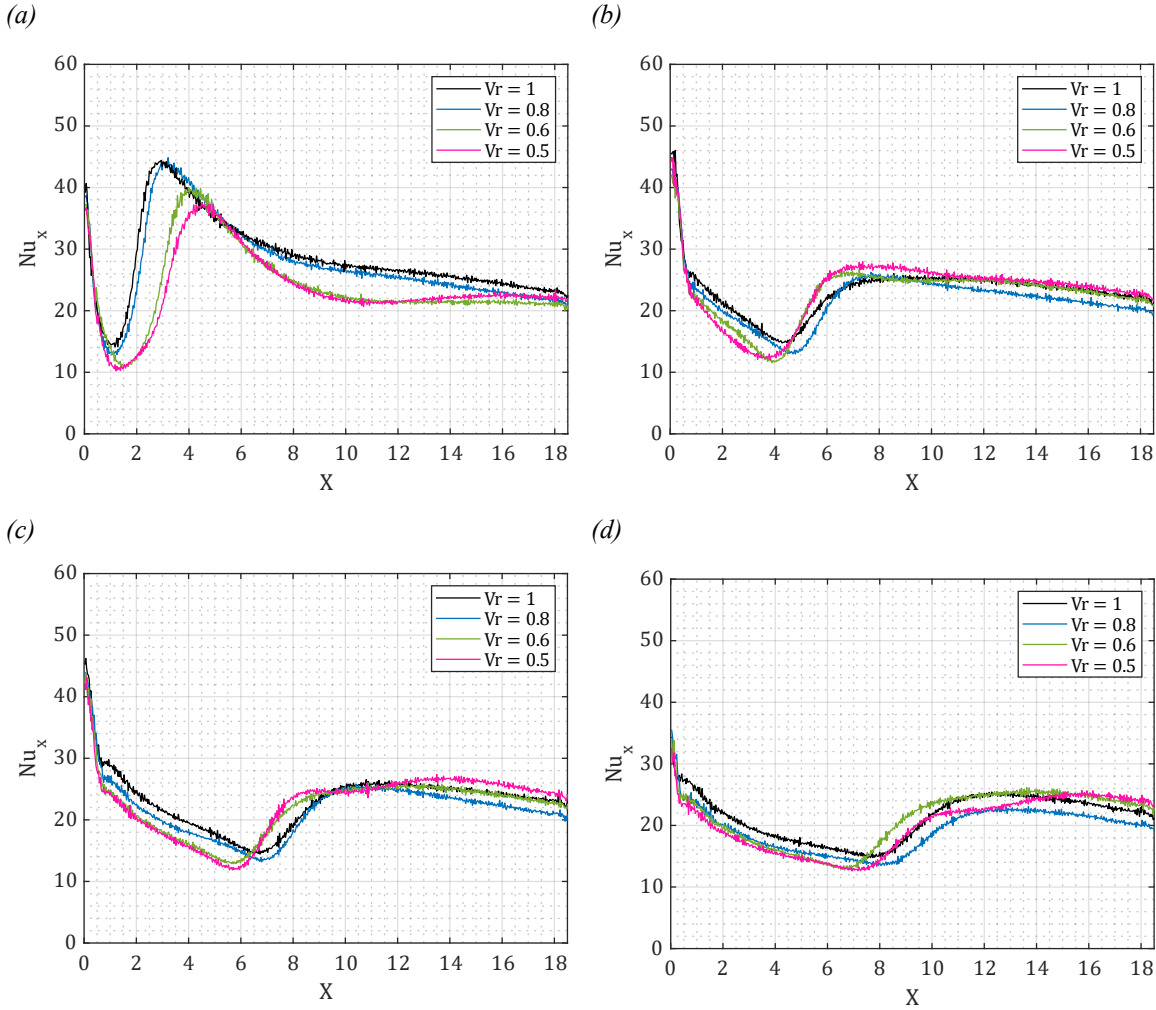


Figure 6.50: Effect of decreasing V_r for a dual jet of constant mass flow rate, i.e., $Re_{tot} = 16000$, and varying offset ratio, where $OR = (a) 1, (b) 3, (c) 5$ and $(d) 7$.

The effect of increasing V_r above 1 for $Re_{tot} = 16000$ is presented in Figure 6.51 for $OR = 1, 3, 5$ and 7 , where a similar difference between the apparent trends is observed depending on OR . For $OR = 1$, an increase in V_r increases the value of $Nu_{x_{min}}$, while moving it further downstream. More notably, the characteristic ‘peaking’ that occurs at $Nu_{x_{max}}$ for lower OR values becomes increasingly suppressed with increasing V_r to the extent that it no longer occurs. This behaviour is as a direct result of the reduced deflection of the wall jet, whereby the separation of the wall jet from the solid wall boundary becomes significantly less with increasing V_r , therefore minimising the prominence of both $Nu_{x_{min}}$ and $Nu_{x_{max}}$. By $V_r = 2$, the Nu_x profile starts to resemble that of a single wall jet, which is consistent with that previously observed regarding the flow behaviour, where the Nu_x profile no longer exhibits characteristics of flow separation but instead a rise in Nu_x due to the laminar to turbulence transition in the wall boundary layer.

For the remaining OR , increasing V_r leads to an increase in the value of $Nu_{x_{min}}$ while generally moving it further upstream, which can again be attributed to the reduced separation of the wall jet from the boundary. For $OR = 5$ and $OR = 7$, the Nu_x profile starts to resemble that of a single wall jet, which indicates that for

$V_r \geq 2$, the presence of the offset jet has very little impact of the induced Nu_x profile. For $OR = 3$, however, the closer proximity between the jets means the Nu_x profile at $V_r = 2$ is still affected by the offset jet flow, and a distinct flattening effect is observed in the far downstream region. The reasoning for this can likely be derived from the streamwise rate of decay of u_m , where, as previously discussed, increasing V_r leads to a slower rate of decay in the far downstream region.

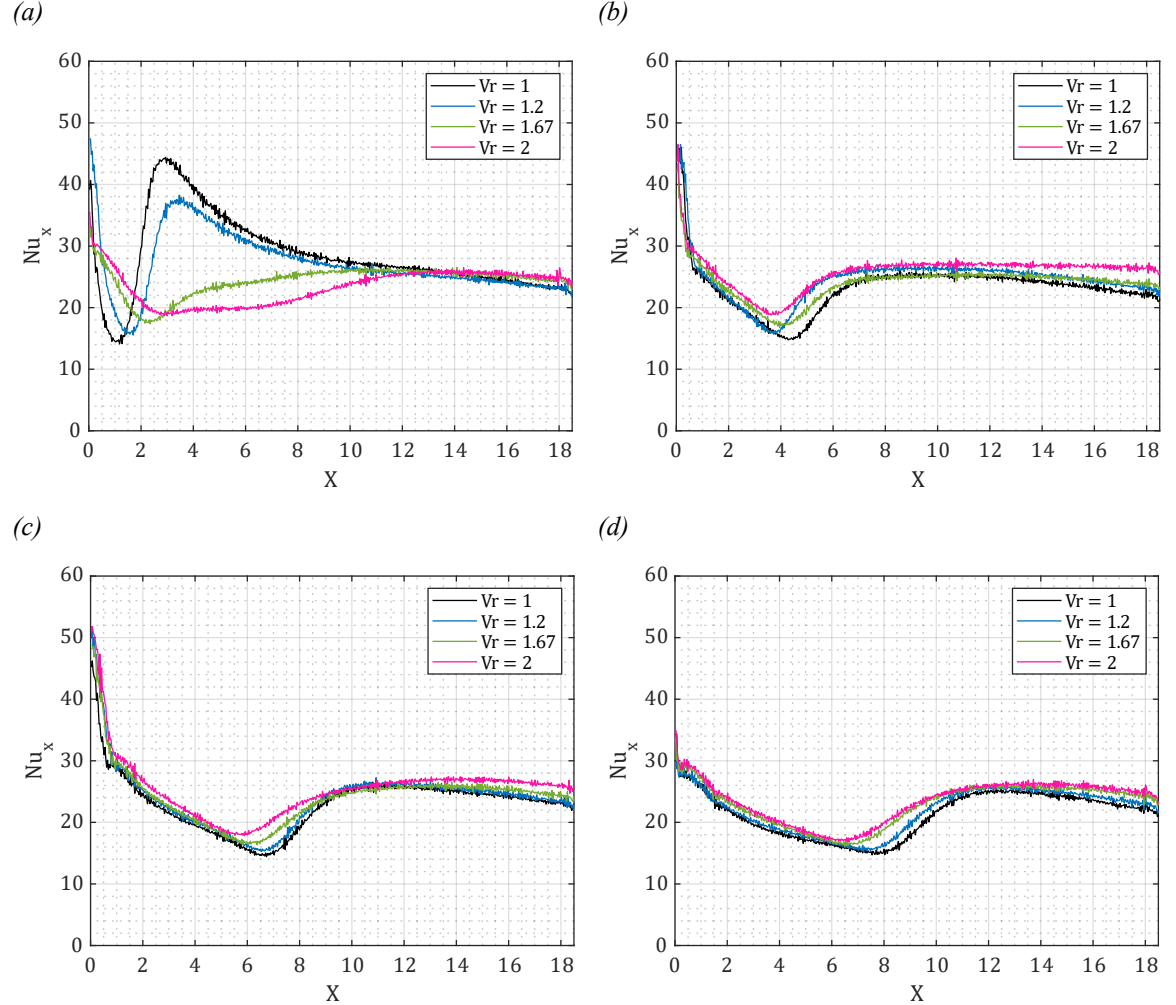


Figure 6.51: Effect of increasing V_r for a dual jet of constant mass flow rate, i.e., $Re_{tot} = 16000$, and varying offset ratio, where $OR = (a) 1, (b) 3, (c) 5$ and $(d) 7$.

The variation in the Nu_x profile as acquired by Vishnuvardhanarao and Das [28] through numerical means using the $k - \varepsilon$ turbulence model is presented in Figure 6.52, which appears considerably more sporadic compared to that of the present investigation. While Vishnuvardhanarao and Das [28] examines a much longer downstream region, the trends acquired close to the nozzle exit generally do not agree with the trends presented thus far in relation to changing V_r . As the Nu_x data of Vishnuvardhanarao and Das [28] appears to contain a local minimum point near the jet nozzle only for $V_r < 0.5$, a direct comparison of the trends acquired is not possible. This thus highlights the requirement for reliable experimental data for dual jet flows to provide the necessary means of validation for experimental studies, where the results of the present investigation act as a step towards addressing this issue.

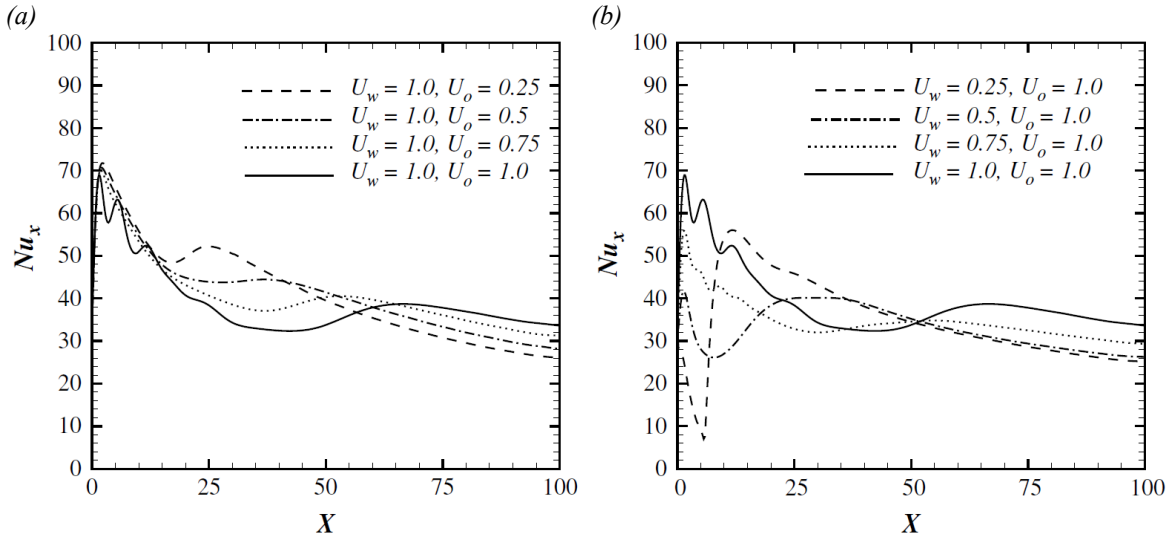


Figure 6.52: Variation of time-averaged Nu_x profile for changing V_r as reported by Vishnuvardhanarao and Das [28] for $Re = 20000$ and $OR = 1$, where the jet inlet velocities, U_w and U_o , are non-dimensionalised with respect to the maximum inlet velocity.

6.7. Summary

This chapter presented the experimental flow and heat transfer results and discussed the emerging trends with respect to varying Reynolds number, offset ratio, and velocity ratio. When compared to that of Wang and Tan [19], the experimental flow data relating to a dual jet with $Re = 10000$ and $OR = 1$ for $V_r = 1$ demonstrated similar flow characteristics, from both a time-averaged and time-resolved perspective, which included the overall flow structure, the streamwise positions of various flow features of interest, and the vortex shedding frequency. This enabled the validation of the dual jet flow behaviour as provided by the experimental test facility, from which the resulting heat transfer findings could be considered credible. In doing so, the dual jet was found to induce a distinct Nu_x profile in the solid wall boundary; an initial decrease in Nu_x where the wall jet was deflected from the wall, a subsequent rise in Nu_x as the flow became reattached to the solid boundary and, followed by a decline in Nu_x over the remaining downstream distance as the mean jet velocity slowed.

Increasing the jet Reynolds number was found to primarily affect the size of the recirculation zone, while all other flow features remained relatively consistent. The trend associated with the length of the recirculation zone was dependent on OR , whereby a reduction was observed for $OR = 1$ and $OR = 2$, while $OR = 3$ was considerably more consistent. Across all OR cases, increasing Re was found to decrease the streamwise rate of decay of maximum velocity, and a greater loss in momentum was noted after the recirculation zone for the lower Re cases. When compared to the heat transfer data, increasing Re was found to enhance Nu_x at all downstream locations while preserving the overall trend, however a greater dispersion in the plots was noted further downstream. The streamwise positions of $Nu_{x_{min}}$ and $Nu_{x_{max}}$ remained consistent for $OR = 1$, however both were found to move downstream with increasing Re for $3 \leq OR \leq 7$, which was consistent with an elongation of the separation region. For all OR examined, the surface-averaged Nusselt number increased with increasing Re in a linear manner, the slope of which declined with increasing OR .

The effect of increasing the offset ratio was a distinct increase in the size of the recirculation zone, which moved the streamwise position of the merge point further downstream. Inside the recirculation region, the time-averaged flow regime was generally consistent across all OR , *i.e.*, two counter rotating vortices, however, the time-resolved analysis revealed a decrease in the Strouhal number associated with the vortex shedding frequency. The heat transfer data indicated that the effect of OR is primarily noted near the jet exit, whereas the Nu_x profiles were more closely aligned in the far downstream region. The streamwise positions of $Nu_{x_{min}}$ and $Nu_{x_{max}}$ were found to move downstream with increasing OR , and a ‘peaking’ effect in the vicinity of $Nu_{x_{max}}$ was observed for lower OR values, which did not occur for $OR \geq 3$. The difference between successive Nu_x profiles for changing OR therefore became more obvious with increasing Re , where the extent of the peaking became exacerbated and moved upstream for the lower OR values, while $Nu_{x_{max}}$ for $OR \geq 3$ continued to move downstream with increasing Re . The effect of increasing OR was hence a general decline in $\overline{Nu}_x$. However, the rate of decrease was found to be greater at lower OR values and approached a more linear relationship with increasing OR . The relationship between $\overline{Nu}_x$, Re and OR is hence summarised by equation 6.2, however further work is required to enable the inclusion of V_r in a final correlation.

Finally, altering the velocity ratio was found to induce two distinctly different flow behaviours depending on whether $V_r < 1$ or $V_r > 1$. For $V_r < 1$, decreasing V_r was found to pull the wall jet farther from the solid wall and reduced the extent of the offset jet deflection, which increased the size of the separation region and the increased the size of the lower vortex with respect to the upper vortex. This resulted in a faster rate of decay of u_m with respect to streamwise distance and self-similarity was not achieved within the scope of the experimental setup. The opposite was observed for $V_r > 1$, where increasing V_r enabled the wall jet to remain attached to the solid wall boundary and increased the offset jet deflection, which reduced the rate of downstream decay of u_m and allowed the flow to reach self-similarity by $X = 8$. The time-resolved analysis showed a rise in St with increasing V_r for $OR = 2$ and $OR = 3$, however, a general decline was observed in the case of $OR = 1$. In addition to this, a limiting V_r range was identified for each OR value, within which the flow exhibited periodic behaviour. With regard to the heat transfer behaviour, the change in the Nu_x profile was found to differ depending on OR . For $OR = 1$, decreasing or increasing V_r with respect to 1 increased the streamwise positions of $Nu_{x_{min}}$ and $Nu_{x_{max}}$, and the peaking effect at $Nu_{x_{max}}$ was increasingly suppressed at higher V_r values. For $OR \geq 3$, the streamwise positions of $Nu_{x_{min}}$ and $Nu_{x_{max}}$ moved upstream when V_r was altered with respect to 1, and a secondary peak in the Nu_x profile emerged for $V_r = 0.5$ when $OR = 5$ and $OR = 7$.

Chapter 7

Numerical Investigation

Following the completion of the experimental investigation, the next stage is the development of a numerical dual jet model that can replicate the observed trends to a reasonable degree of accuracy. The purpose of this chapter is to provide the context behind the methodologies employed during the numerical study with respect to the fundamentals of computational fluid dynamics and the published literature. The first section therefore provides a brief overview of the various considerations required for an extensive numerical study, including the underlying theory, meshing requirements, turbulence modelling, and model verification and validation. A review of numerical modelling in published dual jet studies is hence conducted, following which the rationale behind the design of the present numerical model is described in detail.

7.1.Review of Computational Fluid Dynamics

The objective of this section is to review the fundamentals of computational fluid dynamics (CFD) and highlight its relevance with respect to the current investigation. The use of CFD for the purpose of fluid flow and/or heat transfer analysis is more commonly termed a numerical investigation, and it has numerous associated advantages over traditional experimentation, such as reduced costs and the ability to access an expanded range of parameters that would not be feasible otherwise. The operation of CFD is such that the flow domain of interest is discretised into a finite number of control volumes, which enables the three fundamental equations which govern all fluid flow scenarios to be converted from their original differential or integral form to an equivalent algebraic form [165]. In doing so, an inherent error is introduced, whereby the discretised solution will always differ from the exact solution. As a result, careful consideration must be given to how the flow domain is discretised, as well as to other assumptions and numerical methods adopted for the numerical investigation, to ensure the results produced are within an acceptable range from the exact solution and are credible with respect to the real-life scenario.

7.1.1. Fundamental Equations

Fluid flow and associated heat transfer is governed by three basic principles: the conservation of mass, the conservation of momentum, and the conservation of energy. The principle of mass conservation for an infinitesimal fluid element fixed in space is described by equation 7., *i.e.*,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = S_m \quad (7.1)$$

which states that the net flow of mass flow out of the element must equal the rate of mass decrease inside the element with respect to time [165]. In equation 7., $\mathbf{U} = u\mathbf{i} + v\mathbf{j} + w\mathbf{k}$ denotes the fluid velocity vector, where $\mathbf{i}$, $\mathbf{j}$ and $\mathbf{k}$ represent the unit directional vectors in a three-dimensional Cartesian coordinate system, ρ is the fluid density, and the source term S_m describes the addition of mass to the system from a separate user defined source [166]. The principle of momentum conservation is defined by equation 7.2, also referred to as the Navier Stokes equations, where

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u \mathbf{U}) = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left(\lambda \nabla \cdot \mathbf{U} + 2\mu \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \right) + \frac{\partial}{\partial z} \left(\mu \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) \right) + \rho f_x \quad (7.2a)$$

$$\frac{\partial(\rho v)}{\partial t} + \nabla \cdot (\rho v \mathbf{U}) = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left(\mu \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \right) + \frac{\partial}{\partial y} \left(\lambda \nabla \cdot \mathbf{U} + 2\mu \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial z} \left(\mu \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right) \right) + \rho f_y \quad (7.2b)$$

$$\frac{\partial(\rho w)}{\partial t} + \nabla \cdot (\rho w \mathbf{U}) = -\frac{\partial p}{\partial z} + \frac{\partial}{\partial x} \left(\mu \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) \right) + \frac{\partial}{\partial y} \left(\mu \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right) \right) + \frac{\partial}{\partial z} \left(\lambda \nabla \cdot \mathbf{U} + 2\mu \frac{\partial w}{\partial z} \right) + \rho f_z \quad (7.2c)$$

The principle of momentum conservation is derived using Newton's second law, whereby the LHS of equation 7.2 essentially represents the mass multiplied by the acceleration and the RHS equates to the sum of the pressure forces, viscous forces, and body forces on the fluid element [165]. In this equation, λ represents the second viscosity coefficient, where $\lambda = -2/3 \mu$ and μ is the dynamic viscosity of the fluid. Finally, the principle of energy conservation is denoted by equation 7.3, such that

$$\frac{\partial}{\partial t} \left(\rho \left(e + \frac{\mathbf{U}^2}{2} \right) \right) + \nabla \cdot \left(\rho \mathbf{U} \left(h + \frac{\mathbf{U}^2}{2} \right) \right) = \nabla \cdot \left(k_{eff} \nabla T - \sum_j h_j J_j + \bar{\epsilon}_{eff} \cdot \mathbf{U} \right) + S_h \quad (7.3)$$

where the rate of energy change inside the fluid element is equated to the net heat flux into the element due to conduction, species diffusion and viscous dissipation, respectively, plus the volumetric heat source term S_h [165, 166]. In equation 7.3, k_{eff} is the effective conductivity, *i.e.*, the sum of the fluid conductivity k and the turbulent conductivity k_t , e is the total internal energy and h is the enthalpy.

For Ansys Fluent 2023, the fundamental equations are solved at the centre of each fluid element within the domain by means of the finite volume method and the cell interface values are acquired by means of an appropriate discretization scheme, such as the first-order upwind scheme, the central differencing scheme, the second order upwind schemes, or the QUICK scheme, *etc* [167, 168]. When using the pressure-based solver, which is recommended for incompressible flows such as that encountered in the present study, a suitable pressure-velocity coupling scheme is used to relate and correct the output from the momentum and continuity equations, where it must be ensured that the combination of the pressure and velocity fields satisfy mass conservation on a per cell basis [166]. Solving the fundamental equations in CFD is therefore an iterative process based on a specified initial condition, where the final solution is reached when residuals between subsequent solution iterations fall below a user specified convergence limit.

7.1.2. Meshing Considerations in CFD

The process by which the flow domain is discretised is referred to as meshing, where the flow domain is subdivided into a series of much smaller areas (2D), or volumes (3D), commonly referred to as the mesh elements. In doing so, the mesh is classified as either structured or unstructured, or a hybrid of both, where the term ‘structured’ indicates a geometric uniformity across all mesh elements while ‘unstructured’ implies an irregular division of the flow domain into mesh elements [165]. Both can prove advantageous for different flow scenarios, where a structured mesh can be more useful when used in conjunction with simple geometries, in terms of ease of implementation, accuracy and CPU time, whereas an unstructured mesh can be more readily adapted to suit a complex flow domain [168, 169]. The element shape is also a customisable feature when meshing the flow domain. Triangular or quadrilateral element shapes are generally adopted for 2D simulations, while tetrahedral and hexahedral shaped element volumes are common in 3D meshes [168].

While the chosen meshing method and element type both contribute towards the success of the final solution, it is the number of mesh elements, *i.e.*, the mesh density, which ultimately dictates the overall solution accuracy. As the number of mesh elements is increased, the numerical solution, at least theoretically, approaches the exact solution, while simultaneously inducing a higher computational cost. A degree of prudence must therefore be exercised when meshing a given flow domain, where the incremental gain in accuracy from the use of a greater mesh density should be justified with respect to increased computational costs. Therefore, to attain an adequate solution accuracy while minimising the computational burden, it is generally recommended to concentrate more mesh elements in regions where high gradients in the flow field properties exist, such as regions of sudden fluid acceleration/deceleration, streamline curvature, or high temperature gradients [165, 170]. Examples of where this can occur include geometrical changes in the flow domain, free shear or mixing layers in jet flows, and wall boundary layers [170].

Of these examples, wall boundary layers generally require the finest mesh resolution, as viscous forces present in the near-wall region can induce steep velocity and thermal gradients [168]. This is demonstrated in Figure 7.1 for the boundary layer growth for flow past a flat plate, where the fluid particles in direct contact with the flat plate are brought to rest as a result of the no-slip condition, which subsequently slows adjacent fluid particles. This layer of fluid affected by the presence of the solid boundary is the boundary layer, the thickness of which, δ , is generally defined as the wall-normal distance to the point at which the local streamwise flow velocity is 99% of the freestream velocity, *i.e.*, $u(\delta) = 0.99u_\infty$ [154]. Multiple flow regimes exist within the boundary layer, where a laminar region is expected in the initial stages of the boundary layer growth, followed by a short transition region at some point downstream and, finally, a turbulent region. Inside the turbulent region, viscous forces continue to dominate immediately adjacent to the solid wall, leading to the creation of a viscous sublayer within which turbulent vortical structures cannot exist [168]. Outside of the viscous sublayer and the intermediate buffer zone exists a log layer, inside which viscous and inertial forces are of equal importance. The remainder of the turbulent boundary layer above the log layer is primarily inertia-dominated [168].

To accurately capture the complex flow inside the fully turbulent boundary layer in CFD, it is recommended to have at least 10 mesh element layers across the boundary layer thickness δ in the wall normal direction (y), where the criterion for the first layer height Δy_1 is usually dictated by the dimensionless y^+ value, *i.e.*,

$$y^+ = \frac{\Delta y_1}{\nu} \sqrt{\frac{\tau_w}{\rho}} \quad (7.4)$$

where ν is the fluid kinematic viscosity, ρ is the fluid density, and τ_w is the wall shear stress, such that

$$\tau_w = \frac{1}{2} C_f \rho u_\infty^2 \quad (7.5)$$

The value for the skin friction coefficient C_f is obtained using an appropriate empirical correlation [168, 170]. Based on equation 7.4, it is assumed that $y^+ < 5$ indicates a first layer height within the viscous sublayer, while $30 \leq y^+ \leq 500$ suggests a first layer height coinciding with the log layer [168]. For most turbulent wall-bounded flows, it is generally recommended to choose a Δy_1 value which ensures $y^+ < 1$ to accurately *simulate*, as opposed to *model*, the near-wall behaviour. However, the final y^+ value should ultimately be chosen based on the selected turbulence model and other constraints.

It should be noted that in the case where the user intends to capture the transition from the laminar to turbulent regime inside the boundary layer, a much finer mesh resolution is required in the transition region compared to the fully turbulent boundary layer. While $y^+ < 1$ is still required, a cell expansion ratio ER in the wall normal direction of less than 1.1 is also advised, *i.e.*, $ER = \Delta y_{j+1}/\Delta y_j < 1.1$. In addition to the mesh resolution in the wall-normal direction, consideration must also be given to the streamwise mesh resolution, where $\Delta x = 1\delta$ is recommended inside the transition region [170].

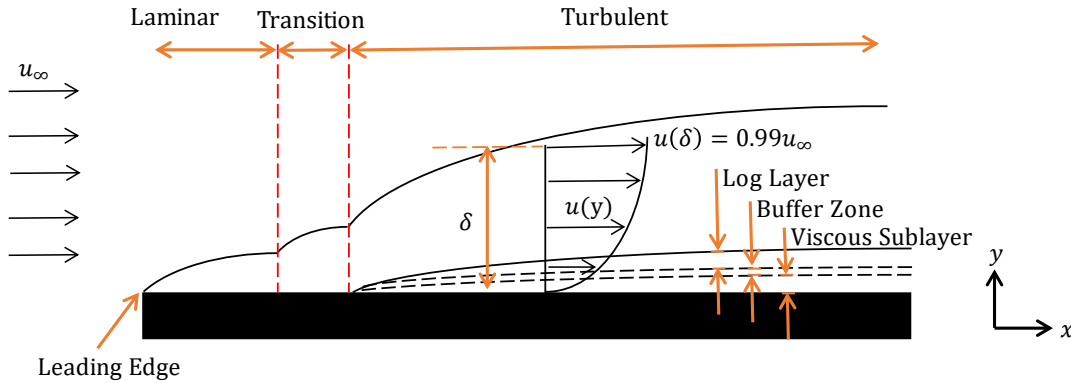


Figure 7.1: Typical boundary layer growth for flow past a flat plate.

7.1.3. Turbulence Modelling

Due to the complexity of turbulent flow, it is often not practical to simulate all turbulent structures present in the flow due to the associated computational costs. Within any given turbulent flow domain, there exists vortices or eddies of varying length and time scales, where the smallest vortices are often several orders of magnitude smaller than the largest eddies [167]. In addition to this, an energy cascade effect has been shown to take place inside turbulent flows, where the kinetic energy contained within larger eddies is transferred to successively smaller eddies as they break down due to inherent instabilities, before finally being dissipated in the form of heat at the smallest length scale. In CFD, directly solving the Navier Stokes equations to obtain information down to the smallest vortical length scale, such as that found within the turbulent boundary layer, is referred to as direct numerical simulation (DNS) [165]. In practice however, this method proves unfeasible due to the grid refinement (and time step for transient studies) required to capture such length scales, and the

computational costs are extortionate as a result. Therefore, a certain degree of turbulence modelling is employed to ease the computational burden. An example of such is large eddy simulation (LES). In this case, the larger eddies within the flow are simulated directly by means of DNS, whereas the effects of the smallest eddies are modelled. Although substantial savings can be made using LES compared to DNS, the computational costs still remain relatively high [167].

Based on this, it is considered more practical to use the Navier Stokes equations to just solve for the average flow properties and subsequently model turbulent eddies within the flow. This is most commonly achieved through the use of the Reynolds Averaged Navier Stokes (RANS) equations, which decompose the fluid properties into an mean and fluctuating component, *i.e.*, $\mathbf{U} = \bar{\mathbf{U}} + \mathbf{U}'$ and $p = \bar{p} + p'$ *etc.*, where the turbulence intensity I is defined as the ratio of the fluctuating velocity component to the mean velocity, *i.e.*, $I = \mathbf{U}'/\bar{\mathbf{U}}$ [167]. In doing so, a Reynolds stress term is introduced to the Navier Stokes equations, which is essentially a fictitious stress tensor which couples the mean and fluctuating components of the velocity field, where $\tau_{ij} = -\rho \overline{\mathbf{U}'_i \mathbf{U}'_j}$ for $i, j = 1, 2, 3$ [166, 167]. Owing to the Boussinesq approximation, the Reynolds stress components can be expressed in terms of a turbulent dynamic viscosity or kinematic viscosity value, μ_T and ν_T , respectively. A RANS-based turbulence model is hence required to determine the turbulence viscosity and subsequently close the RANS equations.

RANS-based turbulence models operate on the assumption that the turbulent viscosity can be related to an appropriate turbulent velocity scale u and length scale l using a proportionality constant C_v , such that

$$\nu_T = C_v \frac{l^2}{t} = C_v u l \quad (7.6)$$

In equation 7.6, the turbulent velocity scale is usually equated to the square root of the turbulent kinetic energy per unit mass, k , which is defined as half the trace of the Reynolds stress tensor, *i.e.*, $k = \frac{1}{2} \overline{\mathbf{U}'_i \mathbf{U}'_i}$ [167]. The turbulence model hence proposes a set of transport equations to solve for k and l , and, while zero- and one-equation turbulence models do exist, the use of a two-equation turbulence model is generally preferred as it solves for the turbulent kinetic energy and length scales independently using separate transport equations [167]. While the first equation solves for the transport of the k , the second equation usually describes the transport of some other property that can easily be related to turbulent length scale. Examples include the energy dissipation rate $\varepsilon = k^{\frac{3}{2}}/l = \nu \frac{\partial u'_i}{\partial x_j} \frac{\partial u'_i}{\partial x_j}$, *i.e.*, the rate of conversion of turbulent kinetic energy to heat, or the specific dissipation rate $\omega = k^{\frac{1}{2}}/C_\mu l \propto \varepsilon/k$, *i.e.*, the rate of dissipation of kinetic energy per unit volume and time, where C_μ is a constant of proportionality [166, 167, 170, 171].

One of the most popular two-equation models is the standard $k - \varepsilon$ model, which solves the respective transport equation for k and ε , from which the turbulent kinetic viscosity can be derived using the relation $\nu_t = C_\mu k^2/\varepsilon$ [167]. While the robustness and simplistic nature of the standard $k - \varepsilon$ model are major factors contributing to its popularity, it may not always be the most appropriate choice for a particular flow configuration. As a result of adopting Boussinesq's hypothesis for isotropy, the standard $k - \varepsilon$ model does not provide accurate results for streamline curvature, swirling flows, axisymmetric jets, and near wall flows [167]. This model is therefore more suited towards high Reynolds number flows within which the turbulence can be described as nearly isotropic. As such, many improvements have been made to improve the accuracy of the

standard $k - \varepsilon$ model over a wider range of fluid flows, such as the Re-Normalisation Group (RNG) or realizable $k - \varepsilon$ model. The use of a wall function in conjunction with the $k - \varepsilon$ model is also recommended when simulating high- Re near wall flows to avoid integrating the model equations all the way to the solid boundary, where a first cell height within the turbulent boundary log layer is preferable, approximately $30 \leq y^+ \leq 50$ [167, 168].

An alternative two-equation turbulence model is the $k - \omega$ model, which respectively solves each transport equation for k and ω , and relates both parameters to the turbulent kinematic viscosity using the expression $\nu_T = k/\omega$ [167]. The primary advantage of the $k - \omega$ model compared to the $k - \varepsilon$ models is its ability to perform in low turbulence, viscous dominant flows, thus providing superior near-wall accuracy, most notably inside viscous sub-layers and regions of boundary layer separation [167, 168]. As the flow is solved all the way to the solid boundary, a fine mesh is required near the wall with a first layer height inside the viscous sublayer, such that $y^+ \approx 1$. The solution provided by the $k - \omega$ model, however, is very sensitive to the freestream values of k and ω . As the values of the turbulent kinematic viscosity becomes indeterminant as ω tends towards 0, an assumed value for ω must be specified for the freestream flow, which reduces its performance in these regions [168]. The use of the $k - \omega$ model is therefore not recommended for flows far from a solid boundary [172].

The Shear Stress Transport (SST) $k - \omega$ model offers as solution in this regard, as it combines the advantages of the $k - \omega$ and $k - \varepsilon$ models to produce a superior model suited to most industrial applications [170]. This hybrid model employs the $k - \varepsilon$ model in the freestream but switches to the $k - \omega$ model in the near wall regions using a suitable blending function, therefore maintaining a consistent level of accuracy across the entire the flow field [168]. The SST $k - \omega$ can be used to accurately model a wide range of flows, including adverse pressure gradients and separated flows [166]. Although the SST $k - \omega$ model tends to overpredict turbulence in regions of large normal strain, it is generally the most recommended model due to its suitability to most flow configurations. Similar to the $k - \omega$ model, a first cell height corresponding to $y^+ \approx 1$ is required at solid boundaries.

Despite the numerous advantages proposed by the SST $k - \omega$ model, limitations do arise as a result of the assumption that the wall boundary layer is in the fully turbulent region. There are few available turbulence models that accurately capture the laminar-to-turbulent transition inside the boundary layer, which is primarily due to the uncertainty which surrounds the onset of transition and the wide range of mechanisms through which transition can occur, *e.g.*, flow instabilities, freestream turbulence, or flow separation [173]. Further to this, the use of Reynolds-averaging in turbulence modelling eliminates any effects of linear disturbance growth, meaning they cannot be readily used to model transition without modification. This may lead to inaccuracies when simulating near wall flows, and in particular for heat transfer applications.

The $\gamma - Re_\theta$ transition model, which is commonly referred to as the Transition SST model, is an example of a reputable turbulence model which can accurately predict the onset of transition in the boundary layer by coupling two additional transport equations with that of the SST $k - \omega$ model [166]. In doing so, two further parameters are introduced to the SST $k - \omega$ model, namely the intermittency γ and momentum thickness Reynolds number Re_θ . The first transport equation solves for the intermittency value, which is essentially an indicator of the current flow regime, where $\gamma = 0$ for a laminar flow regime and $\gamma = 1$ in the fully turbulent

region, which is always the case for the freestream flow. When coupled with the SST $k - \omega$ model, γ is used to ‘turn on’ the production term of the turbulent kinetic energy downstream of the transition point [173]. The transition onset criterion is therefore dictated by Re_θ , which is a local boundary layer Reynolds number calculated based on the local momentum thickness. The value of the transition Reynolds number Re_{θ_t} is determined by the second additional transport equation, where the intermittency value is set to initially increase when the local Re_θ value exceeds a critical value Re_{θ_c} . This occurs upstream of Re_{θ_t} , where the difference between Re_{θ_c} and Re_{θ_t} is defined through empirical correlation [166, 173]. As aforementioned, the use of a transition model has a much finer mesh requirement in the near wall transition region, where $y^+ < 1$ is required to ensure that the transition point is not moved upstream [166].

7.1.4. Verification and Validation in Numerical Modelling

Following the acquisition of a numerical solution, the credibility of the results are confirmed by means of both verification and validation assessment. Verification is the process by which the model implementation is corroborated as an accurate representation of the conceptual model, which includes both a verification of code and a verification of process [174]. The verification of code involves examining the input parameters to the numerical model to ensure they are correct, such as the boundary conditions or fluid assumptions, as well as analysing the code for the presence of bugs. The verification of process subsequently involves defining a mesh uncertainty value for the final solution and assessing whether it falls inside acceptable limits, where the timestep uncertainty must also be accounted for in transient studies. This is commonly achieved by means of a grid convergence study, which compares the output solution from three increasingly fine meshes for a given flow simulation to identify whether the incremental solution changes are converging asymptotically towards the true solution. For a selected parameter of interest, b , the grid convergence ratio r_g is defined as

$$r_g = \frac{b_3 - b_2}{b_2 - b_1} \quad (7.7)$$

where $b_3 - b_2$ is the difference between the solution output for finest and medium mesh and $b_2 - b_1$ is the difference between the medium and most coarse mesh. For $0 < r_g \leq 1$, the difference between successive mesh iterations is reducing with increasing mesh density, meaning the solution is monotonically converging towards the true solution, while $-1 \leq r_g < 0$ implies oscillatory convergence. Outside of these limits, *i.e.*, $|r_g| > 1$, the solution is assumed to be diverging, implying that the chosen simulation parameters or mesh are insufficient for the flow configuration of interest.

The apparent order of accuracy p of a monotonically converged solution can be quantified using equation 7.8, *i.e.*,

$$p = \frac{\ln\left(\frac{b_2 - b_1}{b_3 - b_2}\right)}{\ln(r)} \quad (7.8)$$

where r is grid refinement ratio calculated based on the cell size h of the coarse, medium, and fine meshes, such that $r = h_3/h_2 = h_2/h_1$ [175]. The grid error from the generalised Richardson Extrapolation (GRE) for the finest mesh can thus be defined as

$$\delta_{GRE}^* = \frac{b_3 - b_2}{r^p - 1} \quad (7.9)$$

and is interpreted as how far the solution is from the final asymptotic solution. Finally, the grid uncertainty for a 95% confidence interval can be determined using the Factor of Safety (FS) method, *i.e.*,

$$U_{FS} = f(x) = \begin{cases} (2.45 - 0.85P') \times |\delta_{GRE}^*|, & 0 < P' \leq 1 \\ (16.4P' - 14.8) \times |\delta_{GRE}^*|, & P' > 1 \end{cases} \quad (7.10)$$

where $P' = p/p_{th}$ and p_{th} is the theoretical order of accuracy of the chosen discretisation scheme, *e.g.*, $p_{th} = 2$ for a second order discretisation scheme [176].

Validation is hence the procedure by which the numerical data is compared with experimental results to ensure its congruency with real world observations [177]. As such, the set of numerical results are considered validated if they lie within a certain range of uncertainty from a set of experimentally determined values, *i.e.*, the combination of the experimental uncertainty and the grid uncertainty by means of the RSS method. Should a numerical model produce results which can be considered both verified and validated, it can be assumed that the numerical model is an accurate representation of the desired fluid flow configuration and the results produced can be deemed credible. The numerical model can subsequently be used to analyse the fluid flow setup over an expanded range of parameters, far beyond the realms of physical experimentation.

7.2. Numerical Modelling in Dual Jet Studies

A summary of the published dual jet numerical models to date can be seen in Table 7.1, where subsequent studies using the same numerical model were removed for clarity. In each case, the chosen turbulence model is specified, as well as the adopted method of model validation. While a certain degree of variability is evident, the standard $k - \varepsilon$ model appears to be the most commonly used turbulence model for dual jet investigations, with the $k - \omega$ model being the second most popular. Considering the prominent solid boundary present in a dual jet study, the choice of the standard $k - \varepsilon$ model in most cases is questionable due to its acknowledged inaccuracies in the near wall region. Similarly, as the use of the $k - \omega$ model is generally not recommended, its use in conjunction with the dual jet flow configuration is problematic.

A number of studies reported conducting a comparison between the performance of various turbulence models before selecting which model was best suited for dual jet flow. Li *et al.* [92] was the first study to report such, where a separate set of results were acquired using the realizable $k - \varepsilon$ model, the SST $k - \omega$ model, the $k - kl - \omega$ model, the transition SST model and the $v^2 - f$ model. The results revealed that the realizable $k - \varepsilon$ model was the most effective through a comparison with the experimental flow data of Wang and Tan [19] and was hence adopted for the remainder of the investigation. Kumar [83] subsequently identified the standard $k - \varepsilon$ model as the best predictor of dual jet flow when compared to the RNG $k - \varepsilon$ model and the Reynolds stress model (RSM), whereas Hnaïen *et al.* [10] considered the standard $k - \omega$ model to be superior when compared to a similar selection. In further contrast, Assoudi *et al.* [26] considered the RSM to be the most accurate turbulence model for use with dual jet flows. Based on these findings, it is clear to see that there is significant disagreement amongst the relevant authors regarding the turbulence model that most accurately predicts the behaviour of a dual jet, and thus requires further examination.

Li *et al.* [2] was the only investigation to use LES and, in doing so, compared the effectiveness of two small scale eddy models, *i.e.*, the Dynamic Smagorinsky-Lily Model (DSLM), and the Dynamic Kinetic energy Subgrid-scale Model (DKSM). The DKSM was identified as better at predicting the mean streamwise velocity when compared to the experimental findings of Wang and Tan [19], however no further studies have been conducted since to support this.

Due to the scarcity of published experimental data relating to dual jet flows, many of the numerical studies contained in Table 7.1 have relied on published data relating to single offset jet flows for model validation. However, as noted previously, the flow field associated with single offset jets differs greatly from that of a dual jet, particularly near the jet nozzle exit, which results in distinctly different heat transfer behaviour for non-isothermal flows. Despite this, most numerical studies which explore the heat transfer characteristics of dual jets use thermal data relating to single offset jets as a means of model validation, as highlighted in Table 7.1. In many cases, validation was carried out using the experimental dual jet data published by Wang and Tan [19], however, as this data is related to flow-only parameters, its use is insufficient for studies which aim to define the heat transfer characteristics of dual jet flow. This has resulted in major discrepancies across the published data, where the Nu_x profiles produced from separate investigations for the same jet parameters do not align, as previously discussed in section 3.2 when reviewing the relevant literature. While changes in other model parameters may also contribute towards these discrepancies, such as the chosen boundary conditions, the domain size, the discretisation scheme, and the mesh density, it was noted that these model inputs did not

vary greatly between successive numerical studies and were therefore omitted from Table 7.1 due to space constraints.

Table 7.1: Numerical modelling in past dual jet studies, where the model validation approach is specified for each, which includes the chosen validation parameter, the flow type, and the validation data source.

Authors	Method	OR	V_r	Re ($\times 10^3$)	Model Validation Approach		
					Parameter	Flow Type	Data Source
Vishnuvardhanarao and Das [28]	$k - \varepsilon$	1	0.25 – 4	10–40	- u	- DJ ¹	- Wang and Tan [19]
Li <i>et al.</i> [2]	LES	1	1	10	- u - u_m - $\gamma_{0.5}$	- DJ - DJ - DJ	- Wang and Tan [19] - Wang and Tan [19] - Wang and Tan [19]
Li <i>et al.</i> [92]	$k - \varepsilon$	0.5 – 8	0.25 – 4	10	- u - u_m	- DJ - DJ	- Wang and Tan [19] - Wang and Tan [19]
Farooq and Das [29]	$k - \varepsilon$	2 – 3	1	15	- T_{jet} - Nu_x	- SOJ ² - SOJ	- Holland and Liburdy [178] - Kim <i>et al.</i> [79]
Kumar [83]	$k - \varepsilon$	2 – 14	1	15	- u - T_{jet}	- SOJ - SOJ	- Pelfrey and Liburdy [93] - Holland and Liburdy [178]
Mondal <i>et al.</i> [22]	$k - \varepsilon$	1	1	10	- u_m - C_f - T_{jet} - Nu_x	- SOJ - DJ - SOJ - SOJ ³ - SOJ	- Wang and Tan [19] - Wang and Tan [19] - Pelfrey and Liburdy [93] - Holland and Liburdy [178] - Holland and Liburdy [178] - Kim <i>et al.</i> [79]
Hnaïen <i>et al.</i> [84]	$k - \omega$	4 – 19	1	10 – 40	- u - Nu_x	- DJ - SOJ - SOJ	- Kumar [27] - Pelfrey and Liburdy [179] - Song <i>et al.</i> [81]
Zhao and Sun [89]	BGK	3 – 9	1	56	- u	- FJ ⁴	- Sato and Sakao [180]
Singh and Dewan [24]	SST $k - \omega$	1	1	10 – 40	- u - T_{jet}	- DJ - SOJ	- Wang and Tan [19] - Holland and Liburdy [178]
Assoudi <i>et al.</i> [26]	RSM	1 – 10	0.57 – 1	15	- u - u_m	- DJ - DJ - SOJ	- Wang and Tan [19] - Wang and Tan [19], Mondal <i>et al.</i> [20] - Wang and Tan [19]
Kumar <i>et al.</i> [23]	$k - \varepsilon$	3 – 9	1	5 – 20	- u	- SOJ	- Kumar [27], Pelfrey and Liburdy [179]
Hnaïen <i>et al.</i> [10]	$k - \omega$	5.5 – 15.5	1	15 – 40	- u - Nu_x	- DJ - SOJ - SOJ	- Kumar [27] - Pelfrey and Liburdy [179] - Song <i>et al.</i> [81]
Singh <i>et al.</i> [87]	$k - \varepsilon$	6	1	15	- u - T_{jet}	- SOJ - SOJ	- Song <i>et al.</i> [81] - Holland and Liburdy [178]
Rathore and Verma [108]	$k - \varepsilon$	4 – 12	1	10 – 25	- u_m - u - T_{jet}	- DJ - DJ - SOJ - SOJ	- Wang and Tan [19], Kumar [27] - Kumar and Das [1] - Pelfrey and Liburdy [93] - Holland and Liburdy [178]
Mondal <i>et al.</i> [102]	$k - \omega$	4 – 10	1	15	- u - Nu_x	- SOJ - SOJ	- Pelfrey and Liburdy [93] - Song <i>et al.</i> [81]
Hnaïen <i>et al.</i> [103]	$k - \omega$	4	1	15	- u - Nu_x	- DJ - SOJ	- Kumar [27] - Song <i>et al.</i> [81]

¹Dual Jet, ²Single Offset Jet, ³Single Wall Jet, ⁴Free Jet

7.3.Numerical Approach

For the present investigation, the computational model was built using ANSYS Fluent version 2023 R2, where the geometric and flow conditions were chosen to best replicate the experimental setup. The following sections therefore outline each aspect of the final numerical model setup and the reasoning behind the various model decisions.

7.3.1. Model Assumptions

The following assumptions are adopted for the final numerical model:

- 1) Gravitational effects are negligible.
- 2) The flow is steady.
- 3) The flow field is two-dimensional.
- 4) The bulk flow is fully turbulent.
- 5) The working fluid, air, is incompressible.
- 6) Air properties vary with temperature.
- 7) The heated foil section in the experimental setup can be adequately represented by a uniform wall flux (UWF) boundary condition.
- 8) Natural convection effects are negligible.
- 9) Viscous heating effects are negligible.

Assumption (1) allows for the effect of gravity to be removed from the computational model. Prior to doing so, the results from computational model with and without gravity were compared, which revealed a negligible difference between the two solutions. The increased computational effort through the inclusion of gravitational effects was thus deemed unnecessary and removed from the dual jet computational model.

Regarding assumption (2), the assumption that the flow has reached a steady-state condition implies that any start-up effects associated with dual jet flow are not of interest and that sufficient time has passed to allow any decaying transient effects to dissipate entirely. This assumption, however, does not negate the possibility of von Kármán-like shedding, as reported by Wang and Tan [19], but suggests that this flow phenomenon would occur at a fixed rate and, when time-averaged, produce a consistent flow field result.

Assumption (3) is derived from the planar flow assumption inside the experimental flow field. As described in previously in section 4.1.1, the purpose of the airflow conditioning components and enclosure sidewalls implemented in the experimental was to create a uniform flow velocity across the jet nozzle exit and ensure predictable, 2D behaviour along the midplane of the flow domain. While 3D flow structures may otherwise be present, it was assumed that the implemented measures sufficiently suppressed such effects in the experimental setup. If successful, the two-dimensional assumption of the numerical model should be an accurate representation of the experimental flow field. In additional, the use of a two-dimensional model allows for a fair comparison with the results of previously published numerical dual jet studies, the vast majority of which assume a 2D flow field.

As jet flow is, by its nature, quite turbulent, assumption (4) seems the most prudent option. As no definition of the transitional Re range for dual jet flow could be found in the available literature, and since large-scale mixing behaviour is intrinsic to dual jet flow, it is reasonable to assume the primary flow regime inside the dual jet flow domain would be turbulent. To further validate this assumption, the use of a laminar viscous model was trialled initially, however, the results did not compare favourably with the experimental data. The flow is also

assumed to be incompressible, *i.e.*, assumption (5), the justification for which is based on the Mach number Ma , such that

$$Ma = \frac{u}{c} \quad (7.11)$$

where u is the flow velocity and c is the speed of sound through the working fluid. A gaseous flow can be considered incompressible if flow velocity is significantly less than the speed of sound through the medium, *i.e.*, $Ma < 0.1$ [166]. For the range of experimental Reynolds numbers investigated, the Mach number varies from 0.036 to 0.083 for $c \approx 343 \text{ m/s}$ at 20°C , meaning the flow can be considered incompressible. An incompressible solver, *i.e.*, the pressure-based solver, is therefore correctly adopted for the present flow simulation. Having said this, the air fluid properties are expected to deviate with temperature, particularly in the near wall region, where the air film temperatures are anticipated to vary by up to 30°C depending on the test case, *i.e.*, assumption (6), where the varied fluid properties include density, specific heat capacity, thermal conductivity and dynamic viscosity.

Regarding the heat transfer aspects of the numerical model, it is assumed that the uniform heat generation within the test foil can be replicated in the computational domain using a uniform wall flux boundary condition of $q'' = 1670 \text{ W/m}^2$, *i.e.*, assumption (7). While slight differences may exist between the calculated convective heat flux to the working fluid based on the foil energy balance and the uniform wall heat flux applied in the numerical model, the effect on the Nusselt number was found to be negligible based on the findings from a sample conjugate study, where the foil was modelled as a thin solid region with uniform volumetric heat generation and heat losses from the underside of the foil were enabled. As the difference between the local Nusselt number profile for the uniform wall flux case and the conjugate model case were found to be insignificant, the increased computational effort brought about through the use of a more complex conjugate model was deemed unnecessary. The uniform wall flux boundary condition is thus considered an adequate approximation of the foil test section.

Assumption (8) states that heat transfer in the dual jet configuration occurs primarily through convection and natural convection or buoyancy effects are negligible, as previously shown using the Biot number for the experimental investigation in section 4.3.1.1. Finally, viscous heating effects are considered minor for the present numerical study, as stated in assumption (9), and are thus neglected. This assumption is justified by means of the Brinkman number Br , such that

$$Br = \frac{\mu u^2}{k \Delta T} \quad (7.12)$$

where u is the flow characteristic velocity and ΔT is the temperature difference between the bulk jet flow and the solid boundary [166]. Viscous heating can be considered negligible if $Br \ll 1$, which is satisfied for the current investigation for the experimental Re range and average ΔT value, where $0.006 \leq Br \leq 0.032$.

7.3.2. Computational Domain

A sketch of the computational domain is shown in Figure 7.2. The domain consists of a wall jet and offset jet, each of equal width $w = 7 \text{ mm}$, and a quiescent fluid domain $ABCD$, the dimensions of which correspond to the enclosure as per the experimental setup, such that $X \times Y = x/w \times y/w = 100 \times 35$. The jet inlet plane, solid bounding wall, enclosure roof, and the flow outlet plane correspond to the AB , AD , BC , and CD planes, respectively, and the origin of the computational domain is located at point A, *i.e.*, the intersection of the wall

jet inlet with the solid bounding wall. The offset distance between the jets, denoted by $OR \times w$, is adjustable depending on the desired OR value. Each jet inlet is extended upstream by an additional 35 mm ($= 5w$) length, which corresponds to the streamwise length between the contraction exit and jet nozzle exit plane in the experimental setup. In doing so, it was found that the inclusion of the additional upstream length provided a better representation of the flow conditions at the jet exit compared to the experimental study, in terms of the velocity profile, the turbulence intensity distribution, and the boundary layer growth, *etc.* As a result, the simulated Nu_x profiles more closely aligned with their corresponding experimental Nu_x profile, in particular close to the jet exit, where it was noted that the initial Nu_x decline from the jet exit was much sharper than its experimental counterpart without the 35 mm inlet extension.

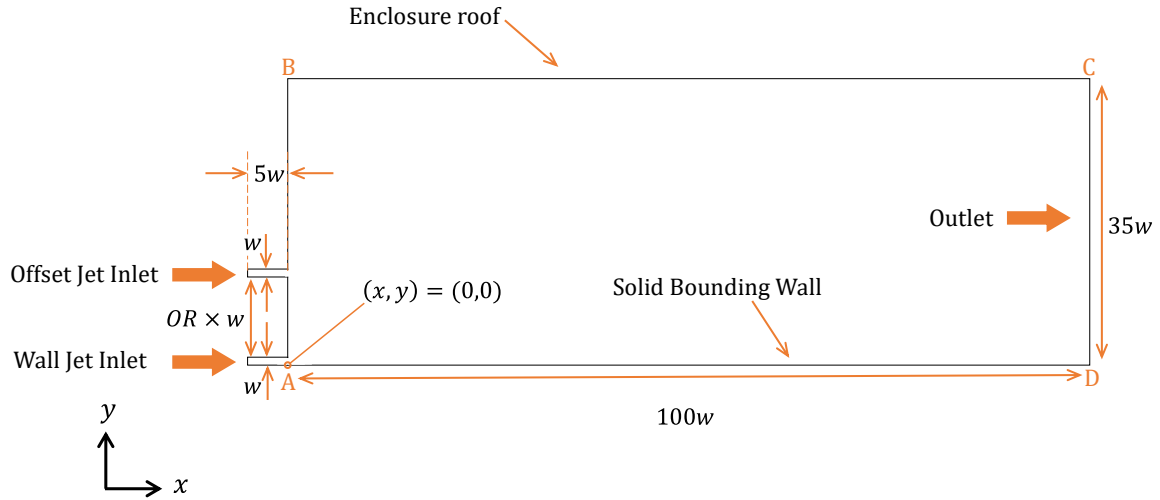


Figure 7.2: Computational domain design for the numerical study, where $w = 7 \text{ mm}$.

7.3.3. Turbulence Model

Due to the complexity of dual jet flow, such as the large-scale mixing, near wall influence, and potential for boundary layer separation, it was decided that the use of the standard $k - \varepsilon$ model would be insufficient, despite its popularity in previously published dual jet studies. Similarly, the $k - \omega$ model was deemed inadequate due to its lack of useability outside the wall boundary layer. While the SST $k - \omega$ model presents numerous advantages in comparison, its assumption of a fully turbulent boundary layer may not be representative of the associated experimental conditions. Due to the wall-bounded nature of dual jet flow, it is important to consider the development of the boundary layer, as it is the near wall flow behaviour that directly influences the local heat transfer characteristics, as opposed to the bulk flow conditions. Further to this, as the laminar-to-turbulent transition greatly influences the heat transfer characteristics of the single wall jet, it is unlikely that this is not also the case for the dual jet. Finally, due to the vortex shedding phenomenon that can occur inside dual jet flow, boundary layer separation is expected to some extent, the location of which is again dependent on the flow regime inside the wall boundary layer. It was thus decided to primarily focus on the transition SST turbulence model for the present investigation, as its ability to predict the laminar-to-turbulent transition in the wall boundary layer is advantageous for the dual jet flow configuration. The use of the transition SST model in conjunction with a turbulent jet was also adopted by Alimohammadi *et al.* [181], who noted that modelling the laminar-to-turbulent transition in the wall jet region of an impinging jet flow was crucial when compared to the experimental heat transfer results. The transition SST model is therefore selected for the present flow configuration, where its performance is compared to other turbulence models in section 8.1.1 to confirm it as

the most appropriate option when modelling dual jet flows. It should be noted that the turbulence production limiters are enabled by default in Ansys Fluent 2023R2, while the curvature and corner correction options are disabled. As varying these options is deemed unnecessary for the present flow configuration, all are maintained at their respective default states.

7.3.4. Governing Equations

The governing conservation equations for the numerical model are derived from the mass, momentum, and energy equations previously presented in section 7.1.1, but simplified to reflect a 2D, incompressible flow, and Reynolds-averaged to account for turbulent conditions. For a now two-dimensional flow field, the direction vector and velocity vector are each reduced to two components, such that $\mathbf{X} = (x_1, x_2) = (x, y)$ and $\mathbf{U} = (u_1, u_2) = (u, v)$. The continuity equation is rewritten as

$$\frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) = 0 \quad (7.13)$$

where the source term S_m is omitted as no additional mass is introduced to the system through phase change or otherwise and the temporal term is removed to indicate the use of a steady state solver. The momentum conservation equations are modified to correspond with the 2-dimensional Reynolds Averaged Navier Stokes (RANS) equations, with gravitational body force term neglected, such that

$$u \frac{\partial}{\partial x}(\rho u) + v \frac{\partial}{\partial y}(\rho u) + \frac{\partial p}{\partial x} = \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) + \mu_t \left(\frac{\partial u}{\partial x_j} + \frac{\partial u_j}{\partial x} \right) \quad (7.14a)$$

$$u \frac{\partial}{\partial x}(\rho v) + v \frac{\partial}{\partial y}(\rho v) + \frac{\partial p}{\partial y} = \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) + \mu_t \left(\frac{\partial v}{\partial x_j} + \frac{\partial u_j}{\partial y} \right) \quad (7.14b)$$

and $j = 1, 2$. Finally, the energy conservation equation is adapted to remove the species diffusion and viscous heating terms.

$$\nabla \cdot \left(\rho \mathbf{U} \left(h + \frac{\mathbf{U}^2}{2} \right) \right) = \nabla \cdot k_{eff} \nabla T + S_h \quad (7.15)$$

To determine the turbulent viscosity value, the transition SST model provides four transport equations, two of which are adapted from the SST $k - \omega$ model. The first transport equation derived from the SST $k - \omega$ model solves for the k value and is modified to allow the intermittency value γ to turn on or off the turbulence production, such that

$$\frac{\partial}{\partial \mathbf{X}_i}(\rho k \mathbf{U}_i) = \frac{\partial}{\partial \mathbf{X}_j} \left(\Gamma_k \frac{\partial k}{\partial \mathbf{X}_j} \right) + \gamma_{eff} \tilde{G}_k - \min(\max(\gamma_{eff}, 0.1), 1.0) Y_k + S_k \quad (7.16)$$

In equation 7.16, $\tilde{G}_k$ and Y_k respectively denote the production and destructive terms, Γ_k is the diffusivity of k , S_k is a user-defined source term and $i = 1, 2$. The second transport equation from the SST $k - \omega$ solves for the specific turbulent dissipation ω , *i.e.*,

$$\frac{\partial}{\partial \mathbf{X}_i}(\rho \omega \mathbf{U}_i) = \frac{\partial}{\partial \mathbf{X}_j} \left(\Gamma_\omega \frac{\partial \omega}{\partial \mathbf{X}_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega \quad (7.17)$$

where, G_ω , Y_ω , Γ_ω , S_ω again denote the production, destruction, diffusivity, and source terms, respectively. The transition SST model therefore introduces two additional transport equations, the first of which solves for the local intermittency value, *i.e.*, equation 7.18. In this equation, P_{γ_1} is the transition source term, the value of

which is controlled by $Re_{\theta c}$ and $Re_{\theta t}$ through empirical correlation, $P_{\gamma 2}$ is the destruction or re-laminarisation source term, and C_{e1} and C_{e2} are constants with values of 2 and 1, respectively.

$$\frac{\partial}{\partial \mathbf{X}_j}(\rho \mathbf{U}_j \gamma) = P_{\gamma 1}(1 - C_{e1}\gamma) + P_{\gamma 2}(1 - C_{e2}\gamma) + \frac{\partial}{\partial \mathbf{X}_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\gamma} \right) \frac{\partial \gamma}{\partial \mathbf{X}_j} \right] \quad (7.18)$$

The final transport equation solves for the transition momentum thickness Reynolds number, such that

$$\frac{\partial}{\partial \mathbf{X}_j}(\rho \mathbf{U}_j Re_{\theta t}) = P_{\theta 1} + \frac{\partial}{\partial \mathbf{X}_j} \left[\sigma_{\theta t}(\mu + \mu_t) \frac{\partial Re_{\theta t}}{\partial \mathbf{X}_j} \right] \quad (7.19)$$

where $P_{\theta 1}$ is the source term, again controlled through empirical correlation, and $\sigma_{\theta t} = 2$. Further information on each transport equation can be found in the Ansys Fluent 2023 R2 manual [166].

7.3.5. Boundary Conditions

The boundary conditions imposed on the computational domain are summarised in Figure 7.2. Fluid enters the domain at the X_{inlet} plane at 22°C through the application of a velocity inlet boundary condition, which develops along the jet inlet channel before eventually entering the region $ABCD$. A no-slip, adiabatic wall boundary condition is applied to the vertical walls along the AB plane, as well as the horizontal boundary along the BC plane, to correspond with the smooth enclosure roof. The use of a pressure outlet along the BC plane was also trialled to investigate the effect of the enclosure roof absence. A pressure outlet boundary condition is also created along the CD plane, with backflow enabled, where an outlet gauge pressure of 0 Pa is specified to suggest ambient conditions. Finally, a no-slip wall boundary with a uniform heat flux of 1670 W/m^2 is applied to the horizontal AD plane to simulate an electrically heated test foil spanning the total solid wall boundary, as this allows for further investigation of the heat transfer characteristics along the solid wall beyond the limitations of the experimental setup.

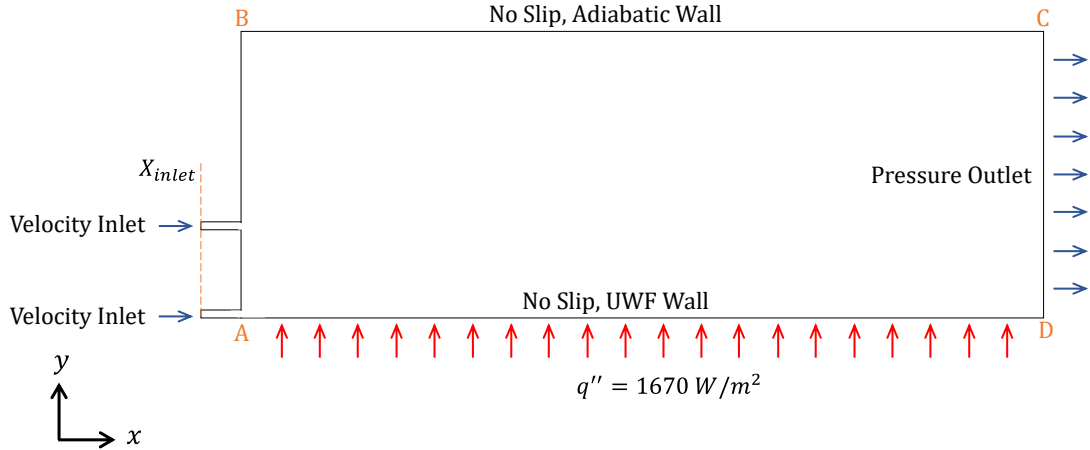


Figure 7.3: Boundary conditions imposed on numerical model

The velocity inlet boundary condition for each jet is defined based on the inlet velocity profile, the turbulent intensity, the turbulent viscosity ratio, and the intermittency value. The specification of the turbulence viscosity ratio, *i.e.*, the ratio of the turbulent viscosity to the fluid viscosity μ_t/μ , is recommended when using the transition SST model, as opposed to the turbulent length scale, to control the rate of turbulence decay from the flow inlet and ensure the desired turbulent intensity is acquired in the region of interest [166]. A low turbulent viscosity ratio in the range $1 \leq \mu_t/\mu \leq 10$ is recommended in the Ansys Fluent 2023 R2 manual, as the accuracy of the transition SST model cannot be ensured for μ_t/μ values outside of this range [166]. The results

of the numerical model were found to be sensitive to all velocity inlet parameters, and careful tuning was therefore required to adapt the inlet conditions to suit the experimental dual jet flow configuration. Further information on the tuning procedure is discussed in section 8.1.3, along with details of the velocity inlet profile.

7.3.6. Mesh Design

Due to the shape of the computational domain, it is possible to adopt a highly structured, quadrilateral mesh type, a sample of which is shown in Figure 7.4. To improve the mesh efficiency, the mesh cells are biased towards the jet inlet and the solid wall boundary, resulting a finer mesh in the upstream, near-wall region and coarser mesh elements downstream and near the enclosure roof, where the flow behaviour is of less importance. In doing so, a larger number of elements are concentrated in the expected jet interaction region, allowing for large velocity and pressure gradients and complex flow structures to be captured. In addition, a maximum first layer height of $3 \times 10^{-5} \text{ m}$ is enforced at the solid wall boundary to ensure $y^+ \leq 1$ inside the anticipated transition region for all Re investigated, while Δx_1 is restricted to a maximum value of $5 \times 10^{-4} \text{ m}$ in this region. Finally, the expansion ratio is limited to $ER \leq 1.1$ as per the recommendation for the transition SST model. To ensure the suitability of the final mesh design for the numerical investigation, a mesh verification and validation study is carried out using the procedure described in section 7.1.4, which resulted in a final mesh design consisting of 1031000 elements.

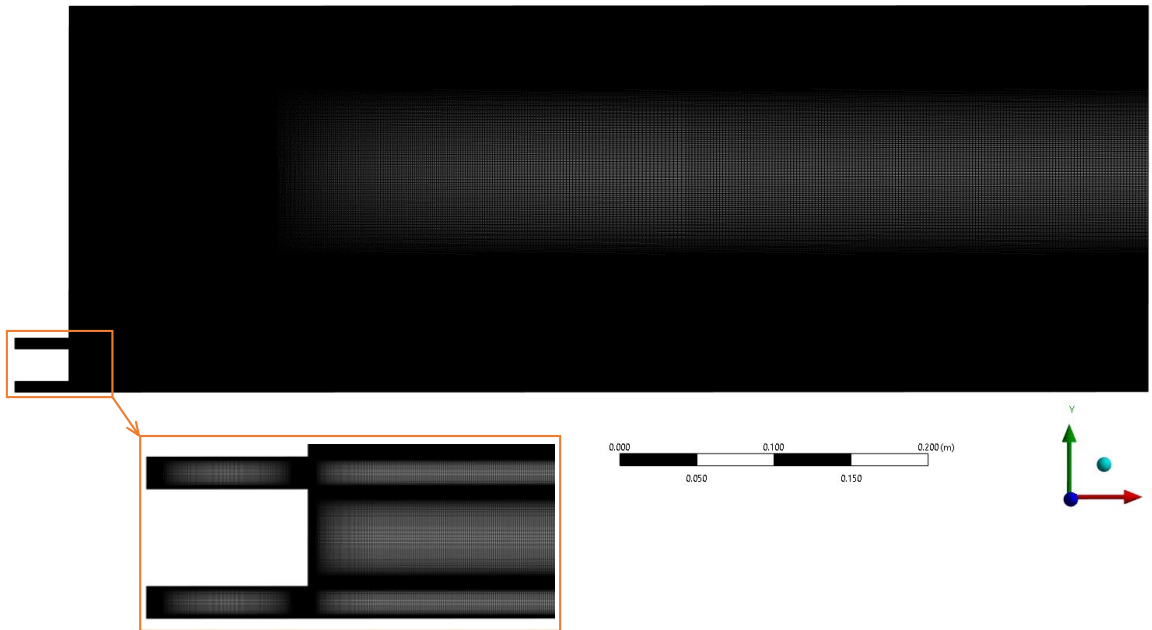


Figure 7.4: Final mesh design for numerical dual jet study consisting of 1031000 cells.

7.3.7. Discretisation Scheme

The discretisation scheme refers to the manner by which the flow quantities at cell interface locations are interpolated from cell centroid values. For Ansys Fluent 2023 R2, the interpolation of the diffusive term for each quantity of interest is achieved through central-differencing to a second order accuracy. The convective terms use an upwind interpolation scheme, where cell interface quantities are only derived from upstream cell centroid values, and the order of accuracy is user defined [166]. The first order upwind scheme is the simplest option, whereby the flow quantity at the interface of interest is equated to the upstream cell centroid value, producing a stable solution of limited accuracy. The use of a second order upwind scheme is generally the recommended option, where the value of the flow quantity at the interface location of interest is extrapolated

based on the cell centroid values of two cells upstream. In doing so, flow quantity gradients are more accurately captured, which is of particular importance for more complex flows, however, there is a risk of generating an unbounded or diverging solution.

Ansys Fluent 2023 R2 offers a first-to-higher-order blending function, which allows the user to solve for an initial converged solution using a first order scheme and subsequently improve the solution accuracy by means of a higher order scheme. This is achieved using a blending factor, the value of which can be varied by the user from 0 to 1, where a value of 0 implies a fully first order discretisation scheme and 1 ensures a solution accuracy to the order of choice [172]. For the present investigation, it was decided to therefore implement a first-to-second order blending for the discretisation scheme, which improves the convergence of the solution while maintaining the required level of accuracy. While higher order discretisation schemes are available in the software, their use did not induce a justifiable improvement in the solution accuracy compared to the increased computational effort, hence the use of second order upwind scheme is deemed sufficiently accurate. In a similar manner, the second order scheme is selected for the pressure interpolation scheme due to the sufficiently high level of accuracy attained in the final pressure field solution with respect to the required CPU time.

7.3.8. Pressure-Velocity Coupling

Ansys Fluent 2023 R2 provides the option of five pressure-velocity coupling algorithms, namely the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm, the SIMPLE Consistent (SIMPLEC) algorithm, the Pressure Implicit with Splitting of Operators (PISO) algorithm, the Fractional Step Method (FSM), and the Coupled algorithm. As the FSM is only recommended for time-dependent flows using non-iterative time advancement, it is not considered for the present investigation.

The SIMPLE, SIMPLEC, and PISO algorithms are classified as segregated algorithms, implying the pressure and velocity solutions are updated sequentially by means of an initial predictor step followed by the necessary correction [166]. Conversely, the Coupled algorithm solves for the pressure and velocity solutions simultaneously, which can be advantageous for simulations of poor mesh quality or transient flows with a large timestep requirement. While the Coupled solver can offer a more robust solution and fewer iterations until convergence, the segregated solvers generally require less computational time per iteration, which may lead to faster simulation times depending on the flow configuration, especially for simulations with a small time-step requirement [172]. According to the Ansys Fluent 2023 R2 user guide [172], the SIMPLEC method can lead to faster convergence compared to the SIMPLE algorithm for basic steady-state flows due to the increased under-relaxation of the pressure correction term, however, this difference can become negligible for more complex flows involving turbulence. Finally, the PISO model is recommended for transient flows with a larger timestep, where its increased efficiency can decrease the number of iterations to convergence, but can, however, induce an increased computational expense when used in conjunction with too small a timestep. For the present computational model, when the performance of each of the available pressure-velocity algorithms were initially compared, the Coupled algorithm was identified as the most suitable option, as it successfully acquired monotonic residual convergence inside the fastest computational time.

7.3.9. Turbulent Prandtl Number

The diffusive term in the energy conservation equation, *i.e.*, equation 7.15, is dependent on the effective conductivity value, k_{eff} , the purpose of which is to account for the apparent increase in the fluid thermal conductivity due to turbulent motion [182]. The effective conductivity is therefore equated to the sum of the fluid conductivity and the turbulent conductivity, such that $k_{eff} = k + k_t$, and the turbulent conductivity can hence be related to the turbulent viscosity by means of a turbulent Prandtl number, *i.e.*,

$$Pr_t = \frac{c_p \mu_t}{k_t} \quad (7.20)$$

where c_p is the specific heat capacity of the working fluid. The turbulent Prandtl number hence represents the ratio of the turbulent diffusive transport of heat and momentum and is of particular importance in the near-wall region when assessing local heat transfer behaviour [182]. In a study carried out by Alimohammadi [183], it was noted that the value of Pr_t directly affected the heat transfer results when simulating an impinging jet flow and was therefore tuned accordingly.

In Ansys Fluent 2023 R2, the default value of Pr_t is 0.85 but can be tailored to suit different flow configurations. The value of Pr_t is generally assumed to be close to unity, where the use of $Pr_t = 0.9$ is common as it appears to complement most flow types [182]. With wall bounded flows, however, Pr_t can vary throughout the wall boundary layer, where the greatest value of Pr_t is expected at the wall boundary, decreasing close to the outer edge of the boundary layer. Multiple empirical correlations have been proposed, where Pr_t was found to vary with Re , Pr , and distance from the wall, however the suggested value of Pr_t varies greatly across the published literature [183]. In a review conducted by Chambers *et al.* [184], it was noted that the turbulent Prandtl number in the fully turbulent region in shear flows varied in the range $0.5 \leq Pr_t \leq 1$ across the published literature, however, Divoky and Mayer [185] observed a slightly lower range of values of $0.42 \leq Pr_t \leq 0.83$. Conversely, a value close to 1 is generally expected in the log layer of the boundary layer, *e.g.*, Launder [186]. For the present investigation, as a simple solution for Pr_t does not exist, it is necessary to tune the input Pr_t with respect to the experimental heat transfer data, where the turbulent Prandtl number is varied inside the limits $0.3 \leq Pr_t \leq 1$. Further details of the tuning procedure are provided in section 8.1.4.

7.3.10. Solution Approach

The numerical flow field for each jet configuration is solved using the steady-state solver to obtain an approximation of time-averaged flow field quantities. The computational domain is initialised based on the jet inlet parameters and the blend factor is initially set to 0 to acquire an approximate flow field solution for a first order upwind scheme. The blend factor is hence increased to 0.5 and, finally, 1, obtaining a converged solution each time, with the final converged solution to second order accuracy. Solution convergence is primarily observed through the residual output, where a convergence criterion of 1×10^{-6} is defined for all residual values except energy, which requires a convergence criterion of 1×10^{-9} . In addition, the maximum wall temperature is monitored to ensure the heat transfer characteristics reach a steady-state condition, as well as the maximum turbulent viscosity ratio as per the Ansys Fluent 2023 R2 recommendation for the convergence of turbulent flows.

Chapter 8

Numerical Model Results

This chapter presents the output from various dual jet numerical models and assesses their performance with respect to the experimental data. The procedure by which the model input parameters were tuned is initially discussed for one jet configuration, *i.e.*, $Re = 5500$ and $OR = 3$ for $V_r = 1$, followed by the model verification and validation study to evaluate the model accuracy and determine the grid uncertainty value. The numerical data is hence compared to the experimental data over a wider range of jet parameters, from which the credibility of the model when predicting the flow and heat transfer characteristics of a dual jet is assessed.

8.1. Tuning of Model Parameters

To ensure the results from the numerical model are representative of the experimental findings for dual jet flow, a certain degree of tuning is required. The tuning procedure is discussed in the following sections, where focus is given to a single test case of $Re = 5500$ and $OR = 3$ for $V_r = 1$, where the model sensitivity to various input parameters is analysed for the purpose of optimisation. The tuning procedure is discussed with respect to the choice of turbulence model, the discretisation scheme, the inlet boundary conditions and the turbulent Prandtl number. Although the reasoning behind certain choices has already been discussed in 0, their respective influences are presented here for completeness.

As the experimental heat transfer results from the present study are distinctly different from those reported previously in published numerical studies, the primary aim of this numerical study is to show that this heat transfer behaviour can be accurately simulated for a dual jet flow field using established CFD methods. The model is therefore tuned based on the local Nusselt number profile induced in the solid wall boundary, where it is assumed that if the heat transfer characteristics are accurately replicated, the corresponding flow characteristics should also be representative of the experimental flow behaviour.

8.1.1. Turbulence Model

As previously discussed in section 7.3.3, the transition SST model was assumed to be the most appropriate turbulence model when simulating dual jet flows for the chosen Re , OR , and V_r ranges due to its ability to predict the laminar-to-turbulent transition in the wall boundary layer. This is of particular importance in the event of separated flow from the solid boundary as the onset of separation depends largely on the flow regime inside the boundary layer, where separation generally occurs sooner in the case of a laminar boundary layer but is delayed to a further downstream location in the case of a more turbulent boundary layer. The

experimental data indicates a definite separation of the wall jet from the solid wall boundary for most dual jet flow configurations investigated. Since this undoubtedly influences the resulting heat transfer characteristics, it is clear that a transitional turbulence model is required.

It should be noted, however, that the effectiveness of a transitional model is somewhat limited as the onset of turbulence production is ultimately predicted based on empirical correlation. In Ansys 2023R2, the empirical correlations implemented in the Transition SST turbulence model are modified from Menter *et al.* [187], who notes that the transport equations do not attempt to model the exact physics of the transition process but instead try to capture these processes through empirical correlation. Therefore, so long as an appropriate correlation is available, all mechanisms of transition can theoretically be modelled for all geometries. As such, the empirical correlations implemented in Ansys 2023R2 can only suit a specific set of transition modes and geometries, *i.e.*, standard by-pass transition with a minor correction to account for separation induced transition, and further user inputted empirical correlations are required to adapt the model to suit flow regimes which fall outside this. For the present investigation, it is understood that the dual jet flow setup falls broadly within this category, however, the accuracy of the solution could potentially be improved through generating an empirical correlation that could better replicate the actual laminar-to-turbulent transition inside the dual jet flow field. However, as this experimental data is not available, it is necessary to implement the Transition SST turbulence model with the default correlations for the present investigation.

To ensure the suitability of the transition SST model, the performance of the model with respect to the experimental data is compared to that of other ‘out-of-the-box’ turbulence models in Figure 8.1, all of which have been previously used in conjunction with dual jet flows in the published literature. It must be noted, however, that the results of the standard $k - \epsilon$ model are omitted from Figure 8.1 as the use of this model failed to provide an adequately converged solution for the present numerical setup.

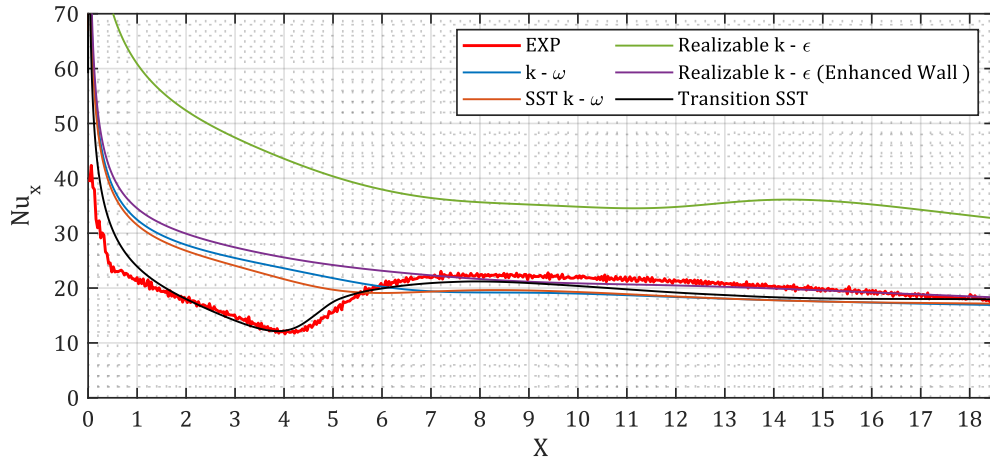


Figure 8.1: Comparison of the performance of various turbulence models when used to simulate the experimental Nu_x profile for a dual jet with $Re = 5500$ and $OR = 3$.

From the data presented in Figure 8.1, it is clear that all models which assume a fully turbulent boundary layer cannot replicate the initial dip in the experimental Nu_x profile, however, a closer agreement is observed further downstream for the majority of cases. The realizable $k - \epsilon$ model appears to grossly exaggerate Nu_x across the entire downstream domain, which is to be expected as $k - \epsilon$ based models perform poorly in the boundary layer regions. Nevertheless, this model has been used previously by Li *et al.* [92] to simulate a dual jet flow field despite being clearly inadequate. The use of the enhanced wall treatment offers a significant improvement

in the far downstream region, though the steady decay of Nu_x with streamwise distance still fails to capture the existence of the local minimum point near the jet exit. The results produced using the $k - \omega$ and SST $k - \omega$ models are relatively similar, where a consistent streamwise decay of Nu_x is evident, however neither provide an accurate representation of the experimental results. As the only model which successfully captures initial dip in the Nu_x profile is the transition SST model, it seems at this point that it would be the most appropriate turbulence model to select when simulating a dual jet flow. Its effectiveness, however, relies heavily on the chosen inlet conditions, the optimisation of which are discussed in detail in section 8.1.3.

8.1.2. Discretisation Scheme

As discussed in chapter 7.3.7, as highly turbulent jet flows generally face convergence issues, a first-to-second order blending function is adopted for the present numerical study, which improves the convergence of the solution while maintaining the required level of accuracy. This is achieved through the use of a blending factor, which is initially set to a value of 0 to acquire a converged solution to first order accuracy. This blending factor is hence increased to 0.5, *i.e.*, a blend of first and second order accuracy, and finally 1 to achieve a fully converged second order accurate solution, where the use of the intermediate solution for a blend factor of 0.5 improves convergence through reducing the step change in the solution. For the purpose of completeness, the outputs from the numerical model using blend factors of 0, 0.5 and 1 are shown in Figure 8.2, alongside the relevant experimental Nu_x profile, where it is clear that increasing the blend factor enables the model to incrementally approach the experimental values. While discrepancies are still present, the second order accurate solution provides the closest alignment with the experimental data and reducing the order of accuracy increases the deviation of the numerical data from the experimental findings, particularly in the region downstream of $Nu_{x_{min}}$.

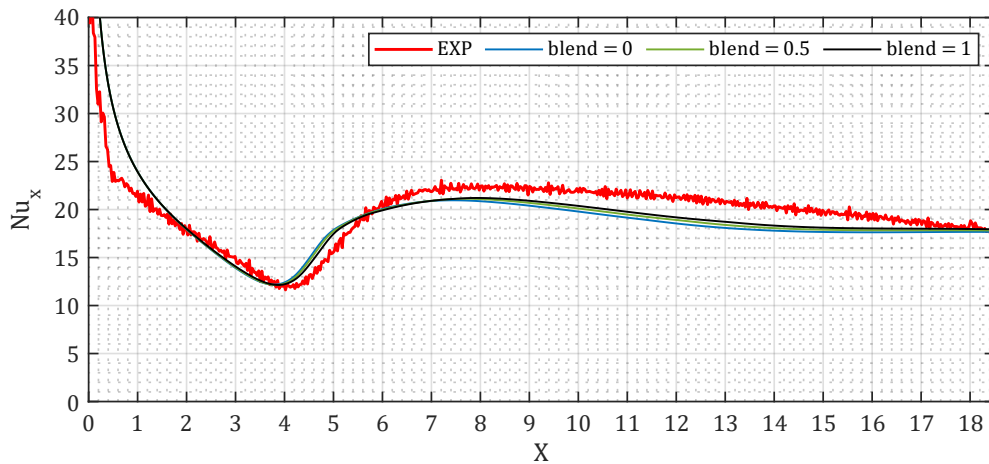


Figure 8.2: Comparison of the output from the numerical model using a blend factor of 0, 0.5, and 1 when used to simulate the experimental Nu_x profile for a dual jet with $Re = 5500$ and $OR = 3$.

8.1.3. Inlet Boundary Conditions

The output from the numerical model was found to be extremely sensitive to the inlet conditions, where the values chosen for each inlet parameter were found to directly affect the extent of the wall jet separation, which included the inlet velocity profile, the turbulence intensity, the viscosity ratio, and the intermittency value. The model tuning procedure therefore focussed primarily on identifying appropriate values for each inlet parameter, which could require altering depending on OR or Re . The following sections discuss the effect of varying each

parameter with respect to the experimental data and the identification of the most suitable set of parameter values for the dual jet configuration in question.

8.1.3.1. Inlet Velocity Profile

To obtain a more accurate representation of the inlet velocity profile as per the experimental setup, a separate numerical model was generated to better replicate the flow conditions at the outlet of the contraction inside the airflow conditioning chamber. The model domain is presented in Figure 8.3, the dimensions of which correspond to the dimensions of the airflow conditioning chamber downstream of the final mesh screen. A uniform velocity inlet was specified at the entrance to the flow domain, where the velocity in this region was assumed to equate to $u_{jet}/10$, *i.e.*, the contraction ratio, and a pressure outlet to ambient was selected at the exit. The flow domain was meshed using tetrahedral mesh consisting of 1954996 cells to best capture the curvature of the contraction wall, and an inflation layer of $\Delta y_1 = 8.6 \times 10^{-7}$ and expansion ratio of $ER = 1.1$ was created at the solid boundary. For simplicity, the energy equation was neglected and constant fluid properties corresponding to air at 22°C were implemented. To remain consistent with the dual jet numerical model, the transition SST turbulence model was adopted and the final results were obtained using the steady solver.

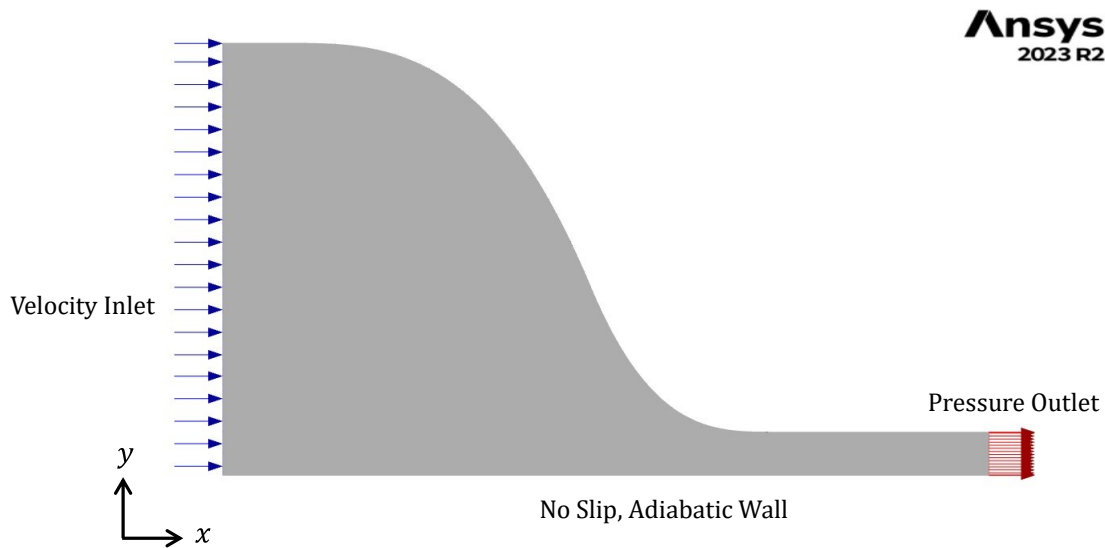


Figure 8.3: Computational domain to obtain the outlet velocity profile from the contraction inside the airflow conditioning chamber of the dual jet experiment.

A sample velocity profile obtained at the nozzle exit for $Re = 12000$ is shown in Figure 8.4(a), which appears relatively flat with evidence of boundary layer growth at the extremities, but somewhat asymmetric as a result of the one-sided contraction. The corresponding turbulence intensity profile is presented in Figure 8.4(b), the maximum value for which exists just offset from the centre of the channel and tends towards 0 at both solid boundaries. As the inlet to the dual jet flow domain corresponds to a location 35 mm upstream of the nozzle exit, the velocity and turbulence intensity profiles were also extracted at this location, as shown in Figure 8.4, and could hence be applied directly to the inlet of the dual jet flow domain. However, as the turbulence conditions at the inlet to the contraction were unknown, in addition to the fact that the maximum turbulence intensity at the nozzle exit appeared very low in comparison to the experimental findings, it was decided not to adopt the simulated turbulence intensity profile but instead apply a uniform turbulence intensity at the inlet to the dual jet flow domain, the value of which was tuned as described in section 0. As such, only the simulated velocity profile acquired 35 mm upstream of the nozzle exit was applied to the inlet of the dual jet flow domain.

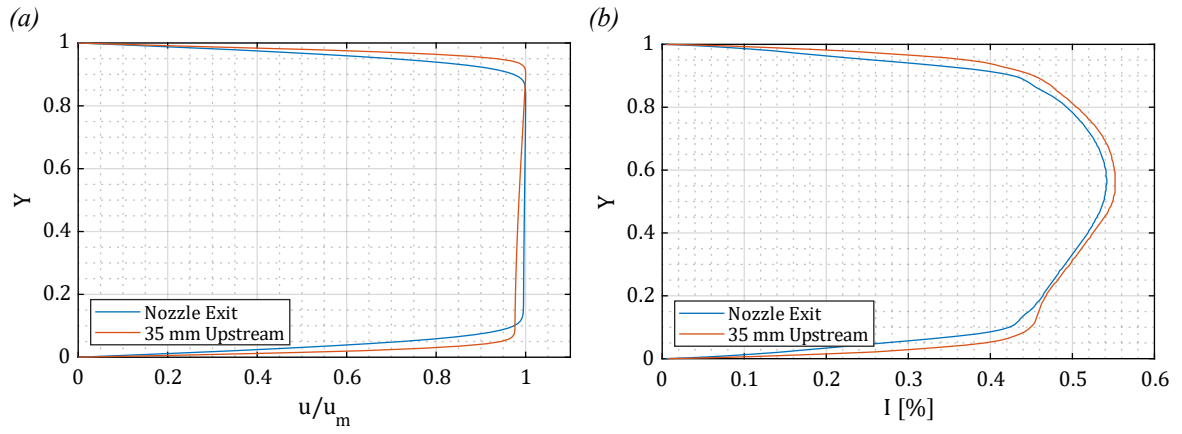


Figure 8.4: (a) Velocity profile and (b) turbulence intensity profile at the nozzle exit and at a location 35 mm upstream of the nozzle exit as per the standalone contraction simulation.

A comparison of the Nu_x profiles acquired using a uniform velocity at the inlet of the dual jet flow domain and applying the velocity profile produced using the separate contraction simulation are presented in Figure 8.5, alongside the relevant experimental dataset. In doing so, a close alignment is observed between both Nu_x profiles, which indicates that the residual flow behaviours arising from the use of the contraction in advance of the nozzle exit are minimal. Slight differences are evident downstream of $Nu_{x_{min}}$, where the use of a uniform inlet profile appears to better replicate the rate of increase of Nu_x in advance of $Nu_{x_{max}}$, however, a greater underprediction of Nu_x is observed over the remaining downstream distance. Conversely, while the use of the velocity profile from the contraction outlet induces a sharper rise in Nu_x downstream of the local minimum, the comparative enhancement in Nu_x over the remaining downstream distance provides a closer alignment with the experimental data in this region.

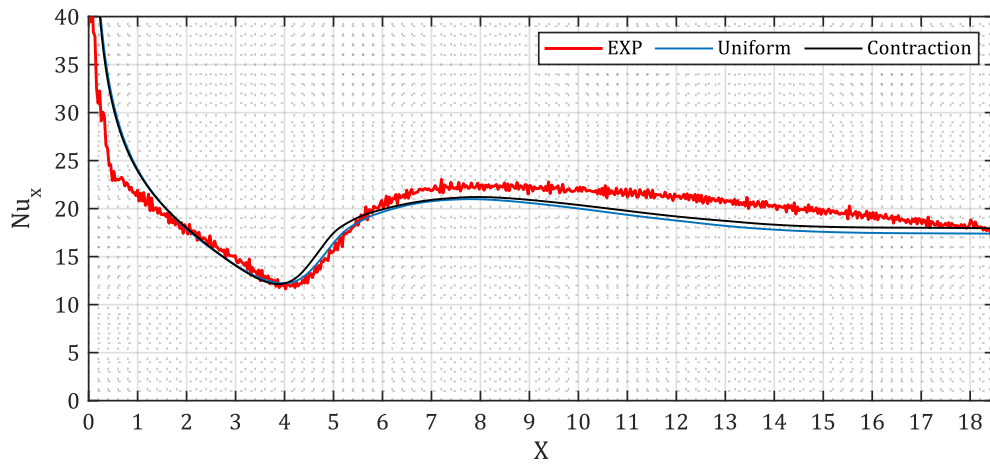


Figure 8.5: Comparison of the Nu_x profile output from the numerical model for both a uniform velocity inlet profile and the velocity inlet profile obtained using the airflow contraction simulation, for a dual jet with $Re = 5500$ and $OR = 3$.

The differences between the respective Nu_x profiles can be attributed to the changes in the wall boundary layer at the point at which the wall jet separation occurs, where the boundary layer in the case of the contraction outlet velocity profile would be further developed at this position due to its start point being located further upstream. As a result, the slightly greater turbulence levels inside the wall boundary layer enable the wall jet to reattach to the solid wall boundary marginally sooner, thus inducing a sharper rise in Nu_x in this region, while also maintaining a higher rate of heat transfer over the remaining downstream distance.

While the differences between the Nu_x profiles obtained using both inlet velocity profiles are minimal, the choice of the most appropriate inlet velocity profile is somewhat arbitrary in the case of a dual jet with $Re = 5500$ and $OR = 3$. Having said this, the difference between the respective Nu_x profiles was found to be exacerbated at higher Re values, where the use of a uniform inlet velocity profile became increasingly insufficient with increasing Re , while that derived from the outlet of the contraction performed reasonably over the entire Re range. For that reason, the contraction outlet velocity profile was selected as the most appropriate for the present dual jet numerical study and was thus implemented for all remaining dual jet configurations to ensure consistency.

8.1.3.2. Turbulence Intensity

The effect of varying the turbulence intensity (I) at the velocity inlet on the numerical Nu_x profile is presented in Figure 8.6 for $3\% \leq I \leq 7\%$, where it is clear that the turbulence intensity directly controls the extent of the local minimum and local maximum in the Nu_x profile. For lower turbulence intensity values, e.g., $I = 3\%$, the wall jet can be pulled from the solid boundary with greater ease, meaning the deflection of the wall jet near the nozzle exit is much greater, thus increasing the thickness of separation region. This results in a sharper decline in Nu_x from the jet exit and reduces the value of $Nu_{x_{min}}$ where the wall jet separation is greatest. However, as the streamwise positions of $Nu_{x_{min}}$ and $Nu_{x_{max}}$ are constant across all I , it suggests that the length of the separation region remains consistent with increasing thickness. This subsequently enables the jet to re-impinge the solid boundary from a more direct angle, thus enhancing the value of $Nu_{x_{max}}$ and inducing the characteristic ‘peaking’ effect. However, this is not consistent with the experimental data for $OR = 3$, where peaking was only observed for lower OR flows.

Increasing the turbulence intensity reduces the deflection of the wall jet, which leads to a less steep decline in Nu_x from the nozzle exit and increases the value of $Nu_{x_{min}}$ in an incremental fashion. As a higher turbulence intensity reduces the thickness of the separation region, the flow can re-impinge on the solid boundary from a direction that becomes increasingly aligned with the streamwise dimension, thereby dampening the peaking effect at $Nu_{x_{max}}$. This reduction in value of $Nu_{x_{max}}$ is only apparent, however, for the lower values of turbulence intensity which already induce peaking, whereas, for higher I values, $Nu_{x_{max}}$ remains seemingly unaffected for increasing I . The data presented in Figure 8.6 suggests that this is the case for $I \geq 6\%$ for a dual jet with $Re = 5500$ and $OR = 3$.

Based on the data shown, it is clear that $I = 6\%$ provides the closest alignment with the experimental data for $Re = 5500$ and $OR = 3$. The most appropriate turbulence intensity value did not remain consistent across all dual jet parameters considered, where a lower value was generally required for higher Re values. This finding was considered reasonable, however, as the magnitude of fluctuations present in the flow should not necessarily scale linearly with increasing flow velocity. In addition, a reduction in OR generally required a lower turbulence intensity, where it is postulated that local fluctuations in the flow may be slightly suppressed due to the closer jet proximity. A summary of the final turbulence intensity values for each jet configuration considered is presented in Table 8.1.

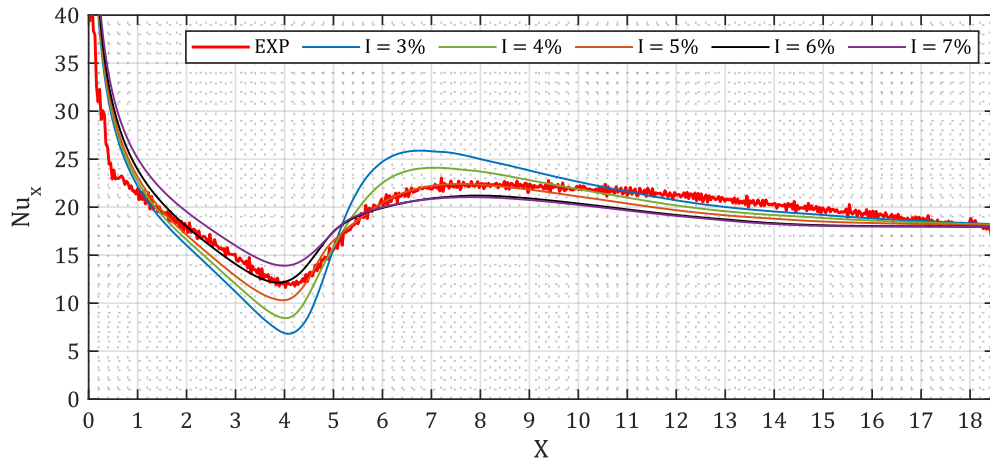


Figure 8.6: Effect of varying the turbulence intensity at the velocity inlet on the Nu_x profile output from the dual jet numerical model for $Re = 5500$ and $OR = 3$.

8.1.3.3. Turbulent Viscosity Ratio

The effect of varying the turbulent viscosity ratio is presented in Figure 8.7 for $1 \leq \mu_t/\mu \leq 5$. Varying this ratio essentially varies the rate of decay of turbulence intensity from the velocity inlet. As stated in the Ansys Theory Guide [166], increasing the turbulent viscosity ratio generally decreases the rate of decay of turbulence intensity, which appears to be the case in Figure 8.7. As μ_t/μ is increased, the rate of decline of Nu_x from the nozzle exit is reduced, which results in an increase in $Nu_{x_{min}}$. This is consistent with increased turbulence levels in this region, which, in a similar manner to increasing the turbulence intensity, acts to reduce the separation of the wall jet from the solid wall boundary. Moreover, the position of $Nu_{x_{min}}$ moves slightly downstream with increasing μ_t/μ which insinuates the delayed onset of wall jet separation. The value of $Nu_{x_{max}}$ appears to remain relatively unaffected by changing μ_t/μ , as well as values of Nu_x in the far downstream region. This can be attributed to the fact that as μ_t/μ is increased, the direction at which the jet re-impinges on the solid boundary ultimately remains consistent as the separation region experiences a reduction in both thickness and length.

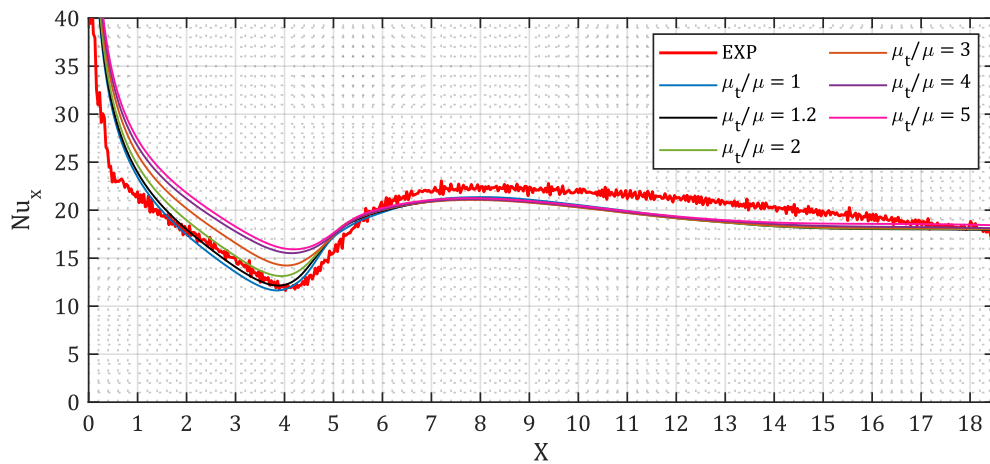


Figure 8.7: Effect of varying the turbulent viscosity ratio at the velocity inlet on the Nu_x profile for $Re = 5500$ and $OR = 3$.

From the data presented in Figure 8.7, a turbulent viscosity ratio of $\mu_t/\mu = 1.2$ appears to provide the closest alignment with the experimental data for $Re = 5500$ and $OR = 3$. Similar to the turbulence intensity value, the most suitable value for μ_t/μ tended to vary with both Re and OR , where a higher Re value generally

required a lower value of μ_t/μ for $OR = 3$, while the opposite was observed for $OR = 1$. The full set of turbulent viscosity ratios for each jet configuration considered is presented in Table 8.1.

8.1.3.4. Intermittency

The effect of varying the intermittency at the velocity inlet is presented in Figure 8.8, where the extreme values of 0 and 1 are presented only. By default, the intermittency is set to a value of 1 in Ansys 2023 and decreasing the intermittency to a value to 0 did not appear to offer a particular advantage for the present numerical study. As a reduction in γ acted to worsen the alignment between the numerical and experimental datasets through a reduction of the wall jet separation, it was decided to maintain an intermittency value of $\gamma = 1$ at the velocity inlet across all dual jet configurations considered.

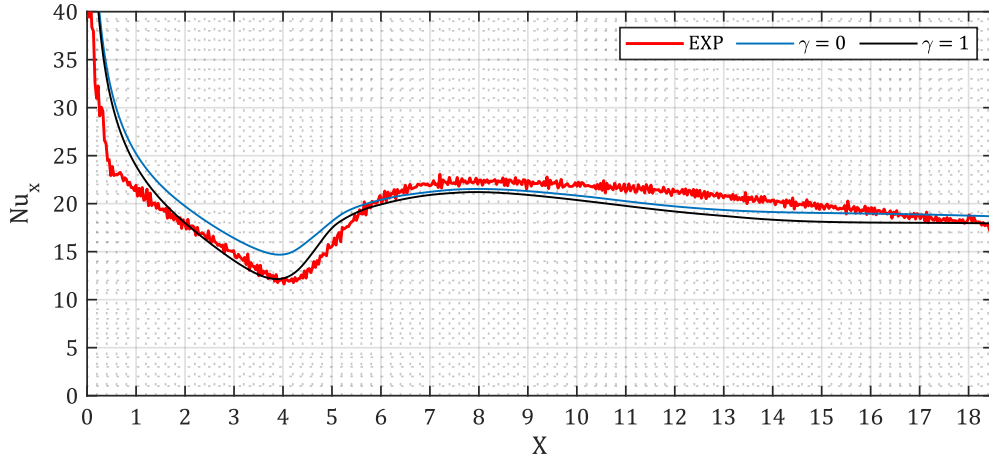


Figure 8.8: Effect of reducing the intermittency value at the velocity inlet on the Nu_x profile for $Re = 5500$ and $OR = 3$.

8.1.4. Turbulent Prandtl Number

Another parameter found to directly influence the output of the numerical model is the turbulent Prandtl number (Pr_t). As previously discussed in section 7.3.9, Pr_t is indicative of the ratio of momentum diffusivity to thermal diffusivity in the flow due to the presence of turbulent eddies. The effect of varying Pr_t on the Nu_x profile is displayed in Figure 8.9 for $0.36 \leq Pr_t \leq 0.6$, where it is clear that its influence is primarily realised in the far downstream region. This can be attributed to the higher level of turbulence in this region, whereas the turbulence intensity in advance of the separation region is significantly lower, meaning an enhancement in the heat transfer due to the presence of turbulent eddies is minimal. Downstream of the separation point, however, the flow sees a sudden injection of turbulence as the boundary layer reattaches and the jets begin to interact, meaning turbulent eddies play a more prominent role and thus bring about an associated enhancement in the heat transfer.

In the far downstream region, a decrease in Pr_t induces a proportional enhancement in Nu_x across this entire region, which is logical considering it corresponds to an increase in the turbulent thermal diffusivity while the turbulent kinematic viscosity remains consistent. For a dual jet with $Re = 5500$ and $OR = 3$, it appears that a turbulent Prandtl number of $Pr_t = 0.36$ provides the closest alignment with the experimental data, which is slightly less than the expected range for a plane jet of $0.4 \leq Pr_t \leq 1$. However, considering a dual jet experiences considerably higher levels of turbulence compared to a traditional plane jet, a value of Pr_t lower than 0.4 does not seem particularly unreasonable. The tuning of Pr_t again varied depending on Re and OR ,

which has been noted by previous studies as discussed in section 7.3.9, and a summary of the values obtained for each set of dual jet parameters can be found in Table 8.1. In general, a higher Reynolds required a higher Pr_t , which could possibly be attributed to the lower values of ν_t at higher Re , where an increase in Pr_t is required to maintain α_t at a somewhat constant value.

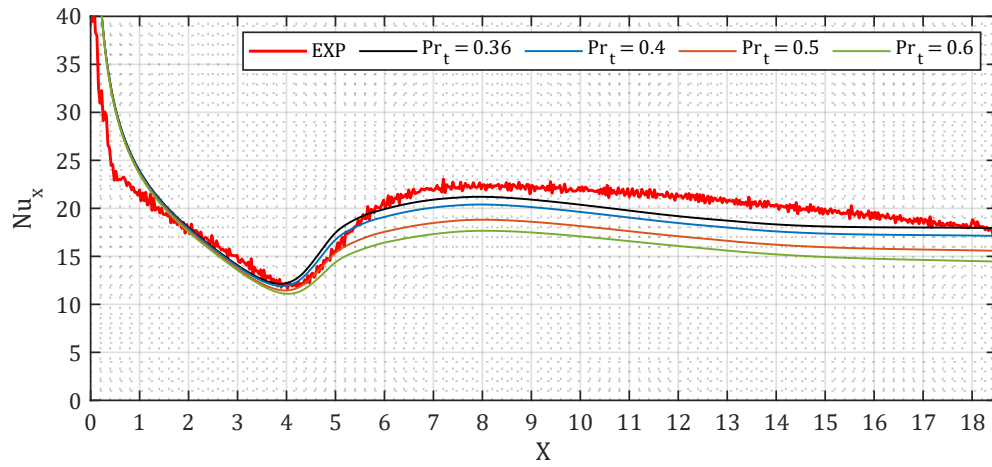


Figure 8.9: Effect of varying Pr_t on the Nu_x profile output for $Re = 5500$ and $OR = 3$.

8.1.5. Summary of Tuning Parameters

The full set of tuning parameters for each test case considered as part of the numerical investigation can be found in Table 8.1, *i.e.*, $Re = 5500, 9000$, and 12000 for $OR = 1$ and 3 , while $V_r = 1$ throughout.

Table 8.1: Summary of final set of tuning parameters for each dual jet configuration used as part of the numerical study.

Re	OR	I	μ_t/μ	Pr_t
5500	1	2.5%	1	0.33
9000	1	1.8%	2	0.42
12000	1	1.5%	5	0.55
5500	3	6%	1.2	0.36
9000	3	5%	1.1	0.5
12000	3	4.5%	1	0.65

8.2. Assessment of Model Implementation

To ensure the numerical model(s) provide a credible representation of the flow and heat transfer behaviours observed within the experimental test facility, it is necessary to assess the overall model implementation in terms of the mesh quality, the verification of the numerical output, and the validation of the numerical results. The following sections detail these procedures and their respective findings in full.

8.2.1. Assessment of Mesh Quality

As previously noted in section 7.3.6, a highly structured, quadrilateral mesh consisting of 1,031,000 cells is adopted for the present numerical investigation, where a biasing is implemented to concentrate more cells near the jet nozzle exit and towards the solid wall boundary. This section therefore analyses the quality of the mesh with respect to adhering the limitations imposed by the chosen turbulence model and relevant mesh metrics.

Regarding the turbulence model limitations, the transition SST model requires a first layer height equivalent to a y^+ value of less than or equal to 1 along the solid boundary of interest to place the first cell height within the viscous sublayer, as well as an expansion ratio of no greater than 1.1. These are needed to accurately capture the steep gradients near the wall region and to correctly predict the onset of transition. For similar reasoning, the streamwise dimension of cells in the region where transition is expected to occur should approximately equate to the boundary layer thickness. For the present investigation, a first cell height of $1 \times 10^{-5} \text{ m}$ and expansion ratio of 1.09 are implemented along the solid wall boundary. This ensures a y^+ value of less than 1 across the total downstream domain for the range of Re investigated, as demonstrated in Figure 8.10, placing the first cell comfortably within the viscous sublayer. Further to this, a maximum streamwise cell spacing of $4 \times 10^{-4} \text{ m}$ is implemented upstream of $X = 18.5$, which is comparable to the expected boundary layer thickness in this region and should allow for the onset of transition to be accurately captured should it occur at any point along this flow domain of interest. Therefore, from the perspective the of the turbulence model requirements, the mesh is of a sufficient scale to enable the numerical model to accurately simulate the dual jet flow field. As the Prandtl number of the working fluid, *i.e.*, air, is less than 1, it is therefore implied that the thermal boundary layer thickness should be greater than the hydrodynamic boundary layer thickness at each streamwise location. As such, the implemented mesh refinement at the solid boundary should also adequately capture the steep thermal gradients in this region.

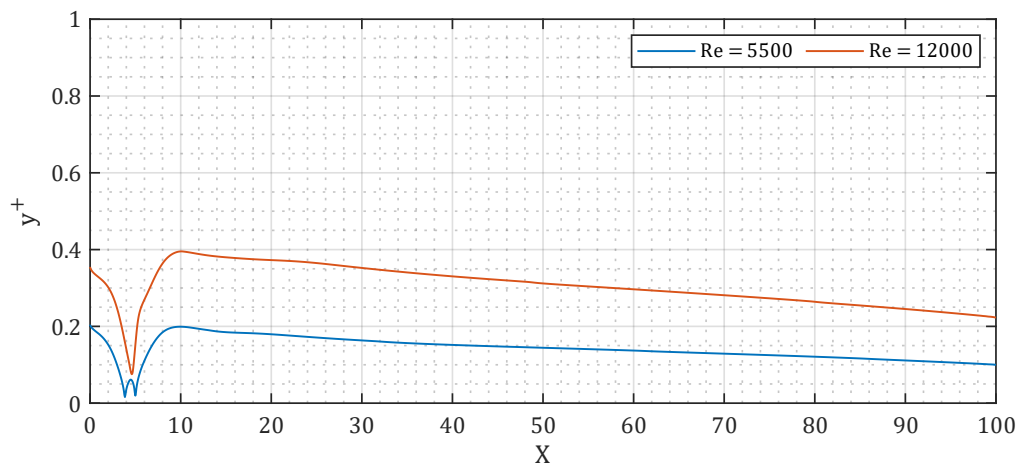


Figure 8.10: Analysis of y^+ along solid wall boundary for $Re = 5500$ and $Re = 12000$ at $OR = 3$.

In terms of the mesh metrics, as a highly structured, quadrilateral mesh is imposed across the entire flow domain, the orthogonality and skewness of the individual mesh cells do not deviate from their ideal values of 1 and 0, respectively. The mesh quality is hence assessed primarily using the aspect ratio, *i.e.*, the ratio of the cell length to the cell height, where values < 5 are preferred in the bulk flow regions or < 10 inside the wall boundary layer [167]. The distribution of the mesh cell aspect ratio values is presented in Figure 8.11, where it is clear the majority of mesh cells are skewed towards the lower end of the spectrum, with 54.06% of cells having an aspect ratio of less than 5 and a further 15.23% of cells possessing an aspect ratio of less than 10. While it is generally preferred to have an aspect ratio of < 5 across 90% of cells, it should be noted most cells in the region of interest, *i.e.*, close to the nozzle exit, do adhere to this expectation. The larger aspect ratio cells primarily exist in the far downstream region, within which the flow is expected to be fully developed but not of particular interest to the present study, therefore enabling the streamwise stretching of cells in this region to reduce unnecessary computational effort. In addition to this, the aspect ratio of many cells adjacent to the solid wall boundary were increased significantly due to the ultra-fine mesh limitations imposed by the transition SST model, which is therefore unavoidable. As increasing the cell density did not produce a justifiable increase in the solution accuracy compared to the increased CPU time, the current mesh was deemed adequate for the present numerical investigation.

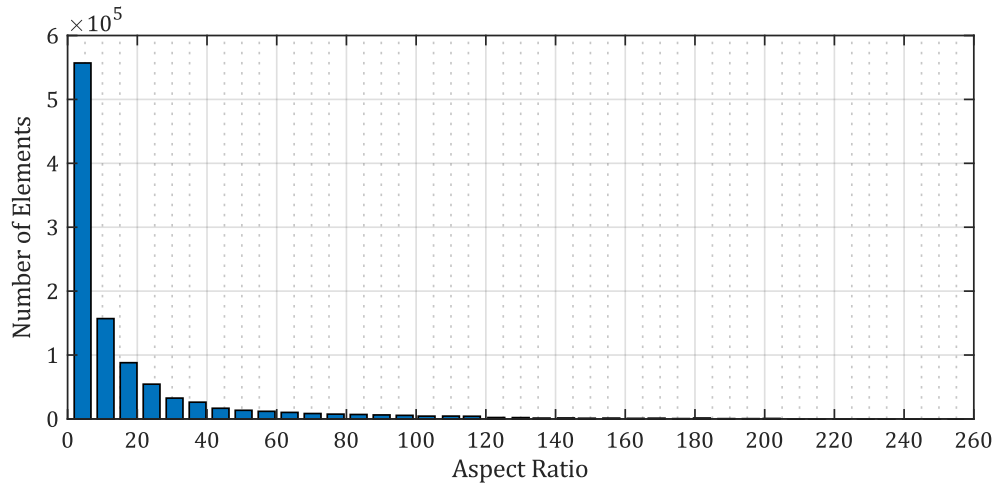


Figure 8.11: Distribution of cell aspect ratios inside the computational flow domain.

8.2.2. Model Verification and Validation

This section describes the numerical model verification and validation procedures used to ensure the credibility of the numerical model with respect to the grid convergence and its associated grid uncertainty, as well as its adherence to physical reality. The data is presented for a single test case of $Re = 5500$ and $OR = 3$ for $V_r = 1$, where the solution verification and validation is based on the local Nusselt number profile in the solid wall boundary sampled at uniform increments of $X = 0.5$.

First, the grid verification study is carried out as per the grid convergence method outlined in section 7.1.4, which involves the production of three incrementally fine meshes and ascertaining whether their respective outputs are converging towards a true solution in an asymptotic manner with increasing cell density. For the present investigation, the coarse, medium, and fine meshes consist of 515,200, 731,350 and 1,031,000 cells, respectively, giving rise to an approximate grid refinement ratio of $\sqrt{2}$. Their respective Nu_x profiles are presented in Figure 8.12(a), where a close alignment can be observed between the successive mesh solutions.

To further examine the convergence of the respective solutions, the full set of verification data is detailed in Table 8.2, where the local Nusselt number at each streamwise sample point is presented for all three meshes, alongside the corresponding grid convergence ratio and an indicator of whether or not convergence is achieved. In doing so, convergence is identified across most sample points, with oscillatory divergence only evident at a single sample point. Though, as the incremental difference in Nu_x between the medium and fine meshes at this diverging point is less than 0.05%, its influence can be considered negligible. Therefore, as the grid convergence ratio across most sample points satisfies $|r_g| \leq 1$, it can be concluded with reasonable certainty that convergence has been achieved using the present numerical model.

Having said this, of the points that have successfully indicated convergence, only half of these points are converging in a monotonic manner, while the remainder exhibit oscillatory convergence. While convergence is still valid at these points, it is not possible to ascertain the grid error or the apparent order of accuracy, p , both of which require monotonic convergence. At the sample points where monotonic convergence is achieved, however, the apparent order of accuracy is always in excess of 2, *i.e.*, the theoretical order of accuracy associated with a second order solver. While these values are much larger than that expected for a second order solver, they do at least indicate that the solution is indeed converging, and the solution produced by the finest mesh is within asymptotic range of the theoretical true solution. Using the generalised Richardson Extrapolation technique followed by the Factor of Safety method, the largest grid uncertainty value is determined at $X = 14$ and equates to $U_{FS} = 0.021$, which is equivalent to 0.1% of Nu_x at that point. It can therefore be concluded with 95% confidence that the numerical solution produced using the finest mesh is within 0.1% of the true solution, which is sufficient for the present investigation, and is thus fully verified. Based on this finding, further mesh iterations of increased mesh density are not required, as it would greatly increase the computational burden without offering any additional insights into the dual jet flow field.

The final stage associated with the assessment of the model implementation is the model validation, which aims to determine whether the simulated flow behaviour aligns with physical reality through comparison with experimental data. The numerical solution is plotted alongside the relevant experimental Nu_x dataset in Figure 8.12(a), where a close alignment between the profiles is clear. The full set of validation data is presented in Table 8.3, where the numerical and experimental Nu_x values are stated for each sample point, *i.e.*, Nu_{x_3} and $Nu_{x_{EXP}}$, respectively, alongside the normalised comparison error E , where $E = (Nu_{x_{EXP}} - Nu_{x_3})/Nu_{x_{EXP}}$. To ascertain the validity of the numerical dataset, the comparison error is compared to the validation uncertainty interval, $\pm U_v$, which consists of the combination of the experimental uncertainty $U_c(Nu_{x_{EXP}})$ and the grid uncertainty U_{FS} using the root sum of squares method at the points where monotonic convergence has been achieved, *i.e.*, $U_v = \sqrt{U_c(Nu_{x_{EXP}})^2 + U_{FS}^2}$, otherwise $U_v = Nu_{x_{EXP}}$. A datapoint can thus be considered validated if E lies within $\pm U_v$.

The full set of normalised comparison error values are presented alongside the normalised validation uncertainty interval in Figure 8.12(b), where the majority of numerical datapoints lie within the acceptable limits of validation. Only two points appear to not meet the requirements of validation, *i.e.*, $X = 0.5$ and $X = 5$, however, both coincide with regions along the solid wall boundary where steep Nu_x gradients were incurred due to the complex nature of the flow field at these locations, meaning a full validation along these regions was expected to be quite difficult. Having said this, the numerical Nu_x profile still appears to quite accurately

simulate the experimental trends in these regions, and, despite not all points acquiring full validation, the general trend is certainly credible. Based on these findings, the numerical model can be considered validated with respect to the experimental data, meaning the simulated flow field aligns with physical reality to an acceptable level of accuracy. As the numerical model satisfies both the verification and validations requirements, it enables the progression of the numerical investigation with a reasonable degree of credibility associated with the simulated results.

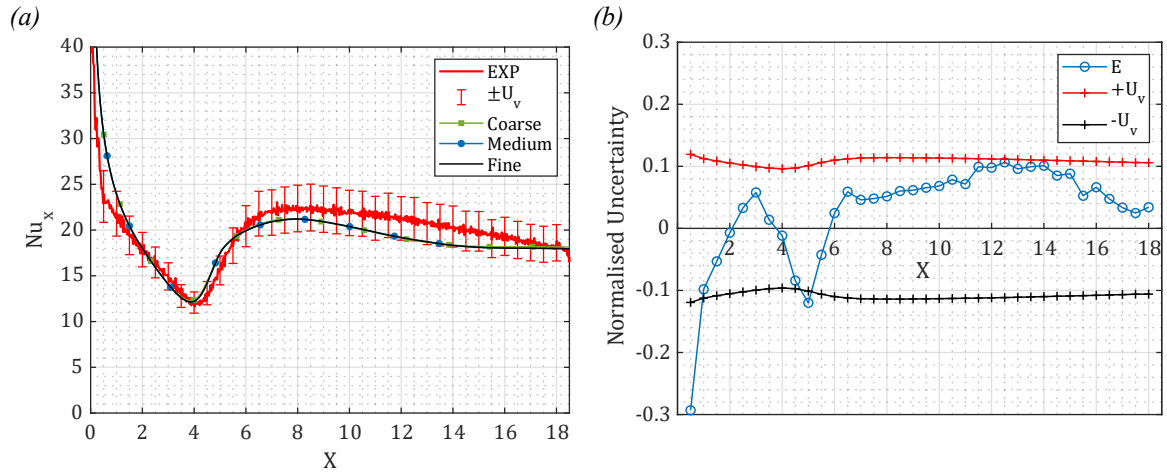


Figure 8.12: Comparison of (a) the Nu_x profiles acquired using the coarse, medium, and fine meshes with the relevant experimental Nu_x data, and (b) the normalised comparison error with respect to the validation uncertainty interval.

Table 8.2: Verification data for the numerical model, where Nu_{x_3} , Nu_{x_2} and Nu_{x_1} correspond to the local Nusselt number data produced by the fine, medium, and coarse meshes, respectively.

X	Nu_{x_3}	Nu_{x_2}	Nu_{x_1}	r_g	Convergence Check	p	δ_{GRE}^*	U_{FS}
0.5	30.61	30.61	30.60	-0.03	Oscillatory Convergence	—	—	—
1.0	23.93	23.93	23.92	0.01	Monotonic Convergence	12.82	-1.2×10^{-6}	1.1×10^{-4}
1.5	20.45	20.45	20.44	0.11	Monotonic Convergence	6.25	-9.1×10^{-5}	3.3×10^{-3}
2.0	17.99	17.99	17.98	0.47	Monotonic Convergence	2.16	-1.5×10^{-3}	4.5×10^{-3}
2.5	15.92	15.92	15.93	-0.29	Oscillatory Convergence	—	—	—
3.0	14.07	14.07	14.12	-0.10	Oscillatory Convergence	—	—	—
3.5	12.61	12.60	12.72	-0.05	Oscillatory Convergence	—	—	—
4.0	12.22	12.22	12.39	-0.02	Oscillatory Convergence	—	—	—
4.5	14.20	14.22	14.41	0.08	Monotonic Convergence	7.16	1.4×10^{-3}	6.3×10^{-2}
5.0	17.46	17.47	17.52	0.32	Monotonic Convergence	3.28	6.9×10^{-3}	8.4×10^{-2}
5.5	19.07	19.08	19.09	0.24	Monotonic Convergence	4.16	1.1×10^{-3}	2×10^{-2}
6.0	19.92	19.92	20.01	0.00	Oscillatory Convergence	—	—	—
6.5	20.51	20.51	20.62	0.01	Monotonic Convergence	12.43	2×10^{-5}	1.8×10^{-3}
7.0	20.91	20.91	21.01	0.03	Monotonic Convergence	9.86	1.1×10^{-4}	7.2×10^{-3}
7.5	21.12	21.13	21.19	0.06	Monotonic Convergence	8.31	2.1×10^{-4}	1.1×10^{-2}
8.0	21.20	21.21	21.21	-2.20	Oscillatory Divergence	—	—	—
8.5	21.11	21.11	21.09	-0.36	Oscillatory Convergence	—	—	—
9.0	20.91	20.92	20.89	-0.26	Oscillatory Convergence	—	—	—
9.5	20.66	20.67	20.63	-0.21	Oscillatory Convergence	—	—	—
10.0	20.38	20.39	20.35	-0.20	Oscillatory Convergence	—	—	—
10.5	20.08	20.08	20.04	-0.15	Oscillatory Convergence	—	—	—
11.0	19.77	19.77	19.72	-0.12	Oscillatory Convergence	—	—	—
11.5	19.46	19.47	19.40	-0.10	Oscillatory Convergence	—	—	—
12.0	19.18	19.19	19.12	-0.08	Oscillatory Convergence	—	—	—
12.5	18.94	18.94	18.87	-0.07	Oscillatory Convergence	—	—	—
13.0	18.72	18.73	18.66	-0.07	Oscillatory Convergence	—	—	—
13.5	18.50	18.51	18.49	-0.22	Oscillatory Convergence	—	—	—
14.0	18.33	18.33	18.36	0.14	Monotonic Convergence	5.57	6.9×10^{-4}	2.1×10^{-2}
14.5	18.20	18.20	18.27	0.05	Monotonic Convergence	8.37	2.1×10^{-4}	1.1×10^{-2}
15.0	18.11	18.12	18.21	0.03	Monotonic Convergence	9.68	1.2×10^{-4}	7.7×10^{-3}
15.5	18.06	18.06	18.18	0.03	Monotonic Convergence	10.50	8.4×10^{-5}	6×10^{-3}
16.0	18.03	18.03	18.17	0.02	Monotonic Convergence	11.32	5.4×10^{-5}	4.2×10^{-3}
16.5	18.02	18.02	18.17	0.02	Monotonic Convergence	12.10	3.5×10^{-5}	3×10^{-3}
17.0	18.01	18.01	18.17	0.01	Monotonic Convergence	13.03	2×10^{-5}	1.8×10^{-3}
17.5	17.99	18.00	18.17	0.01	Monotonic Convergence	14.38	8.3×10^{-6}	8.5×10^{-4}
18.0	17.97	17.97	18.16	0.00	Monotonic Convergence	16.51	2×10^{-6}	2.4×10^{-4}

Table 8.3: Validation data for numerical model using the finest mesh density.

X	Nu_{x_3}	$Nu_{x_{EXP}}$	E	$\frac{U_{FS}}{ Nu_{x_{EXP}} }$	$\frac{U_c(Nu_{x_{EXP}})}{ Nu_{x_{EXP}} }$	$\frac{U_V}{ Nu_{x_{EXP}} }$	Validation Check
0.5	30.61	23.67	-0.29	—	0.12	0.12	Not Validated
1.0	23.93	21.79	-0.10	4.9×10^{-6}	0.11	0.11	Validated
1.5	20.45	19.42	-0.05	1.7×10^{-4}	0.11	0.11	Validated
2.0	17.99	17.86	-0.01	2.5×10^{-4}	0.11	0.11	Validated
2.5	15.92	16.46	0.03	—	0.10	0.10	Validated
3.0	14.07	14.94	0.06	—	0.10	0.10	Validated
3.5	12.61	12.79	0.01	—	0.10	0.10	Validated
4.0	12.22	12.08	-0.01	—	0.10	0.10	Validated
4.5	14.20	13.10	-0.08	4.8×10^{-3}	0.10	0.10	Validated
5.0	17.46	15.59	-0.12	5.4×10^{-3}	0.10	0.10	Not Validated
5.5	19.07	18.29	-0.04	1.1×10^{-3}	0.11	0.11	Validated
6.0	19.92	20.42	0.02	—	0.11	0.11	Validated
6.5	20.51	21.80	0.06	8.2×10^{-5}	0.11	0.11	Validated
7.0	20.91	21.91	0.05	3.3×10^{-4}	0.11	0.11	Validated
7.5	21.12	22.19	0.05	5×10^{-4}	0.11	0.11	Validated
8.0	21.20	22.36	0.05	—	0.11	0.11	Validated
8.5	21.11	22.46	0.06	—	0.11	0.11	Validated
9.0	20.91	22.28	0.06	—	0.11	0.11	Validated
9.5	20.66	22.11	0.07	—	0.11	0.11	Validated
10.0	20.38	21.87	0.07	—	0.11	0.11	Validated
10.5	20.08	21.79	0.08	—	0.11	0.11	Validated
11.0	19.77	21.28	0.07	—	0.11	0.11	Validated
11.5	19.46	21.61	0.10	—	0.11	0.11	Validated
12.0	19.18	21.27	0.10	—	0.11	0.11	Validated
12.5	18.94	21.20	0.11	—	0.11	0.11	Validated
13.0	18.72	20.71	0.10	—	0.11	0.11	Validated
13.5	18.50	20.55	0.10	—	0.11	0.11	Validated
14.0	18.33	20.38	0.10	1.1×10^{-3}	0.11	0.11	Validated
14.5	18.20	19.89	0.09	5.7×10^{-4}	0.11	0.11	Validated
15.0	18.11	19.86	0.09	3.9×10^{-4}	0.11	0.11	Validated
15.5	18.06	19.06	0.05	3.1×10^{-4}	0.11	0.11	Validated
16.0	18.03	19.31	0.07	2.2×10^{-4}	0.11	0.11	Validated
16.5	18.02	18.92	0.05	1.6×10^{-4}	0.11	0.11	Validated
17.0	18.01	18.63	0.03	9.8×10^{-5}	0.11	0.11	Validated
17.5	17.99	18.45	0.02	4.6×10^{-5}	0.11	0.11	Validated
18.0	17.97	18.61	0.03	1.3×10^{-5}	0.11	0.11	Validated

8.3.Comparison with Experimental Results

This section discusses the performance of the numerical model with respect to the experimental findings for an expanded range of dual jet parameters, *i.e.*, $Re = 5500$, 9000 , and 12000 for $OR = 1$ and $OR = 3$. The numerical data is analysed first with respect the flow field characteristics, followed by the corresponding heat transfer behaviours, where the strengths and weaknesses of the numerical model are discussed in depth and recommendations are then made regarding potential model improvements. As the experimental data has been previously shown to be in direct contrast with the published numerical data regarding the heat transfer characteristics associated with dual jet flows, the primary aim of the numerical investigation is to prove that, through making more informed decisions on the model inputs, the experimental trends can be replicated using CFD to a reasonable level of accuracy, hence enforcing the validity of the experimental data.

8.3.1. Flow Characteristics

The steady-state dual jet flow structures obtained using the numerical model for $OR = 1$ and $OR = 3$ are presented in Figure 8.13 by means of the $|U|$ contour plot and the corresponding streamlines. For each OR value, the numerical flow field is demonstrated for the extreme Re values as per the experimental range, *i.e.*, $Re = 5500$ and $Re = 12000$. Across all cases, the numerical flow field presents clear similarities with the experimental results, where both jets initially deflect towards each other upon exiting the jet nozzle, creating a distinct recirculation zone, and eventually merge at some downstream distance. Inside the recirculation zone, two steady counter-rotating vortices are simulated for each case, which is consistent with the time-averaged experimental data. The numerical data also indicates that increasing the jet Reynolds number for a constant OR increases the velocity magnitude across the total flow field while maintaining the overall flow structure, while increasing OR results in a clear growth of the recirculation region; both of which are consistent with the experimental findings discussed previously. The numerical model also extends over a much longer downstream domain compared to the experimental flow domain, where pushing such limitations is the primary advantage of numerical modelling, and, in doing so, demonstrates the jet spreading effects that occur in the far downstream region. Aside from this, no further prominent flow features appear to exist in this region, thus indicating that the experimental setup successfully captured the most prominent time-averaged flow features inside the dual jet flow domain and that extending the experimental field-of-view would have offered no further advantage.

Most notably, however, there is clear evidence of wall jet separation at the solid all boundary for each dual jet case presented, where its occurrence was also reported during the experimental investigation and identified as the primary flow feature which controlled the unique heat transfer behaviour observed. As previously noted, the existence of this separation region has not been reported in any numerical dual jet published to date for $V_r = 1$, however, the present investigation shows the separation region can be successfully simulated in the dual jet flow field though the use of a more appropriate turbulence model, *i.e.*, the transition SST model. This indicates that the many of the published numerical studies employed numerical models not suited to the dual jet flow field, thus inducing significant discrepancies between their respective numerical findings, as well as the dual jet experimental data from the present investigation.

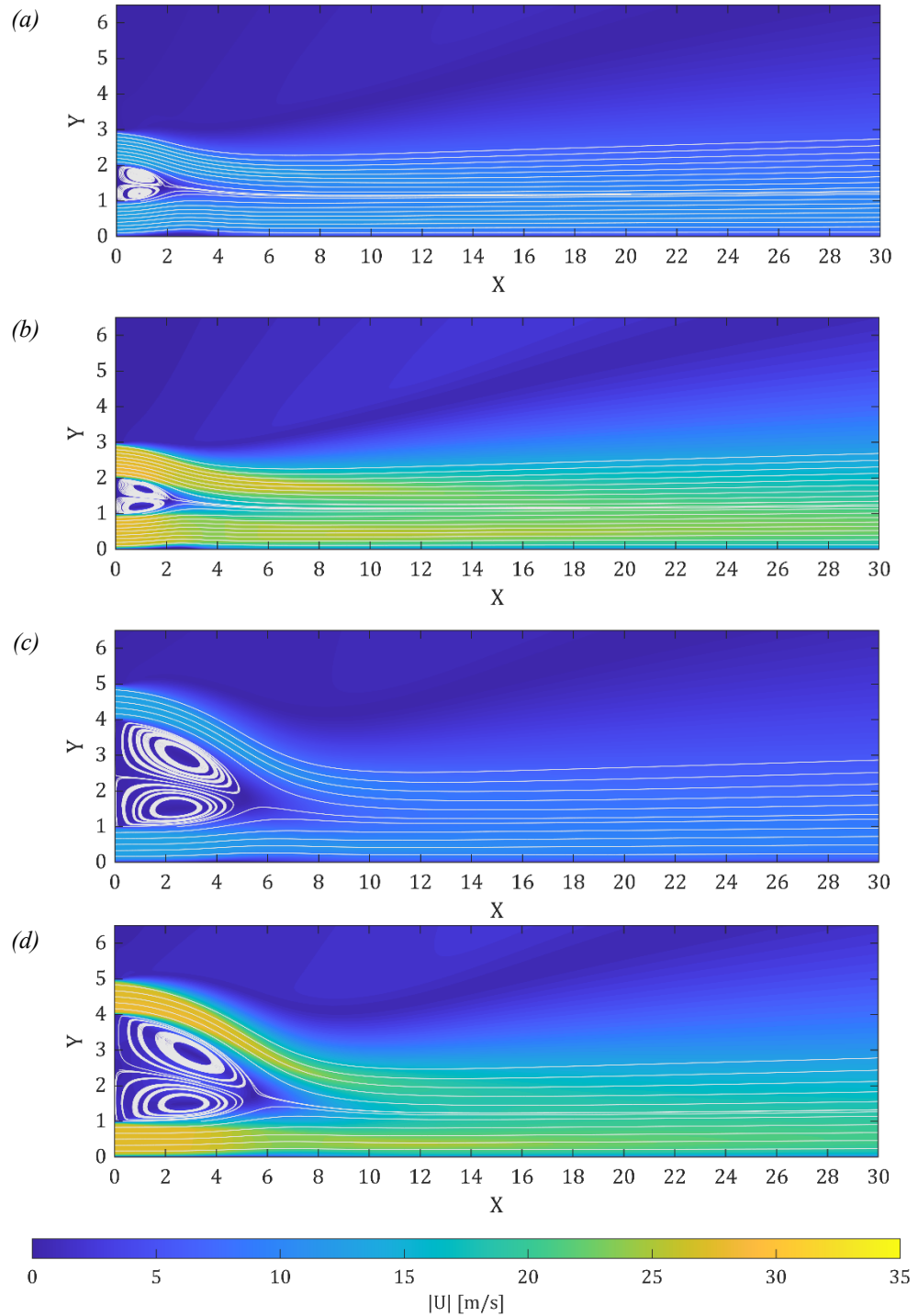


Figure 8.13: Contour plot and streamlines representing the steady-state velocity field as per the dual jet numerical model for $V_r = 1$, where (a) $Re = 5500$, $OR = 1$, (b) $Re = 12000$, $OR = 1$, (c) $Re = 5500$, $OR = 3$ and (d) $Re = 12000$, $OR = 3$.

To further analyse the performance of the numerical model with respect to the experimental flow data, the streamwise and transverse positions of the merge point, combined point, upper vortex centre, and lower vortex centre are plotted in Figure 8.14 for varying Re , where data pertaining to both $OR = 1$ and $OR = 3$ is shown. In doing so, it is clear that the numerical model generally predicts the streamwise locations of each flow feature further downstream than that obtained from the experimental data, indicating that the flow development in the case of the numerical study is slightly slower. Beginning with the merge point, while the X_{mp} values simulated by the numerical model are certainly comparable to the experimental data, they appear to exist slightly further downstream in each case, thus elongating the recirculation zone. This was also noted previously in section 3.3

in relation to other published numerical dual jet investigations, where studies generally reported larger value of X_{mp} compared to the experimental data, such as Singh and Dewan [24] and Mondal *et al.* [20]. This suggests that the numerical models simulate a jet deflection that is marginally less than that observed experimentally, possibly due to limitations in the simulated rate of entrainment inside the recirculation zone, which would act to raise the pressure in this region and subsequently push the merge point further downstream. For the present study, the difference between the numerical and experimental X_{mp} values appear to diminish with increasing OR , where the difference between the respective datasets is slightly less at $OR = 3$ compared to $OR = 1$. In addition to this, the numerical model suggests that the value of X_{mp} increases with increasing Re regardless of OR , however, the experimental investigation found this was only the case for $OR \geq 3$, while a decrease in X_{mp} was observed with increasing Re for $OR \leq 2$. Despite this, the numerical model appears to provide a relatively good estimate for X_{mp} , particularly for $OR = 3$, where future model improvements may attempt to improve the rate of entrainment with the fluid inside the recirculation zone at lower OR , which may increase the extent of each jet deflection and subsequently shorten the recirculation zone.

Regarding the positions of the upper and lower vortex centres, a close alignment is observed overall between the numerical and experimental data. It is interesting to note, however, that the numerical model predicts their respective positions to exist slightly further downstream in the case of $OR = 1$, but further upstream for $OR = 3$. In both cases, the values of X_{UVC} and X_{LVC} appear to increase with increasing Re , which is consistent with the downstream movement of mp , however, it is in opposition of the experimental trend observed. The transverse positions of UVC and LVC are generally close in value with their respective experimental dataset, particularly in the case of Y_{LVC} , however, Y_{UVC} appears to be slightly higher for all Re examined. This can also be attributed to the decreased angle of deflection of the offset jet upon exiting the jet nozzle, thus moving UVC slightly further from the solid wall. Overall, the model performs well as an initial estimator for identifying the positions of features of interest inside the dual jet flow field.

The combined point presents the greatest disparity when compared to the experimental data, where the simulated flow field predicts a value for X_{cp} much further downstream. This suggests a much slower flow development in the case of the numerical model, which appears to worsen with increasing Re . This is generally the case across all published dual jet studies, as previously discussed, where numerical modelling appears to consistently predict a value for X_{cp} much further downstream than that observed physically. In addition to this, the streamwise position of cp in Figure 8.14 appears to move slightly upstream with increasing OR , however, this difference is very slight in comparison to the distinct increase in X_{cp} for increasing OR as per the experimental data. This suggests that the numerical model does not quite replicate the full extent of the mixing between the two jets, but appears to improve somewhat with increasing OR . Related to this, the numerical model generally predicts the dual jet flow should reach self-similarity by $X = 30$, which again indicates a much slower flow development compared to the experimental findings. While it is clear that the numerical model does not quite capture the rate of mixing between the two jets, this may in fact be a limitation of the steady state solver. The use of a transient solver may better replicate the interaction between the two jets, where time-averaging the resulting data could produce a much faster jet development and a combined point located much further upstream.

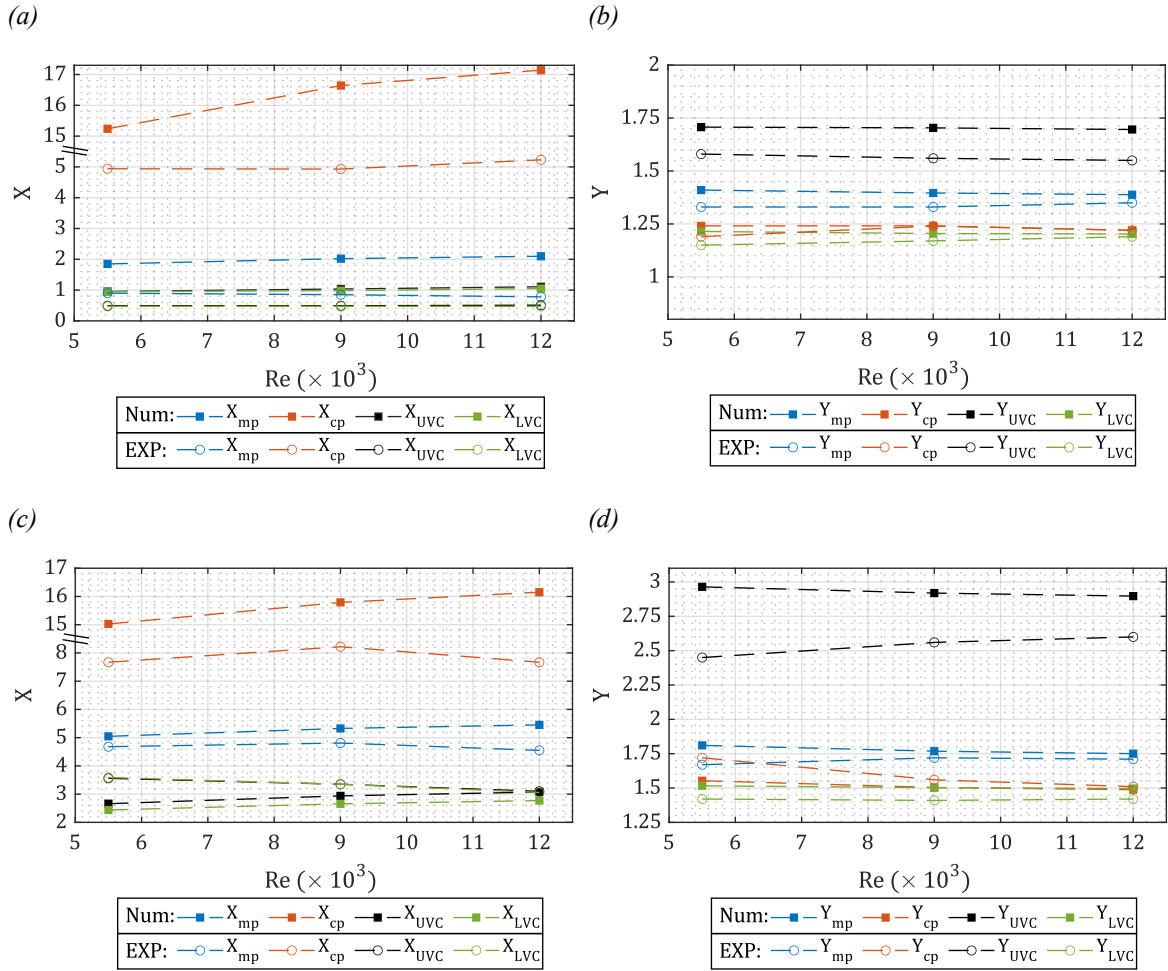


Figure 8.14: Comparison of numerical and experimental data for the X – (left) and Y – (right) locations of mp , cp , UVC and LVC for varying Re and $V_r = 1$, where $OR = (a, b)$ 1 and (c, d) 3.

To complete the analysis of the simulated dual jet flow field, the decay of the normalised maximum streamwise velocity with downstream distance is presented in Figure 8.15 for varying Re and $OR = 1$ and 3. While the profile associated with $OR = 3$ presents similarities with the experimental data, albeit a lesser rate of decay, the profile associated with $OR = 1$ is quite different. Upon exiting from the jet nozzle, the initial decline in u_m appears delayed in numerical model, which implies a delay in the jet deflection and wall jet separation in this region. Further downstream, the numerical profiles exhibit a rise in u_m , which continues for approximately 4 jet widths, at which point u_m decreases rapidly towards a minimum value following which it continues to rise. While this intermediate rise and fall of u_m is inconsistent with the experimental data, it actually corresponds to the region along the numerical downstream domain within which the streamwise offset jet velocity is greater than the wall jet, which is not the case in the experimental data. The sudden decline in u_m hence corresponds to the point at which the wall jet once again possesses the higher streamwise velocity, beyond which the $u_m/u_{m_{max}}$ profile appears to behave in a similar manner to the experimental data in this region. This once again suggests that the extent of the offset jet deflection is much less in the numerical findings, which presents as a higher dual jet streamwise velocity in this region.

Overall, it is clear that the numerical model predicts a much lesser rate of decay of maximum velocity for a dual jet flow, which is somewhat intuitive considering the numerical model assumes an ideal, smooth-wall condition at the solid wall boundary. This is obviously not the case for the experimental setup, where additional friction losses due to wall roughness are expected, not only along the solid wall boundary, but on the insides

of the enclosure walls. Having said this, the reduced rate of decay predicted by the numerical model is not solely due to the reduced friction, but also a suppression of the interaction between the wall jet and offset jet. As a result, the two jets combine at a much slower rate, meaning both jets can possess a higher velocity over a longer downstream distance. As previously suggested, the use of a transient solver may offer a solution in this regard, where successfully simulating the periodic interaction between the jets may result in a more accurate time-averaged flow field, however, this could not be completed during the present investigations due to imposed computational resource and time constraints. Having said this, the numerical model does predict an overall flow field consistent with the experimental findings, meaning the general flow behaviour does not deviate significantly. Therefore, as an initial estimator of the flow behaviour, the numerical model presents significant advantages. Further to this, the use of the transition SST model successfully simulated the separation region between the wall jet and the solid wall boundary, which has not been achieved prior in any published numerical dual jet study.

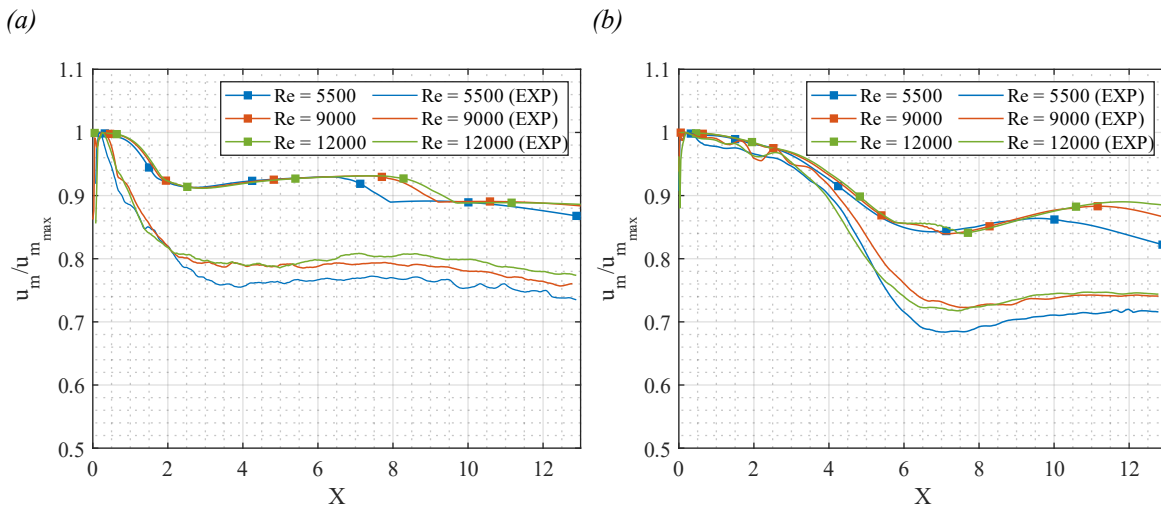


Figure 8.15: Comparison of numerical and experimental data for the normalised maximum velocity for varying Re and $V_r = 1$, where $OR = (a) 1$ and $(b) 3$.

8.3.2. Heat Transfer Behaviour

To analyse the predicted dual jet heat transfer characteristics as per the numerical model, the steady-state Nu_x profiles are plotted in Figure 8.16 alongside the corresponding experimental data for $OR = 1$ and $OR = 3$ at varying Re . In doing so, it is clear that the numerical Nu_x profiles are quite closely aligned with the experimental datasets and provide a credible representation of the overall trends, where only minor deviations exist depending on OR . For $OR = 1$, *i.e.*, Figure 8.16(a), the numerical and experimental Nu_x profiles primarily differ in the region closest to the nozzle exit, where the streamwise positions of both $Nu_{x_{min}}$ and $Nu_{x_{max}}$ are predicted further downstream in the case of the numerical model. However, a very close agreement is observed between the profiles along the subsequent downstream region and the characteristic ‘peaking’ phenomenon is successfully recreated. The existing differences can be attributed to the previously noted elongation of the recirculation zone, where an increase in the streamwise position of the merge point induces the wall jet separation region further downstream. In the region immediately downstream of the nozzle exit, the initial sharp decline in Nu_x appears to be slightly less in the case of the numerical model, which is consistent with the lessened deflection of the wall jet. In doing so, the location at which the thickness of the separation region is greatest, *i.e.*, $Nu_{x_{min}}$, moves slightly downstream, where the reduced value of $Nu_{x_{min}}$ implies the

thickness of the separation region is also increased at this point. The re-impingement of the wall jet consequentially occurs at an increased streamwise position, at which point the flow velocity is slightly less, thereby reducing the value of $Nu_{x_{max}}$.

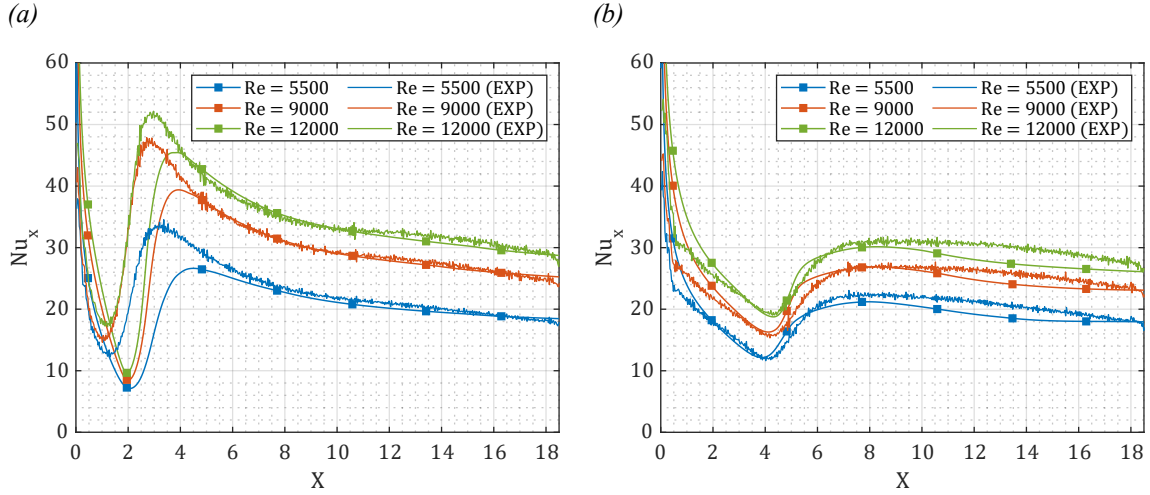


Figure 8.16: Comparison of numerical and experimental dual jet Nu_x profiles for varying Re and $V_r = 1$, where $OR = (a) 1$ and $(b) 3$.

For $OR = 3$, i.e., Figure 8.16(b), a much closer alignment is observed between the numerical and experimental datasets, particularly in the wall jet separation region, where the numerical model appears to accurately identify the magnitude and positions of both $Nu_{x_{min}}$ and $Nu_{x_{max}}$ for each Re examined. Immediately downstream of the nozzle exit, the numerical model captures the rate of decline of Nu_x reasonably well, where slight differences are to be expected due to the discontinuity of the heated section upstream of this point, though the alignment improves as $Nu_{x_{min}}$ is approached. Between $Nu_{x_{min}}$ and $Nu_{x_{max}}$, however, the initial rise in Nu_x is quite sharp in comparison to the experimental data but appears to level off somewhat before reaching $Nu_{x_{max}}$. This may suggest that the re-impingement of the wall jet is slightly sharper in the case of the numerical model, however, the net effect of this on the heat transfer characteristics is ultimately negligible. Further downstream of this point, a slight deviation is observed between the numerical and experimental Nu_x profiles, where a distinct dip exists in the numerical dataset, following which they eventually reconverge. The lower values of Nu_x in this region can most likely be attributed to the slower flow velocities predicted in the vicinity of the solid wall boundary, which comes about as a result of the lack of mixing between both jets. As the combined point is surpassed much earlier in the case of the experimental data, it is to be expected that the combined momentum of both the wall and offset jet would induce higher Nu_x values along this downstream region, while the numerical values of Nu_x would be comparatively less in advance of reaching the combined point as per the numerical model.

Overall, it is clear that, through the use of the transition SST turbulence model, the characteristic dual jet behaviour observed throughout the experimental investigation can be successfully replicated in CFD. While improvements to the model are required, this initial study both validates the findings from the experimental investigation and proves that a reasonable estimate of the heat transfer characteristics of a dual jet can be acquired through numerical modelling. While previous numerical studies report Nu_x profiles related to dual jet flows that vary greatly throughout the published literature, the present investigation shows that a consistent dual jet heat transfer behaviour does exist that can be modelled to a reasonable degree of accuracy through

more informed decisions on the model inputs. This model therefore represents a positive step in the numerical modelling of dual jet flows, where further iterations may improve on the accuracy of the flow field with respect to the experimental data.

8.4. Summary

This chapter presented the approach to modelling the dual jet flow field through numerical means and discussed the flow and heat transfer results with respect to the experimental findings. Through an iterative process, the transition SST model was identified as the most appropriate turbulence model to be used in conjunction with dual jet flows, where the resulting flow field exhibited extreme sensitivity to certain model input parameters, most notably the velocity inlet conditions and the turbulent Prandtl number. Using the experimental heat transfer data, the relevant model input parameters were tuned to suit each specific set of dual jet configurations, where the output from the resulting model was subsequently verified and validated to ensure credibility.

The general steady-state dual jet flow structure predicted by the numerical model was consistent with the time-averaged experimental data, albeit with a slightly slower rate of flow development. For both *OR* presented, the recirculation zone was slightly elongated, thus moving the separation region further downstream, however, the alignment improved with increasing *OR*. The combined point in each case was found to exist much further downstream compared to the experimental findings, which could be attributed to the jet mixing limitations brought about through the use of a steady-state solver. Most notably, however, as the numerical model successfully simulated the wall jet separation from the solid wall boundary, it was possible to reproduce the unique dual jet Nu_x profile discovered during the experimental investigation, where the slight discrepancies could be related to the differences in the flow field discussed previously. The results of the numerical investigation thus indicated that dual jet heat transfer behaviour (i) is both consistent and repeatable, which is in contrast with many published dual jet studies, and (ii) can be modelled to a reasonable degree of accuracy through appropriate validation with experimental data.

Chapter 9

Conclusions

The purpose of this investigation is to characterise the fluid flow and heat transfer behaviour of a wall-bounded dual jet with respect to the jet Reynolds number, the jet separation or offset ratio, and the velocity ratio between the jets. This subject was chosen based on the numerous current and potential applications of dual jets in industry, which include film-cooling, heat exchangers, noise suppression technologies, waste evacuation, and air conditioning. The fundamentals of dual jet flow and heat transfer remain relatively unexplored throughout the published literature. A comprehensive review of the relevant literature reveals significant discrepancies in the published knowledge regarding dual jet flows, the reasoning for which is derived from the severe lack of credible experimental data that could be used to validate the dual jet numerical models and ensure their respective outputs align with physical reality. The present study is thus concerned with the acquisition of accurate flow and heat transfer experimental data relating to dual air jet flows and identifying the effect of varying the jet Reynolds number, the offset ratio, and the velocity ratio in the ranges $5500 \leq Re \leq 12000$, $1 \leq OR \leq 7$ and $0.5 \leq V_r \leq 2$. In doing so, this research represents the first study to obtain experimental flow data for a dual jet where Re , OR , and V_r are varied, as well as the only study to characterise the heat transfer capabilities of a dual jet through experimental means. This data is hence used to develop a validated numerical dual jet model, where the simulated output is consistent with the experimental findings and improves on the dual jet numerical modelling techniques available in the literature.

The experimental setup consists of identical slot jet nozzles of width $w = 7 \text{ mm}$ for the wall and the offset jets. For the heat transfer study, the dual jet flow is subject to uniform heat flux generation of 1670 W/m^2 in the bounding wall using an ohmically heated foil. The temperature distribution in the foil is captured at 25 Hz from below using a FLIR A655sc IR camera. The flow field data is measured using PIV techniques, where movement in the DEHS-seeded airflow is illuminated by means of a Quantronix Darwin-Duo Nd:YLF Laser and captured at 2000 Hz using a Photron FASTCAM SA1.1/HSS 6 high speed CMOS camera. The output from the experimental setup is validated based on published flow field and Nu_x data relating to single wall jets and single offset jets, as well as the flow data provided by the only other experimental study of a dual jet, *i.e.*, Wang and Tan [19].

The time-averaged flow and heat transfer data relating to the dual jet flow reveals a distinct Nu_x profile in the solid wall boundary induced as a result of the wall jet separation, where the resulting Nu_x profile varies greatly from that produced by a single wall jet or single offset jet. For all Re and OR combinations examined, an initial decrease in Nu_x towards a local minimum value $Nu_{x_{min}}$ is observed as the wall jet deflects from the wall.

$Nu_{x_{min}}$ hence exists just downstream of the merge point where the wall jet separation is greatest. Beyond this point, the sharp rise in Nu_x is attributed to the re-impingement of the jet flow on the solid wall boundary, at which point a local maximum Nu_x value is acquired, *i.e.*, $Nu_{x_{max}}$. Beyond this point, entrainment effects from the quiescent surrounding air begin to dominate, leading to a decline in Nu_x over the remaining downstream distance.

By virtue of the increased flow velocity, increasing Re elevates Nu_x across all downstream locations while preserving the overall trend. For all OR , an increase in Re decreases the streamwise rate of decay of maximum velocity, where a greater loss of momentum is evident after the recirculation zone for the lower Re values, which induces a greater dispersion of the Nu_x plots in this region compared to upstream conditions. For $OR \leq 2$, the higher rate of entrainment with the recirculation region with increasing Re causes an increase in the deflection of both jets, which reduces the size of the recirculation region while pulling the wall jet further from the solid wall. This moves positions of both $Nu_{x_{min}}$ and $Nu_{x_{max}}$ further upstream for $OR = 2$, however, their respective positions remain relatively consistent for $OR = 1$ due to the much less pronounced decrease in X_{mp} . This is not the case for $OR = 3$, where an increase in Re does not appear to affect the jet deflection or the position of X_{mp} , however, an elongation of the separation region is noted, which moves $Nu_{x_{min}}$ and $Nu_{x_{max}}$ further downstream. This is also evident in the heat transfer data for $OR = 5$ and $OR = 7$. The surface-averaged data suggests a linear relationship between $\overline{Nu_x}$ and the jet Reynolds number, where increasing Re induces a corresponding rise in $\overline{Nu_x}$ and the rate of increase is higher for lower OR values. Finally, the time-resolved flow data reveals an increase in the vortex shedding frequency with increasing Re , however, the associated Strouhal number remains consistent.

The effect of increasing OR is primarily realised in the regions closest to the jet exit, while flow conditions remain indifferent further downstream, at which point a close alignment exists between all Nu_x profiles. Most notably, increasing OR increases the size of the recirculation zone, where the inevitable increase in X_{mp} pushes $Nu_{x_{min}}$ and $Nu_{x_{max}}$ further downstream. For $OR \leq 2$, a ‘peaking’ effect is evident in the vicinity of $Nu_{x_{max}}$, the extent of which is more exaggerated for higher Re values, which is attributed to the higher velocity at which the wall jet re-impinges on the solid wall for lower OR values. The surface-averaged data displays a general decline in $\overline{Nu_x}$ with increasing OR , where a greater rate of decline is observed in the low OR range. Inside the recirculation region, time-averaged flow data revealed the existence of two counter rotating vortices regardless of OR , the size of which increase with increasing OR . Conversely, the time-resolved data shows clear evidence of periodic vortex shedding at all OR , where increasing OR decreases the associated Strouhal number.

The effect of varying the velocity ratio induces two distinctly different flow behaviours for $V_r < 1$ or $V_r > 1$. For $V_r < 1$, decreasing V_r reduces the deflection of the offset jet and pulls the wall jet farther from the solid wall, which increases the size of the lower vortex compared to the upper vortex. This subsequently induces a higher rate of decay of maximum velocity; however, self-similarity is not achieved until much further downstream. The opposite is true for $V_r > 1$, where increasing V_r enables the wall jet to remain attached and instead increases the offset jet deflection. This reduces the rate of decay of maximum velocity and self-similarity is achieved much sooner. Regarding the periodic shedding of vortices, a general increase in the Strouhal number is clear for $OR = 2$ and $OR = 3$ within a limiting V_r range, however, a decline is observed for $OR = 1$. For $OR = 1$, decreasing or increasing V_r with respect to 1 increases the streamwise positions of

$Nu_{x_{min}}$ and $Nu_{x_{max}}$, therefore indicating a minimum value at $V_r = 1$, and the peaking effect becomes increasingly suppressed at higher V_r values. For $OR \geq 3$, the streamwise positions of $Nu_{x_{min}}$ and $Nu_{x_{max}}$ move upstream when V_r is varied with respect to 1, and a secondary peak in the Nu_x profile emerges for $V_r = 0.5$ when $OR = 5$ and $OR = 7$.

The Nu_x profiles obtained throughout the experimental investigation vary significantly from that provided by any published numerical study, which indicates the use of numerical models not suited to dual jet flow behaviour. Most notably, the presence of a local minimum and local maximum value in the Nu_x profile due to the separation region beneath the wall jet is not reported by any of the published dual jet studies, despite being a trend that is repeated consistently throughout the experimental study. Through the use of the transition SST model and appropriate tuning of the jet inlet parameters, the present numerical investigation successfully simulates this characteristic heat transfer behaviour and adequate model validation is attained using the accepted uncertainty limits. While a slightly slower rate of jet development is predicted by the numerical model, the model still introduces significant improvements to dual jet numerical modelling compared to that available in the published literature, whereby the heat transfer findings can be considered both reliable and representative of physical reality. The results of the numerical investigation indicate that dual jet heat transfer behaviour is both consistent and repeatable, which is in contrast with many published dual jet studies. The numerical results show that heat transfer to a dual jet can be successfully modelled to a reasonable degree of accuracy through appropriate validation with experimental data.

Overall, the results of the experimental investigation reveal a strong influence of Re , OR , and V_r on the Nu_x profile induced in the solid wall boundary. While the general Nu_x profile remains consistent, it is clear that significant alterations can be acquired through adjusting any of the three aforementioned variables, which demonstrates the extent to which the heat transfer characteristics of the dual jet can be tuned. This is achieved through controlling the wall jet separation from the solid wall, where any thickening, elongation, or movement of this region along the streamwise dimension can produce distinct changes in the Nu_x profile, all of which are derived from changes in the size of the recirculation region. Therefore, through manipulating the position of the merge point, the wall jet separation can be adjusted accordingly and the desired Nu_x profile can be acquired. As this heat transfer behaviour can readily be modelled in the numerical environment, the necessary tuning can first be conducted using CFD tools to identify an appropriate range of jet parameters, following which the dual jets can be fine-tuned through physical experimentation. This could prove advantageous for multiple cooling applications, most notably the electronics cooling industry, where controlling the wall jet re-impingent location could easily be used to target specified hotspots on the surface of interest. As this impingement location can be controlled without changing the physical infrastructure of the dual jet arrangement, any drift in the location of the specified hotspot could easily be tracked over time and targeted accordingly, with little to no additional expense.

In conclusion, this research presents an important milestone towards the full characterisation dual jet flow and heat transfer behaviour, as well as furthering the understanding of dual jets in the published literature. This study addresses the gap in the published literature by providing verified, repeatable experimental flow and heat transfer measurements for a dual jet flow over a range of Re , OR , and V_r values, which should aid in the introduction of consistency to future dual jet studies through the provision of a reliable means of model validation.

9.1.Future Work

This study presents the initial stages towards the full characterisation of dual jet flow, where future work could further the experimental investigation over an expanded range of dual jet parameters. For the present investigation, a moderate range of Re values were chosen, however further studies could determine if the observed trends remain consistent at higher Re values, or what changes should be expected for Re values inside the laminar or transitional jet flow regimes. In addition, further work could look towards varying alternative parameters outside of Re , OR , and V_r , such as altering the respective jet nozzle widths, the nozzle shape, the jet temperatures, and the working fluid.

Due to the limitations in the imaging frequency of the thermal camera used during the heat transfer study, it was only possible to obtain time-averaged heat transfer trends associated with dual jet flow, where the IR sampling rate was too low to detect any notable features in the time-resolved heat transfer data. The flow data, however, was captured using a much higher sampling rate of 2000 Hz , which was sufficiently high to capture the periodicity in the dual jet flow field for the range of Re values considered. This thus indicates that analysis of heat transfer data may present some interesting findings if sampled at frequencies $> 2000\text{ Hz}$. While this may not be feasible using infrared thermography, future work could look towards the use of high spatial and temporal resolution heat flux sensors in conjunction with uniform wall temperature boundary condition, such as a hot film sensor, which can achieve sampling rates up to 100 kHz [124, 188].

The results of the present investigation revealed a ‘peaking’ effect that occurs in the vicinity of $Nu_{x_{max}}$, the extent of which became increasingly exaggerated with decreasing OR and increasing Re . In the cases where this peaking effect occurs, a significant increase in Nu_x was observed in the nearby regions, enabling the dual jet to achieve heat transfer rates far in excess of that attainable using a single wall jet or a single offset jet for the same mass flow rate. Future work could examine the limitations of this peaking effect, where the current trends indicate that further decreasing OR should lead to an even greater enhancement of Nu_x in this region and the potential to acquire extreme rates of cooling. Further to this, a simultaneous PIV study could capture the flow characteristics that drive this phenomenon, where a much higher resolution in the vicinity of the recirculation region would be required to achieve this compared to the current study. A higher resolution PIV study may also enable the further characterisation of the wall boundary layer, where the acquisition of data relating to the turbulent statistics of the wall boundary layer could lead to the development of an empirical correlation for use with the Transition SST turbulence model.

Finally, while the numerical investigation produced Nu_x profiles consistent with the experimental findings, the model was quite sensitive to inlet parameters and the downstream flow development was slower compared to its experimental counterpart. Future work could hence look towards introducing further improvements to the numerical model to enhance the extent of the mixing between the jets. This could potentially be achieved through the use of a transient solver, where fully simulating the periodic interaction between the two jets streams may improve their associated mixing properties, therefore moving the positions of all notable flow features further upstream. Through improving the model reliability, the numerical setup could be used to investigate the dual jet flow field over an expanded range of parameters and identify which jet configurations are worthy of further exploration through experimentation.

References

- [1] A. Kumar and M. K. Das, "Study of a turbulent dual jet consisting of a wall jet and an offset jet," *Journal of Fluids Engineering*, vol. 133, no. 10, 2011, doi: 10.1115/1.4004823.
- [2] Z. Li, W.-x. Huai, and J. Han, "Large eddy simulation of the interaction between wall jet and offset jet," *Journal of Hydrodynamics, Ser. B*, vol. 23, no. 5, pp. 544-553, 2011, doi: 10.1016/S1001-6058(10)60148-5.
- [3] T. Mondal, M. K. Das, and A. Guha, "Periodic vortex shedding phenomenon for various separation distances between two plane turbulent parallel jets," *International Journal of Heat and Mass Transfer*, vol. 99, pp. 576-588, 2016, doi: 10.1016/j.ijheatmasstransfer.2016.03.095.
- [4] Q. Liu, D. Baccarella, B. McGann, T. Lee, and H. Do, "Experimental investigation of single jet and dual jet injection in a supersonic combustor," in *2018 AIAA Aerospace Sciences Meeting*, 2018, p. 1363, doi: 10.2514/6.2018-1363.
- [5] I. Aslanidou, V. Kanjirakkad, B. Rosic, and S. Uchida, "Leading edge shielding concept in gas turbines with can combustors," *Journal of Turbomachinery*, vol. 135, p. 021019, 2012, doi: 10.1115/1.4007514.
- [6] S. K. Rathore, A. Pathak, A. Majumdar, and S. Chaudhuri, "Computational investigation of mixed convection heat transfer from laminar offset jet and wall jet," presented at the Fifth International Conference on Computational Methods for Thermal Problems, Bangalore, India, 2018.
- [7] L. Xu, Z. Sun, Q. Ruan, L. Xi, J. Gao, and Y. Li, "Development trend of cooling technology for turbine blades at super-high temperature of above 2000 k," *Energies*, vol. 16, no. 2, p. 668, 2023, doi: 10.3390/en16020668.
- [8] J. Vlachopoulos and N. D. Polychronopoulos, "Understanding rheology and technology of polymer extrusion," ed: Polydynamics Inc, 2019.
- [9] N. Gao, C. Y. Ching, D. Ewing, and J. W. Naughton, "Flow and heat transfer measurements in a planar offset attaching jet with a co-flowing wall jet," *International Journal of Heat and Mass Transfer*, vol. 78, pp. 721-731, 2014, doi: 10.1016/j.ijheatmasstransfer.2014.07.008.
- [10] N. Hnaïen, S. Marzouk, H. B. Aïssia, and J. Jay, "Wall inclination effect in heat transfer characteristics of a combined wall and offset jet flow," *International Journal of Heat and Fluid Flow*, vol. 64, pp. 66-78, 2017, doi: 10.1016/j.ijheatfluidflow.2017.01.010.
- [11] K. Zhao, S. Alimohammadi, P. N. Okolo, J. Kennedy, and G. J. Bennett, "Aerodynamic noise reduction using dual-jet planar air curtains," *Journal of Sound and Vibration*, vol. 432, pp. 192-212, 2018, doi: 10.1016/j.jsv.2018.06.036.

- [12] K. Zhao, P. Okolo, E. Neri, P. Chen, J. Kennedy, and G. J. Bennett, "Noise reduction technologies for aircraft landing gear-a bibliographic review," *Progress in Aerospace Sciences*, vol. 112, p. 100589, 2020, doi: 10.1016/j.paerosci.2019.100589.
- [13] R. Ji, X. Huang, and X. Zhao, "Active jet noise control of turbofan engine based on explicit model predictive control," *Applied Sciences*, vol. 12, no. 10, p. 4874, 2022, doi: 10.3390/app12104874.
- [14] Z. Zhang, X. Wang, and Y. Yan, "A review of the state-of-the-art in electronic cooling," *e-Prime - Advances in Electrical Engineering, Electronics and Energy*, vol. 1, p. 100009, 2021, doi: 10.1016/j.prime.2021.100009.
- [15] S. A. Nada, R. M. El-Zoheiry, M. Elsharnoby, and O. S. Osman, "Enhancing the thermal performance of different flow configuration minichannel heat sink using Al_2O_3 and CuO-water nanofluids for electronic cooling: An experimental assessment," *International Journal of Thermal Sciences*, vol. 181, p. 107767, 2022, doi: 10.1016/j.ijthermalsci.2022.107767.
- [16] S. Sarkar, R. Gupta, T. Roy, R. Ganguly, and C. M. Megaridis, "Review of jet impingement cooling of electronic devices: Emerging role of surface engineering," *International Journal of Heat and Mass Transfer*, vol. 206, p. 123888, 2023, doi: 10.1016/j.ijheatmasstransfer.2023.123888.
- [17] M. Sadeghian, "Technologies for aircraft noise reduction: A review," *Journal of Aeronautics & Aerospace Engineering*, 2020, doi: 10.35248/2168- 9792.20.9.219.
- [18] X. K. Wang and S. K. Tan, "Environmental fluid dynamics - jet flow," *Journal of Hydrodynamics, Ser. B*, vol. 22, no. 5, Supplement 1, pp. 1009-1014, 2010, doi: 10.1016/S1001-6058(10)60067-4.
- [19] X. K. Wang and S. K. Tan, "Experimental investigation of the interaction between a plane wall jet and a parallel offset jet " *Experiments in Fluids*, vol. 42, no. 4, pp. 551-562, 2007, doi: 10.1007/s00348-007-0263-9.
- [20] T. Mondal, M. K. Das, and A. Guha, "Numerical investigation of steady and periodically unsteady flow for various separation distances between a wall jet and an offset jet," *Journal of Fluids and Structures*, vol. 50, pp. 528-546, 2014, doi: 10.1016/j.jfluidstructs.2014.07.009.
- [21] T. Mondal, A. Guha, and M. K. Das, "Effect of bottom wall proximity on the unsteady flow structures of a combined turbulent wall jet and offset jet flow," *European Journal of Mechanics - B/Fluids*, vol. 57, pp. 101-114, 2016, doi: 10.1016/j.euromechflu.2015.12.003.
- [22] T. Mondal, A. Guha, and M. Kumar Das, "Analysis of conjugate heat transfer for a combined turbulent wall jet and offset jet," *Journal of Heat Transfer*, vol. 138, no. 5, 2016, doi: 10.1115/1.4032287.
- [23] M. Kumar, R. Biswal, A. K. Gupta, S. K. Behera, and R. K. Sahoo, "Effect of wall heat flux on fluid flow and thermal characteristics of a turbulent dual jet," *Mathematical Modelling of Engineering Problems*, vol. 7, no. 1, pp. 127-134, 2020, doi: 10.18280/mmep.070116.
- [24] T. P. Singh and A. Dewan, "Heat transfer characteristics of a periodically transient flow for turbulent wall-bounded dual jet," *Engineering Science and Technology, an International Journal*, vol. 36, p. 101146, 2022, doi: 10.1016/j.jestch.2022.101146.
- [25] N. Hnaïen, S. Marzouk, H. B. Aissia, and J. Jay, "Numerical investigation of velocity ratio effect in combined wall and offset jet flows," *Journal of Hydrodynamics*, vol. 30, no. 6, pp. 1105-1119, 2018, doi: 10.1007/s42241-018-0136-0.
- [26] A. Assoudi, N. Mahjoub Saïd, H. Bournot, and G. Le Palec, "Comparative study of flow characteristics of a single offset jet and a turbulent dual jet," *Heat and Mass Transfer*, vol. 55, no. 4, pp. 1109-1131, 2019, doi: 10.1007/s00231-018-2493-1.

- [27] A. Kumar, "Mean flow characteristics of a turbulent dual jet consisting of a plane wall jet and a parallel offset jet," *Computers & Fluids*, vol. 114, pp. 48-65, 2015, doi: 10.1016/j.compfluid.2015.02.017.
- [28] E. Vishnuvardhanarao and M. K. Das, "Study of the heat transfer characteristics in turbulent combined wall and offset jet flows," *International Journal of Thermal Sciences*, vol. 48, no. 10, pp. 1949-1959, 2009, doi: 10.1016/j.ijthermalsci.2009.02.020.
- [29] S. Farooq and M. Das, "Simulation of thermal characteristics of turbulent dual jets," *J Appl Mech Eng*, vol. 3, no. 141, p. 2, 2014, doi: 10.4172/2168-9873.1000141.
- [30] P. J. Murphy, S. Alimohammadi, and S. M. O'Shaughnessy, "Experimental investigation of dual jet flow past a heated surface: Effect of reynolds number," *International Journal of Heat and Mass Transfer*, vol. 218, p. 124786, 2024, doi: 10.1016/j.ijheatmasstransfer.2023.124786.
- [31] P. Murphy, S. Alimohammadi, and S. O. Shaughnessy, "Experimental analysis of heat transfer to a dual jet flow: Effect of reynolds number," in *2022 28th International Workshop on Thermal Investigations of ICs and Systems (THERMINIC)*, 28-30 Sept. 2022 2022, pp. 1-6, doi: 10.1109/THERMINIC57263.2022.9950682.
- [32] P. J. Murphy, S. Alimohammadi, and S. M. O. Shaughnessy, "Experimental investigation of heat transfer to a dual jet flow with varying velocity ratio," in *2023 22nd IEEE Intersociety Conference on Thermal and Thermomechanical Phenomena in Electronic Systems (ITherm)*, 30 May-2 June 2023 2023, pp. 1-8, doi: 10.1109/ITherm55368.2023.10177606.
- [33] I. Namer and M. V. Ötügen, "Velocity measurements in a plane turbulent air jet at moderate reynolds numbers," *Experiments in Fluids*, vol. 6, no. 6, pp. 387-399, 1988, doi: 10.1007/BF00196484.
- [34] C. D. Ungate, D. R. Harleman, and G. H. Jirka, "Stability and mixing of submerged turbulent jets at low reynolds numbers," MIT Energy Lab, 1975.
- [35] B. K. Revill, "Chapter 9 - jet mixing," in *Mixing in the process industries*, N. Harnby, M. F. Edwards, and A. W. Nienow Eds. Oxford: Butterworth-Heinemann, 1992, pp. 159-183.
- [36] D. R. Miller and E. W. Comings, "Static pressure distribution in the free turbulent jet," *Journal of Fluid Mechanics*, vol. 3, no. 1, pp. 1-16, 1957, doi: 10.1017/S0022112057000440.
- [37] I. Wygnanski and H. Fiedler, "Some measurements in the self-preserving jet," *Journal of Fluid Mechanics*, vol. 38, no. 3, pp. 577-612, 1969, doi: 10.1017/S0022112069000358.
- [38] J. Westerweel, C. Fukushima, J. M. Pedersen, and J. C. R. Hunt, "Momentum and scalar transport at the turbulent/non-turbulent interface of a jet," *Journal of Fluid Mechanics*, vol. 631, pp. 199-230, 2009, doi: 10.1017/S00221120090006600.
- [39] E. Gutmark and I. Wygnanski, "The planar turbulent jet," *Journal of Fluid Mechanics*, vol. 73, no. 3, pp. 465-495, 1976, doi: 10.1017/S0022112076001468.
- [40] C. Ball, H. Fellouah, and A. Pollard, "The flow field in turbulent round free jets," *Progress in Aerospace Sciences*, vol. 50, pp. 1-26, 2012, doi: 10.1016/j.paerosci.2011.10.002.
- [41] J. A. Fitzgerald and S. V. Garimella, "A study of the flow field of a confined and submerged impinging jet," *International Journal of Heat and Mass Transfer*, vol. 41, no. 8, pp. 1025-1034, 1998, doi: 10.1016/S0017-9310(97)00205-6.
- [42] M. Soleimani nia, P. Oshkai, and N. Djilali, "Multi-component high aspect ratio turbulent jets issuing from non-planar nozzles," *International Journal of Hydrogen Energy*, vol. 44, no. 29, pp. 15262-15277, 2019, doi: 10.1016/j.ijhydene.2019.04.027.

-
- [43] E. Faghani, R. Maddahian, P. Faghani, and B. Farhanieh, "Numerical investigation of turbulent free jet flows issuing from rectangular nozzles: The influence of small aspect ratio," vol. 80, no. 7, 80, pp. 727-745, 2010, doi: 10.1007/s00419-009-0340-z.
- [44] M. Alnahhal and T. Panidis, "The effect of sidewalls on rectangular jets," *Experimental Thermal and Fluid Science*, vol. 33, no. 5, pp. 838-851, 2009, doi: 10.1016/j.expthermflusci.2009.03.001.
- [45] N. Rajaratnam, *Turbulent jets*. Elsevier, 1976.
- [46] R. Zhang, X. He, G. Doolen, and S. Chen, "Surface tension effects on two-dimensional two-phase Kelvin–Helmholtz instabilities," *Advances in Water Resources*, vol. 24, no. 3, pp. 461-478, 2001, doi: 10.1016/S0309-1708(00)00067-1.
- [47] L. Rosenhead and H. Jeffreys, "The formation of vortices from a surface of discontinuity," *Proceedings of the Royal Society of London. Series A, Containing Papers of a Mathematical and Physical Character*, vol. 134, no. 823, pp. 170-192, 1931, doi: doi:10.1098/rspa.1931.0189.
- [48] M. V. Melander, N. J. Zabusky, and J. C. McWilliams, "Symmetric vortex merger in two dimensions: Causes and conditions," *Journal of Fluid Mechanics*, vol. 195, pp. 303-340, 1988, doi: 10.1017/S0022112088002435.
- [49] C. D. Winant and F. K. Browand, "Vortex pairing: The mechanism of turbulent mixing-layer growth at moderate reynolds number," *Journal of Fluid Mechanics*, vol. 63, no. 2, pp. 237-255, 1974, doi: 10.1017/S0022112074001121.
- [50] W. Dahm and P. Dimotakis, "Measurements of entrainment and mixing in turbulent jets," *AIAA journal*, vol. 25, no. 9, pp. 1216-1223, 1987, doi: 10.2514/3.9770.
- [51] T. L. Bergman, F. P. Incropera, D. P. DeWitt, and A. S. Lavine, *Fundamentals of heat and mass transfer*. John Wiley & Sons, 2011.
- [52] A. Agrawal and A. Prasad, "Properties of vortices in the self-similar turbulent jet," *Experiments in Fluids*, vol. 33, no. 4, pp. 565-577, 2002, doi: 10.1007/s00348-002-0507-7.
- [53] S. Ghosal and M. M. Rogers, "A numerical study of self-similarity in a turbulent plane wake using large-eddy simulation," *Physics of Fluids*, vol. 9, no. 6, pp. 1729-1739, 1997, doi: 10.1063/1.869289.
- [54] S. Hiromasa, S. Hara, and K. Inaoka, "The velocity and thermal fields of two parallel plane jets using simultaneous PIV and two-color lif measurements," *International Journal of Heat and Fluid Flow*, vol. 103, p. 109192, 2023, doi: 10.1016/j.ijheatfluidflow.2023.109192.
- [55] Y. F. Lin and M. J. Sheu, "Investigation of two plane parallel jets," *Experiments in Fluids*, journal article vol. 10, no. 1, pp. 17-22, 1990, doi: 10.1007/bf00187867.
- [56] D. R. Miller and E. W. Comings, "Force-momentum fields in a dual-jet flow," *Journal of Fluid Mechanics*, vol. 7, no. 2, pp. 237-256, 1960, doi: 10.1017/S0022112060001468.
- [57] E. A. Anderson and R. E. Spall, "Experimental and numerical investigation of two-dimensional parallel jets," *Journal of Fluids Engineering*, vol. 123, no. 2, pp. 401-406, 2001, doi: 10.1115/1.1363701.
- [58] Z. Xie, J. Xie, W. Jiang, and W. Du, "Numerical and experimental investigation of near-field mixing in parallel dual round jets," *International Journal of Aerospace Engineering*, vol. 2016, 2016, doi: 10.1155/2016/7935101.
- [59] E. Tanaka, "The interference of two-dimensional parallel jets : 2nd report, experiments on the combined flow of dual jet," *Bulletin of JSME*, vol. 17, no. 109, pp. 920-927, 1974, doi: 10.1299/jsme1958.17.920.
-

- [60] N. Fujisawa, K. Nakamura, and K. Srinivas, "Interaction of two parallel plane jets of different velocities," *Journal of Visualization*, vol. 7, no. 2, pp. 135-142, 2004, doi: 10.1007/BF03181586.
- [61] N. Nassira and M. Amina, "Numerical study of mixed convection of buoyant twin jet," *ASME Journal of Heat and Mass Transfer*, vol. 146, no. 3, 2023, doi: 10.1115/1.4063959.
- [62] D. Lytle and B. Webb, "Air jet impingement heat transfer at low nozzle-plate spacings," *International Journal of Heat and Mass Transfer*, vol. 37, no. 12, pp. 1687-1697, 1994, doi: 10.1016/0017-9310(94)90059-0.
- [63] C. Donaldson and R. S. Snedeker, "A study of free jet impingement. Part 1. Mean properties of free and impinging jets," *Journal of Fluid Mechanics*, vol. 45, no. 2, pp. 281-319, 1971, doi: 10.1017/S0022112071000053.
- [64] K. Jambunathan, E. Lai, M. A. Moss, and B. L. Button, "A review of heat transfer data for single circular jet impingement," *International Journal of Heat and Fluid Flow*, vol. 13, no. 2, pp. 106-115, 1992, doi: 10.1016/0142-727X(92)90017-4.
- [65] J. Eriksson, "Experimental studies of the plane turbulent wall jet," PhD thesis, Department of Engineering Mechanics, Royal Institute of Technology, Stockholm, Sweden, 2003.
- [66] T. J. Craft and B. E. Launder, "On the spreading mechanism of the three-dimensional turbulent wall jet," *Journal of Fluid Mechanics*, vol. 435, pp. 305-326, 2001, doi: 10.1017/S0022112001003846.
- [67] P. Nizou, "Heat and momentum transfer in a plane turbulent wall jet," *Journal of Heat Transfer*, vol. 103, no. 1, pp. 138-140, 1981, doi: doi.org/10.1115/1.3244407.
- [68] R. S. AbdulNour, K. Willenborg, J. J. McGrath, J. F. Foss, and B. S. AbdulNour, "Measurements of the convection heat transfer coefficient for a planar wall jet: Uniform temperature and uniform heat flux boundary conditions," *Experimental Thermal and Fluid Science*, vol. 22, no. 3, pp. 123-131, 2000, doi: 10.1016/S0894-1777(00)00018-2.
- [69] S. Pramanik and M. K. Das, "Computational study of a turbulent wall jet flow on an oblique surface," *International Journal of Numerical Methods for Heat & Fluid Flow*, vol. 24, no. 2, pp. 290-324, 2014, doi: 10.1108/HFF-01-2012-0005.
- [70] A. Kaffel, J. Moureh, J.-L. Harion, and S. Russeil, "Experimental investigation of a plane wall jet subjected to an external lateral flow," *Experiments in Fluids*, journal article vol. 56, no. 5, p. 95, 2015, doi: 10.1007/s00348-015-1969-8.
- [71] C. Fu, M. Uddin, and A. Curley, "Insights derived from CFD studies on the evolution of planar wall jets," *Engineering Applications of Computational Fluid Mechanics*, vol. 10, no. 1, pp. 44-56, 2016, doi: 10.1080/19942060.2015.1082505.
- [72] J. G. Eriksson, R. I. Karlsson, and J. Persson, "An experimental study of a two-dimensional plane turbulent wall jet," *Experiments in Fluids*, vol. 25, no. 1, pp. 50-60, 1998, doi: 10.1007/s003480050207.
- [73] J. C. Akfirat, "Transfer of heat from an isothermal flat plate to a two-dimensional wall jet," in *International Heat Transfer Conference Digital Library*, 1966: Begel House Inc., doi: 10.1615/IHTC3.1890.
- [74] I. Mabuchi and M. Kumada, "Studies on heat transfer to turbulent jets with adjacent boundaries : 1st report, flow development and mass transfer in plane turbulent wall jet," *Bulletin of JSME*, vol. 15, no. 88, pp. 1236-1245, 1972, doi: 10.1299/jsme1958.15.1236.

-
- [75] P. Rajesh Kanna and M. Kumar Das, "Numerical simulation of two-dimensional laminar incompressible offset jet flows," *International Journal for Numerical Methods in Fluids*, vol. 49, no. 4, pp. 439-64, 2005, doi: 10.1002/fld.1007.
- [76] A. Nasr and J. C. S. Lai, "A turbulent plane offset jet with small offset ratio," *Experiments in Fluids*, vol. 24, no. 1, pp. 47-57, 1998, doi: 10.1007/s003480050149.
- [77] S. K. Rathore and M. K. Das, "A comparative study of heat transfer characteristics of wall-bounded jets using different turbulence models," *International Journal of Thermal Sciences*, vol. 89, pp. 337-356, 2015, doi: 10.1016/j.ijthermalsci.2014.11.019.
- [78] R. Gu, "Modeling two-dimensional turbulent offset jets," *Journal of Hydraulic Engineering*, vol. 122, no. 11, pp. 617-624, 1996, doi: doi:10.1061/(ASCE)0733-9429(1996)122:11(617).
- [79] D. S. Kim, S. H. Yoon, D. H. Lee, and K. C. Kim, "Flow and heat transfer measurements of a wall attaching offset jet," *International Journal of Heat and Mass Transfer*, vol. 39, no. 14, pp. 2907-2913, 1996, doi: 10.1016/0017-9310(95)00383-5.
- [80] E. Vishnuvardhanarao and M. K. Das, "Computation of mean flow and thermal characteristics of incompressible turbulent offset jet flows," *Numerical Heat Transfer, Part A: Applications*, vol. 53, no. 8, pp. 843-869, 2007, doi: 10.1080/10407780701715760.
- [81] H. B. Song, S. H. Yoon, and D. H. Lee, "Flow and heat transfer characteristics of a two-dimensional oblique wall attaching offset jet," *International Journal of Heat and Mass Transfer*, vol. 43, no. 13, pp. 2395-2404, 2000, doi: 10.1016/S0017-9310(99)00312-9.
- [82] A. Nasr and J. C. S. Lai, "Comparison of flow characteristics in the near field of two parallel plane jets and an offset plane jet," *Physics of Fluids*, vol. 9, no. 10, pp. 2919-2931, 1997, doi: 10.1063/1.869404.
- [83] A. Kumar, "Mean flow and thermal characteristics of a turbulent dual jet consisting of a plane wall jet and a parallel offset jet," *Numerical Heat Transfer, Part A (Applications)*, vol. 67, no. 10, pp. 1075-96, 2015, doi: 10.1080/10407782.2014.955348.
- [84] N. Hnaïen, S. Marzouk, H. Ben Aïssia, and J. Jay, "CFD investigation on the offset ratio effect on thermal characteristics of a combined wall and offset jets flow," *Heat and Mass Transfer*, journal article vol. 53, no. 8, pp. 2531-2549, 2017, doi: 10.1007/s00231-017-2000-0.
- [85] N. Hnaïen, S. Marzouk khairallah, L. Kolsi, A. Al-Rashed, H. Ben Aïssia, and J. Jay, "CFD investigation on the steady interaction between an offset jet and an oblique wall jet," *Journal of Applied Fluid Mechanics*, vol. 11, pp. 885-894, 2018, doi: 10.29252/jafm.11.04.28214.
- [86] N. Hnaïen, S. Marzouk, L. Kolsi, A. S. Alsagri, H. Ben Aïssia, and J. Jay, "Offset jet ejection angle effect in combined wall and offset jets flow: Numerical investigation and engineering correlations," *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, journal article vol. 41, no. 11, p. 479, 2019, doi: 10.1007/s40430-019-1982-6.
- [87] T. P. Singh, A. Kumar, and A. K. Satapathy, "Heat transfer and fluid flow characteristics of a turbulent dual jet impinging on a wavy surface," *Journal of Thermal Science and Engineering Applications*, vol. 12, no. 4, 2020, doi: 10.1115/1.4045882.
- [88] T. P. Singh, A. Kumar, and A. K. Satapathy, "Role of a sinusoidal wavy surface in enhancement of heat transfer using turbulent dual jet," *Journal of Heat Transfer*, vol. 143, no. 3, 2021, doi: 10.1115/1.4049274.
-

- [89] L. Zhao and J. Sun, "Numerical investigation on a dual-jet consisting of a plane wall jet and a parallel offset jet at low-reynolds number," *Transactions of Nanjing University of Aeronautics & Astronautics*, vol. 35, no. 5, pp. 778-88, 2018, doi: 10.16356/j.1005-1120.2018.05.778.
- [90] V. Sidiropoulos and J. Vlachopoulos, "The effects of dual-orifice air-ring design on blown film cooling," *Polymer Engineering & Science*, vol. 40, no. 7, pp. 1611-1618, 2000, doi: 10.1002/pen.11292.
- [91] N. Gao, S. Li, and D. Ewing, "Investigation of the cooling jets used in the blown film process," *International Polymer Processing*, vol. 20, no. 1, pp. 68-77, 2005, doi: doi:10.3139/217.1866.
- [92] Z. Li, W. Huai, and Z. Yang, "Interaction between wall jet and offset jet with different velocity and offset ratio," *Procedia Engineering*, vol. 28, pp. 49-54, 2012, doi: 10.1016/j.proeng.2012.01.681.
- [93] J. R. R. Pelfrey and J. A. Liburdy, "Mean flow characteristics of a turbulent offset jet," *Journal of Fluids Engineering*, vol. 108, no. 1, pp. 82-88, 1986, doi: 10.1115/1.3242548.
- [94] E. Tanaka, "The interference of two-dimensional parallel jets : 1st report, experiments on dual jet," *Bulletin of JSME*, vol. 13, no. 56, pp. 272-280, 1970, doi: 10.1299/jsme1958.13.272.
- [95] R. E. Spall, E. A. Anderson, and J. Allen, "Momentum flux in plane, parallel jets," *Journal of Fluids Engineering*, vol. 126, no. 4, pp. 665-670, 2004, doi: 10.1115/1.1778717.
- [96] T. Mondal, A. Guha, and M. K. Das, "Computational study of periodically unsteady interaction between a wall jet and an offset jet for various velocity ratios," *Computers & Fluids*, vol. 123, pp. 146-161, 2015, doi: 10.1016/j.compfluid.2015.09.015.
- [97] T. Mondal, M. K. Das, and A. Guha, "Transition of a steady to a periodically unsteady flow for various jet widths of a combined wall jet and offset jet," *Journal of Fluids Engineering, Transactions of the ASME*, vol. 138, no. 7, 2016, doi: 10.1115/1.4032750.
- [98] T. P. Singh and A. Dewan, "Transient flow and thermal transport characteristics of wall-bounded turbulent dual jet with heated undulated wall," *International Journal of Thermal Sciences*, vol. 182, p. 107800, 2022, doi: 10.1016/j.ijthermalsci.2022.107800.
- [99] T. P. Singh, A. Kumar, and A. K. Satapathy, "Effect of wavy wall surface on flow structure and thermal characteristics of a turbulent dual jet comprising of a wall jet and an offset jet," *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, vol. 235, no. 3, pp. 524-545, 2021, doi: 10.1177/0957650920925945.
- [100] T. P. Singh, A. Kumar, and A. K. Satapathy, "Fluid flow analysis of a turbulent offset jet impinging on a wavy wall surface," *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, vol. 234, no. 2, pp. 544-563, 2019, doi: 10.1177/0954406219880209.
- [101] T. P. Singh and A. Dewan, "Performance assessment of different turbulence models for a dual jet flowing over a heated sinusoidal wavy surface," *Journal of Thermal Science and Engineering Applications*, vol. 14, no. 5, 2021, doi: 10.1115/1.4052854.
- [102] T. Mondal, N. Hnaien, M. Ajmi, K. Ghachem, and L. Kolsi, "CFD investigation of thermal characteristics for a dual jet with a parallel co-flow," *ACS Omega*, vol. 7, no. 32, pp. 27864-27875, 2022, doi: 10.1021/acsomega.2c00609.
- [103] N. Hnaien, T. Mondal, M. Ajmi, W. Aich, B. Ayadi, and L. Kolsi, "A CFD modelling on effects of ejection angle of a co-flow on the thermal characteristics for a combined wall and offset jet flow," *Alexandria Engineering Journal*, vol. 68, pp. 83-91, 2023, doi: 10.1016/j.aej.2023.01.015.

- [104] N. Hnaïen, T. Mondal, M. Ajmi, S. Marzouk, W. Aich, and L. Kolsi, "Numerical study on mean flow characteristics for a combined turbulent wall and offset jets with a parallel co-flow," *Journal of Taibah University for Science*, vol. 16, no. 1, pp. 732-748, 2022, doi: 10.1080/16583655.2022.2109936.
- [105] T. Mondal and S. M. O'Shaughnessy, "Numerical investigation of conjugate heat transfer to a turbulent dual offset jet," *International Journal of Thermal Sciences*, vol. 180, p. 107716, 2022, doi: 10.1016/j.ijthermalsci.2022.107716.
- [106] T. Mondal, N. Srivastava, S. M. O'Shaughnessy, and S. Pramanik, "Comparison of the mean flow and turbulence characteristics of a single offset jet and a dual offset jet," *European Journal of Mechanics - B/Fluids*, vol. 98, pp. 161-179, 2023, doi: 10.1016/j.euromechflu.2022.12.003.
- [107] T. P. Singh and A. Dewan, "Improvement in cooling using a sinusoidal wavy surface for a turbulent dual jet: A computational study," *Journal of Thermal Analysis and Calorimetry*, 2022, doi: 10.1007/s10973-022-11695-9.
- [108] S. S. Rathore and S. K. Verma, "Computational investigation on the thermal and flow characteristics of a turbulent dual jet impinging on a heated moving surface," *Journal of Heat Transfer*, vol. 144, no. 9, 2022, doi: 10.1115/1.4054717.
- [109] R. D. Mehta and P. Bradshaw, "Design rules for small low speed wind tunnels," *The Aeronautical Journal*, vol. 83, no. 827, pp. 443-453, 1979, doi: 10.1017/S0001924000031985.
- [110] D. Jackson, "Active fluidic control of a cylinder in a cross-flow using synthetic jet actuation," PhD Thesis, Dept. of Mechanical and Manufacturing Engineering, Trinity College Dublin, 2016.
- [111] V. Kulkarni, N. Sahoo, and S. D. Chavan, "Simulation of honeycomb–screen combinations for turbulence management in a subsonic wind tunnel," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 99, no. 1, pp. 37-45, 2011, doi: 10.1016/j.jweia.2010.10.006.
- [112] K. Ghorbanian, M. R. Soltani, and M. D. Manshadi, "Experimental investigation on turbulence intensity reduction in subsonic wind tunnels," *Aerospace Science and Technology*, vol. 15, no. 2, pp. 137-147, 2011, doi: 10.1016/j.ast.2010.06.009.
- [113] S. H. Amirnordin, S. M. Seri, W. Salim, H. A. Rahman, and K. Hasnan, "Pressure drop analysis of square and hexagonal cells and its effects on the performance of catalytic converters," *International Journal of Environmental Science and Development*, vol. 2, no. 3, pp. 239-247, 2011, doi: 10.7763/IJESD.2011.V2.131.
- [114] J. L. Lumley and J. McMahon, "Reducing water tunnel turbulence by means of a honeycomb," *Journal of Basic Engineering*, vol. 89, no. 4, pp. 764-770, 1967, doi: 10.1115/1.3609700.
- [115] K. Wieghardt, "On the resistance of screens," *The Aeronautical Quarterly*, vol. 4, no. 2, pp. 186-192, 1953, doi: 10.1017/S0001925900000871.
- [116] R. Collar, *The effect of gauze on the velocity distribution in a uniform duct* (Issue 1867 of british Aeronautical Research Committee. Reports and memoranda). H.M. Stationery Office, 1939.
- [117] T. Morel, "Design of two-dimensional wind tunnel contractions," *Journal of Fluids Engineering*, vol. 99, no. 2, pp. 371-377, 1977, doi: 10.1115/1.3448764.
- [118] B. Stratford, "The prediction of separation of the turbulent boundary layer," *Journal of fluid mechanics*, vol. 5, no. 1, pp. 1-16, 1959, doi: 10.1017/S0022112059000015.
- [119] M. Gibbons, C. Howe, P. Di Marco, and A. Robinson, "Local heat transfer to an evaporating sessile droplet in an electric field," in *Journal of Physics: Conference Series*, 2016, vol. 745, no. 3: IOP Publishing, p. 032066, doi: 10.1088/1742-6596/745/3/032066.

- [120] K. Brown, T. Persoons, and D. B. Murray, "Examination of the local and fluctuating heat transfer due to swirling jets impinging upon a heated horizontal surface," in *Proceedings of the 6th Baltic Heat Transfer Conference*, Tampere, Finland, August 24-26 2011.
- [121] H. Nakamura, "Frequency response and spatial resolution of a thin foil for heat transfer measurements using infrared thermography," *International Journal of Heat and Mass Transfer*, vol. 52, no. 21, pp. 5040-5045, 2009, doi: 10.1016/j.ijheatmasstransfer.2009.04.019.
- [122] Special Metals. Inconel® alloy 625 [Online] Available: <https://www.specialmetals.com/documents/technical-bulletins/inconel/>
- [123] D. Lee, Y. Chung, and D. Kim, "Turbulent flow and heat transfer measurements on a curved surface with a fully developed round impinging jet," *International Journal of Heat Fluid Flow*, vol. 18, no. 1, pp. 160-169, 1997, doi: 10.1016/S0142-727X(96)00136-1.
- [124] T. S. O'Donovan, T. Persoons, and D. B. Murray, "High-resolution hot-film measurement of surface heat flux to an impinging jet," *Measurement Science and Technology*, vol. 22, no. 10, p. 105402, 2011, doi: 10.1088/0957-0233/22/10/105402.
- [125] K. Brown, "Heat transfer characteristics of swirling impinging jets," PhD, Dept. of Mechanical and Manufacturing Engineering, PhD Thesis, Trinity College Dublin, 2013.
- [126] T. A. McGuinn, "A study of the heat transfer processes and related flows of a synthetic jet impinging perpendicular to a heated surface," PhD Thesis, Trinity College Dublin, 2011.
- [127] S. Roux, M. Fénot, G. Lalizel, L. E. Brizzi, and E. Dorignac, "Evidence of flow vortex signatures on wall fluctuating temperature using unsteady infrared thermography for an acoustically forced impinging jet," *International Journal of Heat and Fluid Flow*, vol. 50, pp. 38-50, 2014, doi: 10.1016/j.ijheatfluidflow.2014.05.010.
- [128] S. W. Chang, S. F. Chiou, and S. F. Chang, "Heat transfer of impinging jet array over concave-dimpled surface with applications to cooling of electronic chipsets," *Experimental Thermal and Fluid Science*, vol. 31, no. 7, pp. 625-640, 2007, doi: 10.1016/j.expthermflusci.2006.06.008.
- [129] S. Fusetto, A. Aprile, P. Malcovati, and E. Bonizzoni, "Readout ic architectures and strategies for uncooled micro-bolometers infrared focal plane arrays: A review," *Sensors*, vol. 23, no. 5, p. 2727, 2023, doi: 10.3390/s23052727.
- [130] A. Lau, "The narcissus effect in infrared optical scanning systems," in *Proceedings of SPIE/SPSE Technical Symposium East*, Reston, United States, 26 September 1977, vol. 0107, in Stray Light Problems in Optical Systems, doi: 10.1117/12.964596.
- [131] FLIR, "User's manual FLIR A6XX series," ed: FLIR systems Wilsonville, OR, USA, 2016.
- [132] M. Raffel, C. E. Willert, F. Scarano, C. J. Kähler, S. T. Wereley, and J. Kompenhans, *Particle image velocimetry: A practical guide*. Springer, 2018.
- [133] B. Patte-Rouland, G. Lalizel, J. Moreau, and E. Rouland, "Flow analysis of an annular jet by particle image velocimetry and proper orthogonal decomposition," *Measurement Science and Technology*, vol. 12, no. 9, p. 1404, 2001, doi: 10.1088/0957-0233/12/9/305.
- [134] A. Assoudi, S. Habli, N. M. Saïd, H. Bournot, and G. Le Palec, "Experimental and numerical study of an offset jet with different velocity and offset ratios," *Engineering Applications of Computational Fluid Mechanics*, vol. 9, no. 1, pp. 490-512, 2015, doi: 10.1080/19942060.2015.1071525.
- [135] S. O'Shaughnessy, "Bubble-induced marangoni convection," PhD Thesis, Department of Mechanical and Manufacturing Engineering, Trinity College Dublin, 2011.

-
- [136] A. Melling, "Tracer particles and seeding for particle image velocimetry," *Measurement science and technology*, vol. 8, no. 12, p. 1406, 1997, doi: 10.1088/0957-0233/8/12/005.
- [137] M. Samimy and S. K. Lele, "Motion of particles with inertia in a compressible free shear layer," *Physics of Fluids A: Fluid Dynamics*, vol. 3, no. 8, pp. 1915-1923, 1991, doi: 10.1063/1.857921.
- [138] D. J. Lockwood, "Rayleigh and mie scattering," in *Encyclopedia of color science and technology*, M. R. Luo Ed. New York, NY: Springer New York, 2016, pp. 1097-1107.
- [139] LaVision GmbH, "Aerosol generator product-manual," 2018.
- [140] B. Wieneke, "PIV uncertainty quantification and beyond," Delft University of Technology, 2017.
- [141] T. Persoons and T. S. O'Donovan, "High dynamic velocity range particle image velocimetry using multiple pulse separation imaging," *Sensors*, vol. 11, no. 1, pp. 1-18, 2010, doi: 10.3390/s110100001.
- [142] R. D. Keane and R. J. Adrian, "Optimization of particle image velocimeters: II. Multiple pulsed systems," *Measurement science and technology*, vol. 2, no. 10, p. 963, 1991, doi: 10.1088/0957-0233/2/10/013.
- [143] A. K. Azad, A. Abid, C. N. Mithun, M. J. Hasan, R. Hossain, and M. M. Rahman, "Effect of richardson number on transient double diffusive mixed convection: A thermo-hydrodynamic study," *International Journal of Thermofluids*, vol. 17, p. 100273, 2023, doi: 10.1016/j.ijft.2022.100273.
- [144] D. B. Donoghue, "Bubble impingement and the mechanisms of heat transfer enhancement," Ph.D Thesis, Dept. of Mechanical and Manufacturing Engineering, Trinity College Dublin, 2014.
- [145] B. Donnelly, "The effects of a sliding air bubble on heat transfer from an inclined surface," Ph.D., Mechanical and Manufacturing Engineering, Trinity College Dublin, 2011.
- [146] M. J. Gibbons, A. I. Garivalis, S. O'Shaughnessy, P. Di Marco, and A. J. Robinson, "Evaporating hydrophilic and superhydrophobic droplets in electric fields," *International Journal of Heat and Mass Transfer*, vol. 164, p. 120539, 2021, doi: 10.1016/j.ijheatmasstransfer.2020.120539.
- [147] R. O. R. Meehan, "Flow structures and heat transfer enhancement in the wakes of sliding bubbles," Ph.D., Mechanical and Manufacturing Engineering, Trinity College Dublin, 2016.
- [148] O. Raghu and J. Philip, "Thermal properties of paint coatings on different backings using a scanning photo acoustic technique," *Measurement science and technology*, vol. 17, no. 11, p. 2945, 2006, doi: 10.1088/0957-0233/17/11/012.
- [149] A. Ianiro, "Fundamentals and recent advances in particle image velocimetry and lagrangian particle tracking: Image pre-processing methods," in *von Karman Institute for Fluid Dynamics Lecture Series 2022-01*, 2021.
- [150] LaVision GmbH. (2018). Davis 10.0 software product-manual.
- [151] F. Scarano, "Fundamentals and recent advances in particle image velocimetry and lagrangian particle tracking: PIV: Background and working principles," in *von Karman Institute for Fluid Dynamics Lecture Series 2022-01*, 2021.
- [152] T. Astarita, "Fundamentals and recent advances in particle image velocimetry and lagrangian particle tracking: PIV image processing techniques," in *von Karman Institute for Fluid Dynamics Lecture Series 2022-01*, 2021.
- [153] LaVision GmbH. (2018). Flowmaster product-manual for davis 10.
- [154] B. Munson, D. Young, and T. Okiishi, "Fundamentals of fluid mechanics," 6th ed, 1998.
- [155] A. E. Mudau, C. Willers, D. Griffith, and F. Roux, "Non-uniformity correction and bad pixel replacement on lwir and MWIR images," in *Saudi International Electronics, Communications and*
-

- Photonics Conference 2011, SIEPCPC 2011*, Riyadh, Saudi Arabia, 2011, pp. 1-5, doi: 10.1109/SIEPCPC.2011.5876937.
- [156] *Guide to the expression of uncertainty in measurement*, 1995.
- [157] L. Kirkup, *Calculating and expressing uncertainty in measurement*. Department of Applied Physics, Faculty of Science, University of Technology, Australia, 2007.
- [158] H. W. Coleman and W. G. Steele, *Experimentation, validation, and uncertainty analysis for engineers*. John Wiley & Sons, 2018.
- [159] S. Adatrao and A. Sciacchitano, "Fundamentals and recent advances in particle image velocimetry and lagrangian particle tracking: PIV errors and uncertainty quantification," in *von Karman Institute for Fluid Dynamics Lecture Series 2022-01*, 2021.
- [160] B. Wieneke, "PIV uncertainty quantification from correlation statistics," *Measurement Science and Technology*, vol. 26, no. 7, p. 074002, 2015, doi: 10.1088/0957-0233/26/7/074002.
- [161] N. Gao and D. Ewing, "Experimental investigation of planar offset attaching jets with small offset distances," *Experiments in Fluids*, vol. 42, no. 6, p. 941, 2007, doi: 10.1007/s00348-007-0305-3.
- [162] M. Nitsche, "Vortex dynamics," in *Encyclopedia of mathematical physics*, J.-P. Francoise, G. L. Naber, and T. S. Tsun Eds. Oxford: Academic Press, 2006, pp. 390-399.
- [163] F. Hussain and J. Jeong, "On the identification of a vortex," *Journal of Fluid Mechanics*, vol. 285, pp. 69-94, 1995, doi: 10.1017/S0022112095000462.
- [164] H. Vollmers, "Detection of vortices and quantitative evaluation of their main parameters from experimental velocity data," *Measurement Science and Technology*, vol. 12, no. 8, p. 1199, 2001, doi: 10.1088/0957-0233/12/8/329.
- [165] D. Anderson John, *Computational fluid dynamics: The basics with applications*. McGraw-Hill Science, 1995.
- [166] ANSYS Inc, *Ansys Fluent 2023R2 theory guide*. Canonsburg, PA 15317, 2023.
- [167] B. Andersson, R. Andersson, L. Håkansson, M. Mortensen, R. Sudiyo, and B. Van Wachem, *Computational fluid dynamics for engineers*. Cambridge university press, 2011.
- [168] H. K. Versteeg and W. Malalasekera, *An introduction to computational fluid dynamics: The finite volume method*. Pearson education, 2007.
- [169] N. P. Weatherill, "Mixed structured-unstructured meshes for aerodynamic flow simulation," *The Aeronautical Journal*, vol. 94, no. 934, pp. 111-123, 1990, doi: 10.1017/S000192400002265X.
- [170] F. Menter, R. Lechner, and A. Matyushenko, "Best practice: RANS turbulence modeling in Ansys CFD," *ANSYS Inc.: Canonsburg, PA, USA*, 2021.
- [171] D. C. Wilcox, *Turbulence modeling for CFD*. DCW industries La Canada, CA, 1994.
- [172] ANSYS Inc, *Ansys Fluent 2023R2 user's guide*. Canonsburg, PA 15317, 2023.
- [173] F. R. Menter, R. Langtry, and S. Völker, "Transition modelling for general purpose CFD codes," *Flow, Turbulence and Combustion*, vol. 77, no. 1, pp. 277-303, 2006, doi: 10.1007/s10494-006-9047-1.
- [174] J. W. Slater. "Verification assessment." NASA. <https://www.grc.nasa.gov/www/wind/valid/tutorial/verassess.html> (accessed 2021).
- [175] P. J. Roache, "Quantification of uncertainty in computational fluid dynamics," *Annual review of fluid Mechanics*, vol. 29, no. 1, pp. 123-160, 1997, doi: 10.1146/annurev.fluid.29.1.123.

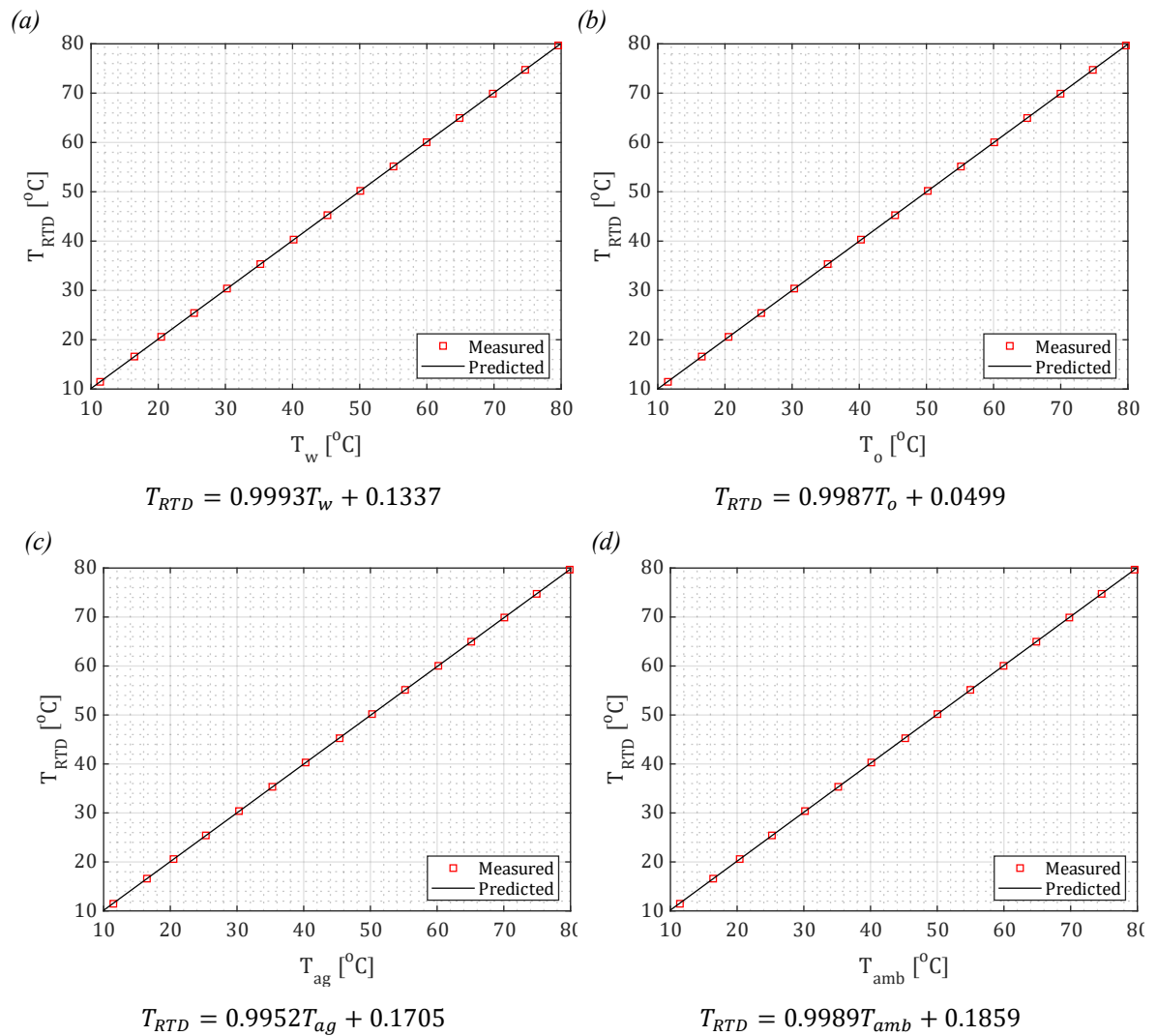
- [176] T. Xing and F. Stern, "Factors of safety for richardson extrapolation," *Journal of Fluids Engineering*, vol. 132, no. 6, 2010, doi: 10.1115/1.4001771.
- [177] J. W. Slater. "Validation assessment." NASA. <https://www.grc.nasa.gov/www/wind/valid/tutorial/valassess.html> (accessed 2021).
- [178] J. T. Holland and J. A. Liburdy, "Measurements of the thermal characteristics of heated offset jets," *International Journal of Heat and Mass Transfer*, vol. 33, no. 1, pp. 69-78, 1990, doi: 10.1016/0017-9310(90)90142-H.
- [179] J. R. R. Pelfrey and J. A. Liburdy, "Effect of curvature on the turbulence of a two-dimensional jet," *Experiments in Fluids*, vol. 4, no. 3, pp. 143-149, 1986, doi: 10.1007/BF00280264.
- [180] H. Sato and F. Sakao, "An experimental investigation of the instability of a two-dimensional jet at low reynolds numbers," *Journal of Fluid Mechanics*, vol. 20, no. 2, pp. 337-352, 1964, doi: 10.1017/S0022112064001264.
- [181] S. Alimohammadi, D. B. Murray, and T. Persoons, "Experimental validation of a computational fluid dynamics methodology for transitional flow heat transfer characteristics of a steady impinging jet," *Journal of Heat Transfer*, vol. 136, no. 9, p. 091703, 2014, doi: 10.1115/1.4027840.
- [182] J. C. Tannehill, D. Anderson, and R. H. Pletcher, *Computational fluid mechanics and heat transfer*. CRC press, 1997.
- [183] S. Alimohammadi, "A validated computational fluid dynamics methodology for characterisation of fluid flow & heat transfer in unsteady jets," PhD Thesis, Trinity College Dublin, 2016.
- [184] A. Chambers, R. Antonia, and L. Fulachier, "Turbulent prandtl number and spectral characteristics of a turbulent mixing layer," *International journal of heat and mass transfer*, vol. 28, no. 8, pp. 1461-1468, 1985, doi: 10.1016/0017-9310(85)90248-0.
- [185] D. Divoky and E. Mayer, "Correlation of intermittency with preferential transport of heat and chemical species in turbulent shear flows," *AIAA Journal*, vol. 4, no. 11, pp. 1995-2000, 1966, doi: 10.2514/3.3830.
- [186] B. E. Launder, "Heat and mass transport," in *Turbulence*, P. Bradshaw Ed. Berlin, Heidelberg: Springer Berlin Heidelberg, 1978, pp. 231-287.
- [187] F. R. Menter, R. B. Langtry, S. R. Likki, Y. B. Suzen, P. G. Huang, and S. Völker, "A correlation-based transition model using local variables—part i: Model formulation," *Journal of Turbomachinery*, pp. 413-422, 2004, doi: 10.1115/1.2184352.
- [188] P. Murphy, T. Persoons, and S. O'Shaughnessy, "The use of hot film sensors for the heat flux measurement of impinging jet flows: A comparison of postprocessing methods," *Heat and Mass Transfer*, 2022, doi: 10.1007/s00231-022-03297-w.

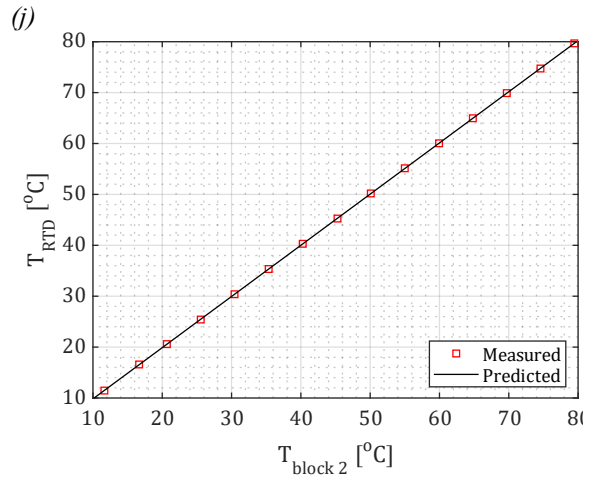
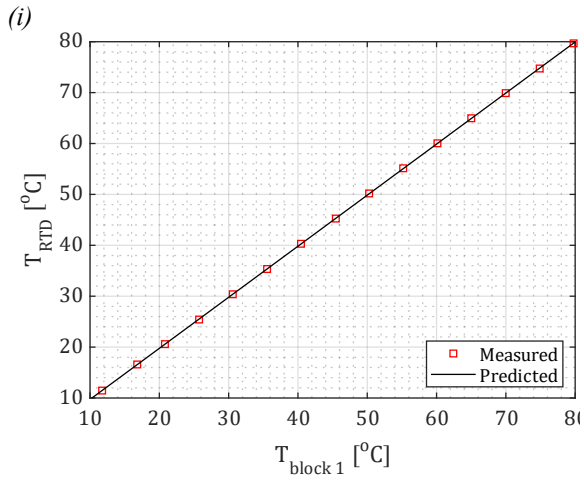
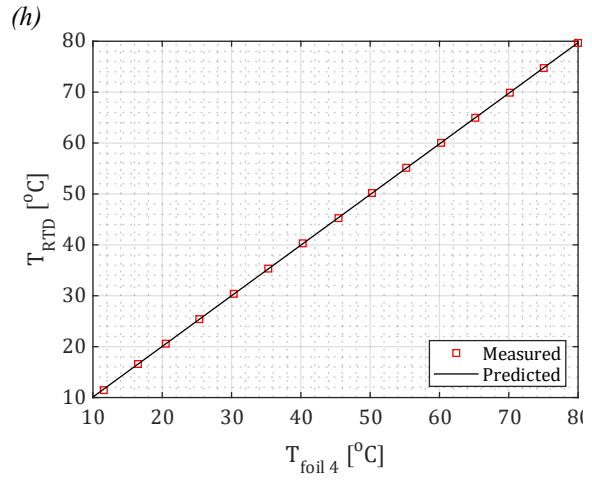
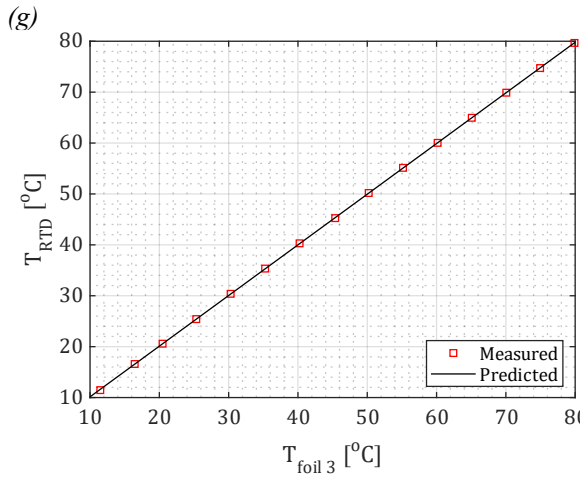
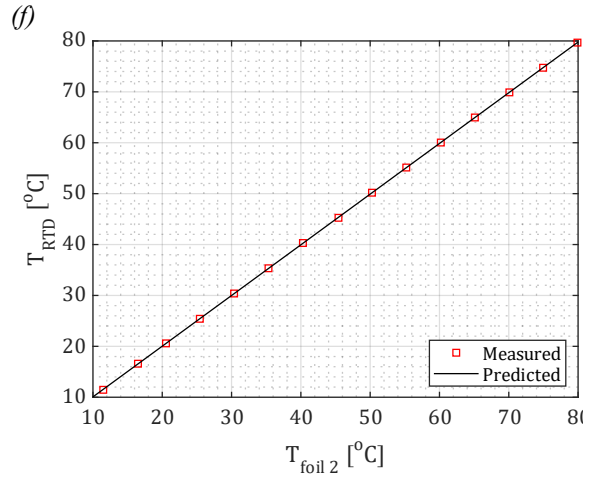
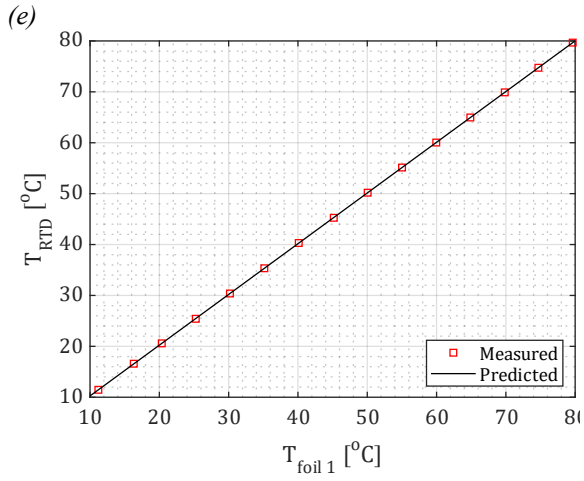
Appendix A

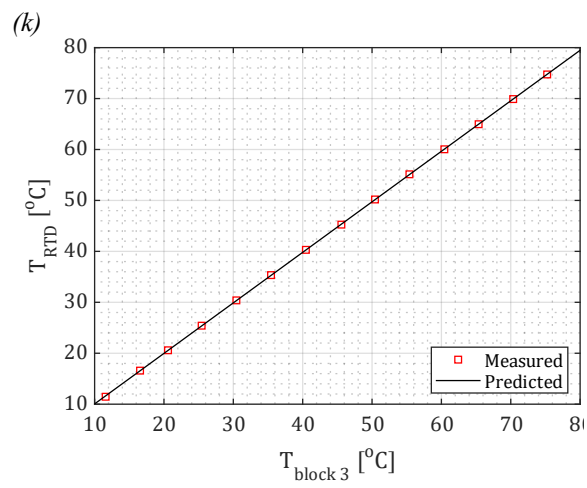
Thermocouple Calibration Curves

Curves

Presented below are the thermocouple calibration curves used during the experimental investigation.







$$T_{RTD} = 0.9917T_{\text{block}_3} + 0.1395$$

Appendix B

Tabulated Data for mp , cp , UVC and LVC

Presented below are the merge point, combined point, upper vortex centre and lower vortex centre location data for the cases discussed in tabulated form.

Table B.1: Locations of mp , cp , UVC and LVC over the range of Re and OR investigated with $V_r = 1$.

Re	OR	V_r	(X_{mp}, Y_{mp})	(X_{cp}, Y_{cp})	(X_{UVC}, Y_{UVC})	(X_{LVC}, Y_{LVC})
5500	1	1	(0.90, 1.33)	(4.94, 1.19)	(0.49, 1.58)	(0.47, 1.15)
6000	1	1	(0.88, 1.33)	(4.99, 1.24)	(0.49, 1.57)	(0.45, 1.15)
7000	1	1	(0.87, 1.34)	(4.86, 1.35)	(0.49, 1.57)	(0.68, 1.21)
8000	1	1	(0.85, 1.34)	(4.78, 1.32)	(0.48, 1.57)	(0.46, 1.17)
9000	1	1	(0.85, 1.33)	(4.93, 1.24)	(0.49, 1.56)	(0.47, 1.17)
10000	1	1	(0.82, 1.35)	(4.96, 1.27)	(0.52, 1.55)	(0.47, 1.18)
11000	1	1	(0.81, 1.34)	(5.63, 1.14)	(0.54, 1.54)	(0.45, 1.17)
12000	1	1	(0.78, 1.35)	(5.23, 1.22)	(0.51, 1.55)	(0.47, 1.19)
5500	2	1	(2.68, 1.64)	(5.79, 1.74)	(2.08, 2.07)	(2.00, 1.44)
6000	2	1	(2.61, 1.62)	(5.13, 1.64)	(2.02, 2.06)	(1.96, 1.43)
7000	2	1	(2.58, 1.64)	(5.11, 1.69)	(1.95, 2.08)	(1.90, 1.43)
8000	2	1	(2.48, 1.64)	(4.85, 1.82)	(1.79, 2.12)	(1.77, 1.44)
9000	2	1	(2.38, 1.68)	(4.73, 1.85)	(1.71, 2.12)	(1.67, 1.45)
10000	2	1	(2.34, 1.65)	(4.81, 1.32)	(1.65, 2.13)	(1.61, 1.45)
11000	2	1	(2.33, 1.67)	(4.72, 1.69)	(1.66, 2.13)	(1.59, 1.45)
12000	2	1	(2.26, 1.66)	(4.62, 1.77)	(1.56, 2.15)	(1.52, 1.43)
5500	3	1	(4.68, 1.67)	(7.67, 1.72)	(3.56, 2.45)	(3.59, 1.42)
6000	3	1	(4.78, 1.69)	(7.98, 1.61)	(3.56, 2.47)	(3.60, 1.42)
7000	3	1	(4.74, 1.75)	(7.79, 1.58)	(3.52, 2.50)	(3.51, 1.42)
8000	3	1	(4.79, 1.71)	(7.97, 1.53)	(3.44, 2.50)	(3.40, 1.41)
9000	3	1	(4.81, 1.72)	(8.22, 1.56)	(3.35, 2.56)	(3.35, 1.41)
10000	3	1	(4.74, 1.73)	(7.99, 1.58)	(3.28, 2.57)	(3.25, 1.42)
11000	3	1	(4.70, 1.72)	(7.89, 1.56)	(3.25, 2.57)	(3.23, 1.42)
12000	3	1	(4.55, 1.71)	(7.67, 1.51)	(3.10, 2.60)	(3.06, 1.42)

Table B.2: Locations of mp , cp , UVC and LVC over the range of V_r and OR investigated with $Re_{tot} = 16000$.

V_r	OR	(X_{mp}, Y_{mp})	(X_{cp}, Y_{cp})	(X_{UVC}, Y_{UVC})	(X_{LVC}, Y_{LVC})
0.5	1	(1.52, 1.58)	(4.11, 0.77)	(1.52, 1.58)	(1.20, 1.31)
0.6	1	(1.64, 1.44)	(4.87, 0.72)	(1.42, 1.54)	(1.14, 1.22)
0.80	1	(1.15, 1.43)	(5.23, 0.56)	(0.93, 1.54)	(0.95, 1.23)
1	1	(0.85, 1.34)	(4.78, 1.32)	(0.48, 1.57)	(0.46, 1.17)
1.2	1	(0.93, 1.28)	(4.48, 1.43)	(0.53, 1.55)	(0.56, 1.16)
1.67	1	(1.58, 1.18)	(3.98, 1.48)	(1.26, 1.40)	(1.40, 1.11)
2	1	(1.63, 1.05)	(3.47, 1.45)	(1.47, 1.35)	(1.62, 1.05)
0.5	2	(3.41, 2.00)	(5.75, 1.24)	(2.60, 2.29)	(2.48, 1.56)
0.6	2	(3.46, 1.83)	(6.35, 0.80)	(2.61, 2.20)	(2.50, 1.46)
0.80	2	(2.94, 1.76)	(5.68, 0.80)	(2.25, 2.15)	(2.13, 1.46)
1	2	(2.48, 1.64)	(4.85, 1.82)	(1.79, 2.12)	(1.77, 1.44)
1.2	2	(2.35, 1.56)	(4.68, 1.85)	(1.72, 2.08)	(1.74, 1.41)
1.67	2	(2.54, 1.44)	(4.35, 1.87)	(1.83, 1.98)	(2.06, 1.33)
2	2	(2.44, 1.35)	(3.99, 1.77)	(1.89, 1.86)	(2.04, 1.27)
0.5	3	(4.27, 2.59)	(6.15, 1.53)	(3.07, 3.02)	(3.46, 2.01)
0.6	3	(4.99, 2.10)	(7.24, 1.19)	(3.55, 2.74)	(3.51, 1.56)
0.80	3	(4.46, 1.94)	(6.57, 1.03)	(3.19, 2.67)	(3.17, 1.51)
1	3	(4.79, 1.71)	(7.97, 1.53)	(3.44, 2.50)	(3.40, 1.41)
1.2	3	(4.17, 1.72)	(6.52, 2.03)	(3.05, 2.53)	(3.07, 1.45)
1.67	3	(3.56, 1.55)	(5.47, 2.06)	(2.58, 2.39)	(2.75, 1.39)
2	3	(3.79, 1.47)	(5.36, 1.74)	(2.69, 2.31)	(2.85, 1.33)

Appendix C

Calibration Certification

The calibration certification corresponding to the ASL F100 Precision Handheld RTD thermometer, the AVT-D-R Air Velocity Transmitter and the Digitron 2000 Series Manometer are, respectively, presented below.

CERTIFICATE OF CALIBRATION

DATE OF ISSUE 23 December 2011 CERTIFICATE NUMBER N7114093T



0013

Trescal Limited

Unit 2

106 Hawley Lane

Farnborough

Hampshire

GU14 8EH

Telephone

01252 533300

Facsimile

01252 533333

Email

ukcms@trescal.com

Website

www.trescal.com

Trescal

PAGE 1 OF 3 PAGES

APPROVED SIGNATORY

☒ J.A.Struthers☐ D.Minton☐☐

Customer :

Address :

HARTEST PRECISION INSTRUMENTS

2 GATTON BUSINESS PARK

WELLS PLACE

RED HILL, SURREY, RH1 3LG

LABQUIP (IRELAND)LTD, 12 THE BUSINESS CENTRE

FONTHILL IND PARK, CLONDALKIN, DUBLIN 22, IRELAND

Order No :

107117

Apparatus Tested -

PRT PROBE

Type No :

Pt100

Serial No :

PO 106095-1-3

Inventory No :

Manufacturer :

HARTEST

Range/Scale :

CALIBRATED RANGE: -50 TO 200 °C

Test Conditions -

Date Instrument Received : 01 December 2011

Date Calibration Completed : 23 December 2011

Ambient Temperature : 20 ± 2 °C

Reference No : 7114093

Certified by

The results given within this certificate only relate to the item calibrated. The uncertainty limits quoted refer to the measured values only, with no account being taken of the instrument's ability to maintain its calibration. The reported expanded uncertainty is based upon a standard uncertainty multiplied by a coverage factor k=2 providing a confidence level of approximately 95%. The uncertainty evaluation has been derived from the EA-402 1999 uncertainty document.

This certificate provides traceability of measurement to recognised national standards, and to units of measurement realised at the National Physical Laboratory, other recognised national standards laboratories or approved ratio techniques. This certificate may not be reproduced other than in full, except with the prior written approval of the issuing laboratory.

22-005-079/010/MAR2008

CERTIFICATE OF CALIBRATION

UKAS Accredited Calibration Laboratory No. 0013

CERTIFICATE NUMBER

N7114093T

PAGE 2 OF 3 PAGES

PROBE LENGTH TOTAL : 70 mm

TIP DIAMETER: 3 mm

SERIAL NO: PO 106095-1-3

passed 4300

THE PLATINUM RESISTANCE THERMOMETER WAS CALIBRATED AGAINST TWO LABORATORY STANDARDS WHICH ARE TRACEABLE TO NATIONAL STANDARDS.

THE INSTRUMENT WAS CALIBRATED BY USING THE COMPARISON METHOD.

THE RESISTANCE WAS MEASURED BY PASSING A CURRENT OF 1 mA THROUGH THE PLATINUM RESISTANCE THERMOMETER.

THE PROBE WAS IMMERSSED TO A MINIMUM DEPTH OF 200 mm INSIDE A HERMETICALLY SEALED TUBE.

TEST TEMPERATURE	MEASURED RESISTANCE OF PROBE	EQUIVALENT TEMPERATURE BS EN 60751:1996
°C	OHMS	°C
-0.012	100.0070	0.018
-49.921	80.3483	-49.894
-0.003	100.0092	0.023
100.090	138.5556	100.132
199.980	175.8981	200.115
-0.017	100.0062	0.015

THE ABOVE RESULTS WERE USED TO CALCULATE THE FOLLOWING COEFFICIENTS.

R0 = 100.01266

A = 3.906623 D-03

B = -5.623143 D-07

C = -1.044285 D-011

THE TEMPERATURE SCALE IN USE IS THE INTERNATIONAL TEMPERATURE SCALE OF 1990, ITS - 90.

THE UNCERTAINTY ASSOCIATED WITH THIS MEASUREMENT IS ± 0.05 °C.

THE UNCERTAINTY OF THE RESISTANCE MEASUREMENT ON THE PROBE IS INCLUDED IN THIS UNCERTAINTY STATEMENT.

THE ABOVE UNCERTAINTY REFERS TO THE MEASUREMENT AND IS NOT INTENDED TO INDICATE THE SPECIFICATION OR REPEATABILITY OF THE INSTRUMENT.

Test Engineer



22-005-081/005/MAR2008

CERTIFICATE OF CALIBRATION

UKAS Accredited Calibration Laboratory No. 0013

CERTIFICATE NUMBER

N7114093T

PAGE 3 OF 3 PAGES

EQUIPMENT USED LIST

REFERENCE NUMBER	CAL DATE DUE
180990	July 2012
181119	December 2011
181121	April 2012
181139	December 2011
181140	June 2012
181163	June 2012
181183	June 2012
181252	June 2012
181308	June 2012
263861	December 2011

Test Engineer



22-005-081/005/MAR2008



CALIBRATION CERTIFICATE

27.4.2021

Serial Number: 37203

Device type: AVT-D-R
 Range: from 0 m/s
 to 10 m/s

Input	Display	Output 0-10 V			Output 4-20 mA		
Air Velocity (m/s)	AVT (m/s)	Out (V)	Error (V)	Error (%FS)	Out (mA)	Error (mA)	Error (%FS)
0	0	0,02	0,02	0,2	4,13	0,13	0,8
5	5,3	5,18	0,18	1,8	12,38	0,38	2,4
10	10	10,02	0,02	0,2	20,10	0,10	0,6

Conditions: Temperature 22,7 °C

Instruments used:

- x Schmidt Flow sn:205000050
- x Schmidt Flow sn:205000051
- x Fluke Multimeter 87V

Signature: Aatu Mäntymäki



Valid for 5 years*

*unit calibrated at 22,7 °C. This certificate does not include temperature shift or any inaccuracy caused by external disruption.

HK INSTRUMENTS Keihästie 7 FIN-40950 MUURAME FINLAND	Tel.	+ 358 14 337 2000	VAT: FI08730729
	Fax:	+ 358 14 337 2020	
	Email:	info@hkinstruments.fi www.hkinstruments.fi	

 Calibration Certificate - 2000 Pressure		Form 05 Rev. 2 04/04/2017 Page 1 of 1
Registered Address: Rototherm Group Kenfig Industrial Estate Margam, Port Talbot, SA13 2PW, United Kingdom Tel: +44 (0) 1656 740 551 Registered in Wales No. 2570730 W: www.digitron.com	Customer Address: TRINITY COLLEGE DUBLIN SARA DOHERTY - DEPT MECH ENG. PARSONS BUILDING TRINITY COLLEGE DUBLIN IRELAND	
Date of Issue	22 April 2021	Certificate Number
		41349

Customer Details

Company:	TRINITY COLLEGE DUBLIN	Customer/ Works Order No.:	E12949
Customer PO Number:	PC140421GByr		

Instrument Details

Product Code:	2082P7	Description:	2000 Series Differential Manometer
Serial Number:	4605110532	Range:	0 - 2 Bar
Customer ID No. (if req'd):	N/A	System Accuracy:	20°C to 30°C 0.1% rdg + 0.1%fs +1 digit -10°C to 50°C 0.15% rdg + 0.15% fs +1 digit
Condition of UUT:	Good.		

Traceability

Measurements were taken in these conditions:	
Temperature (°C)	21.5
Atmospheric Pressure (Mbar)	1012

Measurements were taken using listed test equipment in table below:		
Test Equipment Description	Test Equipment ID	UKAS Certificate No.
MENSOR CPC 6000	EQ.3299	20461 & N28162 to N28169

Test Results

Ref mbar/bar	UUT Rdg mbar/bar	PASS/FAIL	Allowable Tolerance mbar/bar
0.0	0.0	PASS	2.1
30.0	29.8	PASS	2.1
190.0	189.7	PASS	2.3
1000	999	PASS	4
1950	1949	PASS	5

The calibration was conducted in a temperature controlled environment maintained at 21 ± 2°C in accordance with the applicable calibration procedures of Rototherm Group using UKAS certified test equipment			
Tested By:	H SPAIN	Calibration Date	22 April 2021
Signature:	<i>H SPAIN</i>	Position Held	Calibration Technician
Checked by Competent Person Supervisor, Line Manager, QA, etc.		M Jones	