Numerical study on converging jets

Nguyen, Quang Chien

2015

Nguyen, Q. C. (2015). Numerical study on converging jets. Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/62947>

<https://doi.org/10.32657/10356/62947>

NUMERICAL STUDY ON CONVERGING JETS

NGUYEN QUANG CHIEN

School of Civil and Environmental Engineering

A thesis submitted to the Nanyang Technological University
in partial fulfilment of the requirement for the degree of
Doctor of Philosophy

2015

Acknowledgements

In writing this report I have been receiving various support and encouragement; without them the work would not likely to be completed.

First and foremost, I would like to appreciate Associate Professor Tan Soon Keat for guiding me during a long time period. Research work involves many small tasks, some are interesting, some are lengthy and may disappoint the PhD student if they failed to obtain good results. Fortunately, my supervisor sketched the big picture for this research topic and helped me resolve subtle bottlenecks and keep me away from misleading directions in research, so I could manage to put together all results obtained during the past five years.

No research is possible without being funded. NTU had provided generous stipend and offered learning facilities in an excellent campus environment, so I was able to concentrate on the work for such a long time. I would also thank NEWRI for provide continual funding, from conference fee to the expense for computer resources. In accordance to NEWRI research activities, professors and lecturers in the school have been very approachable whenever I need to clarify difficult concepts.

In addition to the method, tools (either physical equipments, mathematical equations, or computer codes) are essential for conducting successful research. Here the role of the computer program Gerris is indispensable for the completion of the current work. So I wish to express my gratitude to the Gerris development team, and particularly, to Dr. Stéphane Popinet (NIWA, New Zealand) for the powerful numerical tool that allows users effectively exploit its functionalities.

It was my pleasure to have occasional discussions with professors from other universities. Prof. Joseph H.-W. Lee (HKUST) provided critical comments on the idea of numerical simulation of free jets, which helped me revise and improve the method.

My colleague in NTU gave me support now and then. Dr. Wang Xikun helped me in setting up experiment for a slot jet and discussed technical details in lab experiments. Zheng Jia Xin provided me dataset and discussed results of experiments involving mutual impacts between two / three jets. My Vietnamese colleagues in the same research group gave me feedback to numerical results in this study.

Singapore's ACRC-A*STAR offered an enjoyable workshop on scientific computing in 2011 and for which I would be so grateful. The staff in NTU High-Performance Computing Centre were also ready to give me a hand in setting up numerical simulations.

And lastly, my personal thank to my family and friends. Their encouragement is a source for my effort in completing the work. So my thank to them all!

Contents

1	Introduction	1
1.1	Motivation for numerical study of converging jets	1
1.2	Research objectives	4
1.3	Layout of the report	5
2	Free jet dynamics and modelling tools	9
2.1	Characteristics of jet flows	9
2.2	Mathematical basis	11
2.2.1	General equations	11
2.2.2	Integral forms	13
2.2.3	Application of boundary layer theory	13
2.2.4	Application of dimensional analysis	14
2.2.5	Empirical and heuristic approaches	15
2.2.6	Previous studies on turbulent round jets	16
2.3	Modelling turbulence	16
2.3.1	Direct Numerical Simulation	17
2.3.2	Mixing length models	17
2.3.3	Two-equation models	18
2.3.4	Large Eddy Simulation	19
2.3.5	Turbulent mixing	20
2.4	Numerical codes	21
2.4.1	Solver	21
2.4.2	Grids	21
2.4.3	Artificial viscosity	24
2.5	Computational techniques and issues	25
2.5.1	Octree grid manipulation	25
2.5.2	Accuracy and speed	25
2.5.3	Parallel computing	26

3	Simulation of a non-buoyant jet	29
3.1	Simulation set-up (Gerris)	29
3.1.1	Geometry and scaling	29
3.1.2	Datasets available from the literature	30
3.1.3	Data sampling and output interrogation	32
3.2	Inflow hydrodynamic condition	34
3.3	Influence of grid cell size on simulated jet flow	36
3.4	Parameter selection	37
3.4.1	Provisional choice of viscosity	38
3.4.2	Parameter tuning	40
3.5	Simulation results	42
3.5.1	Time-averaged velocity	42
3.5.2	Turbulence level	46
3.6	Summary	51
4	Modelling two impinging jets: bulk flow properties	53
4.1	Dynamics of impinging jets	53
4.1.1	Jet impingement on solid, smooth surface	54
4.1.2	Mutual impingement of two round jets	55
4.1.3	Shear flow layer formed by two jets	55
4.2	Mathematical formulation	57
4.2.1	The zone of impact	57
4.2.2	Double shear layer flow	58
4.3	Flow characteristics found based on simulation	61
4.3.1	Simulation set-up	61
4.3.2	Extent of the zone of impact	62
4.3.3	Layer thickness and flow velocity at the boundary of double shear layer	63
4.3.4	The double shear layer in a plan view	66
4.4	Solution for the double shear layer equations	67
4.5	Modelling results compared to simulation output	67
4.5.1	A note on parameter selection	68
4.5.2	Comparison with a simulation of $\alpha = 15^\circ$	68
4.5.3	Simulation based on a symmetry plane	69
4.6	Summary	70

5	Equivalent jet systems and time dependent behaviour of jets	73
5.1	An equivalent form for a two-jet system	73
5.1.1	Motivation	73
5.1.2	Formulation	74
5.1.3	Comparative simulation of equivalent jet systems	75
5.1.4	Result from NTU laboratory experiment	76
5.2	Turbulence modelling	79
5.2.1	Implementation	80
5.2.2	Dynamic grid adaptation	81
5.2.3	Test cases	81
5.3	Time dependent behaviour of two-jet system	81
5.3.1	Observations on turbulence structure in a plane jet	82
5.3.2	Instantaneous flow field	83
5.3.3	Analysis of velocity fluctuation	83
5.4	Summary	87
6	Flow field formed by three co-planar, converging jets	89
6.1	Jets in close proximity	89
6.2	Simulation of three jets	91
6.2.1	Cross-sectional shape and width of the compound jet	92
6.2.2	Cross-sectional velocity distribution	96
6.2.3	Probability distribution for location of the velocity peak on a cross-section	97
6.2.4	Velocity decay in the longitudinal direction	99
6.2.5	Intensity of turbulent fluctuations	100
6.3	Summary	102
7	Mixing and dilution of effluent for various nozzle alignments	105
7.1	Entrainment and mixing in a multiple-jet alignment	105
7.1.1	Indication of mixing	106
7.1.2	Tracer distribution and dilution	107
7.2	Simulation set-up	108
7.3	Analysis of simulation results	109
7.3.1	Dilution	110
7.4	Comparison of intermixing among jet configurations	113
7.5	Reynolds stresses	115
7.6	Summary	116

8	Conclusion and recommendations	119
8.1	Conclusions	119
8.2	Recommendations	121
A	Running Gerris	135
A.1	Syntax of input file	135
A.2	Scenarios	136
A.2.1	Single, horizontal jet	136
A.2.2	Multiple, inclined jets	140
A.2.3	A check on the diagonal flow near the orifices	144
B	Postprocessing	145
B.1	Extraction and transformation of Gerris output data	145
B.1.1	Line integral of flow variables	146
B.1.2	Data extraction to determine characteristics of a double shear layer	146
B.2	Extraction of data produced by experiments using PIV	148
C	Experimental data of single jet	149
D	Miscellaneous problems	153
D.1	Determining the boundary condition for a double shear layer flow . .	153
D.2	Implementation of the double shear layer model	154
D.3	Finding the distribution of the velocity peak on a cross-section	155
D.4	Interpolating the maximum on a grid of samples	157
D.5	Evaluation of intermixing between three jets	160

Summary

Using turbulent jet flows is an important means to dilute pollutant released to ambient water (e.g. in ocean outfalls). Conventional engineering design rules are based on semi-empirical formulae, which involve time-averaged flow quantities. Jet nozzles are commonly aligned parallel to each other assuming minimal momentum impact in the merging process of the component jets. A potential alternative is to have jets emitting diagonally to each other—mutually-impinging jets—where momentum exchange plays a key role in shaping the turbulent core of the compound jet, thus affecting the mixing capability at high Reynolds number (Re).

This study's target is to formulate practical equations to characterise the turbulent jet flows from three impinging jets, thereby elucidating the advantage of such flow configuration as opposed to a single outlet, or an array of parallel aligned jets. To achieve this target, the author carried out incremental research from a single free jet to a system of two interacting jets and finally three co-planar jets. The open source computer program Gerris has been applied to study high- Re viscous flows. The solver functionality is extended with a simple LES model based on Smagorinsky's formula.

Results of single jet simulations are compared to Papanicolaou (1984)'s experimental data, where Re ranges from 5000 to 10000. The simulated velocity profiles follow a typical Gaussian curve, with peak velocity being underestimated about 10% to 20%. The input viscosity parameter (to counter the effect of grid coarseness) is found to be an order of magnitude larger than the physical viscosity.

A mathematical model for two impinging jets has been suggested. The double shear layer flow field formed by these two jets is assumed to radiate from the zone of impact. The velocity profiles of this double shear layer is characterised by the mean velocity, U , and the layer thickness, h . This model is only valid for small values of the angle 2α between the two jets. The simulated flow field was compared to experimental results by Jia Xin (NTU/MRC study group).

For 3-jet cases, simulations with a typical Re of 10^5 has been performed with various jet configurations: $2\alpha = 15^\circ, 30^\circ, 45^\circ$ and an discharge-equivalent single jet. The cross-sectional shape of the turbulent core in the compound flow shows tendency to

become circular from a distance of about $25d$ (nozzle diameter). There is no apparent functional relationship between the shape characteristics and α .

Finally, the mixing capability of a 3-jet configuration has been examined through simulation. Larger α and larger in-between nozzle spacing, s , had both shown more favourable dilution. Increasing s leads to an improvement in entrained flow into the jet core; a relationship between the entrainment rate and s had been proposed. The correlation between entrainment rate $\Delta Q/\Delta x$ and s^2 has been found to be reasonable, with a correlation coefficient of $R^2 \approx 0.85$.

List of Figures

1.1	Layout of thesis	5
2.1	Schematization of a round jet	9
2.2	Photo of a turbulent jet, captured instantaneously, and with long exposure	10
2.3	Coordinate systems used in problems involving round jets	11
2.4	Cells involved in discretisation on a regular grid and on a octree grid .	23
2.5	Partition of computational domain for simulation using 4 CPU cores shown on three orthogonal planes	27
2.6	Performance of computing system for different number of CPU cores used	28
3.1	Geometric dimensions of the problem, including two unit cubes . . .	29
3.2	Two sampling layouts on a jet cross-sectional plane	33
3.3	Sampling locations for the single-jet simulation	33
3.4	Turbulent intensity of cross-sections $x_* = 0.55$, and 1.55 compared to laboratory data at $x_* = 0.05$ by Mi et al. (2001)	36
3.5	Comparison of mean velocity and turbulent intensities for two cases with same $Re = 8000$	37
3.6	Schematic time series of computed viscosity	39
3.7	Mean axial velocity U_* , circumferentially-integrated for various cases	43
3.8	Velocity profiles for the cases $Re = 10937$ at various cross-sections; results of simulation (♦) compared to empirical formula (solid lines) .	43
3.9	Profiles of mean axial velocity, normalised, and for various Reynolds numbers	44
3.10	The decay of centreline mean velocity and the increase in jet width with longitudinal distance	45
3.11	Turbulence level of axial velocity u_{rms}/U_{CL} , circumferentially-integrated for various cases compared to experiment data	47

3.12	Turbulence level of radial velocity v_{rms}/U_{CL} , circumferentially-integrated for various cases compared to experiment data	48
3.13	Reynolds shear stress $\overline{u'v'}/U_{CL}^2$, circumferentially-integrated for various cases	49
3.14	Turbulence level of axial velocity u_{rms}/U_{CL} , circumferentially-integrated for various cases: simulation (thin solid line) compared to experiment data (Papanicolaou 1984 (■); and Wygnanski and Fiedler 1969 (thick solid line))	50
3.15	Turbulence level of radial velocity $v_{r,\text{rms}}/U_{CL}$, represented similarly to Figure 3.14.	50
3.16	Turbulence level of radial velocity $\overline{u'v'}/U_{CL}^2$, represented similarly to Figure 3.14.	50
3.17	Centreline turbulent intensity (symbols: present simulations) compared to (lines: results by Bogey and Bailly (2009)).	51
4.1	A normally-impinging jet, $\alpha = 90^\circ$ (adapted from O'Donovan, 2005)	54
4.2	Conceptual diagram of coplanar impinging jets in air ambient with a liquid sheet, and in water with a double shear layer	56
4.3	To find a possible distribution of radial velocity along the edge of zone of impact for an oblique round jet	58
4.4	Double shear layer, bands, and a volume element	59
4.5	To determine the extent of the zone of impact based on the velocity field	62
4.6	To determine contour of the zone of impact by tracking along the direction perpendicular to velocity vector field	62
4.7	Velocity profile extracted from five sample points on a vertical line at a point on the upstream boundary of double shear layer	63
4.8	Distribution of $\mathcal{U}$ and h along the contour of the zone of impact for Case $\alpha = 15^\circ$	64
4.9	Velocity profiles along the contour of the zone of impact	66
4.10	Streamlines of time-averaged flow velocity on the symmetry plane . .	66
4.11	Comparison between full-domain simulated and modelled radial velocity and layer thicknesses	69
4.12	Comparison between half-domain simulated and modelled representative velocities along the centreline	70
5.1	Equivalence in terms of momentum between a two-jet and a three-jet configuration	74

5.2	Top view comparison of U_* between two-jet ($\hat{\alpha} = 15^\circ$) and three-jet ($\alpha = 18.4^\circ$) systems	75
5.3	Front view comparison of U_* between two-jet ($\hat{\alpha} = 15^\circ$) and three-jet ($\alpha = 18.4^\circ$) systems	76
5.4	Front view comparison in velocity distribution for the case of equivalent systems: two-jet in full domain and three-jet in half-domain (Contour interval $\Delta U_* = 0.05$)	76
5.5	Relationship between various flow quantities in a 2-jet and its equivalent 3-jet system. Length values are multiples of d , and velocity values are multiples of U_o	77
5.6	Time-averaged velocity on cross-section $(x - x_{\text{conv}})_* = 3.8$. Above: NTU experiment. Below: Simulation output.	78
5.7	Cross-sectional vorticity plot of vector field	79
5.8	Time-averaged and instantaneous vector flow fields of a plane jet, measured using PIV instrument	83
5.9	Velocity component normal to the double shear layer, $\text{Re} = 1.09 \times 10^4$	83
5.10	Pressure distribution with velocity vectors of two impinging jets, $\text{Re} = 1.09 \times 10^4$	84
5.11	Time record of longitudinal velocity	84
5.12	Intermittency factors at various transects of the double shear layer.	85
5.13	Double velocity correlations in various directions: longitudinal, vertical, and lateral.	86
6.1	Multiple jet merging model (adapted from Wood et al., 1993)	90
6.2	Converging trend of two initially parallel-aligned jets in simulation ($\text{Re} = 1.1 \times 10^4$), with contours of U_*	91
6.3	A compound jet in translated coordinate system	92
6.4	Schematic velocity distribution for a compound jet cross-section and the representative contour, together with definitions of d_{max} , $d_\perp$, and d'	93
6.5	Contours indicating generic shapes of the distribution for U at various cross-sections, for the case of one jet and three jets of various angles α	94
6.6	Characteristic diameter (left) and flattening factor (right) of axial velocity contours against x_*	95
6.7	Comparison between axial velocity profile (U_*) formed by three impinging jets with $\alpha = 45^\circ$ ($x_* = 8.56, 18.56$) and one equivalent jet ($x_* = 2$).	97

6.8	Probability distribution and location of instantaneous velocity peaks on two cross-sectional planes for three jets, $\alpha = 15^\circ$ and for one jet	98
6.9	Changes in mean and standard deviation in normal distributions of $U_{\max}$ versus x_* for various values of α	99
6.10	Decay of centreline velocity and change of cross-sectional average velocity in longitudinal direction for 3-jet configurations	100
6.11	Contours indicating distribution of axial turbulent intensity	101
6.12	Comparison of simulated flow velocity and turbulent intensities for single jet (solid line) and three-jet system with $\alpha = 30^\circ$ (dash line).	102
7.1	Schematisation of entrainment prior to the formation of compound jet	105
7.2	Schematization of a standard diffuser	106
7.3	Comparison between spreading of time-averaged concentration against velocity	109
7.4	Ratio between concentration spreading rate and velocity spreading rate for various α	110
7.5	(Left) Average dilution ratio and (Right) centreline concentration against axial distance of a compound jet	110
7.6	Average dilution ratio of component jets before merging	111
7.7	Correlation between the increase in discharge and the area enclosed between two consecutive sections	112
7.8	Distribution of three tracers at the convergence cross-section, $x_* = 0$ ($I_{\text{mix}} = 0.310$) and at $x_* = 5$ ($I_{\text{mix}} = 0.895$) in Case $\alpha = 45^\circ$	114
7.9	Longitudinal distribution of the mixing index for various 3-jet alignments	115
7.10	Turbulent intensity of compound jet for various α compared to that of single jet	116
A.1	Comparison in axial velocities between a jet emitted at an angle of 30° and a horizontal jet.	144
D.1	Cumulative distribution and fitting normal probability curves to the data representing displacement of axial velocity peak from centroid of cross-section	156
D.2	Shaded region to contain the maximum; to locate the maximum value along the j^{th} -grid line	157

List of Tables

3.1	Selected experiment by Papanicolaou (1984) to be reproduced in this study	31
3.2	Initially prescribed viscosity, ν_{init} , and computed (mean) viscosity, ν_{comp} , for the case no turbulence model is implemented.	40
3.3	Prescribed values for the viscosity source term in the numerical model, with comparison to the estimated numerical viscosity	41
4.1	Functional forms of R_i^* , h_i^* and $\mathcal{U}_i^*$ for various α	65
5.1	Computed viscosity for different cases, without and with subgrid scale turbulence modeling using different values of C_s	82
6.1	Relationship between the width of a component jet and that of a single jet	95
D.1	Sum of square errors in fitting CDF to normal distributions. Values of mean and standard deviation are also shown in Figure 6.9.	156

List of notations

Acronyms

CDF	Cumulative distribution function
CFD	Computational Fluid Dynamics
CPU	Central Processing Unit
DNS	Direct Numerical Simulation
LES	Large Eddy Simulation
LIF	Laser-Induced Fluorescence
MPI	Message Passing Interface
p.d.f.	Probability Distribution Function
PIV	Particle Imaging Velocimetry
PTV	Particle Tracking Velocimetry
RANS	Reynolds-Averaged Navier-Stokes
TKE	Turbulent Kinetic Energy
ZEF	Zone of Established Flow
ZFE	Zone of Flow Establishment

Symbols

α	Angle of diagonally-aligned jet w.r.t the x-axis
A	Area of jet cross-section, area bounded between centrelines of two diagonally-aligned jets
β	Entrainment coefficient
b	Half-width of the cross-sectional velocity distribution of the jet, = radial distance from jet centreline to where $U/U_{CL} = 1/e \approx 0.368$
$b_{n\%}$	Radial distance from jet centreline to where $U/U_{CL} = n/100$
b_c	Half-width of the cross-sectional concentration distribution
c	Instantaneous concentration (of a chemical species) [g m^{-3}]
C_1, C_2	Coefficients in the continuity and momentum equations for double shear layer
C	Time-averaged concentration [g m^{-3}]
$C_\mu, C_{1\varepsilon}, C_{2\varepsilon}$	Calibration parameters in standard k - ε model
d	Diameter of jet orifice
d'	Equivalent diameter of a compound jet

δ	Thickness of boundary layer; radius of turbulent cone for a round jet
E_{kin}	Kinetic energy of system [m^2s^{-2}]
ε	Dissipation rate of turbulent kinetic energy k
$\mathbf{e}_x, \mathbf{e}_y, \mathbf{e}_z$	Unit vectors on x, y, z -directions
Γ	Turbulent diffusivity [$\text{m}^2 \text{s}^{-1}$]
γ	Intermittency factor
g	Gravity acceleration [m s^{-2}]
h	Computational grid cell size, half-thickness of the double shear layer formed by two impinging jets
I_{mix}	Mixing index [—]
i, j, k, n	Indices of x, y, z - grid lines and time layers
$\mathcal{K}_1, \mathcal{K}_2$	Constants of integration
k	Kinetic energy of fluctuation motion (per unit mass), $k = \frac{1}{2} (\overline{u'^2} + \overline{v'^2} + \overline{w'^2})$
l_{mix}	Mixing length [m]
l_M	Momentum length of a buoyant jet
M	Momentum of the jet [$\text{m}^3 \text{s}^{-1}$]
ν	Kinematic viscosity [$\text{m}^2 \text{s}^{-1}$]
ν_T	Turbulent/eddy viscosity [$\text{m}^2 \text{s}^{-1}$]
$O(\dots)$	Order of magnitude
p	Pressure
$\mathcal{P}$	Production term of turbulent kinetic energy
ϕ	A scalar quantity (concentration, temperature. . .) angle, in the symmetry plane of a double shear layer
ϕ_{limit}	Radial angle (in a double shear layer) beyond which flow is not accounted for
ρ	Density of fluid [kg m^{-3}]
Q	Discharge, or mass flux of the jet [$\text{m}^3 \text{s}^{-1}$]
q_E	Entrainment rate [$\text{m}^2 \text{s}^{-1}$]
Re	Reynolds number of jet flow at the orifice, $= U_o d / \nu$
R_i	Radius of the zone of impact
R_{uu}	Double velocity correlation (for u component).
s	Spacing between centre points of adjacent orifices
S	Dilution ratio
S_{ij}	$= \frac{1}{2} (\partial U_i / \partial x_j + \partial U_j / \partial x_i)$, average strain [s^{-1}]
$\sigma_\varepsilon, \sigma_k$	Calibration parameters in standard k - ε model (default values 1.3 and 1.0)
t	Time
T_L	Lagrangian time scale
τ	Shear stress [$\text{kg m}^{-1}\text{s}^{-2}$]
u, v, w, v_r, v_θ	Velocity components in the x, y, z, r, θ -directions
$ \mathbf{u} $	$= \sqrt{u^2 + v^2 + w^2}$, velocity magnitude
U, V, W, V_r, V_θ	Time-averaged velocity in the x, y, z, r, θ -directions

$\mathcal{U}$	Characteristic averaged velocity in the τ -direction
$\tilde{U}$	Velocity component (τ -direction) at each transect of double shear layer, as a function of y .
$\bar{u}, \bar{v}, \bar{w}$	Filtered velocity in x, y, z -direction (in LES method)
η	Exponent in the power decay $U_{CL} \propto x^{-\eta}$
ζ	Enstrophy [s^{-2}]
$\propto$	Proportional to
Operators	
$\frac{D}{Dt}$	$= \frac{\partial}{\partial t} + \bar{u} \frac{\partial}{\partial x} + \bar{v} \frac{\partial}{\partial y} + \bar{w} \frac{\partial}{\partial z}$, material derivative
∇	$= \frac{\partial}{\partial t} + \mathbf{e}_x \frac{\partial}{\partial x} + \mathbf{e}_y \frac{\partial}{\partial y} + \mathbf{e}_z \frac{\partial}{\partial z}$, nabla
∇^2	$= \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} + \frac{\partial^2}{\partial z^2}$, Laplacian
Superscripts and subscripts	
$(\dots)'$	Turbulent fluctuation
$\langle \dots \rangle$	Ensemble average
$\overline{(\dots)}$	Filtered, subgrid-scale spatial averaged; time averaged
$(\dots)_\infty$	Condition far from outfalls/obstacles, e.g. free stream
$(\dots)_o$	At the outfall exit
$(\dots)_*$	Dimensionless quantity
$(\dots)_{av}$	Cross-sectional average
$(\dots)_{CL}$	At the centreline of a fluid jet
$(\dots)_{conv}$	At the convergence point of multiple jet centrelines
$(\dots)_i$	Along the contour of zone of impact
$(\dots)_{pl}$	Of a plane jet
$(\dots)_{ref}$	Reference quantity
$(\dots)_{rms}$	Root-mean square value
$(\dots)^{SGS}$	Sub-grid scale, as for stress
$\widehat{(\dots)}$	For dynamic-equivalent

Chapter 1

Introduction

1.1 Motivation for numerical study of converging jets

Using turbulent jet flows is an important means to dilute pollutant concentration issued from contaminant sources into ambient water. Studies on single turbulent jets have been carried out extensively using various approaches, including experimental, field measurement and numerical modelling. Here the knowledge of fluid mechanics plays a crucial role. In 1822 the Navier-Stokes equations were established to describe the motion of fluids. The process to solve these equations, however, is so complicated that approximation techniques are typically called for. One such successful method is based on the concept of boundary layer theory, which was proposed in 1905 by Prandtl. When this theory is applied to jets, there are mathematical solutions, see for example Tollmien and Görtler methods (Rajaratnam, 1976, p. 36).

To verify the jet theory, in the first place there are experiments based on physical models. Usually, fluid (air or water) is pumped through a pipe nozzle, or an orifice into a chamber or a tank. Early in the history of fluid mechanics, observation of the fluid velocity fluctuations can only be made at specific points using hot-wire anemometer. Later, with the use of laser techniques such as Particle Image Velocimetry (PIV) (Adrian, 1991) and Particle Tracking Velocimetry (PTV) (Adamczyk and Rimai, 1988), researchers are now able to obtain spatial flow characteristics readily. These methods have become popular and are universally applied today.

Probably at the same time when laser imaging techniques were first applied, numerical methods were also being developed for turbulent flows. Two-equations models such as the k - ϵ (Launder and Spalding, 1974) have been developed and implemented on computers for industrial applications. Direct Numerical Simulation have gained serious considerations while performance of supercomputers have advanced many-

fold. Large Eddy Simulation also benefits from increasingly powerful computers: this technique is finding its way into flow and combustion science and engineering while playing a fundamentally important role in geophysical fluid dynamics.

As a fundamental topic in turbulence, shear flow, and particularly jet flow, has gained significant attention in the scientific community, not only due to its beautiful physical insights but also because this non-trivial problem is a good benchmark for verifying and validating computational and instrumentation techniques.

The practice of hydraulic design is quite different, however. Turbulence is usually neglected in favour of bulk flow quantities such as flow rate and momentum flux. A notable application of jet flow is in the design of ocean outfalls. From an engineer's perspective, this design task is conventionally based on semi-empirical formulae. Such equations describe the trajectory of the jet centreline, the spreading rate of the jet, and the dilution rate. In summary, only average properties of the jet are required. Physical quantities relating to the turbulence characteristics of the jet are of secondary concerns. This omission is justifiable: (i) the criteria on which water quality is evaluated are conveniently expressed in minimum, maximum, or averaged values, e.g. mean concentration in a certain zone, and (ii) the condition of ambient water significantly affects the mixing process, so effort should also be expanded to provide good hydrodynamic parameters of the ambient flow.

Although there have been extensive studies on the dynamic behaviours of single jets using different approaches (Abramovich, 1963; Lee and Chu, 2003; Rajaratnam, 1976), the behaviour of multiple jets has mainly been investigated with the aids of physical models. Some of the important results from physical models are generalized into some form of knowledge, such as empirical formulae, which help improve the predictivity of software packages, for example CORMIX (Jirka et al., 1991).

The need to investigate more complicated cases involving multiple jets is for serving practical requirements to improve the performance of multiple jets as opposed to a single jet. Converging jets are especially relevant to aerospace engineering, where jets of fuel and air are directed toward each other in an optimal angle so that the highest degree of mixing can be achieved. Research on multiple converging jets in hydraulic engineering is probably not as extensive as that in the aerospace engineering community. Mathematical formulation such as that proposed by Hall et al. (1981) is not commonly adopted in studies involving multiple water jets. Recent experimental studies often deal with converging jets as a whole system to reproduce the flow field far from the convergence zone.

The author believes that converging jet has certain value in engineering design and it is the aim of the study to assess the effectiveness of certain alignment patterns of

jet nozzles in diluting pollutants in the recipient water. As a first step towards making the assessment, the author conducts numerical studies and numerical experiments to elucidate and demonstrate the mixing characteristics of multiple converging jets.

From the scientific point of view, the behavior of converging jets poses a complex problem involving interacting shear flows with nonhomogeneous turbulence. The results obtained in the numerical studies will also serve as a potential source for benchmarking numerical solvers.

Therefore, the aims of this study are to (i) obtain intensive mixing rate and (ii) retain the entrainment in the vicinity of outlet ports. These are achieved, generally, via arranging an array of ports along the outfall tunnel diffuser. The preferable mixing condition can be expressed in technical terms, such as, “completely mixed condition”, where “no measurable difference in the concentration of a pollutant exists across a transect of the water body (e.g., does not vary by more than 5%)” (USEPA, 1994). If high mixing rate is achieved, the concentration measured at selected control points would decrease and more likely meet the water quality standard.

An ocean outfall system often includes long pipes to direct wastewater from inland populated/industrialized areas across the surf zone to the deeper part of the sea. At the zone of release, the water depth should be adequate to dilute the pollutants before they are dispersed by the tidal currents.

In many conventional designs, the outfall jets are aligned parallel to each other (Roberts, 1979) and this array of jets immediately forms a two-dimensional flow field at some distance from the nozzles. The behaviour of this flow field is similar to that of the line plume, which implies assuming minimal momentum interactions between the jet components.

Alternatively, one can have jets emitting at an inclined angle to each other, i.e. mutually-impinging jets, where momentum exchange plays a key role in shaping the turbulent core of the resulting flow field, thereby affecting the mixing capability of the jet system on any solutes presented. The dynamics of such impinging water jets at high Reynolds number has not been systematically examined in literature.

This study attempts to fill the gap of knowledge in the topic on multiple impinging jets. The target is to formulate practical equations to characterise the turbulent jet flows from three co-planar converging jets, with the emphasis placed on elucidating the advantage of such flow configuration as opposed to a single outlet, or an array of parallel jets.

To investigate this problem from the hydrodynamic perspective, and with the aid of numerical simulations, the author adopted an incremental approach, from a single free jet to a system of two interacting jets and finally the practical application involving

three co-planar jets. The three co-planar jets are arranged symmetrically on the same plane, i.e. the central jet remains straight and the two side jets are inclined towards the central jets at the same angle. This configuration allows strong mixing of the jets by means of momentum impact and turbulent flow generated. Numerical simulation is the important tool used to provide additional data beside experimental ones, thus providing benchmark features of the flow structure in cases involving multiple jets. The final approach is somehow inductive: from a large collection of simulation output data, a useful set of equations are generalised, featuring bulk properties of the flow without defining the details.

Although the study focuses on hydrodynamics of the impinging jets, it occasionally refers to tracer concentrations. This does not bear any implication that the model can predict concentration of the solute in reality. It is only that the distribution of tracer provides a clue or means to estimate the travel distance of water particles emitted from the jet orifice. Tracer plays the role of a “passive” species: the flow advects the solutes and diffuses them (through eddy diffusion), and that the solute concentration is redistributed does not alter the density and flow condition. In all simulation we assume the fluid is homogeneous in both density and viscosity. Although this is simplified from reality where stratification is an important factor, this assumption facilitates the interpretation of results obtained from simulation. Additional input factors to the model may be readily implemented in further research.

1.2 Research objectives

With the above research approach, the five objectives to be completed in this study are outlined as follow:

1. Set up simulation for the case of a single jet based on an open-source, general-purpose CFD solver and verify its output against lab experiment data available in literature;
2. Investigate, with the support of numerical simulation, the physical phenomenon that takes place when two equal jets intersect, i.e. two jets having the same discharge and momentum flux with centrelines crossing each other. The centreline of the jets lie on the same plane.
3. Approximate the double shear layer produced by two converging jets in the presence of a symmetry plane;

Figure 1.1: Layout of thesis

4. Simulate three co-planar, converging jets; the output data concerning shape of the jet cross-section, velocity and turbulent intensity are compared with those of a single jet of equivalent discharge, and;
5. To establish a preliminary configuration of three co-planar jets, with given mass flux and momentum flux, which yields favourable mixing capability in terms of dilution.

1.3 Layout of the report

Figure 1.1 shows the broad organization of this thesis report. Chapter Two contains reviews of the physical aspects of jet dynamics and techniques of modelling, which serve as the basis for the ensuing numerical simulations. Most techniques have been extensively reviewed in the literature, although some of them are easier to implement with respect to the current research. Whenever appropriate, quantitative examples will be drawn from the study in subsequent chapters to illustrate the concept. A section of this chapter is devoted to introduce Gerris, the open source solver (Popinet, 2003) that is used for numerical solution.

The simulation for a single free jet is investigated in Chapter Three, and the results obtained are compared to experimental data found in literature. Although single jet simulation may not reveal any new physical insights, this process is important because it provides the chance for the author to compare his simulation output with the data published in the literature. For multiple jet dynamics that is addressed later, the available data are mostly collected in different experimental conditions: e.g. air jet or low-Reynolds water jet to the air, and are not directly comparable to simulation results. The comparison presented in this chapter forms a foundation to develop the study on multiple jets in the subsequent chapters.

Chapter Four starts by presenting a simplified case of the problem—the impact of a jet on a hard surface, moving along the flow direction. This case is relevant because when two identical jets intersect each other, there exists, in theory, a symmetry plane between these two jets. If such an ideal plane exists then the phenomenon can be reduced to that of a single jet. Since the problem of obliquely-impinging jet is no longer axisymmetric, or two-dimensional in general, numerical simulation is necessary for obtaining solution for the flow field in such case. In this chapter only the time-averaged flow velocity is considered; turbulent flow behaviour will be addressed in the next chapter.

Chapter Five presents a model of equivalent two-jet and three-jet systems, which is based on conservation of mass and momentum. This chapter also investigates the instantaneous condition of the simulated flow behaviour. An implementation of a simple eddy-viscosity model has been added to the Gerris flow solver and relevant parameters of this model is assessed.

Chapter Six contains the results of simulation for a system comprising three coplanar, converging jets. The cross-section of this compound jet is examined, with the variation of dilution against the distance from the jet orifice. A comparison with a single jet is made. The comparison serves as a basis to compare the performance and advantage (if any) of using multiple jets. In this chapter, by generalising the data obtained, several formulae are derived regarding “bulk” parameters of the compound jet, namely: the shape and size of jet cross-sections, the distribution of velocity on each cross-section, the velocity decay and the dilution rate of pollutant along the jet centreline. Also, a preliminary estimation for the displacement of the peak velocity distribution from the centroid of a jet cross-section has been carried out.

Chapter Seven presents the simulation results for nine potential layouts of jet orifices, which are combinations of three jet angles ($\alpha = 15^\circ, 30^\circ, 45^\circ$) with three nozzle spacings ($s/d = 10, 20, 30$). Comparison is carried out with respect to the jet geometries and structures at the point of convergence. The dilution rates for various

jets within the merging zone are evaluated. A semi-empirical formula is suggested to compute the dilution based on the planar area bounded by centrelines of three component jets. The condition of mixing is also examined by using a separate tracer for each orifice. Finally, there will be a remark on a favourable configuration among those potential arrangements.

Conceptually, the main material of this report is divided into three “blocks” involving (1) single jet, (2) double jets, and (3) triple jets. It is advantageous to move the focus of study by the order 1–2–3 in the number of jets, with the possible exception of Chapter Four. Chapter Four requires more elaboration on the mathematics and the progress can be made from Chapter Three to Six, skipping Chapter Four, without noticeable discontinuity.

Finally, Chapter Eight summarises of the content in this thesis report, deliberates on the pertinent results obtained, highlights the simplifications and shortcomings of the method used. A list of suggestions is also provided and point to possible improvements to the method adopted in this study.

The supplement materials for this report can be found at http://162.250.127.146/numerical_study_converging_jets/, which will be maintained at least one year from the submission of this thesis report.

Chapter 2

Free jet dynamics and modelling tools

2.1 Characteristics of jet flows

In this work we consider pure turbulent jets, which implies that the only physical source to drive the liquid jet is its momentum. The ambient water body is considered to be infinitely large and tranquil. Any effects due to the difference between the density of the jet and that of the ambient water are neglected, so a jet centreline is straight as expected. Water is considered incompressible. For practical purposes only round nozzles with diameter d , are examined, which for a single jet, exhibits an axisymmetrical dynamic system.

A turbulent round jet is characterised by an expanding region of turbulence resembling a cone shape (Figure 2.1). This turbulent region is separated from its ambient with a meandering interface which continually evolves with time. Such an interface captured in a time instance snapshot is shown in Figure 2.2 (left subfigure). Although the instantaneous location of this interface cannot be determined precisely due to ran-

Figure 2.1: Schematization of a round jet, featuring potential core, expanding turbulent cone, “thickness” of boundary layer, zone of established flow and self-similar velocity profiles

Figure 2.2: Photo of a turbulent jet, captured instantaneously (left) and with long exposure (right, implying time-averaged flow condition) (adapted from the report by Weersink (2010)).

domness, its time-averaged location is more likely to be predictable (Figure 2.2, right). The highly-distorted curvature of this interface allows even more ambient fluid to be entrained into the turbulent core. Therefore the “thickness” of this turbulent region, denoted δ , increases with x . In reality, δ appears to increase linearly with distance x (specifically, $x/\delta \sim 5$ and almost unchanged).

An interesting feature in jets is that, although the velocity decreases with distance from the jet nozzle, the velocity profiles are very similar in form (usually approximated as Gaussian, as observed by Albertson et al. (1950)). Formally, we can write $U(x, r)/U_{CL}(x) = f(r/\delta)$ where r is the radial distance to the jet centreline. In fact, this self-preserving feature of the jet can be observed far from the jet orifice, $x/d \gtrsim 70$ (see e.g. the paper by Wygnanski and Fiedler (1969)).

The vortices in the turbulent core tend to turn “away from the jet centreline”, due to higher momentum in the jet core compared to that of the ambient fluid. These vortices, or eddies, are responsible for the diffusion of momentum and scalar quantities (heat, concentration of tracer). Large eddies is formed by taking the kinetic energy from the mean flow, and transfer its energy to smaller eddies. This process, energy cascade, continues to very small eddies until the fluctuation kinetic energy is completely dissipated by fluid viscosity.

The turbulent jet core is conventionally divided into two zones (Abramovich, 1963; Lee and Chu, 2003; Rajaratnam, 1976), and they are: (i) zone of flow establishment (ZFE), from the nozzle to the cross-sectional plane $x/d \approx 6$ where the centreline velocity remains equal to that at the jet nozzle; and (ii) zone of established flow (ZEF) where the potential core vanishes and the entire cross-section of the jet becomes turbulent, see Figure 2.1.

In each cross-section, the magnitude of velocity fluctuation generally decreases as one moves away from the jet centreline. At the edge of the turbulent core, turbulence is intermittent: within the time series of flow velocity for such a location, alternating

Figure 2.3: Coordinate systems used in problems involving round jets

patterns of vigorous and mild fluctuation co-exist. Also, at this edge, viscous eddies continually draw the ambient fluid into the turbulent core and this new fluid itself becomes turbulent. Consequently the thickness δ increases downstream.

2.2 Mathematical basis

2.2.1 General equations

For convenience one may adopt the following coordinate notations. The origin, O , is located at the centre of the outlet, which has a cross-sectional area of A . The x axis goes streamwise, the y axis together with x axis form the plane of the paper page, the z axis is perpendicular to the page. (For the case of multiple co-planar jets, which are dealt with later, the x - y plane is chosen to contain all jet centrelines.) This Cartesian coordinate system (Figure 2.3 left), which is useful for analysis of multiple jets and representation of input-output for the numerical solver. For single jet analysis, a cylindrical system (r, θ, x) , where $r = \sqrt{y^2 + z^2}$, is conceptually more suitable (Figure 2.3 right). In this context, the water motion is considered mainly longitudinal, with velocity U ; and slightly lateral with velocity V_r . The second velocity component is calculated as the projection of $\mathbf{U}$ on the radial direction, $V_r = \text{proj}_{\mathbf{r}} \mathbf{U} = \mathbf{U} \cdot \mathbf{r}/|\mathbf{r}|$, or $V_r = (yV + zW)/\sqrt{y^2 + z^2}$. The swirl component, V_θ , is negligible, hence the θ coordinate is usually irrelevant.

Upon entering the ambient water body, the jet takes some time to reach dynamic equilibrium. Since wastewater outfalls are typically designed to operate for a long period of time, the equilibrium state, or steady state, is more relevant in the current context as compared to unsteady motion of fluid. Mathematically, steady condition means all time derivatives, $\partial U/\partial t$, $\partial V/\partial t$, and $\partial W/\partial t$, are zero.

The second simplification in this preliminary analysis is to seek the time average of velocity components. By this one introduces mean flow variables, U, V, W in the dynamic equations for the jet.

Another approximation is the use of eddy-viscosity model which was formalised by Boussinesq in 1877. Here the turbulent stresses, such as $-\overline{u'v'}$, are replaced by

$\nu_T \left(\frac{\partial U}{\partial y} + \frac{\partial V}{\partial x} \right)$ and $-\overline{u'^2}$ by $2\nu_T \frac{\partial U}{\partial x}$. For most practical cases, $\nu_T \gg \nu$, or $\nu + \nu_T \approx \nu_T$. A numerical simulation (Durbin and Medic, 2007, p. 238) indicated that $\nu_T = 2300\nu$.

The third simplification arises from the specific flow configuration of the problem setting of this study. The system of a single round jet, as mentioned before, is axisymmetric. Hence the roles of y and z are equivalent. One may assume the flow to be radially-oriented, or $|V| = |W|$. When considering the magnitude, the flow is longitudinally dominant, $U \gg V, W$. The gradient of flow characteristics, however, exists mainly in the lateral direction rather than longitudinal, i.e. $\partial(\cdot)/\partial y \gg \partial(\cdot)/\partial x$.

Based on these assumptions, the following steady-state Reynolds-averaged Navier-Stokes equations can be derived:

- in a Cartesian coordinate system:

$$\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} + \frac{\partial W}{\partial z} = 0 \quad (2.1)$$

$$U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} + W \frac{\partial U}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu_T \left(\frac{\partial^2 U}{\partial x^2} + \frac{\partial^2 U}{\partial y^2} + \frac{\partial^2 U}{\partial z^2} \right); \quad (2.2)$$

- in a cylindrical coordinate system:

$$\frac{1}{r} \frac{\partial(rV_r)}{\partial r} + \frac{\partial U}{\partial x} = 0 \quad (2.3)$$

$$V_r \frac{\partial V_r}{\partial r} + U \frac{\partial U}{\partial x} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \frac{\nu_T}{r} \frac{\partial}{\partial r} \left(r \frac{\partial V_r}{\partial r} \right). \quad (2.4)$$

When modelling fluid flows (both physically and mathematically) it is convenient to express the spatial extent relative to some reference length l_{ref} , which in this study is chosen as the orifice diameter, d . Similarly the speed of flow may be set relative to a reference velocity $U_{\text{ref}} = U_o$. From this, one can derive other quantities such as nondimensional axial distance, $x_* = x/d$; radial distance, $r_* = r/d$; jet width, $b_* = b/d$; nondimensional velocity $U_* = U/U_o$. The definition of U_{ref} leads to $t_{\text{ref}} = l_{\text{ref}}/U_{\text{ref}} = d/U_o$.

Navier-Stokes equations in terms of non-dimensional parameters would read very similarly to the above set of equations but with “starred” symbols and with ν replaced by $1/\text{Re}$, where $\text{Re} = U_o d/\nu$. Hence, Reynolds number is the single most important behaviour characterising the pure-momentum jet problem.

2.2.2 Integral forms

Although the aforementioned formulae describe well the problem at hand, in practice they are usually integrated and thus provide intuitive meaning to the flow dynamics.

The momentum flux

$$M = \rho \int_0^\infty 2\pi r U^2 dr \quad (2.5)$$

is conserved in the x -direction: $dM/dx = 0$.

The volume flux or jet discharge (also constant in the x -direction)

$$Q = \int_0^\infty 2\pi r U dr \quad (2.6)$$

is not constant in the x -direction: as the flow evolves in the streamwise direction, an additional volume of water is entrained into the turbulent region. Denote q_E as the entrainment rate of ambient water, the conservation of mass requires:

$$\frac{d}{dx} \int_0^\infty 2\pi r U dr = q_E. \quad (2.7)$$

The change of Q against x shows the capability of jets in diluting pollutants. The average dilution ratio of the jet at a cross-section x is: $\bar{S}(x) = Q(x)/Q_0$.

In special cases involving strict water quality requirement, the maximum concentration is of concern. The centreline dilution denotes how many times the centreline concentration at a certain cross-section is reduced with respect to the source concentration. This ratio is equal to the centreline dilution, S :

$$S \equiv C_0/C_{CL}. \quad (2.8)$$

More sophisticated integral forms of Eqs. (2.5)–(2.7) exist, such as the second-order form proposed by Wang (2000). In the previous reference, turbulent mass and momentum fluxes are introduced into the integral equations, instead of the corresponding mean quantities, in order to obtain a more accurate second-order closure.

2.2.3 Application of boundary layer theory

To solve the nonlinear equations (2.1) and (2.2), approximations must be introduced. An important method is the boundary layer approximation, for which the momentum equation in cylindrical coordinate reads:

$$U \frac{\partial U}{\partial x} + V_r \frac{\partial V_r}{\partial r} = \nu_T \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial V_r}{\partial r} \right). \quad (2.9)$$

Here the following simplification is made: $\tau_{rz} = \rho \nu_T (\partial U / \partial r)$ and then $\nu_T = c_1 b U_{CL}$ where b is the half-width of the (plane) jet and c_1 is a constant, one can derive formulae for velocity components U and V_r . For example, by using an empirical spreading rate value $db/dx = 0.215$, Davidson (2004, pp. 154–156) derived the relationships $\nu_T = 6.2 \times 10^{-3} \times 2\delta(x) U_{CL}(x)$ and $U(r)/U_{CL} = \left[1 + 8.65 \left(\frac{r}{2\delta} \right)^2 \right]^{-2}$.

More general formulae are mentioned in the text by Schlichting (1960):

$$U = \nu_T \frac{2x}{r^2} \frac{\xi^2}{(1 + \xi^2/4)^2} \quad V_r = \frac{\nu_T}{r} \xi \left[\frac{\xi - \xi^3/4}{(1 + \xi^2/4)^2} \right] \quad \text{where} \quad \xi = \frac{1}{4\nu_T} \sqrt{\frac{3M}{\pi\rho}} \frac{r}{x}. \quad (2.10)$$

Specifically, the centreline velocity is

$$U_{CL} = \frac{3M}{8\pi\rho\nu_T x}, \quad \text{or} \quad \frac{U_{CL}}{U_o} = \frac{3}{32} \text{Re}_T \frac{d}{x}, \quad \text{where} \quad \text{Re}_T = \frac{U_o d}{\nu_T}. \quad (2.11)$$

So given a turbulent jet issuing at constant rate, the centreline velocity [corresponding to $\zeta = 0$] is inversely proportional to the longitudinal distance to the outlet, $U_{CL} \propto x^{-1}$. This is in accordance with experimental results by Papanicolaou (1984), $U_{CL}/U_o = 6.71(x/d)^{-1}$.

2.2.4 Application of dimensional analysis

By reasoning that there must be some relationship between pairs of nondimensional quantities $U_{CL} Q_o / M_o \sim Q_o^2 / (M_o x^2)$, $b/x \sim Q_o^2 / (M_o x^2)$, and $C_{CL} \sim Q_o / (\sqrt{M_o} x)$, Wood et al. (1993) derived the following formulae (empirical coefficients are based on Papanicolaou, 1984):

$$\frac{U_{CL}}{U_o} = 7.57 \sqrt{\frac{\pi}{4}} \left(\frac{x}{d} \right)^{-1} \quad (2.12)$$

$$b/x = 0.11 \quad (2.13)$$

$$\frac{C_{CL}}{C_o} = 6.06 \sqrt{\frac{4}{\pi}} \left(\frac{x}{d} \right)^{-1}. \quad (2.14)$$

The empirical formulas that describe the variation of U_{CL} against x are based on the dimensional analysis, which leads to the conclusion that U_{CL} is inversely proportional

to x . (A derivation can be found in the text by Wood et al. (1993).) The coefficient 6.2 is found by considering first the conservation of momentum for the jet flow. The jet width is assumed to increase linearly downstream. Based on extensive laboratory measurements by Albertson et al. (1950) the spreading rate of a round jet, db/dx , was found to be 0.114. A substitution of this value into the integral formula for round jets (which is derived from the conservation of momentum) yields the value of 6.2. This value has been suggested in several reported works, including Baines and Chu (1996) and Lee and Chu (2003).

Other values for this coefficient represent different spreading rates; for example Fischer et al. (1979) cited a number of experimental values, which have an average value of 0.107. With the latter value, the coefficient can be found to equal to 6.61 instead of 6.2.

Experimental results from other authors added more variability to this coefficient. Papanicolaou (1984) cited a value of 6.71. Wygnanski and Fiedler (1969) reported a value of 5.67 which was applicable for $x/d > 30$.

2.2.5 Empirical and heuristic approaches

There are other approaches to solve the problem, in some simpler way to achieve the quantities of reasonable accuracy that would interest the engineers. Such methods often involve some kind of assumptions.

In the Lagrangian method, a “volume element”—a cylindrical slice of the jet core is tracked for its movement in time. Within this slice a simple top-hat velocity profile is specified, such that the mass and momentum fluxes are equivalent to those of an empirical Gaussian profile. Another assumption is that there is no interaction between the “slices”. The method is useful in analysis of inclined buoyant jets, and is implemented in the CORJET software package (Lee and Chu, 2003). By introducing such assumptions into the model, the computation is more straightforward and easier to implement, but any application involving multiple jets would require elaborate mathematical analysis.

Another example is the length-scale method used in RSB (Roberts et al., 1989) and CORMIX (Jirka et al., 1991) models where the jet flow condition is categorised based on the relative magnitudes between momentum, buoyancy and geometry of the orifice. Since the target of this thesis is to make use of computational and numerical tools to explore the behaviour of multiple interacting jets, the length scale method is introduced here for reference only.

2.2.6 Previous studies on turbulent round jets

Jet flows applications could be found many applications in the aerospace, mechanical and environmental industries, and many experiments had been conducted with either air or liquid jets. Corrsin (1943) measured turbulent characteristics of air jet emitted from a wind tunnel at $Re \sim 10^5$. Corrsin and Kistler (1953) conducted measurements to investigate the intermittency of “turbulent front” separating the jet flow with the nonturbulent ambient fluid. Wygnanski and Fiedler (1969) also conducted jet study at $Re = 10^5$, specifically in the self-preservation region. Papanicolaou (1984) conducted experiments with round jet in water, including both momentum-driven and buoyant jets. Panchapakesan and Lumley (1993) measured air jet at $Re = 1.1 \times 10^4$ to examine higher-order moments of the turbulent fluctuation signals.

At the turn of the century, Wang and Law (2002) used the DPIV (Digital Particle Image Velocimetry) and PLIF (Planar Laser-Induced Fluorescence) techniques to study the turbulent characteristics of buoyant jets. Another source of data, as reported by Ninomiya and Kasagi (1993), contains timeseries measurements for a water jet at $Re = 2566$. Weisgraber and Liepmann (1998) used DPIV to study the turbulent behaviour in the region $x_* = 15 \rightarrow 30$ for two cases, $Re = 5500$ and 16000 . (Here x_* is the longitudinal distance represented as multiples of nozzle diameter, d .)

Numerical studies in turbulent jet have become more popular owing to the recent advances of cluster computing. Some of these studies are reviewed in Section 2.3. Numerical simulation has been shown to be powerful tools that helps and provides great insight in the investigation of complex flow structures such as counter-rotating vortex pairs in jets issuing into a crossflow, It is also noted that numerical simulation has not been successful in modelling complex flow systems where the mathematical descriptions of the flow are still inadequate. For examples, most shear flow phenomena are being investigated using experimental techniques, leveraging strongly on advanced instrumentations including high speed photography and laser imaging devices and velocimetry.

2.3 Modelling turbulence

Turbulence plays an important role in the evolution of jet flows. It is the driving force for diffusing momentum, heat, or other scalar properties within the flow. Turbulence can be simulated directly, as with Direct Numerical Simulation, solving the full Navier-Stokes equation for every microscopic motion of the fluid, or using a model to approximate the “degree” at which fluid is mixed. An example for a simple model is the

“mixing length theory”, which has application for a few classes of idealised shear flow. Among the most widely-used model classes are the Two-equation models and Large-Eddy Simulation (LES).

2.3.1 Direct Numerical Simulation

The brute-force approach, Direct Numerical Simulation (DNS), is computationally expensive. The number of grid cells required to carry out a 3-D DNS is in the order of $\text{Re}_L^{9/4}$ (Tennekes and Lumley, 1972), where Re_L is the Reynolds number associated with the velocity fluctuation and $\text{Re}_L \sim 0.01 \times \text{Re}$ (of macroscopic flow). That means for typical turbulent flow in engineering where $\text{Re} \geq 2000$, the number of grid cells required is $\geq 10^9$. Even with the help of powerful computers we are only able to apply DNS for some specific classes of flow problems, especially flows in simple geometric domains with low Reynolds numbers. Creeping flows and grains falling under Stokes (laminar) regimes have been the topics of interest in DNS, see e.g. the paper by Yang and Shy (2003). In recent years, studies have been made using computer clusters to model more realistic turbulence in simple geometric domains (for a detailed review, see Moin and Mahesh, 1998). Babu and Mahesh (2005) simulated a turbulent jet with 3.8×10^6 hexahedral elements at the start of simulation. Whereas in the case of Muppidi and Mahesh (2007) for jet interaction with cross-flow near a wall, the mesh contains 11×10^6 hexahedral elements.

2.3.2 Mixing length models

In this method, the eddy viscosity is related to the velocity gradient:

$$\nu_T \propto \ell_m^2 \left| \frac{\partial U}{\partial y} \right|. \quad (2.15)$$

The mixing length ℓ_m is intuitive; it represents the distance needed for a fluid parcel moving perpendicular to the shear flow to blend with the (motion of) surrounding fluid. Interestingly, this length is found to be proportional to the thickness, δ , of the shear layer. For round jets, $\ell_m/\delta = 0.075$ and for plane jets, $\ell_m/\delta = 0.09$.

Despite the conciseness of Eq. (2.15), the determination of $|\partial U/\partial y|$ still requires investigation of the specific flow structure. To further simplify this equation, Prandtl¹

¹Prandtl (1942), “Bemerkungen zur Theorie der freien Turbulenz”, *ZAMM* **22**(5), pp. 241–243. Reviewed in (Rodi, 1993, p. 20).

suggested a linearised formula for the eddy viscosity:

$$\nu_T = C\delta|U_{\max} - U_{\min}|, \quad (2.16)$$

where for jets $U_{\max} = U_{CL}$ and $U_{\min} = U_{\infty}$, C is an empirical constant; $C = 0.011$ for round jets, 0.014 for plane jets and 0.019 for radial jets. The formula is a generalisation for that presented in Section 2.2.3 (where $\nu_T \propto bU_{CL}$).

While the mixing length approach cannot be used for complex phenomena such as the turbulent flow at the convergence zone of impinging jets, it provides a conceptual model for approximating certain flow features, for example the double shear layer discussed in Chapter 4.

2.3.3 Two-equation models

In the two-equation models one has to solve, in addition to Navier-Stokes equations, two equations related to turbulent characteristics of the flow. For example, in the k - ε model (Launder and Spalding, 1974), the turbulent kinetic energy (TKE), k , and its dissipation rate, ε , are represented by two simultaneous partial differential equations:

$$\frac{\partial k}{\partial t} + (\mathbf{u} \cdot \nabla)k = \frac{\partial}{\partial x_i} \left[\left(\nu + \frac{\nu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + \mathcal{P} - \varepsilon \quad (2.17)$$

$$\frac{\partial \varepsilon}{\partial t} + (\mathbf{u} \cdot \nabla)\varepsilon = \frac{\partial}{\partial x_i} \left[\left(\nu + \frac{\nu_T}{\sigma_k} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} \mathcal{P} - C_{2\varepsilon} \frac{\varepsilon^2}{k} \quad (2.18)$$

with the eddy viscosity modelled as $\nu_T = C_\mu k^2/\varepsilon$; the TKE production rate, $\mathcal{P} \equiv -\overline{u'_i u'_j} (\partial u_j / \partial x_i) = \nu_T S^2$ where $S = \sqrt{2S_{ij}S_{ij}}$, in which $S_{ij} = \frac{1}{2} (\partial U_i / \partial x_j + \partial U_j / \partial x_i)$ is the strain of flow velocity. This model has five constants: $C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ , σ_k , and σ_ε . In total we have a closed system of equations containing six unknowns $u, v, w, p, k, \varepsilon$. The k - ε model is reported (Rodi, 1993) to be successfully applied to turbulent flow. Compared to DNS, k - ε model requires much fewer grid cells in the domain. An example is a simulation of jet emission to crossflow using the k - ε model (Lee et al., 2002) where $92 \times 26 \times 36$ grid cells were used. However we are not going to study the behaviour of turbulent jet flows by using this type of model due to implementation issues: the computational code to be utilised in the following chapters is not readily adjustable by introducing coupled equations containing k and ε . Hence, for practical purposes one will only consider the LES method for turbulence modeling.

2.3.4 Large Eddy Simulation

The basic idea of LES model lies in filtering the Navier-Stokes equations. In the filtered version of the model, flow characteristics are replaced with their filtered quantities. For a physical quantity u , the filtered quantity $\bar{u}$ is calculated through a convolution:

$$\bar{u} = \iiint_{-\infty}^{+\infty} G(\mathbf{x}, \mathbf{y}) u(\mathbf{y}) d\mathbf{y}, \quad (2.19)$$

where $\mathbf{x}, \mathbf{y}$ are 3-D spatial vectors, G is a smoothing kernel. The frequently used kernels are the box, or top-hat type, the Gaussian type, and the sharp spectral cut-off. Among them, the box type is a natural choice for numerical solvers that implement the finite-volume method:

$$G(\mathbf{x}, \mathbf{y}) = \begin{cases} \prod_{\alpha=1}^3 \frac{2}{\Delta_{\alpha}} & \text{for } |\mathbf{x}_{\alpha} - \mathbf{y}_{\alpha}| < \Delta_{\alpha}/2 \\ 0 & \text{otherwise} \end{cases} \quad (2.20)$$

In terms of filtered variables the full Navier-Stokes equation may be written as:

$$\frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{w}}{\partial z} = 0 \quad (2.21)$$

$$\frac{D\bar{u}}{Dt} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x} + \nu_T \nabla^2 \bar{u} + \frac{\partial \tau_{xx}^{\text{SGS}}}{\partial x} + \frac{\partial \tau_{xy}^{\text{SGS}}}{\partial y} + \frac{\partial \tau_{xz}^{\text{SGS}}}{\partial z} \quad (2.22)$$

$$\frac{D\bar{v}}{Dt} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial y} + \nu_T \nabla^2 \bar{v} + \frac{\partial \tau_{yx}^{\text{SGS}}}{\partial x} + \frac{\partial \tau_{yy}^{\text{SGS}}}{\partial y} + \frac{\partial \tau_{yz}^{\text{SGS}}}{\partial z} \quad (2.23)$$

$$\frac{D\bar{w}}{Dt} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial z} + \nu_T \nabla^2 \bar{w} + \frac{\partial \tau_{zx}^{\text{SGS}}}{\partial x} + \frac{\partial \tau_{zy}^{\text{SGS}}}{\partial y} + \frac{\partial \tau_{zz}^{\text{SGS}}}{\partial z} \quad (2.24)$$

Here τ_{ij}^{SGS} are the subgrid-scale stresses, which arise when we try to filter a product of two flow variables in the convection terms; $\tau_{ij}^{\text{SGS}} = \bar{u}_i \bar{u}_j - \overline{u_i u_j}$, and hence implies the effect of non-linearity of turbulent flow. Using the eddy-viscosity concept, this subgrid-scale stress can be represented through the strain-rate tensor $\bar{S}_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right)$ as:

$$\tau_{ij}^{\text{SGS}} = 2\nu_T \bar{S}_{ij}. \quad (2.25)$$

The turbulent viscosity, ν_T , can be approximated by various models. The simplest one, Smagorinsky–Lilly model (Smagorinsky, 1963), is dependent on the rate of strain:

$$\nu_T = (C_s \cdot \Delta)^2 |\bar{S}| \quad (2.26)$$

where Δ , a representative grid cell size, usually taken as $\sqrt[3]{\Delta x \cdot \Delta y \cdot \Delta z}$ or just l_{cell} for octree grids; and $|\bar{S}| = \sqrt{2\bar{S}_{ij}\bar{S}_{ij}}$. By using this, one assumes that turbulence is isotropic at small scales.

The constant C_s is theoretically derived to be 0.23 (Lilly, 1966). Other studies showed that practical values of C_s are in the range $0.13 \rightarrow 0.21$. Deardorff (1970), through simulating turbulence in channel flow, claimed that the theoretical value of C_s is too high and may cause excessive dissipation; and suggested a value of 0.10 for C_s . Canuto and Cheng (1997) proposed a SGS model to yield $C_s \sim 0.11$. Smagorinsky (2010) listed various values for C_s found in literature, in both original and derived forms.

More complex models such as Germano et al. (1991)'s dynamic model can compute spatially-distributed stress and therefore eliminate the empirical choice of C_s . Such method, however, introduces additional equations to the existing solver, which is not easy to implement on a modular, multi-component software package like Geris. For this reason, this study will confine to the use of simple Smagorinsky–Lilly formulation with C_s determined appropriately for turbulent round jet cases.

2.3.5 Turbulent mixing

A usual way to describe the dispersion of passive substances in fluid is through a diffusivity coefficient, Γ . In the popular k - ε model, Γ in the transport equation is similar to the role of ν_T in momentum equations. The two quantities are linked by a coefficient (see e.g. the text by Rodi (1993)). Applications of turbulent models to environmental and chemical mixing problems are common (e.g. Orszag and Pao, 1974).

For turbulent shear flow in particular, however, the use of Γ has limitation, as reported by Corrsin (1974). To apply the gradient transport approximation, the mean free path of the transport process must be much larger in magnitude than the characteristic length scale of the turbulence; however this is usually not satisfied.

As an alternative to the gradient transport model, an explicit tracer advection procedure can be implemented. In this procedure, a number of imaginary particles are carried along with the flow, and concentration of such tracer will be estimated as the number of particles contained in a grid cell. A certain degree of “random walk” behaviour can be applied to the particles in order to simulate diffusion of this tracer.

2.4 Numerical codes

2.4.1 Solver

The computer program to be used throughout this study is Gerris (Popinet, 2003), a general-purpose CFD solver for incompressible fluid dynamics. The solver is implemented with a fractional time-step method (Chorin, 1968) to solve the incompressible Euler equations:

$$\mathbf{U}_t = -u\mathbf{U}_x - v\mathbf{U}_y - \nabla p \quad (2.27)$$

$$\nabla \cdot \mathbf{U} = 0. \quad (2.28)$$

(For viscous flow, additional diffusive terms will be added as momentum sources to represent viscosity.) The flow variables in the new time step, $\mathbf{U}^{n+1}$, are calculated from the present ones, $\mathbf{U}^n$, through an intermediate estimate, $\mathbf{U}^{**}$ (viz., the Hodge decomposition), based on the following formulae:

$$\mathbf{U}^{**} = \mathbf{U}^n - \mathbf{A}^{n+1/2} \Delta t \quad (2.29)$$

$$\nabla^2 \phi = \nabla \cdot \mathbf{U}^{**} \text{ with } \frac{\partial \phi}{\partial \mathbf{n}} = \mathbf{U}^{**} \cdot \mathbf{n} \text{ on } \partial\Omega \quad (2.30)$$

$$\mathbf{U}^{n+1} = \mathbf{U}^{**} - \nabla \phi. \quad (2.31)$$

Here $\mathbf{A}^{n+1/2}$ is the approximation for the advection term, $[(\mathbf{U} \cdot \nabla)\mathbf{U}]^{n+1/2}$ at the time instance corresponding to half of a timestep. The quantity $\mathbf{A}^{n+1/2}$ is computed using a second-order scheme and then $\mathbf{U}^{n+1}$ is determined using the projection method (Bell et al., 1989) involving the solution of a Poisson equation.

The advection of tracers can be simulated by Gerris. The method of tracking can be simple advection-diffusion of a scalar variable, or incorporating the Volume-of-Fluid technique. The latter is preferable in the case of two fluids with an interface, as was done especially in the researches involving two-phase flows (Tomar et al., 2010; Fuster et al., 2011; Tryggvason et al., 2011).

2.4.2 Grids

In Gerris, the governing equations are solved using finite volume method, in which discretisation is performed on an octree grid. In such a hierarchical system, each com-

putational cell has a cubic shape and can be refined, i.e. split into eight octants called “child cells” (Samet, 1984). The refinement is implemented, splitting the local cells from $level$ to $level_{\max}$ if certain condition is met. There are two cases of refinement:

- Initial refinement: Naturally the flow structure varies within the flow domain. To take into account the effect of geometrical factor, a function of spatial coordinates is usually used to specify the initially desired resolution level of the grid. For instance, in this study the region approximating the turbulent cone is refined as follow: $level = level_{\max}$ if $y^2 + z^2 < 0.04x^2$. For other problems, the initial refinement might be performed near a solid surface, which is useful to simulate wall turbulence.
- Adaptive refinement: During the transient phase, the flow continuously evolves and grid refinement should be expanded to reflect the spatial extent in which strong turbulent mixing occurs. For this purpose the criteria to be used are typically hydrodynamic characteristics such as velocity gradient and vorticity:

$$level = level_{\max} \text{ if } \frac{|\nabla \times \mathbf{u}| \Delta x}{\max |\mathbf{u}|} > \vartheta \quad (\text{vorticity})$$

where ϑ is a predefined threshold value.

The ease to perform refinement on quadtree/octree grids makes them an attractive choice for modelling dynamic systems at multiple scales. For microscale multiphase flow, there have been applications of quadtree grids to Lattice-Boltzmann simulations (Tölke et al., 2006), motion of droplets Agbaglah et al. (2011), granular flows (Staron et al., 2012), and laminar fountains (Srinarayana et al., 2008). In environmental flow modelling, despite the prevalence of unstructured (triangular) grids, the quadtree grid has been applied for hydrodynamic modelling of lake circulation (Gáspár and Józsa, 1992; Gáspár, 1996), river flood inundation (Liang et al., 2004, 2008), marine pollution (Borthwick et al., 2000; Copeland et al., 2003). There are reported works on wind wave prediction (Tsai et al., 2013), coastal wave dynamics (Park and Borthwick, 2001), tsunami wave propagation (Popinet, 2012), and ocean circulation modelling (Popinet and Rickard, 2006). For some applications, the development of quadtree algorithm for CFD has been carried out (Yiu et al., 1996; Greaves and Borthwick, 1998).

In the Gerris computer program, solving the Poisson equation is the main task; this involves discretisation of the Laplacian terms. For the simple case of a regular Cartesian grid, with grid spacing h (Figure 2.4, left), the gradient of a field variable ϕ at the right face of a cell $\mathcal{C}$ is:

Figure 2.4: Cells involved in discretisation on a regular grid and on a octree grid [plan view] (Adapted from Popinet, 2003.)

$$\nabla_E \phi = \frac{\phi_E - \phi}{h} = -\frac{1}{h}\phi + \frac{\phi_E}{h}.$$

In generalisation to octree grid with different cell sizes, one would expect a similar linear formula: $\nabla_E \phi = \alpha_E \phi + \beta_E$. This relationship holds, and Popinet (2003) illustrated for a case where cell $\mathcal{C}$ is half the size of its east neighbour cell, with $\alpha_E = -\frac{1}{3h}$ and $\beta_E = \frac{1}{h} \left(-\frac{1}{5}\phi_W + \frac{1}{2}\phi_E - \frac{1}{20}\phi_{SE} + \frac{1}{12}\phi_{NE} \right)$. The solution of Poisson equation proceeds by iteratively calculating a relaxation operator (i.e. a corrector):

$$\phi \leftarrow \frac{h \nabla \cdot \mathbf{U}^{**} - \sum_d \beta_d}{\sum_d \alpha_d} \quad (2.32)$$

wherein the corrected velocity $\mathbf{U}^{**}$ is updated using the newly computed values of ϕ .

Gerris uses collocated arrangement of velocity vectors on the octree grid. The advection terms, $\mathbf{A}^{n+1/2}$ at a half-timestep of a cell are computed through the weighted sum of velocities at the centres of every cell faces, $\mathbf{U}_{N/S/E/W}^{n+1/2}$. To obtain these velocities, Gerris solves a Riemann problem using Godunov method (Bell et al., 1989) with an upwind scheme. To maintain a second-order accuracy, the determination of a flux from a smaller neighbour cell to cell $\mathcal{C}$ would incorporate a gradient of that flux in the transverse direction.

2.4.3 Artificial viscosity

False diffusion is inherent to numerical computations; its side-effect is creating an additional diffusive term to those discretised equations describing the flow. Alternatively, the simulated fluid flow behaves as if it has an additional viscosity, ν_{num} . Two factors of a numerical grid that can affect the viscosity include (a) how coarse or fine are the grid cells, and (b) how are the grid lines arranged w.r.t. velocity vectors.

Grid resolution. Fine grid cells cost simulation run-time; halving the grid size would extend the simulation time by 16 for a 3-D explicit solver like Gerris. On the other hand, finer grids give more detailed flow patterns and produce lower artificial viscosity. The second issue is important, since it can influence the reproducibility of simulations. Finding a suitable viscosity and C_s is an approximate process because when subgrid-scale viscosity is introduced, the flow condition changes and this modifies the grid, which is adapted based on the hydrodynamic condition. The numerical viscosity previously estimated is just a reasonable value.

Artificial viscosity can be estimated in various ways depending on the specific flow condition. As an example, for a 2-D oscillatory wave, Lin and Philip (1998) estimated the artificial viscosity as

$$\nu_{\text{num}} \approx \frac{|U|l_{\text{cell}}}{2} \left(\alpha' - \frac{|U|\Delta t}{l_{\text{cell}}} \right) \quad (2.33)$$

where α' is a weighting coefficient between the central differential scheme and the upwind differential scheme, $|U|\Delta t/l_{\text{cell}} < \alpha' \leq 1$. For our case the most critical part is near the nozzle where $|U|\Delta t/l_{\text{cell}} = 0.8$ (Courant-Friedrichs-Lewy condition), leading to $0.8 < \alpha' \leq 1$. Hence, a rough estimate of local numerical viscosity is $\nu_{\text{num}} \leq 0.1|U|l_{\text{cell}}$.

Grid alignment. The existence of diagonally-oriented velocities in an upwind numerical scheme is known to cause false diffusion (e.g. Ferziger and Perić, 2001, p. 88). In such cases the 2-D flow system exhibits an effective diffusivity given by the formula $\Gamma_{\text{eff}} = U \sin \theta \cos \theta (\Delta x \cos \theta + \Delta y \sin \theta)$, where θ is the angle between the velocity vectors and gridlines. An alternative formula suggested by Mallinson and de Vahl Davis (1972) is:

$$\Gamma_{\text{false}} = \frac{\rho U \Delta x \Delta y \sin 2\theta}{4 (\Delta y \sin^3 \theta + \Delta x \cos^3 \theta)}.$$

In principle, the alignment of a boundary cross-section diagonally to the flow direction is not recommended. For simulation of multiple jets, however, it is inevitable because among two jets intersecting at an angle, at least one of them must be diago-

nally aligned relative to the gridlines.

2.5 Computational techniques and issues

2.5.1 Octree grid manipulation

The adaptive nature of octree grids provides the users with adequate computational resolution in the turbulent region of the jet which attracts much of our attention, while keeping the resolution of the outer region at a relatively low level to save computing resources. Compared to the conventional Cartesian grid, the octree grid can reduce a large amount of computer memory, and in some cases, computational time. Consider the simulation of a single, non-buoyant jet in Section 3.1, where 3.3×10^5 grid cells are used. Had a stretch cylindrical grid with similar number of grid cells been used, the number of grid lines in each dimension would have been around $n_x = 230$, $n_r = 30$, $n_\phi = 48$. Such a structured cylindrical mesh would, however, yield grid cells of very different sizes depending on the distance r from the jet axis, which is not suitable for modelling the flow within ZEF.

From the computing point of view, both structured and unstructured grids have certain advantages. For structured grids, the conventional matrix indexing scheme is fast: it takes a constant amount of time to access each element, and thus implies a computational complexity of $O(1)$. On the other hand, a hierarchical octree can be designed appropriately as a fully-threaded tree (Khokhlov, 1998) to yield an access time of $O(N \log_2 N)$ where N is the total number of leaf cells. Despite the computational cost for accessing neighbour cells, which is important in e.g. estimating gradients (see Section 2.4.2), an octree grid is capable of fast adaptive refinement. Grid refining or coarsening is performed simply by inserting new cells or deleting old ones and alter the corresponding connections with neighbour cells. In contrast, similar changes in a structured grid will affect all elements from that location toward the end of every matrix that stores the flow variables.

2.5.2 Accuracy and speed

The Gerris program has been tested by the software development team. For the iterative Poisson solver (cf. Eq 2.32), a test to compare the simulation results with a known analytical solution showed that the maximum residual of solution decreases exponentially with CPU time (Popinet, 2003). Also the error norms ($\|\phi\|_n = \frac{\sum_i |\phi|^n h^2}{\sum_i h^2}$, where $n = 1, 2, \infty$ corresponding to the 1-, 2-, and ∞ -norm) were found to decrease

exponentially with the grid resolution level, and the convergence order approaches 2 as this level increases. Another test for the convergence of the Godunov advection scheme also showed that the error norms decreases exponentially with grid level and the order of convergence is about 2.2.

The speed of computation in a numerical grid is usually indicated by the number of “lattice [node] updates per second” (LUPS), assuming that the cost of calculation for each cell or node is similar. Gerris measures this speed in “node-timestep/s”. For various simulations with different problem sizes, the simulation durations differ, but the speed should remain stable and characterises the hardware system. At the time of writing, four CPU cores (of the Intel Sandy Bridge architecture) operating at 3.5 GHz typically perform at 45,000 node-timestep/s. This speed was attained when modelling the flow and tracer transport with turbulence generated using Smagorinsky’s method (see Section 5.2).

The time consumed by various subprocesses within the solver may be estimated as follows: around 55% is devoted to solving the cell-centred velocities (advection-diffusion), 20% for solving Poisson equation, 14% for approximate projection, 13% for the Marker-and-Cell (MAC) projection, 12% for predicting velocity at cell interfaces, and the remaining for grid adaptation, advection of tracer, and other input/output tasks. Therefore, the addition of any computation elements to the existing solver, such as the turbulence approximation implemented in Chapter 5, has to be evaluated carefully because this will cause significant increase to the runtime of the program ($\sim 30\%$ in the present simulation).

2.5.3 Parallel computing

To leverage on computing power, parallel computation on multiple CPU cores is supported using Message Passing Interface (MPI).² The computational domain is divided into several contiguous parts, in each subdomain calculations are performed by one CPU core. Figure 2.5 shows an example for a two-jet simulation, wherein each colour denotes a region responsible for a CPU core. At each time step, the flow variables need to be transferred between two adjacent domains in order to compute the flow condition at the common interface. Because every data transfer between CPU cores takes a significant amount of time compared to a calculation performed by one CPU core, the speed-up ratio by using more cores is not as much as the ratio of increase in the cores used.

As an example, the initial development stage of two impinging jets is simulated

²The Message Passing Interface (MPI) standard www.mcs.anl.gov/mpi/, retrieved 30-Jul-2013.

Figure 2.5: Partition of computational domain for simulation using 4 CPU cores shown on three orthogonal planes

using various number of CPU cores of a desktop system and of the cluster. Figure 2.6 shows the peak performance of each system. It can be seen that for both the desktop and cluster computer systems, doubling the number of CPU core used will probably lead to around 1.7 times increase in performance (in terms of node·timestep/s).

The computer systems used for simulation in this study all have parallel processing capabilities. The desktop PCs feature Intel Sandy Bridge/Ivy Bridge³ architecture operating at 3.3→3.5 GHz and each PC is capable of running on four CPU cores simultaneously. The NTU High Performance Computing cluster features a multiconnected network of computer resources that users can log in and submit jobs (computer programs) that may require up to, theoretically, 64 CPU cores simultaneously. Each computing node in this cluster consists of 8 CPU cores (Intel Westmere architecture) operating at 2.66 GHz. The `qsub` software utility is used to submit batch jobs of simulation on the cluster. Despite the availability of CPU cores and memory, the cluster system requires waiting time for resource allocation. Moreover, output data must be downloaded to local PCs before being visualised.

³Intel, Sandy Bridge, Ivy Bridge and Westmere are registered trademarks of Intel Corporation.

Figure 2.6: Performance of computing system for different number of CPU cores used

Chapter 3

Simulation of a non-buoyant jet

3.1 Simulation set-up (Gerris)

3.1.1 Geometry and scaling

In the numerical model for a single jet, the flow domain is composed of two unit cubes aligned side-by-side. A circular jet inflow is defined at one end and a free outflow boundary is prescribed at the opposite end. All other sides of the system are considered as rigid, free-slip boundaries (Figure 3.1). The maximum cell level is $l_{\max}$, so the smallest cell size is $1/2^{l_{\max}}$, where 1 is the size of unit cell. The diameter of the jet is chosen as $6/512$, equivalent to 6 cell size of level $l = 9$. Far from the orifice, the grid size is as coarse as $1/8$ (i.e. level 3).

The domain should be wide enough to keep the turbulent flow away from being affected by the four lateral walls. A rule of thumb is that the jet is practically unaffected if the diameter of jet turbulent core, 2δ , is less than 40% the space between lateral walls. As a comparison to other simulation cases found in literature, Babu and Mahesh (2005) specified a grid extending $50d$ both laterally and longitudinally, and containing 30 cells across the jet nozzle diameter. Muppidi and Mahesh (2007) set up a domain with dimension $36d \times 64d \times 64d$, and 72 cells across the nozzle. Lee et al. (2002)

Figure 3.1: Geometric dimensions of the problem, including two unit cubes

constructed one half of a symmetric domain with dimension $64d \times 42d \times 32d$, and the jet nozzle diameter is 36 cells.

The domain is refined to the maximum level, $l_{\max} = 9$ at the vicinity of the inlet, $\{0 < x < 6d; r < d/\sqrt{2}\}$ (which is approximately one potential core length). Within the turbulent core, a refinement level of 7 is used for the region relatively far from jet orifice, $\{x > 12d; r/x < 0.2\}$, and a refinement level of 8 for the buffer region in between, $\{6d < x < 12d; r/x < 0.2\}$. For the ambient non-turbulent flow domain the coarsest cells are at level 3. This resulted in 3.3×10^5 grid cells, of which the majority are level 7 (3×10^5 cells).

No grid adaptation is made during this simulation, in order to keep the set of input parameters simple. The refinement criteria and thresholds are omitted at this stage. Although this might cause some loss of local turbulent motions, the resulting bulk characteristics of flow is still comparable with measured data in the literature.

To facilitate intercomparison between flow quantities of different magnitudes from various scenarios, nondimensional quantities e.g. x_* , U_* are adopted (see Section 2.2.1). Note that since $d = 0.75$ cm for all laboratory experiments, a predefined reference time makes the fluid viscosity constant. Specifically for $d = 6/512$, $\nu = 2.44 \times 10^{-6}$. However this is practically a crude value for the parameter, because the effective viscosity is altered by the presence of artificial viscosity due to finite grid cell size. So a better value for ν might be obtained through tuning and comparing simulation results with experiment data. Numerical diffusion related to the choice of grid resolution is discussed elsewhere (Section 2.4.3).

Simulations were performed from the initial state when the entire ambient water was still, until turbulence had been developed up to the outflow boundary. Time-averaging was performed for 4000 timesteps. During the simulation, the timestep was automatically chosen based on a default CFL (Courant-Friedrichs-Lewy) coefficient, $\Delta t \left(\frac{u}{\Delta x} + \frac{v}{\Delta y} + \frac{w}{\Delta z} \right)$, of 0.8.

3.1.2 Datasets available from the literature

Due to large demand on measurements of jet flows in the aerospace industry, many experiments had been conducted with air jets. Corrsin (1943) measured turbulent characteristics of air jet emitted from a wind tunnel at $\text{Re} \sim 10^5$. Corrsin and Kistler (1953) arranged further measurements to investigate the intermittency of “turbulent front” separating the jet flow with the nonturbulent ambient fluid. Wygnanski and Fiedler (1969) also conducted jet study at $\text{Re} = 10^5$, specifically in the self-preservation region. Panchapakesan and Lumley (1993) measured air jet at $\text{Re} = 1.1 \times 10^4$ to examine

higher-order moments of the turbulent fluctuation signals.

Among the publicly available data on water jet flows, the set of experimental data produced by Papanicolaou (1984) is a good source. The experiments were conducted with round jets in water including both pure jet and buoyant jet. Several pure jet experiments are shown in Table 3.1. In fact, these experiments exhibited slight negatively buoyant jets ($g' = \frac{\Delta\rho}{\rho_a}g \sim 10^{-3} \text{ m/s}^2$). However, these test cases reflect most clearly the “momentum-dominated” flow condition selected from Papanicolaou (1984)’s experiments, with $x/l_M = 0.13 \rightarrow 0.23$. In this chapter, the data for comparison study with the numerical simulation are taken from this dataset. An extraction of the dataset from Papanicolaou (1984) relevant to the study is presented in Appendix C.

Table 3.1: Selected experiment by Papanicolaou (1984) to be reproduced in this study. For all these cases, $d = 0.75 \text{ cm}$ and $d^{\text{model}} = 1$. Measured flow quantities at cross-section x_* include U , u_{rms} , v_{rms} , $\overline{u'v'}$.

Expt. ID	Re	U_o (cm/s)	U_o^{model}	t_{ref}	ν_*	x_* (meas. data)
PAP16	2459.9	31.98	0.500	2.0	4.065×10^{-4}	28.33
PAP17	5426.9	70.55	1.102	0.907	1.843×10^{-4}	50.0
PAP20	10936.9	142.18	2.221	0.450	0.914×10^{-4}	100.0
PAP21	7303.8	94.95	1.484	0.673	1.369×10^{-4}	70.0
PAP30	10815.3	140.60	2.196	0.455	0.925×10^{-4}	88.0
PAP32	6092.7	79.206	1.238	0.808	1.641×10^{-4}	51.0

Wang and Law (2002) combined the DPIV (Digital Particle Image Velocimetry) and PLIF (Planar Laser-Induced Fluorescence) to study the turbulent characteristics of buoyant jets. Another source of data, by Ninomiya and Kasagi (1993), contains time-series measurements for a water jet at $\text{Re} = 2566$. Weisgraber and Liepmann (1998) used DPIV to study the turbulent behaviour in the region $x_* = 15 \rightarrow 30$ for two cases, $\text{Re} = 5500$ and 16000 .

For the output data generated by the author’s simulation, the following important flow characteristics need to be derived:

- The velocity at the centreline of the jet, U_{CL} , which is supposed to be inversely proportional to the longitudinal distance from the outlet (see Lee and Chu, 2003, p. 37 for a summary of formulae).
- The (half-)width of jet, b , which is expected to rise linearly versus longitudinal distance; such values were derived from lab experiments.
- The velocity profiles in each cross-section of the jet, which are supposed to become flattened as we go further from the outlet; and all collapse to a Gaussian

curve after a proper shape transformation; this self-conservation behaviour is affirmed through experiment by e.g. Wygnanski and Fiedler (1969).

- The turbulent kinetic energy profile, in “cup-shaped” form with highest intensity at the boundary of the jet where vorticity is maximum, and decreasing slightly toward the centreline of the jet, as confirmed by experiments, e.g. those by Hargrave et al. (2006).

Aside from analysing the results in Section 3.5, two factors should be taken into account: (i) the realistic boundary condition at the orifice and (ii) the choice of an appropriate value for ν considering that numerical viscosity already exists in simulation. These factors are not readily available from standard Gerris solver. A rational way of implementating these considerations has to be developed. The issue of inflow hydrodynamics is addressed in Section 3.2, while the existence of numerical viscosity is discussed in Section 3.3. For implementation details (computer input files), see Appendix A.2.

3.1.3 Data sampling and output interrogation

For simulations focusing at steady-state flow condition, the flow quantities (as field variables for the whole domain) can be captured at arbitrary time instances. Accumulated flow variables in a finite time period t includes $\sum u, \sum v, \sum w, \sum u^2, \sum v^2, \sum w^2$, in which $\sum u \equiv \sum_{t_i=0}^{t_i=t} (u_{t_i})$, etc. These variables can be stored in output files during simulation; details can be found in Appendix A. From these data, it is possible to obtain moments such as temporal means and variances, e.g. $U = \frac{\sum u}{t}$, $\text{Var } u = \frac{\sum u^2}{t} - \left(\frac{\sum u}{t} \right)^2$ and the root-mean-square values, or standard deviations, e.g. $u_{\text{rms}} = \sqrt{\text{Var } u}$.

Time series of flow variables at specific locations within the flow domain can be recorded during simulation. However, such an inquiry must be indicated in the input file before the simulation starts. These can be discrete sample points or an array of points for calculating integrals. For the situation involving arrays there is linear sampling and radial sampling (Figure 3.2). The former is used for cases involving compound jets and the latter for single axisymmetric jets. Figure 3.3 shows a radial sampling layout on a cross-sectional area of a jet. On the $z = 0$ plane (plan view), a slice of the octree grid is displayed in the left half and color shading denoting U in the other. On the cross-sectional plane ($x_* = 50$), contours corresponding to values $U_{CL}/20$ (boundary of the jet), U_{CL}/e (half-width of the jet), and U_{max} are shown.

Figure 3.2: Two sampling layouts on a jet cross-sectional plane. Left: linear sampling. Right: radial sampling.

Figure 3.3: Location of sampling points (white) in a jet simulation ($\text{Re} = 7.3 \times 10^3$).

All the output data can be produced with double precision, though exact values can only be obtained at cell centres. Otherwise a trilinear interpolation is performed to determine the values at the required locations.

Unlike temporal integration, spatial integration is not readily available in the program. To perform this operation, the domain for integration should be manually specified before simulation process to be carried out. After simulation, the integral is calculated by summing up the recorded values for the whole domain. As an example, consider the integration of time-averaged velocity components over $\theta = 0 \rightarrow 2\pi$ on a cross-sectional plane perpendicular to Ox . For U it is straightforward to take the average of values of $N(r)$ sample points on each concentric “ring” at a distance r

from O : $U(r) = \frac{1}{N(r)} \sum_{r_i=r} (\sum u)_i$, while for the radial velocity component: $V_r(r) = \frac{1}{N(r)} \sum_{r_i=r} \frac{(\sum v)_i y_i + (\sum w)_i z_i}{r}$. Reynolds shear stresses, which involve averages of a product between two variables, can also be calculated. However an auxiliary accumulator, $\sum uv_r$ where $v_r = (vy + wz)/r$, should be prescribed prior to each simulation run. Then $\overline{uv_r} = \frac{\sum uv_r}{N(r)}$.

3.2 Inflow hydrodynamic condition

Inflow is not only a source of momentum but also a source of turbulence. Despite the accuracy of the procedures to solve the full Navier-Stokes equations, the boundary conditions can only be realised to certain degree of accuracy. This implies that (a) rigid boundaries can be specified by some parameters but can never describe exactly the roughness of solid surfaces in real world, and (b) (more problematic), open boundaries are difficult to approximate in details, and even more difficult with a few bulk parameters such as in a simple turbulence model. Outflow boundaries are more tolerant to the inaccuracies of parameters (Ferziger and Perić, 2001) and can be specified as a free boundary which acts as a border with the non-turbulent surrounding fluid (Rodi, 1993, p. 44).

The inflow condition is characterised by two factors: the profiles of mean velocity and the turbulent intensity. This condition depends on the shape of the nozzle. A pipe nozzle would produce a parabolic velocity profile, whereas a gradually contracted orifice might produce a top-hat velocity profile. The inflow condition in this study is top-hat (theoretically-oriented), as opposed to many experiments and other numerical simulations with pipe flow. Note that those test cases conducted by Papanicolaou (1984) are based on top-hat inflow velocity profiles.

A simple way to specify the turbulent inflow is to impose a random velocity component on the time-averaged power-law velocity profile widely used for pipe flows. For example, Babu and Mahesh (2005) superimposed on the velocity profile a pseudo-random component with amplitude equalling 0.1% of the jet outlet velocity. However, this random approximation has drawbacks because in such a randomly-generated flow field, the correlation of velocity of any two points is essentially zero; whereas despite the apparent randomness, turbulence still exhibits some vortical patterns with a non-zero velocity correlation.

To circumvent this, there are other approaches which numerically generate a col-

lection of vortices in the boundary surface, analogous to the concept that turbulence is the superposition of coherent structures, or vortices. Townsend (1980) suggested a time-stationary periodic pattern of eddies. Davidson (2004) mentioned Townsend's method and extended the discussion to 3-D turbulence. In a different implementation, Jarrin et al. (2006) described the turbulent flow condition at the inlet as a collection of individual turbulence spots. Each spot at a location $\mathbf{x}_i$ has a “shape function” f , with a standard deviation σ satisfying the normalisation condition, $\frac{1}{2\sigma} \int_{-\sigma}^{\sigma} f^2(x) dx = 1$. The total velocity at location x is contributed by all shape functions f_i : $u(x) = \frac{1}{\sqrt{N}} \sum_{i=1}^N \epsilon_i f_i(x - x_i)$, where $\epsilon = \pm 1$ is a random factor. Based on Kraichnan (1970)'s formulae describing random motion of particles using Fourier series, Smirnov et al. (2001) made a modification to apply for the case of 3-D turbulent motion,

$$u_i(x_j, t) = \sqrt{\frac{2}{N}} \sum_{n=1}^N \left[p_i^n \cos(\tilde{k}_j^n \tilde{x}_j + \omega_n \tilde{t}) + q_i^n \sin(\tilde{k}_j^n \tilde{x}_j + \omega_n \tilde{t}) \right], \quad (3.1)$$

where the tilde symbol denotes a normalisation to the length- and time-scales of the turbulence; the terms p, q, k, ω all contain a random factor following the normal distribution. After a proper transformation the resulting velocity field will have a correlation function of $\overline{u'_i u'_j}$, and the turbulent length- and time-scales are equal to the preferred values. Kondo et al. (1997) devised a method for creating velocity fluctuation with the expected correlation and power spectral density, which also possesses very small velocity divergence.

Another, more practical, technique is to introduce a short pipe section at the inlet to make the inflow velocity distribution change from the idealised top-hat type to a typical, fully-developed turbulent flow velocity profile. This method is applied in the work by Muppidi and Mahesh (2007) where a pipe of length $2d$ is included in the computational domain.

In the present version of Gerris, implementing the specification of a spatially-varying random velocity field at the inflow is not straightforward. It is also somewhat not “worth the effort” because turbulence in a pipe is generally simpler than that within a typical open waterbody. On the other hand, if the lead-in pipe section is introduced then some way of invoking perturbation to the flow is needed. For example, one has to make the pipe surface rough, or introduce a certain amount of turbulent viscosity to the inflow (e.g. in Lee et al., 2002), or specify the boundary values of k and ϵ for the turbulent transport model (e.g. in Besbes et al., 2003). Such method is, to some extent, subjective and is not applied in this study.

To overcome these difficulties, the author simply used a top-hat velocity profile for the inflow, but now show that the turbulence state produced by simulation is almost

as observed near an orifice in reality. Simulated profiles of U and turbulent intensity, u_{rms}/U at several cross-sections including $x_* = 0.55$ and $x_* = 1.11$, for a case with $\text{Re} = 1.6 \times 10^4$, were compared with available data from Mi et al. (2001)'s experiments. The simulation results show top-hat velocity profiles, as expected; but turbulent intensity in the middle part of the cross-section (Figure 3.4) is virtually zero compared to about 0.5% of the aforementioned measured data. When x_* increases, this middle part rises slowly but narrows quickly so the agreement between measured and simulation should compromise at either lower part. Nevertheless, as the distance x_* has little influence on u_{rms}/U it is reasonable to consider that $x_* = 1.5$ corresponds to the location of the physical jet nozzle, for the purpose of data comparison. The same remark applies to other cases and since most measurements were taken far from the nozzle ($x_* > 20$), it is reasonable to locate the inflow boundary for simulation corresponding to the location of the physical nozzle or orifice.

Figure 3.4: Turbulent intensity of cross-sections $x_* = 0.55$, and 1.55 compared to laboratory data at $x_* = 0.05$ by Mi et al. (2001)

3.3 Influence of grid cell size on simulated jet flow

As an illustration, a jet flow simulation is performed for two cases with $\text{Re} = 8000$, but having different resolutions. One grid is refined the same way as described in Section 3.1, the other grid contains cells of twice side length.

Results from this simulation at cross-section $x_* = 50$ (Figure 3.5) shows that a coarser grid produces more viscous flow. For the present case this translates to less lateral spreading, less mixing, a peaky velocity profile and weak turbulent motions. In particular, the centreline velocity in the fine mesh is closer to that expected from

Figure 3.5: Comparison of mean velocity and turbulent intensities for two cases with same $Re = 8000$. Solid lines: fine grid. Dash lines: coarse grid.

empirical formula (i.e. $6.2x_*^{-1} = 0.124$). Based on the fact that observed maximum turbulent intensities are practically about 0.05, the fine grid is also better between the two. However, this test does not imply a particular value for the grid cell resolution. The grid may be locally refined to yield realistic flow patterns. Alternatively, viscosity of the flow field can be specified through a source term in Gerris, which can be considered as an input for the model.

For quadtree structures, the grid spacing does not vary continuously but change in a two-fold scale; hence an analysis on the effects of grid spacing to the numerical solution is not appropriate here. Instead, the above computation just served as an example to show that in the present study of free turbulent jets, a maximum grid level of 9 near the orifice and level 7 in the turbulent core is reasonable.

3.4 Parameter selection

Although geometrical dimensions and inflow of the jet are fixed and the governing equations for this flow are assumed to be correct (at least for the ideal case under consideration), there is an important parameter—numerical viscosity—which affects the numerical solution obtained. Section 3.4.1 presents a preliminary method to determine the viscosity based on a physical interpretation, while Section 3.4.2 discusses the adjustment of this parameter as an input of the Gerris program.

3.4.1 Provisional choice of viscosity

The turbulence pattern of a jet needs to be properly simulated to reflect detailed dynamics of the flow field. In order to achieve this, the first consideration is choosing a value of an intrinsic viscosity—that if the flow were laminar, and then add a turbulent term to the momentum equation to the Gerris solver.

Numerical models such as Gerris tend to be dissipative in terms of energy. This behaviour is attributed to false diffusion of momentum, which arises when an upwind scheme is employed in the model. In Section 2.4.3, a preliminary method to estimate the numerical viscosity based on simple parameters, U and l_{cell} , has been discussed. In this section, attention is focused on the derivation of numerical viscosity from energy variation of the system.

Popinet, in an example¹ implementing the projection method by Rider (1995), estimated the viscosity based on the energy degradation of the flow which evolves in a closed box (size L) with no additional source of momentum. A physical viscosity value of zero is prescribed for this flow but due to numerical viscosity, the kinetic energy (E_{kin}) lowers gradually and eventually the fluid becomes stationary. The numerical viscosity being responsible for energy dissipation can then be estimated as

$$\nu_{\text{num}} = \frac{-\log(E_{\text{kin}}/E_{\text{kin}}^0)}{4t(2\pi\mathcal{M})^2},$$

where $\mathcal{M}$ can be seen as the number of oscillation modes per unit length (see a representation of flow structures in Townsend, 1980). This method is specific only for an enclosed box.

For homogeneous turbulence the decline in total kinetic energy is known (Davidson, 2004, p. 300) to relate with the viscosity as follows:

$$-\frac{dE_{\text{kin}}}{dt} = -\frac{d}{dt} \int_V \frac{1}{2} |\mathbf{u}|^2 dV = \nu \int_V |\vec{\omega}|^2 dV. \quad (3.2)$$

This formula provides a means to estimate ν based on the total kinetic energy and vorticity of the system, which can be numerically integrated in a convenient way for computer programs such as Gerris. In this particular problem, to take into account the supply of kinetic energy from the inflow jet, a term $\frac{1}{2}U_{\text{o}}^2 \left(\frac{\pi}{4}dU_{\text{o}}\right)$ representing the energy flux which has to be added to the declined rate of energy.

A preliminary simulation is performed, with zero viscosity as an input. The computed viscosity fluctuates rapidly at first (Figure 3.6) but then converges toward a mean value, with a relatively small standard deviation. This indicates an intrinsic viscosity,

¹<http://gerris.dalembert.upmc.fr/gerris/tests/reynolds.html>

Figure 3.6: Schematic time series of computed viscosity, ν_{comp} (the average is shown as a thick solid curve). The numerical viscosity, ν_{num} , is the excess of ν_{comp} with regard to the initially-chosen ν_{init} for the simulation.

even when apparently the input viscosity is zero. For four different program runs, the deviation between the computed viscosities and the initially prescribed viscosities are listed in Table 3.2. It can be seen that when the initial viscosity is increased, the computed viscosity also rises, but at a slower rate. Then the computed viscosity (a bulk quantity) is the sum of numerical viscosity (another bulk quantity, for now) and the initially prescribed viscosity,

$$\nu_{\text{comp}} = \nu_{\text{num}} + \nu_{\text{init}}. \quad (3.3)$$

To balance the existence of ν_{num} , the value of ν_{init} should be reduced accordingly. The target is to obtain $\nu_{\text{comp}} \approx \nu$, the kinematic viscosity of water (considering the scale of model to preserve the Reynolds number). The procedure to find ν_{init} is iterative. First the kinematic viscosity of water, e.g. 2.44×10^{-6} in Case PAP20, is used as an input for the simulation. After a certain simulation time until the viscosity of system reaches stability, this value of viscosity is compared to the desired value, ν . If $\nu_{\text{comp}} \approx \nu$ then this ν_{init} is chosen; otherwise $\nu_{\text{init}} + (\nu - \nu_{\text{comp}})$ is chosen as the input for the next simulation. The selected initial viscosities to be used as input for different scenarios are listed in Table 3.2. Note that ν_{init} can be negative because it will be added up with ν_{num} to represent a physical quantity.

An estimation using Lin's formula, Eq (2.33), results in a numerical viscosity of $O(10^{-5})$ for the jet core; however the bulk viscosity (averaged for the domain) is lower.

In the procedure previously adopted, ν_{num} is the “net” numerical viscosity. Generally $\nu_{\text{num}} > 0$ and is an indication of the grid effect in numerical modelling. In simulations carried out for this study, the ν_{num} value fluctuates significantly, and has an average value in the order of 1×10^{-5} , an order larger than the kinematic viscosity of

Table 3.2: Initially prescribed viscosity, ν_{init} , and computed (mean) viscosity, ν_{comp} , for the case no turbulence model is implemented.

Case	Re	$\nu_{\text{init}}, \times 10^{-6}$	$\nu_{\text{comp}} \times 10^{-6}$
PAP16	2460	-164.	70.
PAP17	5427	-62.	?.
PAP32	6093	-65.	?.
PAP21	7304	-75.	?.
PAP20	10937	-382.	10.

the water in simulation (2.44×10^{-6}).

The method of using a uniform value of ν_{num} for the entire computational domain is physically not true, as the turbulent viscosity in the jet core may be much greater than that in ambient water. However ν_{num} is still a useful parameter which characterizes the behavior of the system. From that perspective, a more complicated jet system also has a value of ν_{num} , which can be found empirically as previously mentioned (Figure 3.6 and Eq. (3.3)).

The proposed model should be applied to simple cases of a single jet only, as the magnitude of ν_{num} may then be compared to analytical calculation such as that reported by Davidson (2004, pp. 154-156). In such case, the semi-analytical estimation of ν_T reduces to $(6.2 \times 10^{-3} \times (2 \times 0.114) \times 6.2 \times d \times U_o)$ where the value 0.114 represents the expansion rate of jet core and 6.2 is the empirical coefficient associated with centerline velocity decay. For $d = 0.0075$ m and $U_o \approx 1.4$ m/s used in the PAP20 and PAP30 experiments with $\text{Re} \sim 10000$, $\nu_T \approx 8.6 \times 10^{-5}$; the result is considerably greater but is within one order of magnitude compared to that simulated (which averages to 3×10^{-5} , see Table 3.3). The reason for this underestimation of ν_{num} in simulation is that the value of numerical viscosity is for the entire domain including still water whereas the turbulent viscosity is that for the jet core.

3.4.2 Parameter tuning

The value of the numerical viscosity depends on the local flow velocity (which hopefully not to be changed) and local grid cell size (which can be adapted dynamically during simulation). To understand a hydrodynamic system in most studies, except DNS and fine-resolution LES, it is better to control the spatial distribution of l_{cell} rather than introducing dynamic adaptation. This control can be gained by specifying how the grid was refined during the entire simulation. But there are infinitely many ways to refine the grid. In particular, for this study the grids are refined in two regions: a cylindrical volume protruding from the inflow plane, and an expanding cone adjacent

Table 3.3: Prescribed values for the viscosity source term in the numerical model, with comparison to the estimated numerical viscosity

Case	Re	SourceViscosity	ν_{num} , estimated
PAP16	2459.9	0	$3 \times 10^{-6} \rightarrow 4 \times 10^{-5}$
PAP17	5426.9	-3×10^{-5}	$3 \times 10^{-6} \rightarrow 5 \times 10^{-5}$
PAP20	10936.9	-6×10^{-5}	$9 \times 10^{-6} \rightarrow 4.5 \times 10^{-5}$
PAP21	7303.8	-7.5×10^{-5}	$6 \times 10^{-6} \rightarrow 4.5 \times 10^{-5}$
PAP30	10815.3	-6×10^{-5}	$9 \times 10^{-6} \rightarrow 6 \times 10^{-5}$
PAP32	6092.7	-3.2×10^{-5}	$0 \rightarrow 5.5 \times 10^{-5}$

to the cylinder. For this refinement three parameters are involved, namely: radius of the cylinder, length of the cylinder, and slope of the cone shape.

It is normally desirable to constrain the variability of choice in refinement, in order to minimise the number of parameters. This was done by associating the dimensions of the grid regions that need to be refined, with the geometry and characteristics of the flow (the jet diameter at orifice, d ; the length of potential core, $\sim 6d$; and the jet spreading rate, db/dx). So practically there is only level of grid refinement that remains.

But since the target of this parameter selection process is to counterbalance the effects caused by numerical viscosity, it should be preferable if we can change the viscosity of system. This is possible in Gerris by specifying a parameter, `SourceViscosity`, which can be a function of spatial coordinates. However, for simplicity, this value is chosen as a constant in space. `SourceViscosity` can be a negative value, and a collection of such values for various cases is shown in Table 3.3.

It can be seen that, for most cases the chosen `SourceViscosity` appeared to be of the same order of magnitude with ν_{num} . Here the numerical viscosities were estimated locally using Eq. (2.33), with $\alpha' = 0.05$, for the cross-section containing measured data.

These values of `SourceViscosity` are chosen to give a good match between the simulated and laboratory-measured data. Both the mean velocity and turbulent intensities are considered, however no objective criteria are used during this selection process.

The selection of ν in engineering design is difficult because apparently ν has no simple correlation with Re. For the study presented in subsequent chapters, however, only relatively large value of Re is chosen, i.e. Re = 10815 and Re = 10936, corresponding to $\nu = -6 \times 10^{-5}$.

An implementation of Large Eddy Simulation (LES) in Gerris is shown on page 126. LES offers an additional parameter, namely C_s , to specify the turbulent condition

of the flow. However, in the present study, the author finds that using LES effectively reduces the viscosity of the system. This would result in wiggles on the plots of velocity and turbulent level profiles, which is not realistic. Another problem is that implementing LES would double the CPU run time, which is due to the limited time spent for the present study. So the implementation of LES serves as a proof-of-concept for this simulation technique.

3.5 Simulation results

3.5.1 Time-averaged velocity

The mean velocity profiles are recorded at various cross-sections in the zone of established flow, typically at the location of observed profile for each case (Table 3.1). On each cross-section, the sample points are arranged at various r_* on eight line segments radiating from the centre to facilitate circumferential integration of flow quantities. This approach yields smoother distribution of the variables compared to that using individual samples.

A direct comparison with the dataset of Papanicolaou is performed for all six cases with the corresponding measured data at corresponding cross-sections, $x_* = 100, 51, 50, 88, 28.3$, and 70 (Figure 3.7). The computed profiles are generally flatter than those measured, with wider spreads of velocity. It can be seen that generally those parts of the profiles near the inflexion points are closer to Papanicolaou's data points (in terms of absolute difference). Nevertheless, the corresponding velocity profiles from empirical and numerical approaches should yield the same integration volume, which is the mass flux at that particular cross-section.

A common feature is that the simulated centreline velocity, $U_{\max}$, or U_{CL} , is smaller than that measured experimentally for each case. This situation illustrates a high degree of momentum transfer in radial direction. While coarsen the grid may yield a shaper peak such as the case for $Re = 2460$, this approach would lead to lower resolution of the flow field. Having a higher resolution almost always lead to superior quality of simulated turbulent intensities, or Reynolds stresses, as is also reported by Babu and Mahesh (2005), and Muppidi and Mahesh (2007). As a compromise, the candidate chose to reproduce the turbulence and Reynold stress. The central velocity occupies but a small percentage of the flow, and would have less effect interm of mixing of the jet in the recipient water body.

The variation of velocity profiles along the x -direction is depicted in Figure 3.8, for only one specific case of turbulent flow (PAP20). The flattening trend of these velocity

Figure 3.7: Mean axial velocity U_* , circumferentially-integrated for various cases. Horizontal axes denote r_* . As with the subsequent Figures 3.11–3.12, the simulation results are mirrored to give clearer comparison to measurement data (■, Papanicolaou (1984)), which were obtained at only one transect perpendicular to the flow direction.

Figure 3.8: Velocity profiles for the cases $Re = 10937$ at various cross-sections; results of simulation (♦) compared to empirical formula (solid lines)

Figure 3.9: Profiles of mean axial velocity (left:) at various cross-sections, x_* after normalisation for the case with $Re = 1.1 \times 10^4$, (right:) at $x_* = 50$ for various Reynolds numbers

profiles can be identified. Also shown in Figure 3.8 are the idealised Gaussian profiles. The mathematical expression for such a profile is $u(r) = U_{CL} \exp(-r^2/b^2)$, where $U_{CL} = 6.2/x_*$ and $b = 0.114x$. In Papanicolaou's experiments, an equivalent form, $u = U_{CL} \exp(-ar^2)$ was presented, where the coefficient a ranges from 0.1781 at cross-section $x_* = 25$, to 0.0158 ($x_* = 100$), corresponding to the increasing values of b_* from 2.37 to 10.6. After normalising the axial velocities to U_{CL} and radial distances to b , the profiles are shown on the left-handed chart of Figure 3.9. It is apparent that from the centreline to the inflexion point (i.e. $r/b = 1$) the normalised curves collapse well, except that the curve for $x_* = 100$ tends to deviate from the rest.

The right-handed half of the profile shown in Figure 3.9 suggests that the Reynolds number might not have significant influence on the simulated velocity profile obtained from a specific cross-section. The data points obtained from simulation, however, converge only at the part near the inflexion points ($r_* \approx 5$). For most part of the range $0 < r_* < 12$, the variability of U_* is restricted in an envelope of which $\Delta U_* \sim 0.02$.

The relatively low goodness-of-fit is mostly due to points in cross-section further from the orifice. A possible cause is due to the slight influence of the lateral boundary on the jets, which is more pronounced for larger (x/d). In Papanicolaou's experiments, ample space for experimental apparatus allow the lateral walls are located as far apart as $W/d = 150$. In contrary, due to limitation in computing power, this simulation only assumes $W/d = 85$.

Another point worth noting is that, although Wygnanski and Fiedler (1969) considered the flow to be self-preserving for $x_* = x/d > 70$ but in the present numerical study this self-preserving feature appears relatively near, and possibly even for $x_* < 50$.

Figure 3.10: The decay of centreline mean velocity (U_{CL}/U_o , left) and the increase in jet width (b_* , right) with longitudinal distance (x_*) (solid lines correspond to semi-empirical formulae $U_{CL}/U_o = 6.2x_*^{-1}$ and $b_* = 0.114x_*$ by Lee and Chu (2003); dashed line $U_{CL}/U_o = 6.71x_*^{-1}$ and $b_* = 0.104x_*$ by Papanicolaou (1984); dashed & dotted $U_{CL}/U_o = 5.67x_*^{-1}$ by Wygnanski and Fiedler (1969))

The decay of simulated centreline velocity with longitudinal distance is shown in Figure 3.10. The jet width is derived from simulated velocity profile as the distance between two inflexion points, although for certain cases the width is difficult to determine and hence omitted. Compared to the empirical formulae: $b = 0.114x$ and $U_{CL*} = 6.2x_*^{-1}$ (Lee and Chu, 2003), the simulation result underpredicts the centreline velocity and overpredicts the jet width. Specifically, the result overpredicts the jet width by 40% at $x_* \approx 110 \rightarrow 120$. This behaviour, together with the flat shape of mean velocity profiles, suggests the grid effect on flow condition. All measurement points on the cross-sections are in level-7 grid cells, whose size is $1/128$. With a typical velocity from $0.1 \rightarrow 0.2$, a quick estimate of the numerical viscosity for these cells based on Eq. (2.33) is in the order of 10^{-5} , one magnitude greater than the kinematic viscosity of water used in the model, $\nu_*^{\text{model}} = 2.44 \times 10^{-6} L^2$.

The comparison between simulation output and laboratory data obtained from Papanicolaou (1984) is the only available comparison regarding mean velocity profiles for this study. The candidate agrees that it is unsafe to rely on just one data source for comparison. There are a couple of reasons for this compromise: (i) it is not easy to gain access to the original data obtained from other well-established experiments such as those by Albertson et al. (1950), Wygnanski and Fiedler (1969), etc. (ii) Even when data is reproducible from published materials with adequate accuracy, the experimental conditions are different from that used for the flow condition considered in the present study. For example, Wang and Law (2002) investigated round jet but the jet was issued from pipe nozzles rather than orifices; (iii) the sampling location is too far

from the orifice and away from the intense mixing zone.

While most previous studies superimpose data points of various experiments spanning a broad range of Re , the author places strong emphasis on differentiating and presents velocity profiles for various Re values. The reason is for practical issue in simulation: each simulation for a specific value of Re should be parameterized by a suitable value of numerical viscosity, see Subsection 3.4.1.

3.5.2 Turbulence level

Based on the detailed data obtained from the simulation, statistical quantities such as mean value and root-mean-square deviation are calculated. A typical quantity is the axial turbulent intensity, or turbulence level, u_{rms}/U_{CL} . It is apparent from Papanicolaou's dataset that the turbulence level profile is trapezoidal: a high level of turbulent intensity is contained in the part $-1 < r/b < 1$ and this level quickly decreases as r increases. Specifically, a range of axial turbulence levels from 0.18 to 0.27 is found to be valid for a range of $Re \sim 600 \rightarrow 1.1 \times 10^4$. The shape of this normalised turbulent velocity profile does not considerably change with distance; its peak is located near the jet centreline, with a magnitude of $u_{rms}/U_{CL} = 0.25 \rightarrow 0.28$ (Pope, 2000, p. 105). For all cases, the simulated axial intensities are generally in accordance with those obtained from the experiments (Figure 3.11). The tail parts of simulation profile tend to be overpredicted, which might be attributed to the additional viscosity that numerical grids "impose" on fluid, which dampens the local fluid motion in the longitudinal direction.

Figure 3.11: Turbulence level of axial velocity u_{rms}/U_{CL} , circumferentially-integrated for various cases compared to experiment data (■, Papanicolaou 1984)

Compared to the axial turbulent characteristic, the lateral component, $(v_r)_{\text{rms}}/U_{CL}$, is about 50% weaker. Based on the output data (Figure 3.12), the part of velocity profile near the centreline is not well-predicted, but the remaining parts have better agreement with the observed data.

A note when interpreting the results obtained from Gerris simulations: values at $r = 0$ might appear to be singular, due to the way Gerris interpolates this value from either side of plane $y = 0$ (or $z = 0$). The program does not take into account that calculate values should be continuous in the domain and particularly at $r = 0$.

Figure 3.12: Turbulence level of radial velocity v_{rms}/U_{CL} , circumferentially-integrated for various cases compared to experiment data (■, Papanicolaou 1984)

Now consider $\overline{u'v'_r}$, which signifies the correlation between fluctuation motions in the axial and radial directions. The experimental data points show a clear trend: any strong fluctuation in the positive axial direction is associated with a strong fluctuation in the positive radial direction (i.e. the tendency of jet spreading). The quantity reaches zero at $r = 0$ because along the centreline there is no profound radial movement of water particles.

Figure 3.13: Reynolds shear stress $\overline{u'v'}/U_{CL}^2$, circumferentially-integrated for various cases. The simulation result is rotated to be compared with experiment data (■, Papanicolaou 1984).

Figure 3.13 shows periodic variations in the simulated $\overline{u'v'}$ for some cases. Generally the order of magnitude is captured, except for the overprediction in Case $Re = 2460$. However, the turbulent intensities are usually more difficult to reproduce compared to the mean axial velocity.

In addition to the previous comparison, simulation results are compared to the experimental data of jet flow at $Re \sim 10^5$ produced by Wygnanski and Fiedler (1969). The data include decay of U_{CL} versus x , time-averaged velocity and turbulent intensity profiles for $x/d = 50, 60, 75$, and 97.5 .

Beside experimental data, results from two other numerical studies are used to assess the performance of the present model (Gerris). The first numerical data to be used is retrieved from the work of Babu and Mahesh (2004) on low-Reynolds flow ($Re = 2400$) using Direct Numerical Simulation. The second one is from Large Eddy

Figure 3.14: Turbulence level of axial velocity u_{rms}/U_{CL} , circumferentially-integrated for various cases: simulation (thin solid line) compared to experiment data (Papanicolaou 1984 (■); and Wynanski and Fiedler 1969 (thick solid line))

Figure 3.15: Turbulence level of radial velocity $v_{r,\text{rms}}/U_{CL}$, represented similarly to Figure 3.14.

Simulation of a high-Reynolds flow ($Re = 11000$) by Bogey and Bailly (2009).

It can be seen that the results obtained from the case $Re = 10937$ better agree with data of Wynanski and Fiedler in two aspects: the peaky nature of $(v_{r,\text{rms}}/U_{CL})$ profile and the magnitude of $(\overline{u'v'})/U_{CL}^2$. For the case $Re = 10815$, the simulated profile of $(\overline{u'v'})/U_{CL}^2$ shows exaggerated fluctuation.

In the three sets of results observed (u_{rms}/U_{CL} , v_{rms}/U_{CL} , and $\overline{u'v'}/U_{CL}^2$), apparently higher Re causes higher turbulence level. (More intense fluctuation also corre-

Figure 3.16: Turbulence level of radial velocity $\overline{u'v'}/U_{CL}^2$, represented similarly to Figure 3.14.

Figure 3.17: Centreline turbulent intensity (symbols: present simulations) compared to (lines: results by Bogey and Bailly (2009)).

sponds to samples collected further from the orifice, i.e. greater x_* .) In this report, the result could have been improved by using an averaging filter to smooth out the wiggles in the curves. After all, simulation is just one kind of experiment; and experimental results often shows fluctuation (and sometimes, outliers). Another possible approach is to present the results in the form of a band bounded by the lower and upper envelopes. In the latter case, the band would have included almost all the data points.

3.6 Summary

Although not being the final target of this study, the simulation of a single round jet is important. It serves to confirm the reliability of the solver and provides the basis to build a framework for multiple impinging jets in the following chapters. After some considerations regarding issues of inflow boundary condition and artificial viscosity, we compared the numerical results with experimental data in the literature for a range of $Re \sim 5000 \rightarrow 10000$. Generally the simulated mean axial velocity profiles follows the empirical Gaussian curve, with discrepancy in peaks up to 20% (e.g. case $Re = 10937$). The trend of centreline velocity decay is reasonably well-reproduced. Due to lower centreline velocity, the simulated jet width is larger. Calculated turbulent levels were in less accordance with the measured data: a local difference of 100% might occur.

Established formulae e.g. in (Fischer et al., 1979; Baines and Chu, 1996; Lee and Chu, 2003) show no dependence of mean velocity and tracer concentration profiles on the Re number. This would permit arbitrary scaling from measurement data in the laboratory to the design and operation of real-world structures. It is probably true at a macroscopic scale for turbulent flows. Semi-empirical formulae for turbulent flow characteristics confirm the absence of Reynolds number. Durbin and Medic (2007, p. 237) affirmed no remarkable effect of Re once the flow reaches turbulent state. In fact,

it has been found that the Reynolds number has certain influence on the jet behaviour in specific flow conditions. Suresh et al. (2008) conducted physical experiments on plane jets and found the change of centreline velocity, turbulent intensity, and jet width, among other spectral characteristics, as a function of the axial distance in a certain flow condition e.g. $Re \sim 2000$. Since this range of Reynolds number is low relative to that modelled in this study, we shall adopt Gerris simulation results for cases with $Re = O(10^5)$ which are relevant to hydraulic engineering applications.

In this model of a single jet the necessary input parameters are brief and simple. In addition to physical parameters, d , U_o and Re , a viscosity term has been specified for each simulation case. Generally this term has been chosen to counterbalance the effect of numerical viscosity on the solution obtained. Care has been taken to choose the extent of domain large enough and to refine the cells at the inflow orifice so that $d = 6l_{\text{cell}}$. Grid refinement is nontrivial; not only it determines the computational effort but also the flow behaviour. So far, simulation has been performed on level-7 resolution. It does not resolve small eddies in the turbulent zone of the jet, but it can reproduce, to an acceptable degree of accuracy, the mean velocity and turbulent intensity—those are of interest to engineers. The author adopts this simulation approach and extends it to the case of three impinging jets described in Chapter 6.

Comparison of simulation results with different sources of published data may be necessary. There are other sources of data that could be used for the comparison purpose, e.g.

- Wagnanski and Fiedler (1969, *J. Fluid Mech.* **38**, pp. 577–612)
- Hussein, Capp, and George (1994, *J. Fluid Mech.* **258**, pp. 31–75).

These comparisons could be carried out as a follow up development of the present study.

Chapter 4

Modelling two impinging jets: bulk flow properties

4.1 Dynamics of impinging jets

Impinging jets implies momentum transfer between the component jets. In comparison to single jets, there has been less theoretical deliberation for multiple impinging jets. Experimental works can be found in applications of combustion (Gill and Nurick, 1976) where two jets of fuel and air intermix and react. Such a pair of jets has importance in aerospace engineering; an example is a bipropellant rocket engine, where a jet of fuel and another jet of oxidant intersects, causing intense mixing and effective burning of fuel. In all these applications, the ambient environment is air. The problem of our concern is, however, jets in ambient water. Nevertheless, from the literature what happens at the zone of impinging is worthwhile to investigate and compare to the author's study.

Before dealing with mutually impinging jets a simpler class of problems is worth mentioning—jet impingement on a plane, solid surface. When two jets, similar in size, impinge on each other then the symmetry plane in between should somehow act like a non-slip wall. Certain hydrodynamic properties in such interaction, namely the lateral spreading of momentum and shear layer thickness, will be useful for the study. In many idealised situation of impinging jets, the fluid sheet on the solid surface is assumed to be thin, with uniform velocity at each location on the plane, and the jet velocity is large so gravitational force can be neglected compared to the force exerted on the wall.

Figure 4.1: A normally-impinging jet, $\alpha = 90^\circ$ (adapted from O'Donovan, 2005)

4.1.1 Jet impingement on solid, smooth surface

This traditional topic is well reported in literature because of its wide application in the industry (notably heat transfer and dissipation, see e.g. the report by O'Donovan (2005)). The structure of a normally impinging jet (Figure 4.1) consists of three parts: (a) free jet which is similar to that studied in Chapters 2–3, (b) stagnation region (or zone of impact) where the pressure builds up, and (c) wall jet region where water flows in radial direction and exhibit the wall turbulence phenomenon.

Consider a simple case of a normally-impinging round jet. Assume that the velocities that enter, and exit (from the circumference of) the stagnation region are all equal V . The thickness of fluid layer, h_i , at the exit of stagnation region (which is a circle with radius R_i) is then given by:

$$h_i = d_s^2 / 8R_i. \quad (4.1)$$

Another simple case is an oblique but planar jet:

$$h_i = \frac{1}{2} d_s (1 \pm \cos \alpha), \quad (4.2)$$

where α is the angle between the jet and the solid plane ($\alpha = 90^\circ$ for normal jets, $\alpha < 90^\circ$ for oblique jets).

For oblique impingement the flow pattern is no longer two-dimensional. Only one symmetry plane exists (that containing the jet centreline and perpendicular to the solid plane). In this case a stagnation point is located theoretically at point S “upstream” relative to the main flow direction on the flat surface.

Beltaos (1976) suggested a semi-empirical formula which can be used to estimate

h_i in the case of oblique round jet impingement:

$$h_i = \frac{d_s}{\sqrt{\sin \alpha}} (1 \pm \cos \alpha). \quad (4.3)$$

Donaldson et al. (1971) investigated both normal and oblique round jet impingement (wall-jet $Re_w \equiv (v_r)_{\max} \delta / \nu = 1 \times 10^4 \rightarrow 2 \times 10^5$) on various surface shape; with $\alpha = 0 \rightarrow 60^\circ$. Beltaos (1976) experimented with jets of $Re = 3.4 \times 10^4 \rightarrow 1 \times 10^5$, impinging from various distances $H/d = 15.5 \rightarrow 47$ and various angles $\alpha = 20^\circ \rightarrow 60^\circ$. The result showed that the thickness of fluid layer on the plate increases linearly along the radial distance r from the stagnant point. Cooper et al. (1993) confirmed this linear increase of turbulent layer thickness against the radial distance through two experiments for flows with $Re = 23000$ and 70000 . The datasets for these experiments are available from ERCOFTAC repository.¹

4.1.2 Mutual impingement of two round jets

Two opposite round jets after impingement will form a radial jet, in which the jet thickness would be twice that of an impingement on a solid surface. Schwarz (1963) presented a mathematical treatment on radial jets of both laminar and turbulent types, also taking heat transfer into account. Witze and Dwyer (1976) derived empirical formulae to predict the behaviour of a radial jet resulted from impingement, and showed that no theoretical method based on momentum conservation can predict well the analytical solution exists for the case of radial jet resulted from impingement. Hall et al. (1981) conducted experiments on two perpendicular-aligned round jets ($\alpha = 45^\circ$) and showed that while both the resulted forward and backward flows exhibit Gaussian velocity profiles in the $x - z$ plane, the profile of backward flow is elliptical in the $x - y$ plane.

4.1.3 Shear flow layer formed by two jets

The impingement of two obliquely-aligned jets were also studied extensively in literature, since it has prime importance in aerospace engineering. For example, in liquid propellant rockets, two or more jets containing gas and fuel are arranged directly to impinge one another (see e.g. the paper by Cho et al. (2004)). For the case involving two jets, the shape of the resulting fluid sheet had been suggested to be similar to a leaf (Figure 4.2). Choo and Kang (2003) had confirmed this flow configuration

¹<http://bit.ly/UwWMB6>, retrieved 1-Dec-2012.

Figure 4.2: Conceptual diagram of coplanar impinging jets in air ambient with a liquid sheet (Ashgriz, 2011), and in water with a double shear layer (adaptation to the case of submerged jets for the present study)

for the cases of two jets meeting at angles 80° , 100° and 120° , and with $Re = 3350$. When the jet velocity increases, the liquid sheet breaks up and forms a spray. Bush and Hasha (2004) conducted experiments with $Re = 1$ to 2000 and a range of values for surface tension (characterized by the weber number, We), and found a general classification of the intersecting fluid sheet based on the flow velocity, geometry (d), and fluid properties (ν and surface tension, σ). Mathematical analyses had been carried out in the reference to determine the shape of fluid sheet in its stable condition. At higher speed this fluid sheet becomes wavy and ultimately breaks up. Some features of this spray such as droplet size and velocity are investigated by Kang and Poulikakos (1996); Choo and Kang (2003). The mechanics of this type of flow, which involves fluid surface tension, is different from the currently studied double shear layer flow,²

²The term "double shear layer" is used instead of plane jet, since the flow field being considered is three-dimensional.

which is submerged in water. However, certain features of such flow will be helpful conceptually. For example, as will be shown later, the leaf shape is very similar to the flat region with high flow velocity and intense fluctuation. The thickness of the fluid sheet is also analogous to that of a double shear layer. In this chapter, the time-averaged thickness is analysed; and the next chapter provides further details on wavy patterns in the shear layer.

4.2 Mathematical formulation

In this problem, due to symmetry, only half-space $y \geq 0$ is considered when deriving the formulae below. For convenient positioning on the symmetry plane, a coordinate system is setup where the origin is at the convergence point of the two jet centrelines. The position of any point on the horizontal plane is indicated by its radial coordinate r and angular coordinate ϕ with reference to the positive x direction.

4.2.1 The zone of impact

For the zone of impact, which is in proximity to the intersection between two jet centrelines, the flow can be approximated as inviscid, i.e. flow of an ideal fluid. Assuming that the jets are not extending long enough after leaving the orifices, so that the entrained mass flux is negligible, and energy loss is omitted, then for a control volume enclosing the zone of impact previously shown in Figure 4.1:

- Conservation of mass flux gives: $\int_{-\pi}^{\pi} h_i(\phi) \mathcal{U}_i(\phi) R_i(\phi) d\phi = \frac{1}{4} \pi d^2 U_o$
- Conservation of momentum flux gives: $\int_{-\pi}^{\pi} h_i(\phi) \mathcal{U}_i^2(\phi) R_i(\phi) d\phi = \frac{1}{4} \pi d^2 U_o^2$
- Conservation of energy flux gives: $\int_{-\pi}^{\pi} h_i(\phi) \cdot \frac{1}{2} \mathcal{U}_i^3(\phi) R_i(\phi) d\phi = \frac{1}{8} \pi d^2 U_o^3$

Here the zone of impact is assumed as an ellipse with centroid being the intersection between two jet centrelines. To evaluate the left-hand-side integrals in above formulae, the distribution of jet thickness, h_i , and radial velocity, $\mathcal{U}_i$, along the edge of zone of impact must be known. For a normally-impinging jet, these quantities are uniform along the edge, $h_i(\phi) = \text{const}$ and $\mathcal{U}_i = \text{const}$ for all $\phi : -180^\circ \leq \phi < 180^\circ$. For the general case of oblique impingement, their distribution cannot be theoretically derived. Nevertheless intuition might provide some hints.

First, it is realised that the flow field currently studied is somewhere between a radial jet and a plane jet. The first type of jet arises from normal impingement of a round jet, while the second from oblique impingement of another plane jet. (Here it is

Figure 4.3: To find a possible distribution of radial velocity along the edge of zone of impact for an oblique round jet

assumed that α is small so that the backflow is negligible.) Hence the unknown distributions should resemble the combined form of those distributions. Here, a difference between thickness and velocity in plane jets emerges. While the jet thickness is virtually unchanged relative to ϕ as long as the zone of impact is small; the radial velocity as projection of a vector, indeed depends on ϕ . Specifically, $\mathcal{U}_i(\phi) = \mathcal{U}_i(0) \cdot \cos \phi$ (see Figure 4.3). So only the distribution of $\mathcal{U}_i(\phi)$ is worth finding. A few properties of such distribution can be readily perceived :

- $\mathcal{U}_i$ is a continuous function of ϕ , symmetrical about the maximum at $\phi = 0$, i.e. the main direction of flow on the symmetry plane;
- $\mathcal{U}_i$ decreases smoothly when ϕ continually deviates from zero;
- there exists an inflexion point of the function on each half $\phi > 0$ and $\phi < 0$;
- there is a limit angle at which the radial flow is negligible, such that $\mathcal{U}_i \rightarrow 0$ for either $\phi > \phi_{\text{limit}}$ or $\phi < -\phi_{\text{limit}}$.

The mathematical expression for this distribution will be determined using output data from numerical simulations. By the above assumption that $h(\phi)$ is an even function, the integrals $\int_{-\pi}^{\pi} F(h(\phi))d\phi$ can be reduced to $2 \int_0^{\phi_{\text{limit}}} F(h(\phi))d\phi$.

4.2.2 Double shear layer flow

After leaving the zone of impact, the jet flow is confined to a “thin” shear layer with the following features:

- The thickness of this layer (in the y -direction) is small compared to its breadth (z -direction), while the breadth is in the same order with the length (x -direction).

Figure 4.4: (Left) Double shear layer divided into bands; (Right) A volume element

- The layer radiates from the impinging region with an aperture ϕ : $-\phi_{\text{limit}} < \phi < \phi_{\text{limit}}$.
- Whether the thickness of the layer increase or decrease streamwise depends not only on the aperture but also on the entrainment rate of ambient fluid into the double shear layer;
- The flow attains maximum velocity on the symmetry plane rather than either above or below it.
- The layer is different from two wall jets combined: there is no friction along the symmetry plane.

By noticing that the streamlines must follow the radial direction (i.e. no circumferential flow), the shear flow region can be divided into a number of narrow “bands”; in each band the flow velocity varies along the r -direction and unchanged along the θ -direction.

Conventionally, the flow along a band can be analysed using the boundary layer theory. However, it is possible to make use of existing results in literature regarding the velocity profile across a plane/radial jet. In that way, an average velocity $\mathcal{U}$ and a width h can be specified to characterise the local flow condition on the double shear layer. The steady-state flow equations written for a volume element within the layer shown in Figure 4.4 are as following:

Continuity equation:

$$\frac{\partial(\mathcal{U}h)}{\partial \mathbf{r}} = q_E; \quad (4.4)$$

Momentum equation:

$$\mathcal{U} \frac{\partial \mathcal{U}}{\partial \mathbf{r}} = \frac{1}{\rho} \frac{\partial \tau_{ry}}{\partial y}. \quad (4.5)$$

Here the velocity distribution perpendicular to the double layer in each single transect was approximated by a profile typically found in a plane jet, namely $\tilde{U} = U_{CL} \exp(-y^2/b_{pl}^2)$, where the half-width of a plane jet is related to that of a round jet via an almost constant ratio $b_{pl}/b \sim 1.053$. Therefore, to choose appropriate values of U and h for a transect at a radial distance r , the mass flux and momentum flux must remain unchanged after substituting the profile $\tilde{U}(y)$ for characteristics U and h :

- Mass flux: $Uh = U_{CL} \int_0^\infty \exp(-y^2/b_{pl}^2) dy$;
- Momentum flux: $U^2h = U_{CL}^2 \int_0^\infty \exp(-2y^2/b_{pl}^2) dy$.

Given the integral of a Gaussian function $\int_0^\infty \exp(-x^2/a^2) dx = \frac{a}{2}\sqrt{\pi}$, also $\int_0^\infty \exp(-2x^2/a^2) dx = \frac{a}{2}\sqrt{\frac{\pi}{2}}$, it is possible to infer:

$$U = \frac{1}{\sqrt{2}}U_{CL} \quad \text{and} \quad h = \sqrt{\frac{\pi}{2}}b_{pl}. \quad (4.6)$$

Assuming that the flow in each band is similar to a plane jet flow (which will be later verified against simulation results), the terms on right-hand side of Eqs. (4.4) and (4.5) can be approximated as:

$$\begin{aligned} q_E &= \beta U_{pl}^{CL} \quad (\beta \text{ is the entrainment coefficient}) \\ &= \left(\frac{\sqrt{\pi}}{4} \frac{db_{pl}}{d\tau} \right) (U\sqrt{2}) \quad \text{then assuming } db_{pl}/d\tau = 0.12 \text{ (Lee and Chu, 2003, p.32)} \\ &= 0.0752U. \end{aligned} \quad (4.7)$$

$$\begin{aligned} \frac{1}{\rho} \frac{d\tau}{d\tau} &= \nu_T \frac{\partial^2 \tilde{U}}{\partial y^2} \\ &= 0.008 \times 2\delta U_{pl}^{CL} \frac{\partial^2 \tilde{U}}{\partial y^2} \quad \nu_T \text{ calculated as in (Davidson, 2004, p.152),} \\ &\quad \text{and for } \delta = b_{10\%} = 1.5175b_{pl}, b_{pl} = 0.12r, U_{pl}^{CL} = U\sqrt{2} \\ &= 4.12 \times 10^{-3} r U \frac{\partial^2 \tilde{U}}{\partial y^2}. \end{aligned} \quad (4.8)$$

Since $\tilde{U}(y) = U\sqrt{2} \exp\left(-\frac{y^2}{b_{pl}^2}\right)$, the second derivative $\frac{\partial^2 \tilde{U}}{\partial y^2} = \frac{2\tilde{U}(y)}{b_{pl}^2} \left(\frac{2y^2}{b_{pl}^2} - 1\right)$, whose magnitude represents the curvature of the radial velocity profile, attains maximum at $y = 0$ and tends to zero at $y \rightarrow \infty$. By choosing a representative value [which is also the maximum] for the second derivative at $y = 0$, it is possible to obtain

$$\partial^2 \tilde{U} / \partial y^2 = -\sqrt{2}\pi U / h^2 \quad (4.9)$$

Substitution of Eqs. (4.7)–(4.9) into Eqs. (4.4) and (4.5) yields the following semi-empirical equations for the flow in the double shear layer:

$$\frac{\partial(\mathcal{U}h)}{\partial \tau} = \mathcal{C}_1 \mathcal{U}, \quad (4.10)$$

$$\mathcal{U} \frac{\partial \mathcal{U}}{\partial \tau} = \mathcal{C}_2 \tau \frac{\mathcal{U}^2}{h^2}. \quad (4.11)$$

Equations (4.10) and (4.11) are nonlinear differential equations which can be solved numerically for h and $\mathcal{U}$, as is addressed in Section 4.4. The empirical constants $\mathcal{C}_1 = 0.0752$ and $\mathcal{C}_2 = -1.83 \times 10^{-2} (= -4.12 \times 10^{-3} \times \sqrt{2}\pi)$ are nondimensional and originate from the approximation process of the double shear layer with a plane jet. They are not dependent on the boundary conditions such as U_o and α .

4.3 Flow characteristics found based on simulation

To solve Eqs. (4.10) and (4.11), the boundary condition (h_i and $\mathcal{U}_i$) surrounding the zone of impact must be given. This information can be extracted from the output data of simulation. Only then can we process to the solution (Section 4.4).

4.3.1 Simulation set-up

The simulation domain to be considered (hereafter referred to as “full domain”) comprises two cubic boxes. The two jets are with elliptical inflow cross-sections. The centroids of these ellipses are equidistant ($s = 3d$) to the horizontal mid-plane $y = 0$. The ellipses are purposely elongated in y -direction: $\frac{(y \pm s)^2}{\cos^2 \alpha} + z^2 = 1$, to provide a round velocity profile when being projected on a plane perpendicular to the flow. Jet 1 directs upward and Jet 2 downward, i.e. $V_1 = U_o \sin \alpha$ and $V_2 = -U_o \sin \alpha$. The specification $U_{in} = U_o \cos \alpha$ completes the inflow boundary condition.

The computational grids were refined at level 9 for the oblique jet core and the impinging region, $(x - x_o)/s < 8/3$ but only with $U > 0.1U_o$. A level-8 refinement was applied to the region just above the bottom plane and near the impingement point, specifically $y/s < 5/3$ and $-8/3 < z/s < 8/3$; this region extends up to $(x - x_o)/s = 35/3$. Far from the zone of impact, where the shear flow structure has been smeared out, and grid level 7 is used for the zone enclosed in the cone shape $r < 0.116x$.

Unlike single jets having centrelines aligned with x -axis, in the current case which involves impingement, the jet centrelines are aligned diagonal relating to grid lines. To verify that the effective artificial viscosity has little influence to the flow, comparisons

Figure 4.5: To determine the extent of the zone of impact based on the velocity field

Figure 4.6: To determine contour of the zone of impact by tracking along the direction perpendicular to velocity vector field (only half of the zone with $\phi > 0$ is shown)

have been made using a jet at $Re = 5360$ emitting at different angles α relative to grid-lines. Results (Appendix A.2.3) show that the cross-sections of jet core and velocity contour lines do not deviate much from those of a jet aligned parallel to the gridlines.

4.3.2 Extent of the zone of impact

Based on the available simulation output data, the extent of the zone of impact can be determined. Since the backflow is negligible, a threshold in velocity magnitude, specifically $0.05|U|$, helps identify the backward limit of a zone of impact. The limit in forward direction ($\phi = 0$) is set at the vertical line where flow velocity vectors have almost turned parallel to the symmetry plane ($|V/U| < 0.05$, see Figure 4.5).

To estimate the extent of the zone of impact along an arbitrary radial direction ($\phi \neq 0$), it is possible to start from the forward limit point, and draw a contour line orthogonal to velocity vectors on the symmetry plane (Figure 4.6). This contour is the boundary separating the zone of impact (upstream) and the zone occupied by the double shear layer (downstream).

Figure 4.7: Velocity profile extracted from five sample points on a vertical line at a point on the upstream boundary of double shear layer, from which the characteristic velocity $\mathcal{U}_i$ and thickness h_i are derived

4.3.3 Layer thickness and flow velocity at the boundary of double shear layer

In order to obtain the flow condition at the boundary, data must be extracted from the simulation output. Each point on the contour line is also a data point constituting the boundary condition. However, the actual data ($\mathcal{U}$ and h) of this data point must be calculated based on a number of sample points on the vertical line through this data point.

From the simulation output data extracted for each such sample point, the radial velocity component are calculated $\mathcal{U}(\mathbf{r}, \phi) = [\mathcal{U}(x - x_{\text{centroid}}) + Wz] / \sqrt{(x - x_{\text{centroid}})^2 + z^2}$. The velocity vector is parallel to the horizontal plane so y does not present in the preceding formula.

The thickness h of the layer and $\mathcal{U}$, the transect averaged velocity can then be determined from discrete sample points on the velocity profile at limit of the zone of impact. While $\mathcal{U}$ can be readily calculated from the centreline velocity, $\mathcal{U}_{CL} = \mathcal{U}|_{\phi=0}$ (Eq. 4.6), the value of h should be found through b_{pl} (Eq. 4.6) by interpolation from these samples. Figure 4.7 shows $\mathcal{U}$, h , and b at the forward limit ($\phi = 0$) of the zone of impact. These values were derived from the velocity profile, which was itself interpolated from sample points. Sampling results from simulation also show that the velocity profile becomes flatter when moving along the contour while increasing ϕ .

By examining the distribution of $\mathcal{U}_i$ and h_i (upper part of Figure 4.8) it can be seen that the schematised profile of $\mathcal{U}_i$ applies well here. A maximum value of ~ 0.3 for $\mathcal{U}_i$ at the central axis and $\phi_{\text{limit}} \sim 0.4\pi$ had been identified. The thickness of flow layer along the contour of the zone of impact remains almost constant, which is in

Figure 4.8: Distribution of $\mathcal{U}$ and h along the contour of the zone of impact for Case $\alpha = 15^\circ$

accordance to the previously-mentioned assumption, except for the edge of this layer where also the unit flow rate q is very small. The unit momentum flux is almost a quadratic function of $\mathcal{U}_i$ and is negligible for $\phi \gtrsim \frac{1}{4}\pi$.

Based on the results above, the velocity distribution can be represented with a Gaussian curve

$$\mathcal{U}_i = 0.3177U_o \exp(-\phi^2/0.062474), \quad (4.12)$$

where $(0.3177U_o)$ is the peak value of $\mathcal{U}_i$ in Figure 4.8 and 0.062474 is found by curve-fitting with the Gnuplot³ computer program. In determining similar distributions for the radius and layer thickness, the conservation of mass, momentum, and energy must be taken into account. There is no exact solution for the problem regarding such constraint; and an approximation has been adopted (See Appendix). Thus for a single impinging jet with $\alpha = 15^\circ$, the flow condition along the edge of zone of impact can be given in nondimensional form as follow:

$$\begin{aligned} R_i^* &= 6.6488 + 0.3870\phi \\ h_i^* &= 0.710 + 0.1505\phi^2 \\ \mathcal{U}_i^* &= 0.3177 \exp(-\phi^2/0.06274). \end{aligned}$$

The above process is then applied to Cases $\alpha = 30^\circ$ and 45° , and approximated

³Gnuplot home page, <http://www.gnuplot.info/>. Retrieved 29-Jul-2013.

functions are derived for R_i , h_i , and $\mathcal{U}_i$, which are listed in Table 4.1. An analysis covering the variation of these flow characteristics on a continuous range of α is out of scope of the current study. However, some qualitative remarks on the results can be drawn.

Table 4.1: Functional forms of R_i^* , h_i^* and $\mathcal{U}_i^*$ for various α

α	ϕ_{limit}	R_i^*	h_i^*	$\mathcal{U}_i^*$
15°	0.4π	$6.6488 + 0.3870\phi$	$0.710 + 0.1505\phi^2$	$0.3177 \exp(-\phi^2/0.06274)$
30°	0.33π	$2.5 + 0.1740\phi^2$	$0.1164 + 0.0692\phi^2$	$0.6532 \exp(-\phi^2/0.0951)$
45°	0.43π	$3.28 + 0.126\phi^2$	$0.188 + 0.092\phi^2$	$0.50 \exp(-\phi^2/0.2710)$

First, being derived from tracking in velocity vector field on the symmetry plane $y = 0$, the radius R_i is almost constant for all ϕ : $-\phi_{\text{limit}} \leq \phi \leq \phi_{\text{limit}}$. Second, the range of ϕ widens for larger α , though it is not likely that ϕ_{limit} exceeds 90° for $\alpha \leq 45^\circ$. The thickness of a double shear layer at the edge of the zone of impact can usually be approximated as a quadratic function of ϕ , symmetrical and with a minimum at $\phi = 0$. The radial velocity distribution exhibits a decreasing trend when ϕ increases, and can be approximated as a Gaussian distribution. The flow is fastest along the symmetry axis, with a velocity of $\mathcal{U}_{i,0}$. This velocity increases when α rises from 15° to 30° . The reason is that a larger impinging angle causes the double shear layer to spread wider and becomes thinner, which translates to higher velocity. However, when α increases from 30° to 45° this maximum velocity decreases. A possible explanation is as follow. After transition from “oblique” to “parallel” to the symmetry plane, the flow attains high velocity in a thin layer close to the symmetry plane and exhibits supercritical flow condition. A hydraulic jump might occur at the edge of the zone of impact, for cases with large α . The presence of “backward” flow—in this case, radially inward flow—alters the velocity distribution. Specifically, in Figure 4.9, the velocity profiles corresponding to large ϕ , i.e. $\phi \rightarrow \phi_{\text{limit}}$, shows inward flow for $y_* > 0.4$. Such a flow distribution is similar to that observed from an oscillating bed (see e.g. Nielsen, 1992, p. 7), which appears when a flat bed suddenly shifts horizontally in an otherwise still waterbody.

The above remark on the shape of velocity profiles in cases of large α (e.g. 30° , as in Figure 4.9) suggests that the model should apply only for small values of α , i.e. the component jets have slight momentum interaction. Hence, for the subsequent sections in this chapter, the double shear layer is intended for jet impingement at small angles, $\alpha \lesssim 15^\circ$.

Figure 4.9: Velocity profiles along the contour of the zone of impact

Figure 4.10: Streamlines of time-averaged flow velocity on the symmetry plane; approximated zone of impact shown as a thick contour line

4.3.4 The double shear layer in a plan view

Now that the boundary conditions are obtained, before proceeding to the solution, a check for the flow pattern on the horizontal symmetry plane would be useful. From the streamlines of time-averaged velocity shown on Figure 4.10, it can be seen that the flow tends to bend and move along the x direction at some distance downstream of the zone of impact. This is a supportive evidence for the assumption that double shear layers are somewhat similar to plane jets. Also, this implies justification in considering the “double shear layer bands”, running parallel and having constant widths (see Section 4.2.2).

4.4 Solution for the double shear layer equations

In the previous section, by noticing that the flow field under analysis is intermediate between a radial jet and a plane jet, the distribution $\mathcal{U}_i \sim \phi$ is determined, but only along the contour of the zone of impact. The entire (two-dimensional) flow field (i.e. $\mathbf{r} \geq R_i$) is now analysed. This requires solving Eqs. (4.10) and (4.11), which can be rewritten as:

$$\mathcal{U} \frac{\partial h}{\partial \mathbf{r}} + h \frac{\partial \mathcal{U}}{\partial \mathbf{r}} = C_1 \mathcal{U} \quad (4.13)$$

$$\frac{\partial \mathcal{U}}{\partial \mathbf{r}} = C_2 \frac{\mathbf{r} \mathcal{U}}{h^2}. \quad (4.14)$$

The nonlinear nature of these equations requires numerical solution. Discretisation is performed using a forward difference scheme:

$$\mathcal{U}_{i,j} \frac{h_{i+1,j} - h_{i,j}}{\Delta \mathbf{r}} + h_{i,j} \frac{\mathcal{U}_{i+1,j} - \mathcal{U}_{i,j}}{\Delta \mathbf{r}} = C_1 \mathcal{U}_{i,j} \quad (4.15)$$

$$\frac{\mathcal{U}_{i+1,j} - \mathcal{U}_{i,j}}{\Delta \mathbf{r}} = C_2 \frac{\mathbf{r}_{i,j} \mathcal{U}_{i,j}}{h_{i,j}^2}. \quad (4.16)$$

where $\Delta \mathbf{r} = \mathbf{r}_{i+1,j} - \mathbf{r}_{i,j}$. From Eqs. (4.16) and (4.15), the unknowns $\mathcal{U}_{i+1,j}$ and $h_{i+1,j}$ are calculated as

$$\begin{aligned} \mathcal{U}_{i+1,j} &= \mathcal{U}_{i,j} + C_2 \frac{\mathbf{r}_{i,j} \mathcal{U}_{i,j}}{h_{i,j}^2} \Delta \mathbf{r} \\ h_{i+1,j} &= h_{i,j} \left(2 - \frac{\mathcal{U}_{i+1,j}}{\mathcal{U}_{i,j}} \right) + C_1 \Delta \mathbf{r} \end{aligned}$$

The boundary conditions, corresponding to $i = 0$, are given in Eqs. (4.12) and (D.5). The solution is implemented using computer code shown in Appendix D.

4.5 Modelling results compared to simulation output

To verify the simplified model previously established, it is preferable to compare the results obtained with real measured data. However these type of experiment setting is rarely found in literature. For example, Hall et al. (1981) conducted experiments on two impinging jets at $\alpha = 45^\circ$, where axial velocity distribution at certain cross-section were measured. This angle is considered large for the model to be valid. Furthermore, all those probes are located upstream of the jet intersection point; hence the results are not comparable to that reproduced by this simple model. For that reason, the results

from the double shear layer model is compared to the numerical output from computer simulation.

4.5.1 A note on parameter selection

In order to achieve agreement between the simulated and measured data, certain parameters relating to $\mathcal{C}_1$ and $\mathcal{C}_2$ must be adjusted. Several factors determine the values of these two coefficients, as are considered below.

Firstly, noted that in Eq. (4.9), the coefficient is derived by adopting the location of centreline, $y = 0$, for estimating the second derivative $\partial^2 \tilde{U} / \partial y^2$ (the diffusive term). This term should be negative to have a physical meaning, but not necessary be the local maximum attained at $y = 0$. In other words, the requirement is $-\sqrt{2\pi} \frac{U}{h^2} \leq \frac{\partial^2 \tilde{U}}{\partial y^2} < 0$. This leads to $-1.83 \times 10^{-2} \leq \mathcal{C}_2 < 0$. Secondly, in deriving Eqs. (4.7) and (4.8) both the spreading and entrainment hypotheses are based on the case of a plane jet. A modification to the present case would somewhat include the behaviour of a radial jet. Specifically, the spreading rate, db/dx , for a radial jet is 0.106 whereas for a plane jet, a spreading rate of 0.12 was adopted. Furthermore, the entrainment rate for the radial jet is very different from that of a plane jet. Using a round jet as an intermediate case for comparison, Guo and Sharp (1996) had shown that, theoretically, $\frac{\beta_{\text{radial}}}{\beta_{\text{round}}} = \frac{18.77}{\sqrt{x_*}}$. Alternatively, one could make use of the well-established boundary layer theory to infer $\frac{\beta_{\text{pl}}}{\beta_{\text{round}}} = \frac{\sqrt{\pi}}{2}$. Hence $\frac{\beta_{\text{radial}}}{\beta_{\text{pl}}} = \frac{37.54}{\sqrt{\pi x_*}} \sim 4 \rightarrow 7$ for the range of x_* here. These crude approximations show that, if a radial jet has been used instead of a plane jet, the coefficients would be $0.30 \leq \mathcal{C}_1 \leq 0.53$ and $-1.83 \times 10^{-2} \leq \mathcal{C}_2 < 0$.

Since the double shear layer being considered is an intermediate case between a plane jet and a radial jet, it is reasonable to choose $0.0752 \leq \mathcal{C}_1 \leq 0.53$, although a smaller value (more plane jet-like) of $\mathcal{C}_1$ would be more preferable. By fine-tuning these two coefficients, the agreement between the modelled and simulated velocity and thickness of the double shear layer flow can be achieved.

4.5.2 Comparison with a simulation of $\alpha = 15^\circ$

In this simulation ($\text{Re} = 5360$) the domain is refined at level 9 from the two jet orifices up to near the point of convergence ($x_{\text{L9}} \leq x_{\text{conv}} - 4d$). The part of double shear layer to be analysed is contained within the level-8 refinement zone ($x_{\text{cond}} - 4d < x_{\text{L8}} \leq x_{\text{conv}} + 48d$). Refinement at level 7 takes place in the subsequent zone in a truncated cone shape with a slope of $dr/dx = 0.1414$.

Figure 4.11: Comparison between full-domain simulated (■) and mathematically formulated (—) radial velocity and layer thicknesses. Upper row: along the central band ($\phi = 0$) and lower row: along the band starting at position $\phi = 30^\circ$ on the boundary.

The flow velocities obtained from simulation along the centre line, or “band”, agree with those calculated based on the mathematical model (Figure 4.11). The sampled thickness data is generally more irregular and therefore has lower goodness-of-fit. For the band starting from $\phi = 30^\circ$, there is a lead-in section ($r_* \sim 4$) where the flow changes direction from “expanding” to “parallel” (see Figure 4.10), and the sampled velocity does not match the model output. Although the thickness is underpredicted, it can be seen that a more accurate specification of h_* at the boundary (i.e. $r_* = 0$) would improve the prediction capability of the model. In the present case, C_1 is left as its predefined values (0.0752) and C_2 is set as -1.6×10^{-2} .

4.5.3 Simulation based on a symmetry plane

At this point, a natural question would be, “Why not simulate on half the domain, as the problem is symmetrical?” To answer this question, a similar simulation to that performed in Section 4.3.1 is implemented, except that the bottom panel is set to free-slip type, an alternative to specifying a symmetry plane. The inflow boundary condition

is prescribed as $U_{\text{in}} = U_o \cos \alpha$ and $V_{\text{in}} = -U_o \sin \alpha$.

Figure 4.12: Comparison between half-domain simulated (■) and modelled (—) representative velocities and layer thicknesses, along the central band ($\phi = 0$)

It can be seen, from Figure 4.12, that the flow characteristics obtained from this half-domain simulation do not quite match the values from the calculations following the scheme presented in Section 4.4. The simulated flow tends to be fast and confined in a thin layer close to the symmetry plane. This observation suggests that the use of a symmetry plane as in this case will be less likely to reproduce realistic mutual jet impact which is obtained from a full-domain simulation.

4.6 Summary

The time-averaged flow field resulted from two converging round jets are considered in this chapter. By intuition, it can be inferred that the flow structure should be a double shear layer with mirror symmetry about the vertical plane $z = 0$.

A simplified mathematical model for the double shear layer has been suggested; the thickness of such a layer increases radially from the zone of impact. Within this layer, the flow is predominantly along the radial direction, r . Two quantities characterising the flow in each cross-section are the thickness h and the radial velocity U . In deriving the continuity and momentum equations, the phenomena of entrainment and layer spreading are approximated as in those of a plane jet.

From the time-averaged velocity profiles of the shear layer it is seen that the model is suitable for two jets converging at small angle α so that the momentum impact is small compared to the momentum transport along the dominant flow direction, x . An implementation using a computer program and solved for the case $\alpha = 15^\circ$ has been performed. The solution was compared with simulation results. By using the computational scheme defined in Section 4.4, it is shown that one parameter needs to be

adjusted in order to obtain good prediction for the velocity, and reasonable prediction for the thickness of a double shear layer.

Using a symmetry plane is apparently a reasonable way to model the two-jet impingement process. However it turns out that this simplification omits important features of the time-averaged flow field. Comparing to the simulation of two impinging jets on a full domain, the model yields a thinner shear layer, presumably due to the lack of capability in describing vertical motion of water particles.

The double shear layer model is simple yet useful as it provides a conceptual tool to study the flow field formed by two jets. It is used in Chapter 5 to compare a group of three converging jets to a two-jet system. Such pair of system is equivalent if they produce similar double shear layers. This might lead to possible replacements using three jets instead of two jets or vice versa.

Chapter 5

Equivalent jet systems and time dependent behaviour of jets

5.1 An equivalent form for a two-jet system

5.1.1 Motivation

Section 4.5.3 shows that the introduction of a symmetry plane into the computational domain leads to less realistic simulation results. The output flow field tends to be confined in a layer, thinner than half the double shear layer thickness in the case when the symmetry plane is removed. To overcome the adverse effect of the symmetry plane in suppressing the vertical motion of water particles near it, an adjustment is needed to promote the development of the double shear layer developed at the downstream-side of the zone of impact. This can be done by, e.g.:

- Retarding the flow along the symmetry plane by modify the boundary condition on the symmetry plane by a certain way, e.g. imposing a moving boundary, specifying roughness of the plane to enhance turbulence;
- Adding a jet to the zone of impingement to increase the thickness of the existing double shear layer formed by two jets. This added jet flows along the symmetry plane.

The first method requires an inspection of the flow field on the plane $y = 0$ in the full domain simulation. With this information one can prescribe appropriate boundary conditions for the symmetry plane in a half-domain simulation. The second method might be more difficult since it involves more interaction between shear layers. However it provides an analogy between a two-jet and a three-jet configuration. In the scope of this study, the second approach will be examined in subsequent sections.

5.1.2 Formulation

The equivalence between a two-jet and a three-jet configuration is based on the decomposition of momentum vectors (Figure 5.1). Suppose that two jets of equal strength (momentum $\rho\hat{Q}_o$) make the same angle $\hat{\alpha}$ with the symmetry plane. Each of these two jets can then be decomposed into a component along the main centreline and another component making an angle α to this centreline. Hence, if the full domain is considered then a three-jet configuration will be obtained.

Figure 5.1: Equivalence in terms of momentum between a two-jet and a three-jet configuration

The relationship between α , Q_o and $\hat{\alpha}$, $\hat{Q}_o$ can be determined as follows. First, to satisfy the requirement that three jets are of equal strength, the component of momentum along the centreline on each half of the three-jet configuration must be $\frac{1}{2}\rho Q_o$. The presence of the centreline jet therefore decreases the initial outflow rates by one-third, i.e. $Q_o = \frac{2}{3}\hat{Q}_o$. If the outflow velocities are assumed unchanged then the jet diameter of this three-jet configuration must be reduced accordingly, $d = \sqrt{\frac{2}{3}}\hat{d}$.

The conservation of momentum leads to an expression relating α and $\hat{\alpha}$. Indeed, consider the linear momentum equation projected on the x -axis:

$$\begin{aligned} 2\rho\hat{Q}_o \cos \hat{\alpha} &= \rho Q_o + 2\rho Q_o \cos \alpha \\ 3 \cos \hat{\alpha} &= 1 + 2 \cos \alpha. \end{aligned} \quad (5.1)$$

Despite the apparent nonlinear form of Eq. (5.1), the relationship between α and $\hat{\alpha}$ is practically linear, $\alpha = 1.243\hat{\alpha} - 0.226$, with a Pearson coefficient $R^2 = 0.999$.

To validate the method mentioned above, a comparison of the double shear layers produced by two cases is presented in Section 5.1.3. (Hereafter the 3-jet configuration is called “equivalent” to the 2-jet system with angle $\hat{\alpha}$.) The comparison is now mainly based on the distribution of U instead of the characteristic quantities $\mathcal{U}$ and h previously defined in Chapter 4. Also, only part of the double shear layer near the zone of impact is considered. The cross-sections of flow structure far from this zone in a 3-jet system are investigated in Chapter 6.

Figure 5.2: Top view comparison of U_* between two-jet ($\hat{\alpha} = 15^\circ$) and three-jet ($\alpha = 18.4^\circ$) systems

5.1.3 Comparative simulation of equivalent jet systems

Simulation is performed with a two-jet system of $\hat{\alpha} = 15^\circ$ and a three-jet system of $\alpha = 18.40^\circ$, satisfying Eq. (5.1). In simulating the three-jet system, the geometrical set-up is straightforward, and readily accomplished by combining those of a single-jet, and a two-jet case of equal angle. The refinement process is as described in Section 4.5.2 for the flow at $Re = 5360$.

The comparative results from the top view (Figure 5.2) show that the extents of double shear layers (marked by the contours $U_* \approx 0$) are roughly similar. However the distributions of U within two double shear layer differs: the three-jet configuration results in faster flow in most part of its double shear layer (10% higher than corresponding flow formed by the two-jet system).

The front view of longitudinal velocity contours (Figure 5.3) provides a hint about thickness of the two double shear layers. The three-jet systems again yields faster flow along the x -axis. It gives higher U_{CL} ; this is likely due to the momentum exerted by the straight jet. The two-jet system apparently produces a thicker flow layer at the constricted transect, but the presence of a strong shear force in the three-jet system allows the double shear layer to expand downstream.

The three-jet equivalent system has shown its effectiveness in producing a thick double shear layer. However, it is not much helpful for the case of modelling with a symmetry plane. Figure 5.4, the three-jet system modelled in a half-domain produce a thinner shear layer even when compared with the two-jet system.

The result from simulation shows that the equivalence of three-jet and two-jet systems is generally accepted in practice (with a difference of 10% in flow velocity). The flow formed by simulation with a symmetry plane, however, does not compare well with a that of a full-domain simulation, as is easily noticeable through the thickness of the flow layer along the central axis. Therefore it is not appropriate to approximate

Figure 5.3: Front view comparison of U_* between two-jet ($\hat{\alpha} = 15^\circ$) and three-jet ($\alpha = 18.4^\circ$) systems

Figure 5.4: Front view comparison in velocity distribution for the case of equivalent systems: two-jet in full domain and three-jet in half-domain (Contour interval $\Delta U_* = 0.05$)

the flow produced by 2 jets in a full domain by an equivalent three-jet system in a half-domain.

The comparison results in Figures 5.2, 5.3, 5.4 are summarized and presented in Figure 5.4b. It can be seen that the radial extent of velocity contours on the plan view show reasonable agreement between computed and measured values. The thicknesses of the shear layers are similar between the 2-jet and 3-jet systems. On the other hand, the average velocity of the middle layer (confined between the planes $z = \pm 2d$) shows significant difference between the two systems which is to be expected as a result of the presence of the middle jet. The average flow velocity is significantly higher in the 3-jet system.

5.1.4 Result from NTU laboratory experiment

Several experimental studies on momentum-driven submerged jet impact were conducted at NTU by Zheng (2013). In this section the original dataset obtained from PIV system is presented and compared with the numerical results.

Zheng used a number of short copper pipe nozzles with equal diameter $d = 0.8$ cm. The nozzle outlet might exhibit a small degree of contraction, due to mechanical cut-

Figure 5.5: Relationship between various flow quantities in a 2-jet and its equivalent 3-jet system. Length values are multiples of d , and velocity values are multiples of U_o .

ting process. Zheng performed experiments for one, two and three jets. The 2-D flow field had been captured by a CCD camera and processed by DaVis software (LaVision Flow Master PIV Product-Manual, 2007).

After testing with various scenarios, Zheng (2013) remarked that for achieving reliable time-averaged flow field, the number of data images [samples] must be sufficient (i.e. 1050 images). With a recording frequency of 15 Hz, this implied a time window of 70 s.

One difficulty in comparing between simulation and physical experiment lies in the difference in geometrical setting of the problem. Numerical simulations usually contain simple geometric features thus physical experiments hardly reproduce the ideal flow condition. For example, a standard-type contraction nozzle is required to produce an ideal uniform inflow jet velocity. On the other hand, the laboratory experiment conducted by Zheng (2013) involved pipe nozzles so the turbulent intensity at the inflow cross-section might be much higher than that simulated numerically. Errors may also arise from the measuring process involved, e.g. the flow velocity might be estimated from the flow rate Q and area of cross-section. Therefore, the candidate only chooses a subset of Zheng (2013)'s dataset and performs only a modest comparison.

For two-jet experiments, Zheng aligned the nozzles at an angle $\alpha = 30^\circ$ and with spacing $s_* = 3$ (hence $x_{\text{conv}}^* = 5.2$). The flow condition ranges from $\text{Re} = 4240$ to 8000. Available lab measurement data contains mean and turbulent flow characteristics ($V, W, v_{\text{rms}}, w_{\text{rms}}$) at seven equidistant cross-sections $x_* = 9, 13, \dots, 33$. The sample spacing is $\Delta = 2.72 \text{ mm} \approx d/3$.

Figure 5.6: Time-averaged velocity on cross-section $(x - x_{\text{conv}})_* = 3.8$. Above: NTU experiment. Below: Simulation output.

When comparing flow field formed by two jets, there are large differences between experimental and simulation results (Figure 5.6). First, there is a difference in number of vortices. Four vortices, almost symmetrical, emerged in Zheng's work, while apparently there were eight vortices in Gerris simulation. The flow directions are also different. In Zheng's experiment, the flow pattern shows a clear tendency of changing direction: from centreline-oriented to laterally-directed. A flow feature similar to a double shear layer can readily be discerned from the vector plot.

Figure 5.7: Cross-sectional vorticity plot of vector field

Since the only available NTU laboratory data samples were collected on the y - z plane (i.e. side view), the possibility to compare the flow fields in experiment and simulation is somewhat limited. Due to the complex structure of a vortex core, especially that formed by different jets, the comparison of both vector fields is likely to show dissimilarities. The author believes that the NTU dataset reflects more accurately the time-averaged flow field, because the vector plot shows clearly the flow structure, which is symmetrical and with four vortices. In the graphical plot of simulation result, on the other hand, only two right-handed vortices are clearly shown. A flow current directs from the centre to the right side where it meets the entrained flow from the opposite direction.

To facilitate intercomparison, vorticity can be used as a criterion. Figure 5.7 shows the corresponding distribution of vorticity in the cross-section in simulation. The magnitude of vorticity is higher in simulation and the extent of high-vorticity region is smaller compared to that of Zheng's experiment. The similar feature, however, between two results is that alternating regions of low and high vorticity presents.

5.2 Turbulence modelling

So far, the simulations have been performed using a constant value for turbulent viscosity for each simulation. This techniques has proved to be applicable in simulation

of a single jet. However the validity of a uniform ν_T for the entire domain in the case of multiple-jet impact is questionable. As there have been no analytical solution to mutually-impact jets and very few laboratory experiment data found in literature, at this point it is difficult to determine whether an inclusion of turbulence can improve the prediction of multiple-jet behaviour. The purpose of this study is to make some preliminary comparison of the jet flow behaviour between two cases: (i) using a constant value for ν_T and (ii) using a simple method to estimate the time- and space- varying values of the Reynolds stresses. It is not intended to study in depth the mechanism of turbulent modelling in the present work.

5.2.1 Implementation

In the current version of Gerris solver, turbulence is described through the motion of infinitesimal fluid parcels, which is the DNS approach. Tomar et al. (2010) used such a method to study the atomisation process of liquid with high surface tension. A turbulence model is currently not implemented (informal communication),¹ and LES is currently considered.² Popinet et al. (2004) conducted a preliminary study on the turbulent wind around a vessel following the LES approach but did not include subgrid-scale viscosity, with an explanation (originally suggested by Boris et al., 1992) that the numerical viscosity generated would well compensate for the eddy viscosity being absent from the model.

In this section, a Smagorinsky–Lilly turbulence model is tested, see Eqs. (2.25) and (2.26). Instead of the turbulent viscosity ν_T , it is necessary to specify the momentum source terms, $\partial\tau_{ij}^{\text{SGS}}/\partial x_k$ (where i, j, k are spatial indices). To do this, first the spatial derivatives, then the shear strain rates S_{ij} , and Smagorinsky numerical viscosity are computed. The implementation below for Gerris input data file is straightforward, as this computer program supports users in defining field variables which are spatial derivatives of primitive variables (u, v, w, p , and the tracer concentration c). (For a detailed input file, refer to Appendix A.2.)

- Input: data of flow field $\mathbf{u} = (u, v, w)$
- Compute gradient of velocity components, e.g. $\partial u/\partial x$, which can be specified in Gerris as `dudx = dx("U")`
- Compute second-order derivatives, e.g. $\partial^2 u/\partial x \partial z$, specified as `d2zdudx = dz("dudx")`

¹<http://bit.ly/16zB9Vf>, retrieved 12-Oct-2011.

²<http://bit.ly/MWkC8D>, retrieved 10-Jun-2012.

- Compute gradient of subgrid shear stress as a source of momentum, e.g., $\frac{1}{\rho}(\frac{\partial}{\partial y}\tau_{xy}) = \frac{\partial}{\partial y}(\overline{u'v'}) := \sqrt{2}C_s^2(\Delta L)^2 \frac{\partial}{\partial y}(\bar{S}_{xy}|\bar{S}_{xy}|)$, specified as $2.828 * C_s * C_s * dL * dL * \text{abs}(dudy + dvdx) / 2 * (d2udydy + d2vdx dy) / 2$ where abs is the absolute value function.

The choice for C_s in practice is much dependent on specific problems. For turbulent round jets, Pope (2000) recommended that $C_s = 0.17$. Ilyushin and Krasinsky (2006) performed LES on a $360 \times 132 \times 24$ grid, and observed that an increase in C_s will lead to the lengthening of the jet potential core.

5.2.2 Dynamic grid adaptation

Grid adaptation is used in this simulation case to enhance the level of details in the double shear layer. By applying adaptive refinement, a higher grid resolution is generated at any local region satisfying a predefined flow condition. For example, a condition $0.2 < C < 0.5$ can be predefined where C is the tracer concentration. In case of a simulation using a pure jet without any tracer concentration, a velocity-based refinement criterion, e.g. $0.1 < U_* < 0.11$, can be used. The choice for refinement criterion is apparently ad-hoc, but simulation practice shows that, for high-resolution grids, only minor differences between results obtained using these criteria. In this case, refinement is made at level 10 for locations where vorticity strength $||\Omega||$ exceeds 250. This is based on observation of the vorticity field generated by simulations.

5.2.3 Test cases

A test with a range of C_s values has been carried out, and with appropriate values of viscosity chosen as input for the simulations. The procedures for finding ν_{comp} has been presented in Section 3.4.1. The results listed in Table 5.1 show that, for higher C_s the computed viscosity, ν_{comp} , tends to rise slowly (through the increase of ν_{SGS}). However the difference in ν_{comp} between the cases with and without modelling is large. It should be noted that ν_{comp} is a bulk quantity characterising the system, derived from the kinetic energy and vorticity of the *whole system*.

5.3 Time dependent behaviour of two-jet system

This section interprets the manifestation of instability in a double shear layer. To achieve this, data must be collected continuously during a time period for every sample points. On the other hand, snapshots of the state for the whole system should be

Table 5.1: Computed viscosity for different cases, without and with subgrid scale turbulence modeling using different values of C_s

# jets	α	Re	ν_{init} ($\times 10^{-6}$)	$\nu_{\text{comp}} (\times 10^{-6})$ for various values of C_s below					
				No	0.09	0.11	0.13	0.17	0.21
One	-	2460	-164.	70.0		-	193.	203.	210.
	-	5427	-62.	-		-	-	-	-
	-	6093	-95.5	10.0		-	-	-	-
	-	7304	10.3	14.6		-	-	-	-
	-	10815	10.7	23.0		-	-	-	-
	-	10937	11.7	30.0		-	-	-	-
Two	15°	5360	2.19						
	30°	5360	2.19		-		-		
	45°	5360	2.19		-	-	-		

recorded periodically so that certain features of the flow, e.g. oscillation and wave propagation, can be recognised.

5.3.1 Observations on turbulence structure in a plane jet

The following example does not represent two-jet mutual impingement. Rather, it shows how the asymmetric nature of the jet flow affects its stability. A plane jet is considered; this is somewhat related to the double shear layer in its “parallel flow stage” mentioned in Section 4.3.4. The jet is formed through a slot (1 cm wide) near a wall.³ The flow field is essentially two-dimensional with nonuniform distribution of inflow velocity. This inflow velocity is greater on the downward side of the plane jet, and also slightly deflected toward this side. (See the mean flow field on the left subfigure of Figure 5.8. The difference in dynamic pressure between two sides of the plane jet probably causes a Kelvin–Helmholtz instability, which is observable through the meandering centreline of the jet (subfigure on the right).

In the instantaneous flow field, at the vicinity of jet slot there is a vortex street on both sides of the jet. In the time-averaged colour plot, however, the vortices are smeared out into a pair of stripes with high vorticity. This suggests that the vortices continually move downstream. The alignment of counter-rotating vortices can be resembled as two vortex sheets. This was inferred by Goldschmidt et al. (1983) through flow visualisation.

³Experiment conducted at Maritime Research Centre, NTU

Figure 5.8: Time-averaged and instantaneous vector flow fields of a plane jet, measured using PIV instrument; colour backgrounds indicate the strength of vorticity.

Figure 5.9: Velocity component normal to the double shear layer, $Re = 1.09 \times 10^4$

5.3.2 Instantaneous flow field

While the change in U on the boundary is gradual in the double shear layer, as shown in Figure 5.2, the spatial variation of V is more prominent. Figure 5.9 shows that the flow takes the form of alternating up and down “jumps” which results in a ripple pattern in the V -velocity distribution on this symmetry plane. Each “ripple” is curved and apparently travels outward, away from the zone of impact.

The upward/downward flow velocities directed to stagnant ambient fluid and gives rise to pressure. This is shown in Figure 5.10, where alternate circular regions with high/low pressure are marked. Once the jet flow hits a small region with high pressure, it is deflected and this occurs repeatedly. Eight such deflections can be seen from the figure. These deflections are apparently magnified, becoming asymmetric and deviate from the symmetric plane.

5.3.3 Analysis of velocity fluctuation

In order to analyse the fluctuation of flow velocity, a time series for each velocity component is stored during simulation. The CFL condition restricts the simulation time

Figure 5.10: Pressure distribution with velocity vectors of two impinging jets, $Re = 1.09 \times 10^4$

step to typically 10^{-3} s (for $U_o = 1$ and minimum grid size = $\frac{1}{512}$). This temporal resolution is similar to those obtained from hot-wire anemometers. Figure 5.11 shows the instantaneous velocity recorded at four location points in the double shear layer. It can be seen that on a given transect, the points nearer to the central axis has both greater mean velocity and fluctuation; and this discrepancy becomes less when travelling downstream.

Figure 5.11: Time record of longitudinal velocity. Left chart: at transect $(x - x_{conv})_* = 5$. Right chart: $(x - x_{conv})_* = 10$. Upper solid line: $y_* = 0$ (central axis). Lower solid line: $y_* = 2$. Dotted lines: $\bar{u}$.

Intermittency. The intermittency of a turbulent flow can be defined as the probability that the flow at a specific location in a specific time instance, is turbulent. Since it is difficult to collect an ensemble of flow states, this probability can be approximated as the percentage in time that turbulence is observed at that location. To quantify whether the fluid is “turbulent” or not, a threshold of vorticity can be defined (Lee and Chu,

Figure 5.12: Intermittency factors at various transects of the double shear layer.

2003, p. 41).

Townsend (1980, p. 211) stated that the probability that a sample point⁴ $(x, y, 0)$ lies in the turbulent fluid, or the local intermittency is $\gamma(x, y) = \int_y^{+\infty} \mathfrak{P}(y_1, x) dy_1$, where y_1 is the vertical position of the interface separating the turbulent shear layer and the nonturbulent fluid above it. For free turbulent flows Corrsin and Kistler (1953) showed that $\mathfrak{P}$, the probability density function of y_1 , follows a Gaussian error function with a mean value μ and a standard deviation σ for the random variable y_1 .

Gutmark and Wygnanski (1976); Wygnanski and Fiedler (1969) adopted a formula based on the kurtosis of the distribution of u' ,

$$\gamma = \left(\frac{\overline{u'^4}}{(\overline{u'^2})^2} \right)_{CL} \bigg/ \left(\frac{\overline{u'^4}}{(\overline{u'^2})^2} \right) \quad (5.2)$$

for the region inside the dihedral angle $y/x \leq 0.16$ bounding a plane jet. It is well known from literature that the intermittency factor on a transect decreases from 1.0 in the centreline of the jet to almost zero outside the turbulent zone. A value of 0.5 is usually chosen for γ to separate between the turbulent layer and nonturbulent ambient fluid.

Based on the observation that the variation in tracer concentration is more regular than the distribution of momentum, in this study a threshold regarding the probability that $c > c_{\text{threshold}}$ is adopted.

The distribution of intermittency factors at two transects, $x'_* = (x - x_{\text{conv}})_* = 5, 10$ for the case $\alpha = 15^\circ$ and other two transects (15, 20) for the case $\alpha = 45^\circ$ is

⁴Here only the half-space above the symmetry plane is considered, $y \geq 0$.

Figure 5.13: Double velocity correlations in various directions: longitudinal, vertical, and lateral.

summarised in Figure 5.12. Different values for the threshold concentration have been chosen ($c_{th} = 0.05$ for the former case and $c_{th} = 0.10$ for the latter). This threshold, however, should not be chosen arbitrarily. An objective choice for c_{th} might aim for maximum likelihood between the intermittency factor distribution produced with a Gauss error function.

Double velocity correlation. Correlation coefficients between the axial velocity fluctuation, at an origin point $(0, 0, 0)$ in three directions are defined as

$$\begin{aligned}
 R_{uu}(\Delta x, 0, 0) &= \overline{u'(0) u'(\Delta x)} / \left[\overline{u'(0)^2} \cdot \overline{u'(\Delta x)^2} \right]^{1/2} \\
 R_{uu}(0, \Delta y, 0) &= \overline{u'(0) u'(\Delta y)} / \left[\overline{u'(0)^2} \cdot \overline{u'(\Delta y)^2} \right]^{1/2} \\
 R_{uu}(0, 0, \Delta z) &= \overline{u'(0) u'(\Delta z)} / \left[\overline{u'(0)^2} \cdot \overline{u'(\Delta z)^2} \right]^{1/2}
 \end{aligned} \tag{5.3}$$

For this study, the origin points are chosen on the central axis (x varies, $y = 0$, $z = 0$). The results (Figure 5.13) show R_{uu} for various locations of origin point along the central axis. In longitudinal direction (left subfigure), a good velocity correlation R_{uu} can only be achieved over a very short distance, i.e. $\Delta x \sim 3d$. This is different from observations of typical shear flows, where $R_{uu}(\Delta x, 0, 0)$ decays exponentially but remains positive. For the correlations in y and z directions, there is an original point ($x'_* = 20$) for which the correlation curve notably differ from the rest. Generally the remaining curves qualitatively agree with the typical curves in theory.

Although a quantitative comparison of R_{uu} with experimental data is not performed in this study, there are relevant data in literature. For example, Grant (1958) conducted experiments on a wall jet and plotted R_{uu} , R_{vv} , R_{ww} for both origin points inside and outside the boundary layer. The former point, located where the velocity U satisfied

$U_\infty - U = \sqrt{\tau_0/\rho}$ (here τ_0 is the wall stress), may be used to compare to the present results.

5.4 Summary

This chapter has been broadly divided into two parts. The first part deals with time-averaged flow condition. Some simulation results are reused from Chapter 4, but not the simplified double shear layer model. In this part a new method has been proposed, to provide a three-jet system which is equivalent to the two-jet system in terms of mean flow field. The method is derived based on equating both momentum and mass fluxes of two systems.

Simulation is then performed based on this newly proposed three-jet system, and the output shows reasonable agreement with the two-jet system. The same applies to simulation with a symmetry plane (i.e. half-domain simulation). However, a three-jet system simulated in half-domain is unable to reproduce the flow field in a two-jet system.

In the second part of this chapter, the detailed structure of the flow resulted from two impinging jets has been described. A simple Smagorinsky–Lilly model is implemented in the simulation. This model effectively raised the value of computed viscosity, ν_{comp} , of the system. Due to restriction in resources, however, this study does not cover any verification of this turbulence model.

The computed flow field (double shear layer) produced by a two-jet system ($\text{Re} = 5360$) has been examined. At any time instance, the double shear layer is wavy and not planar as can be seen in the result from time-averaged simulations. This waviness is more pronounced for larger ν . An intermittency factor based on a threshold concentration value, c_{th} , is adopted. However its use to determine the boundary between the turbulent flow and ambient fluid largely depends on the choice of c_{th} .

The velocity correlation functions, R_{uu} , for the points along the central axis show that the correlation in u' between nearby points tends to decrease rapidly when the distance between them increases. Typically R_{uu} diminishes when the distance exceeds $2d$, and shorter such distance in the lateral (z) direction.

Chapter 6

Flow field formed by three co-planar, converging jets

6.1 Jets in close proximity

A practical situation one frequently encounters in environmental engineering is the use of an outfall with multiport diffuser to disperse the pollutant load into the ambient water. Most conventional approaches of a outfall design have been those that neglect dynamic interaction among the jets, and assume that small round buoyant jets merge at some elevation above the diffusers to form a bigger plane plume. In such cases, the jet momentum is often neglected and buoyancy is considered the dominant driving force. The formulae developed are typically not for calculating flow velocity directly, but rather for establishing relationships between the dilution ratio to x_* and Fr, the Froude number at the orifice (Rouse et al., 1952; Liseth, 1976).

Wood et al. (1993) presented a method to analyse the velocity distribution of merging buoyant jets, assuming that each jet follows a Gaussian velocity distribution (see Figure 6.1). The merging velocity distribution by superposition will have two peaks and a thicker tail in the intersection zone of two contributing jets while the Gaussian profile is preserved in the transverse direction (z in this case). Far from the jet nozzles, the peaks become less prominent and the flow distribution becomes almost uniform. The general formula is:

$$U(x, y, z) = U_{CL}(x) \exp(-z^2/b^2) \frac{\sum_i \exp\left[-\left(\frac{y/s+i}{b/s}\right)^2\right]}{\sum_i \exp\left[-\left(\frac{i}{b/s}\right)^2\right]}, \quad (6.1)$$

[where the symbols are defined in Figure 6.1] from which Wood derived three shape

Figure 6.1: Multiple jet merging model (adapted from Wood et al., 1993)

functions for the mass, momentum and buoyancy fluxes. By adding these fluxes he obtained a solution for merging jets, which was validated using experimental results of plane jets (Cederwall, 1971; Kotsovinos, 1975).

Lee and Chu (2003, p. 53), based on Reichardt's (1943) hypothesis, suggested a "rule of thumb" that the square of longitudinal velocity for multiple jets can be obtained by superpositioning the squares of longitudinal velocities from each jet on one another,

$$U^2(x, y) = U_{CL}^2(x) \sum_i e^{-2(y-y_{o,i})^2/b^2(x)}. \quad (6.2)$$

Hodgson et al. (1999) adopted the same approach in the treatment of pollutant, and with validation based on laboratory measurement of tracer concentration in a jet flow ($Re = 4900$). Although this method was quantitatively different from Wood's (1993), the assumptions were similar: momentum interactions among the component jets were neglected.

The approaches described thus far are ideal for an array of infinite, identical, and equally-spaced jets. Symmetric planes are formed (Figure 6.1, right) and the mass flux from each nozzle is restricted in the space between two symmetry planes. Therefore the flow condition produced by all these jets are similar. All the jet centrelines are

Figure 6.2: Converging trend of two initially parallel-aligned jets in simulation ($Re = 1.1 \times 10^4$), with contours of U_* starting from 0.1 with increment of 0.1. The dot-dashed lines are straight.

straight. In reality, however, the number of nozzles is finite, and the outermost jet tends to “bend” towards the inner jets. The reason might be the difference in flow entrainment on either side of each outermost jet. The phenomenon is more pronounced in experiments involving thermal or buoyant plumes (see e.g. Pera and Gebhardt, 1975).

In a simulation performed by the author, three initially parallel jets ($Re = 1.1 \times 10^4$) emitted from orifices with a spacing of $s_* = 2.5$. Very slight convergence of two outermost jets can be observed as reflected in the inward shift of the notches in the corresponding velocity contours (Figure 6.2).

The velocity profile formed by multiple jets evolves toward a single-peak profile, somewhere between $x/s = 6.5$ to 9.5 in this simulation. Analogously, for merging plumes Kaye and Linden (2004) reported a value of $x/s = 4.8$ calculated from a mathematical model (or 4.1 from experiments). Plumes exhibit shorter merging height compared to jets because the former have larger entrainment rates β , while x/s is inversely proportional to β by the model.

By means of mathematical analysis it is possible to predict the behaviour of such parallel-aligned jets. For converging coplanar jets, however, momentum interaction between the jets make the prediction practically infeasible. Instead, the analysis is based on simulation output.

6.2 Simulation of three jets

For the present simulation, it is interesting to compare the output with single jet cases. To achieve this, the same grid refinement is applied for the segment of the jet cone shape beyond the merging point of three jets. Specifically, for $x_* > 18$, refinement is made at level 7 for all points where $r/x \leq 0.15$ (i.e. narrower than the case with a single jet). For the zone of jet merging, $4 < x_* \leq 18$, refinement is made at level 9 for the region bounded by the cylindrical zone $r_* = 2.5$. In the vicinity of orifices

($x_* \leq 4$), and similar to the 2-jet case, a refinement level of 9 is again used. The jet flow is simulated for $Re = 1.09 \times 10^4$ and various types of nozzle configuration chosen include $\alpha = 15^\circ, 22.5^\circ, 30^\circ, 37.5^\circ$, and 45° , all with $s_* = 2.5$; and a single orifice for comparison. A constant value of viscosity is chosen, $\nu_* = 2.44 \times 10^{-6} L^2$.

The flow variables have been sampled at various cross-sections of x_* . To facilitate intercomparison of the resulting flow field between different α , the zero-point for longitudinal distance is chosen and set at the geometric convergence of three jet centrelines, $x_{0*}(\alpha) = s_* \tan \alpha$. The focus is set on the compound jet from the convergence point downstream (Figure 6.3). The flow structure in this zone can be divided into these sections: (i) impinging section, near the geometrical convergence of three jet centrelines, $x_* = s_* \tan \alpha$, where momentum dominates; (ii) flat section, where the axial velocity spreads in the lateral direction (z); and (iii) restoring section, where the jet turbulent cone becomes less flat and similar to that of a round jet. Collections of superimposed contours at various cross-sections are shown in Figure 6.5 and will be discussed later. The change from section (ii) to (iii) is determined through the variation of cross-sectional shapes of U -contours against x_* , as will be shown later in Section 6.2.1.

Figure 6.3: A compound jet in translated coordinate system

6.2.1 Cross-sectional shape and width of the compound jet

To compare the simulation result between one and three jets with diameter d , the single jet must have the nozzle diameter enlarged to $d_{eq} = \sqrt{3}d$, to yield the same cross-sectional area. It should be noted that since the compound jets are fully turbulent (with no existence of a potential core), it is reasonable to choose the starting point for equivalent single jet within the zone of flow establishment. For this comparison purpose, $x_* = 8$ was chosen as the starting point for a single equivalent jet, which is the point of convergence for a case with $\alpha = 17^\circ$. The choice is somewhat arbitrary but as is shown in Figure 6.6, with $x_* = 8$ the width of a single jet can be compared to

Figure 6.4: (Left) Schematic velocity distribution for a compound jet cross-section and the representative contour, together with (Right) Definitions of $d_{\max}$, $d_{\perp}$, and d' for such a contour.

those of compound jets for different values of α .

Results of various simulations showed that a set of axial velocity contours in each cross-sectional plane tend to be concentric; hence one contour should characterise the spread of velocity for that cross-section. The contour corresponding to an axial velocity of $U_{\max}/e \approx 0.368U_{\max}$ has been selected (Figure 6.4), so that the “radius” bounded by this contour represents the characteristic jet half-width. The contours tend to be elongated in the direction perpendicular to the plane containing centrelines of the component jets. This tendency is more pronounced for the cases with large α , as strong momentum impact suppresses the velocity spreading within the aforementioned plane.

Figure 6.5 shows contours of flow velocity at four equidistant cross-sections: $x_* = x/d_{\text{eq}} = 5, 10, 15$, and 20 , which continually expand and tend to restore axisymmetry as x_* increases. Also, contours are less horizontally symmetrical for larger value of α , which is probably due to the increasing mutual impact among the jets.

The oval-like shape for each of these contours are measured using three scales: major diameter, $d_{\max}$ (arbitrary direction), transverse diameter $d_{\perp}$ (orthogonal to the direction of $d_{\max}$), and characteristic diameter d' (that of a circle which has the same area with the area encompassed by the contour) (see Figure 6.4). In particular, $d' = \sqrt{4A/\pi}$, where A is the area enclosed by the contour and was tracked manually from the digital image produced through computer visualisation. To quantify the degree of deviation of a contour from a circle, the flatness factor, defined as $\text{flatness} \equiv 1 - d_{\perp}/d_{\max}$, is used; where $\text{flatness} = 0$ implies a circle.

The linear increase in jet characteristic diameter is evident from Figure 6.6 (left subfigure). This rate of increase varies from 0.57 for $\alpha = 30^\circ$ to 0.88 for $\alpha = 37.5^\circ$. Good correlations between values of $d'_*(\alpha)$ and x_* were found (with coefficients of

Figure 6.5: Contours indicating generic shapes of the distribution for U at various cross-sections, $x_* = 5, 10, 15, 20$; for the case of one jet and three jets of various angles α . Contour values are $0.368U_{\max}$. Small circles represent orifices.

Figure 6.6: Characteristic diameter (left) and flattening factor (right) of axial velocity contours against x_*

Table 6.1: Relationship between the width of a component jet and that of a single jet

Formulae read: $b_3^* = mb^* + n$, $b_{\max}^* = p_{\max}(b_3^*)^{q_{\max}}$, $b_{\min}^* = p_{\min}(b_3^*)^{q_{\min}}$

α	m	n	$p_{\max}$	$q_{\max}$	$p_{\min}$	$q_{\min}$
$\alpha = 15^\circ$	1.004	-0.328	3.652	0.537	0.250	1.538
$\alpha = 22.5^\circ$	1.010	-0.254	3.056	0.684	0.348	1.322
$\alpha = 30^\circ$	0.841	1.778	7.520	0.298	0.180	1.675
$\alpha = 37.5^\circ$	1.303	1.147	3.995	0.747	0.181	1.463
$\alpha = 45^\circ$	1.008	2.293	14.014	0.122	0.188	1.526

determination typically exceeding 0.99). So apparently, for small to moderate values of α , the “bulk” spreading behaviour characterised by d' is very similar to that of an equivalent single jet.

From the variation of d' against x_* , and as has been defined, $d' = 2b_3$, the spreading rate of a compound jet width can be inferred. Specifically, $db_3/dx \sim 0.3$ for a single jet and three-jet impinging at smaller angle ($\alpha \leq 30^\circ$) while a larger rate $db_3/dx \sim 0.44$ was found for the case $\alpha = 37.5^\circ$. All these values are greater than empirical measurements for single jet, which is 0.116. The reason might be due to the numerical viscosity which acts as extra friction to “pull” water surrounding the jet core into motion. Although this is not quantified in the present study, a similar effect can be observed from simulations using Gerris. When running two otherwise identical simulations with difference values of prescribed viscosity (ν_{init}), the case with higher viscosity would show less mixing behaviour with smoother contours.

Based on empirical results obtained from the simulation, several preliminary formulae can be inferred for three-jet configurations; they are listed in Table 6.1. First, the linear formulae relating b_3^* with b^* show slopes $m = 0.841 \rightarrow 1.303$, implying that b_3^* and b^* have similar magnitude. The intercepts $n > 0$ are not important because when

defining the dimensions of a single equivalent jet, the axial point of origin was shifted. Second, the relations between b^* and $b_{\max}^*$, $b_{\min}^*$ the two semi-axes of the schematised ellipse representing a cross-section of the compound jet.

It can be seen, through the flatness factor represented on the right subfigure of Figure 6.6, that generally for larger α cases, the shape of contour is flatter in vicinity of the orifices. The reason might be due to the momentum impact of two sideways jets to the middle one forming a thin shear layer where the longitudinal velocity is strongly spread out in lateral direction (i.e., the z -direction). In contrast, jets from smaller α configurations will smoothly merge together, and the momentum will be distributed more uniformly in both lateral (z) and transverse (y) directions.

As x_* increases, the flattening factors decrease, showing that the cross-sections tend to reach the circular shape of a single jet, regardless the value of α . The trends of decrease are generally not linear, although certain cases exhibit linearity in a certain range of x_* , specifically from $x_* = 5 \rightarrow 15$ for the case $\alpha = 15^\circ$, $7 \rightarrow 17$ for $\alpha = 30^\circ$, and at least $5 \rightarrow 20$ for $\alpha = 37.5^\circ$. Also, the flattening factor does not indicate the rotation of the cross-sectional velocity profile in three-jet cases, as mentioned in the next section.

6.2.2 Cross-sectional velocity distribution

Comparing d' , and even flatness, against x_* does not reveal the fact that momentum distribution on a cross-sectional plane is more complex in the cases involving three jets. The compound jet flow usually manifests in two peaks in the distribution of axial velocity in every cross-section near the confluence.

The upper left subfigure of Figure 6.7 shows such a typical bimodal velocity distribution at cross-section $x_* = 10$ formed by three jets with $\alpha = 45^\circ$. A saddle surface of velocity distribution can be realised. In contrast, the regular, concentric contours of a single jet (the right subfigure) correspond to a Gaussian-like distribution.

The presence of two peaks in a velocity profile is an important factor contributing to the instability during evolution of the compound jet. A behaviour of such instability is that one peak tends to enlarge while the other depletes. An intuitive interpretation is that of the two parallel, proximate currents, the one which moves faster has lower static pressure and the other current will be attracted to. Furthermore, the compound jet cross-section tends to rotate, albeit slowly, as x_* increases. This is due to a slight asymmetry in cross-sectional vortex motion similar to that shown in the section of two impinging jets. Consider the case of $\alpha = 22.5^\circ$ from Figure 6.5. If we assume that the shape of contours characterises well the distribution of U on the cross-sectional plane,

Figure 6.7: Comparison between axial velocity profile (U_*) formed by three impinging jets with $\alpha = 45^\circ$ ($x_* = 8.56, 18.56$) and one equivalent jet ($x_* = 2$).

then the misalignment of the outermost contour ($x_* = 16.5$) from the vertical axis suggests that a swirl motion effect takes place. Such swirl motion will not be analysed in this study. Rather, suffice it to say now that an increasing α leads to stronger instability and causes the velocity profile to rotate around the x -axis along this longitudinal direction.

6.2.3 Probability distribution for location of the velocity peak on a cross-section

Another feature of instability in three-jet configurations manifests in the dislocation of the maximum point from the centroid of a cross-sectional plane. Ideally, they should coincide (i.e., at $r = 0$); but strong momentum interactions frequently alter the position of this peak. The lower part of Figure 6.8 shows that, for the case of three jets converging at $\alpha = 15^\circ$ (left panel), the locations of peak velocities cluster at two zones on both sides of the vertical axis (y). Also, the distance from the velocity peak to the central axis (i.e. the length of dislocation) enlarges with increasing x . For the case of a single jet (right panel), there is only one cluster around the centroid of the cross-section.

Figure 6.8: Upper: Probability distribution of the velocity peaks at different cross-sectional planes along the downstream direction. Lower: Location of instantaneous velocity peaks on two cross-sectional planes for three jets, $\alpha = 15^\circ$ (left) and one jet (right); also shown are the size of orifices.

The probability (i.e. proportion of time during simulation period) for $U_{\max}$ to locate within an interval of radial distance, $[r, r + dr]$, from the geometric centre of the cross-section, is $dp = p(r)dr$, where $p(r)$ is the probability density function (p.d.f.) for various jet configurations. The normal probability distribution has been assumed for all these p.d.f's. There are cases in which multiple peaks co-exist in a profile. For every such case, only the highest peak was chosen as $U_{\max}$.

Based on the output data sampled in each window sized $\frac{L}{4} \times \frac{L}{4}$ for 4000 timesteps, the radial distance to location of peak, r , has been bilinearly interpolated and formatted as a data series. Empirical cumulative distribution functions are established for such time series and curve-fitting were performed to find the normal distribution density

Figure 6.9: Changes in mean and standard deviation in normal distributions of $U_{\max}$ versus x_* for various values of α

functions, $p(r) = \frac{1}{\sqrt{2\pi}\sigma} \exp\left(-\frac{(r - \bar{r})^2}{2\sigma^2}\right)$, where $\bar{r}$ and σ are parameters to be adjusted. More details on this curve-fitting process can be found in Appendix D.3. The upper part of Figure 6.8 shows that the distribution of peak velocity tends to spread out and at the same time shift away the centreline in radial direction.

A summary of the distribution parameters are depicted in Figure 6.9. First, it can be seen that for each α value, both the mean and standard deviation of the peak dislocation increase, in linear trends, with x . This corresponds to the enlargement of clusters and the rise in the length of dislocation, as noted above. Second, the dislocation appears to be larger for larger α , as in such cases the flow is not restricted in the y -direction.

6.2.4 Velocity decay in the longitudinal direction

Figure 6.10 (left chart) shows that the centreline velocity varies inversely to x_* as expected. The rates of decay, η , are essentially universal for all α . Here η varies from 0.498 for $\alpha = 30^\circ$ to 1.274 for $\alpha = 15^\circ$. The cases $\alpha = 45^\circ$, single equivalent jet, and $\alpha = 15^\circ$ give values of η near the empirical value $\eta = 1$ for single round jets; although none of them are close enough. On the other hand, generally the values of $U_{\max}^*$ all converge at $x_* \sim 20$. This is an indication that far from the orifices, a compound jet tends to behave similarly to a single round jet.

Although the centreline velocities of all 3-jet configurations are lower than that of the single jet, the cross-sectional average velocity shows only slight changes regarding x_* for all cases with various α (Figure 6.10, right chart). For most of the time, a 3-jet system gives lower average velocities than an equivalent single jet, except for

Figure 6.10: (Left) Decay of centreline velocity; trend lines show power functions, $U_{max}^* \propto x_*^{-\eta}$. (Right) Change of cross-sectional average velocity in x -direction.

case $\alpha = 15^\circ$ which is quite close. Each cross-section velocity U_{av}^* was calculated as the average of all positive velocities from sample points in the cross-section, divided by U_{max} . The samples on every cross-section were taken similarly to that for finding U_{max}^* —a uniform grid of points on a window sized $\frac{L}{4} \times \frac{L}{4}$.

6.2.5 Intensity of turbulent fluctuations

The Reynolds normal stress $\overline{u'^2} = u_{rms}^2$ is an important characteristic of jets in hydraulic engineering. Figure 6.11 represents the distribution of u_{rms} on each cross-section, normalised by the maximum velocity of that cross-section for easier comparison between different cross-sections. A contour value of 0.2 is chosen because that is roughly the value of u_{rms} near the “sharp point” on a graph of u_{rms} against r for the case of a single round jet (see also Figure 3.11), which characterises the extent of a region with high intensity of fluctuation. Unlike the mean axial velocity earlier shown on Figure 6.5, the intensity of turbulent fluctuation exhibits more irregular distribution with convoluted contour lines. For a large value of α such as 45° and far from the orifices, the contours manifest in complex shapes without any definite characteristic dimensions.

The profiles obtained from single-jet and three-jet simulations exhibit different patterns. It can be seen from Figure 6.12 that the mean velocity profile of the 3-jet system shows different shapes on the front (x - z) and plan (x - y) views as a result of the non-axisymmetrical layout of the nozzles in the 3-jet system. The intersection of three smaller jets tends to expand the velocity profiles laterally. Therefore wider profiles are observed in the plan view. This is also in accordance with the contours on the side views (Figures 6.5 and 6.11).

Figure 6.11: Contours indicating distribution of intensity of turbulent fluctuation u' (normalised by the peak velocity of each cross-section); the values are $u_{\text{rms}}/U_{\text{max}} = 0.2$ for various cross-sections $x_* = 5 \rightarrow 20$

Figure 6.12: Comparison of simulated flow velocity and turbulent intensities for single jet (solid line) and three-jet system with $\alpha = 30^\circ$ (dash line).

6.3 Summary

Three parts of a compound jet flow formed by three round jets are identified: the impinging region where the momentum is dominant; the intermediate restoration region; and the axisymmetrical-asymptotic region beginning from $x_* = 20$ where no remarkable difference in shape of the velocity profiles exists between turbulent flows produced by single jet and three-jet alignments.

Based on the observation that the cross-sectional velocity profile of a compound jet is oval-shaped for most of the time, an attempt to parameterise it using two semi-axis lengths, $b_{\max}$ and $b_{\min}$, had been made. Among the parameters and quantities, b and x_* appear to have strong correlation. Due to the abrupt change in momentum near the orifices, the flatness factor of the oval shape profile does not change linearly to x_* . The parameter which was found to be least related to the others is α . In a set of equations relating $b_{\max}$ and $b_{\min}$ to b , the angle α plays the role of a nominal value to help select the appropriate equation.

For preliminary design, α is chosen among every sixth of a right angle ($15^\circ, 30^\circ, 45^\circ$). The omission to analyse flow characteristics as continuous functions of α , that was presented in this chapter, can therefore be justified.

For all cases of α , the momentum spreading (b) and velocity magnitude ($U_{\max}^*$) tend to converge at $x_* \sim 25$ (Figures 6.6, 6.10). This fact suggests a tendency of the compound jet toward axisymmetric behaviour when travelling downstream. Far from the orifices, the compound jet flow show certain similar characteristics to a single jet: contours of axial velocity on cross-sectional planes are less prolonged; the flow pattern is almost indistinguishable from those of single jets.

In addition to the mean behaviour of a compound flow field, the randomness in location of a velocity peak on a compound jet cross-section had been addressed. Gaus-

sian probability density functions had been found at various x_* for different α values.

The predominant component of turbulence level, $u_{\text{rms}}/U_{\text{max}}$, for various 3-jet configurations were found to be of comparable magnitude with that of a single jet, although there were no clear trends showing any dependency of this turbulence level on α .

Chapter 7

Mixing and dilution of effluent for various nozzle alignments

7.1 Entrainment and mixing in a multiple-jet alignment

The idea behind dividing a given jet discharge is to increase the volume of ambient fluid entrained into the jets. The entrainment takes place through the interface between turbulent jet flow and the (essentially) non-turbulent ambient water. Regarding the case of three co-planar jets, the entrainment between component jets include those water volumes indicated by the shaded regions in Figure 7.1. If only one single larger jet was used, no such zone of fluid entrainment would be observed. Another factor to be considered is that since the entrainment rate q_E is proportional to the product $U_{av}b$ (where “av” denotes cross-sectional average), using 3-jet configuration instead of a single jet might lead to an increase of q_E through increasing b despite the decrease of U_{av} (as in Section 6.2.4). Further details on dilution will be presented in Section 7.3.1.

A typical diffuser comprises a series of ports (nozzles) on each side. Wood et al. (1993) presented a qualitative description for the merging process of buoyant jets emitted from such a system (Figure 7.2). Here the round buoyant jets emitted from the same side of diffuser are assumed to completely merge into a plane plume when they reached

Figure 7.1: Schematisation of entrainment prior to the formation of compound jet

Figure 7.2: Schematization of a standard diffuser: with view along a pipe section (left) and cross-section A-A (right) (adapted with simplification from Wood et al., 1993)

the position M2. Then two plane plumes merge into a single larger plume (M3) and rises vertically (M4). The angle at which the two plane jets merge is small and this may imply that there is no significant momentum interactions between these two plane jets. During the merging process the amount of water contained in the zone bounded by two buoyant jets immediately above the pipe is entrained into the component jets and creates a lower pressure in this entrainment zone. As a consequence, water from outside is continually brought into this zone through the gaps between the small round jets emitted from diffuser ports.

7.1.1 Indication of mixing

The aim of a good outfall configuration is to maximise the expansion of turbulent region generated by the jets so that more ambient fluid is drawn into turbulent motion. In most general circumstances, mixing occurs at two scales: (i) macroscopic, which is associated with shear flow dispersion, and (b) microscopic, which is caused by turbulent fluctuation.

Momentum-driven jets, being the subject of the present study, are capable of both macroscopic and microscopic mixing, which makes them a conventional technique for diluting effluents. On the larger scale, the gradient of U in the radial direction causes a shear effect for which tracer spreads by a rate which can be a million-fold the typical molecular diffusivity of the tracer (Taylor, 1953). On the smaller scale, a jet possesses high turbulent intensity ($\sim 20\%$) in its turbulent core so that any potential species issued from the orifice will be mixed throughout every cross-section of the jet. Specifically,

the turbulent diffusivity, Γ characterises the extent of mixing and can be formulated as $\Gamma = u_{\text{rms}} T_L$. Thus the mixing extent is proportional to the velocity fluctuation, for a given Lagrangian time scale T_L that characterises the persistence of flow condition.

To evaluate the performance of mixing, it is natural to use some measures related to the extent of mixing zone. The *length of mixing* is taken as the longitudinal distance from where turbulent cores of individual jets meet to where the compound jet cross-section has evolved to an approximate circular shape, i.e. the same flow condition that is generated by a single equivalent jet. The *mixing volume* is the amount of space contained in the turbulent core of such a compound jet.

Alternatively, the *centreline concentration*, or maximum concentration in a specific cross-section of a compound jet can be used. Usually this concentration is smaller than that released from the nozzle, except possibly at the point of convergence, where the tracer is concentrated. The centreline dilution is reciprocal of the centreline concentration.

The three quantities above represent mixing in a definitive manner. In addition, if the degree of mixing is of concern, then some derivative physical quantities must be used. Naturally this should associate with the velocity fluctuation, u_{rms} , since the spread of tracer concentration is directly related to this quantity. However, there are also indicators that depend on bulk flow properties. For example, Rupe (1953, 1956) used the momentum ratio between two jets to evaluate the mixing performance of the system.

It should be noted that this type of mixing performance applies for relative mixing between substances issued from each nozzle rather than mixing of the whole jet effluent with ambient water. It will be pointed out later that this kind of evaluation is useful and provide a complementary way to assess the mixing capability of an outfall system.

7.1.2 Tracer distribution and dilution

In most parts of the following discussion, the term “concentration” does not refer to the presence of a certain chemical species, but rather an indication for the spread of turbulent motions. A detailed analysis of a contaminant spreading requires properties of the pollutant, such as diffusivity, specific weight and temperature, which might introduce noticeable buoyancy. Such an analysis will not be pursued in this study.

In the specific problem involving free jets, semi-empirical results indicated that the concentration profile also conforms to a Gaussian curve very similar to the velocity profile, but with a wider spread (specifically, 1.17 times the spread of longitudinal velocity, see e.g. (Baines and Chu, 1996)). To explain why the distribution of concen-

tration is wider, a simple way might rely on the fact that a superposition of velocity involving U_i^2 (Eqs. 6.1, 6.2) where a superposition of concentration just consists of C_i . A quadratic form in the expression for velocity translates to a higher peak and consequently, less spreading.

The average dilution rate is estimated as the ratio between discharges at cross-section x_* and that at the orifice, where the former discharge is determined through circumferential integration of axial velocity on a cross-section. To evaluate dilution in a compound jet, however, the location of converged point regarding the location of orifices should be taken into account. If the orifices are closely located then the component jets are not much developed before impinging, and the compound jet is completely responsible for dilution. Otherwise, dilution takes place in both component jets and the compound jet.

7.2 Simulation set-up

In this chapter, the analysis regarding tracer distribution and dilution will be based on simulated results presented in Chapter 6. The reason for carrying out additional simulations is that the influence of s on the formation of the compound jet need to be further evaluated. It is easy to perceive that an increase in s would lead to an increase in size (b ; and correspondingly a decrease in U) of each component jet at the location just upstream of the point of impingement. This resembles a geometrical scaling and not much difference is expected for the compound jet except that it is scaled up. But there is still a difference—the underlying velocity distribution of component jets. To illustrate possible differences between such situations, nine simulations have been carried out, each with a combination between $\alpha = 15^\circ, 30^\circ, 45^\circ$ and $s_* = 10, 20, 30$.

The grid refinement specification is as follows. The parts in proximity with three orifices ($x_* < 6$), which contain the jet potential cores, are refined at level 9. The next part, the zone of established flow for component jets ($6 < x_* < x_{\text{conv}}^*$) are refined at level 8, but only for locations where $U > 0.1U_o$. Here $x_{\text{conv}}^* = s_* \tan \alpha$ is the axial coordinate of the convergence point. The zone of compound jet flow enclosed in a conical shape ($x \geq x_{\text{conv}}, r < 0.15x$) is refined at level 7. Although this required dynamic grid adaptation, simulation cases in practice show no significant change in grid structure once the steady flow condition has been reached.

The viscosity was prescribed as $-0.05|U|l_{\text{cell}}$. The negative value of viscosity chosen is to compensate the inherent numerical viscosity in the model. While this might raise consideration on choosing a particular mathematical expression for viscosity, it

Figure 7.3: Comparison between spreading of time-averaged concentration (left half of each subfigure) against velocity (right half).

is necessary to produce more realistic flow condition in the ZEF for component jets, as opposed to the use of a constant value for viscosity.

7.3 Analysis of simulation results

The fact that tracer concentration has a similar distribution but wider spread than velocity is evident from the shape of contours resulted from the simulations, see Figure 7.3. Particularly, the concentration spreading rate is greater along the narrower axis of the elliptic contour, which causes the concentration contours to be more circular than the velocity contours.

Also from simulation results, the ratio between the concentration-based and velocity-based jet widths, b_c/b , is generally $1.1 \sim 1.2$ (see Figure 7.4). A slight increase in this ratio is identified along the x -direction, but in all four cases the increasing ratios b_c/b tend to reach certain horizontal asymptotes. Specifically, for the single jet case, b_c/b appears to reach a level about 1.2, just higher than that expected from the empirical formula (which is $b_c/b = 1.17$). Apparently, for increasing α , the mixing is more intense and this ratio becomes larger.

Common passive contaminants exhibits the abovementioned behaviour: the mass diffusion is slightly larger than its momentum diffusion. Therefore the Schmidt number, $Sc \equiv \nu/D$ is smaller than one. In relation to the concentration profile, we may write: $C = C_{CL} \exp[-Sc(r/b)^2]$, where a comparison with the axial velocity profile reveals that $Sc = (b/b_c)^2 = 0.73$.

Figure 7.4: Ratio between concentration spreading rate and velocity spreading rate for various α

Figure 7.5: (Left) Average dilution ratio and (Right) centreline concentration against axial distance of a compound jet

7.3.1 Dilution

First, comparison for dilution rate is made between a single jet and three-jet configuration; the scope is beyond the point of convergence for 3-jet configurations. To estimate Q through a cross-sectional plane, a grid of sample points (64×64 , spacing $\frac{1}{128}L$) was used. The increase in discharge through various cross-sections along the downstream direction is shown in Figure 7.5. Here the dilution ratios, Q/Q_0 , are calculated for various 3-jet configurations at $Re = 1.1 \times 10^4$, and compared to the empirical value for the case of a single jet, $Q/Q_0 = 0.32x_*$.

The simulated results show that generally the rates of increase in dilution ratio of three-jet cases match that of a single jet. Also, this ratio is larger for cases with higher values of α . So a three jet system is certainly advantageous compared to a single jet in terms of dilution.

A complementary view of turbulent jets' capability to dilute tracers is through the

Figure 7.6: Average dilution ratio of component jets before merging

decay of centreline (or cross-sectional maximum value) of concentration. Apparently, the sampled data exhibits an inversely proportional relationship: $C_{\max}/C_o \propto x_*^{-1}$. The comparison is made with an empirical formula, $C_o/C_{\max} = 0.19(x + 6)/d$, where $(x + 6)$ is used instead of x , to neglect the length of the potential core. All the 3-jet alignments yield more favourable decay of tracer concentration compared to that of a single jet. Particularly, increasing the impingement angle α would result in higher decay rate.

Another comparison is carried out among the nine scenarios of various α and s_* values, in order to determine the effect of these two parameters on the dilution of the component jets before the merging process takes place. Four equidistant cross-sections $x = \frac{1}{4}x_{\text{conv}}$, $\frac{1}{2}x_{\text{conv}}$, $\frac{3}{4}x_{\text{conv}}$, and x_{conv} are considered. Values of U are sampled at grid points located on these cross-sections (with grid spacing $\Delta y = \Delta z = \frac{1}{256}L$), from which the amounts of discharge can be obtained through integration.

The results (Figure 7.6) shows that the ambient fluid is rapidly entrained into the turbulent core. Certainly, the ratio Q/Q_o is already high even for small s_* due to the contribution of ambient fluid (shaded region in Figure 7.1) between the component jets. From cross-section $x = \frac{1}{4}x_{\text{conv}}$ to $x = \frac{3}{4}x_{\text{conv}}$, the increase in Q appears to be of exponential rate (linear in this log-graph). Then this trend of increase slows down as $x \rightarrow x_{\text{conv}}$, as in this region the outer edges of two adjacent jets meet each other and the volume of entrainment is limited.

Also from this figure, the spacing s (represented by colour of symbols) appears to have greater effect on dilution, compared to the angle α (represented by grey tones of symbols). While α is an important parameter for compound jet dynamics due to its relation to jet momentum interaction, s is important in the zone of component jet

Figure 7.7: Correlation between the increase in discharge and the area enclosed between two consecutive sections

development because it determines the volume of fluid that is entrained into the jets.

The remark that s influences the dilution behaviour via expansion of the entrained zone is validated by examining the increase in discharge between cross-sections $x/x_{conv} = \frac{1}{4}, \frac{1}{2}, \frac{3}{4}, 1$. Figure 7.7 represents such an increase against parts of the area between each pair of consecutive cross-sections (whose proportions to the total, shaded triangular area in the subfigure are $\frac{7}{16}, \frac{5}{16}, \frac{3}{16}$ and $\frac{1}{16}$, respectively). In this case it is assumed that the entrainment volume of water is proportional to the area A shown in the figure. This 2-D simplification can be justified given the same diameter of orifices.

From the simulation results, it turns out that the discharge increment, ΔQ , and therefore, the dilution ratio can be estimated from simple geometry. In general, for two cross-sections located at x_1 and x_2 ($x_1 < x_2 < x_{conv}$) the area bounded between them can be determined as

$$A = \frac{1}{2} \Delta x [2(x_{conv} - x_1) \tan \alpha + 2(x_{conv} - x_2) \tan \alpha]$$

$$A = 2(x_{conv} - \bar{x}) \Delta x \tan \alpha \quad (7.1)$$

whereby $\bar{x} = \frac{1}{2}(x_1 + x_2)$. From Eq. (7.1) the rate of increase in discharge can be assessed readily using the correlation in Figure 7.7. Based on simulation results of

nine scenarios, the following mathematical relationship can be derived:

$$\frac{\Delta Q}{Q_o} = 0.034 \frac{A}{d^2} + 3.705, \quad (7.2)$$

with a coefficient of determination $R^2 = 0.858$.

On the other hand, since $\Delta Q \propto A$ approximately and $A \propto (x_{\text{conv}} - \bar{x}) \Delta x \tan \alpha$,

$$\begin{aligned} \Delta Q / \Delta x &\propto (x_{\text{conv}} - \bar{x}) \tan \alpha, & \text{or} \\ \Delta Q / \Delta x &\propto s - \bar{x} \tan \alpha. \end{aligned} \quad (7.3)$$

However by definition, $dQ/dx \equiv q_E$, the entrainment rate. Taking the limit of $\Delta Q / \Delta x$ as $\Delta x \rightarrow 0$ and replacing $\bar{x}$ by x , the following expression is obtained:

$$q_E \propto s - x \tan \alpha. \quad (7.4)$$

This equation shows the dependence of the entrainment rate on both the angle of alignment and the spacing between the nozzles. The spacing s has a linear effect on q_E ; the angle α has an intermediate effect through the tangent function. For moderate values of α (i.e. $\alpha \sim 30^\circ \rightarrow 45^\circ$), $\frac{d}{dx} \tan \alpha = (\sec \alpha)^2 = 1.33 \rightarrow 2$. Therefore, given a distance x upstream of the convergence point, the influence of α on the dilution ratio downstream of x is practically bounded. In other words, α has limited influence on the dilution ratio.

7.4 Comparison of intermixing among jet configurations

A way to investigate mixing of mass between the component jets. i.e. intermixing, is to introduce an individual tracer for each component jet and track the distribution of these tracers in the compound jet. Such an implementation has been done for the simulation by assigning three mass tracer sources, C_1, C_2, C_3 to the corresponding velocities U_1, U_2, U_3 . At the inflow boundary, these mass tracer amounts are equal to 1.

From the jet orifices along the three component jets, the individual tracers are bound by the corresponding jet cores and therefore separated, until near the point of convergence where jet boundaries touch each other and the mixing process takes place. From the point of convergence downstream it is expected that individual tracers

Figure 7.8: Distribution of three tracers at the convergence cross-section, $x_* = 0$ ($I_{\text{mix}} = 0.310$) and at $x_* = 5$ ($I_{\text{mix}} = 0.895$) in Case $\alpha = 45^\circ$.

are well mixed across the compound jet core.

Figure 7.8 shows how the tracers are distributed on the two different cross-sections of a compound jet. The sample points were initially generated from a grid of size 64×64 with random displacement of grid points. At each point, only the tracer with the highest concentration, and this concentration must be larger than a threshold of 2%, are shown. For the first cross-section at $x_* = 0$, there are fewer points associated with the tracer originated from the middle orifice (Tracer number 3). The reason is due to the criterion chosen for the purpose of visualisation: while C_3 must be greater than both C_1 and C_2 to be represented as a dot, each of the latter just need to exceed C_3 to appear on the graph. At the second cross-section, $x_* = 5$, the tracers spread in lateral direction and parts of them blend together (1-3 and 2-3). Also, tracer 3 has clearly emerged and has comparable coverage as tracers 1 and 2.

The clustering of similar tracer points indicates poorly-mixed condition. By noting that the distribution of tracer is continuous in space, it is possible to characterise the mixing condition at a point by the product $C_1 C_2 C_3$. Integration across the whole cross-section (area A) yields the degree of mixing in the jet core at that cross-section. To normalise this quantity so that the well-mixed state corresponds to one and the non-mixing state to zero, the following expression for “index of mixing” is used:

$$I_{\text{mix}} = \frac{\left(\int_A \sqrt[3]{C_1 C_2 C_3} dA \right)^3}{\int_A C_1 dA \cdot \int_A C_2 dA \cdot \int_A C_3 dA}, \quad (7.5)$$

or for discrete sample points $\{(i, j)\}$,

Figure 7.9: Longitudinal distribution of the mixing index for various 3-jet alignments

$$I_{\text{mix}} = \frac{\left(\sum_{i,j} \sqrt[3]{C_1^{i,j} C_2^{i,j} C_3^{i,j}} \right)^3}{\sum_{i,j} C_1^{i,j} \cdot \sum_{i,j} C_2^{i,j} \cdot \sum_{i,j} C_3^{i,j}}.$$

A short computer program (see Appendix B) is written to calculate I_{mix} from the output data. The distribution of I_{mix} along x_* is shown in Figure 7.9, from which the degree of mixing appears to be dependent on α : larger impingement angles provide faster mixing rates. As an example, for $\alpha = 45^\circ$ the mixing process is very active just after the formation of compound jets ($x_* < 2.5 \rightarrow 5$), whereas for $\alpha = 15^\circ$, the mixing takes place in a wide range of x_* up to $x_* > 10$. Apparently there is also dependence of I_{mix} on s_* : wider spacing leads to lower mixing condition. This should be due to the decrease in flow velocity and mixing is less intense.

7.5 Reynolds stresses

Profiles of turbulence intensities are less mentioned in literature, compared to those of axial velocity. There is no closed form of mathematical function describing the former profiles, yet laboratory measurements (e.g. Papanicolaou and List, 1988) shows that the general range for $u_{\text{rms}}^2/U_{CL}^2$ is bounded. These maximum and minimum limits are shown as dashed curves in Figure 7.10. Also in this plot, data points from simulations with different α and x_* are included. Apparently the turbulent level does not depend on α . The orders of magnitude of turbulence levels appear to be similar between simulated and measured values, although no accurate assessment can be made for simulation

Figure 7.10: Turbulent intensity of compound jet for various α compared to that of single jet; the bounds are derived from (Papanicolaou and List, 1988)

results due to the large variation in turbulence intensity profile obtained from laboratory experiments.

Among the Reynold stress components of round jets, $\overline{u'u'}$ is dominant, and is still a useful measure to compare three-jet cases with a single jet. Shear stresses are also considered as correlation quantities characterising the flow condition.

7.6 Summary

The distribution and concentration of emitted tracer has been evaluated for the compound jet flow field. As in the case of a single jet, for a compound jet the distribution of concentration is also similar to the axial velocity distribution, but with a wider spread.

The momentum interaction between component jets in 3-jet alignments helps disperse the tracer effectively in comparison to a single jet case. Three-jet alignments with larger impinging angle yield lower centreline tracer concentration as well as higher dilution ratio.

The effect of s to dilution was examined through analysing the zone of established flow of component jets. An increase of s would lead to an expansion in this zone. As simulation result shows, more favourable dilution can be achieved by increasing the area A confined between the component jets. The relation between the rate of increase in discharge, ΔQ , and A can be approximated as linear.

The inter-mixing condition has been visualised through tracer distribution on every cross-section. It can be seen that shortly after the merging process takes place at convergence point, the compound jet is well-mixed. Larger impingement angle promotes

mixing via momentum exchange; while the port spacing only plays a supplement role with α to affect the mixing process through the starting condition of compound jet flow, at the convergence point.

The dominant Reynolds stress component, $\overline{u'u'}$, is found by integration and compared to the value obtained experimentally (Papanicolaou and List, 1988) for single jets. With larger α ($37.5^\circ \rightarrow 45^\circ$) this stress component comparable in magnitude, while for smaller α the stress is larger than that of single jets.

Chapter 8

Conclusion and recommendations

8.1 Conclusions

The investigation of hydrodynamics and potential mixing of coplanar mutually-impinging jets has reveals some interesting features, as are summarised below.

Dynamics of two converging jets

The direct mutual impact of two submerged, round jets has a direct effect of forming a double shear layer on the transverse plane. The flow in such a shear layer develops radially in the vicinity of the zone of impact, then directs virtually parallel to the x -axis. This double shear layer is characterised by a representative thickness, h , and an (vertically-)averaged velocity, $\mathcal{U}$. The behaviour of this double shear layer has an intermediate form between a plane jet and a radial jet.

By making use of empirical formulae for these types of idealised flow, a set of differential equations describing the double shear layer has been derived. The approximated solution obtained from finite-difference of these equations is then compared to the simulation output in terms of radial flow velocity and the thickness of the double shear layer. It is difficult for this mathematical model to reproduce the increase in thickness of the double shear layer, while the velocity can be reproduced by fine-tuning just one parameter.

It is found that the double shear layer model is appropriate only when α is small. This is revealed through simulations: the case $\alpha = 15^\circ$ produces results which are comparable to the mathematical model, while the cases with $\alpha = 30^\circ$ and 45° do not. A possible explanation is that when α increases, the vigorous flow driven by two component jets forms a backflow and thus alters the general structure of a double shear layer.

Based on the conservation of momentum and mass, an equivalent three-jet system is proposed with the aim to reproduce the flow field similar to that generated by two jets. The simulation output from two systems are compared, and the centreline flow velocities are found to differ by about 10%. Two systems produce double shear layers of similar thickness.

Mixing and dilution in three converging jets

Three coplanar jets form a compound flow field beyond the point of convergence. The cross-sectional distribution of longitudinal velocity typically contains two peaks. The contour of this velocity is elliptical in shape and can be represented by a pair of two principal axes, $d_{\max}$ and $d_{\perp}$. An analysis of these dimensions has been carried out for various orifice configurations with different values of α . When moving further downstream the jet cross-section tends to be more circular.

The spreading rate of the compound jet flow is comparable to that of a single round jet, although both simulation outputs exhibit wider spreading compared to experimental results. A set of linear relationships between the cross-sectional breadth of a single jet with those of three jets for various α 's have been derived.

Unlike the case of a single round jet, for three jets there are usually two velocity (U) peaks on a cross-section of the compound jet flow. Consequently, the eccentricity of the higher peak (or dislocation from the jet centreline) is larger compared to that of a single peak for a single jet. Also, for the 3-jet cases the degree of uncertainty of the peak location is greater.

The rate of spread in tracer concentration profiles is greater than that of velocity profiles, similar to empirical observations from single round jets. Higher dilution rates are associated with larger α , which is valid up to $\alpha = 37.5^\circ$. The entrainment flow rate, on the other hand, is usually larger for cases with smaller α (15°). A simple relationship has been established among the rate of entrainment q_E , the port spacing, s , and the angle α . It shows s has direct (linear) influence on q_E whereas the influence of α is limited.

Numerical simulation of jet flows

For submerged jet simulations, conventional computer programs are often based on an assumption of a time-averaged, steady state flow condition. In recent years, numerical simulations incorporating $k-\epsilon$ or LES turbulence modelling have been increasingly popular. Such type of simulations not only offers the opportunity to investigate the

structure of turbulence but also provides averaged flow quantities for engineering design and applications. The dynamically-adapted quadtree grid of the Gerris solver used in this study allows computing at second-order accuracy the near-field flow behaviour of a single jet, or a momentum interaction between two or three co-planar jets.

This study considered a constant value of viscosity accompanying small-scale eddies that are not resolved by the numerical model. Based on the equation regarding the rate of change in kinetic energy, a bulk value for the viscosity of the system is determined. Usually this numerical viscosity is one order of magnitude greater than the kinetic viscosity. It is therefore possible to specify a negative value for the formal viscosity as a parameter of the model. As a proof-of-concept for this approach, simulations for single round jets of various Re numbers have been carried out. The mean axial velocities are found to be comparable with experimental data by Papanicolaou (1984). Turbulent intensities are of the same order of magnitude with this set of data.

Aside from time-averaged properties of the flow, the velocity fluctuation u' of the shear layer flow has also been considered. The distribution of intermittency factor has been found to be of similar form with that in turbulent boundary layers. The longitudinal double velocity correlations, derived from simulation results, resemble those observed by Grant (1958). The structure of vortices is identifiable as a vortex street downstream near the point of convergence. The structure is complex when looking at a cross-section of the compound jet flow, where counter-rotating vortex pairs manifest.

Beside the advantages, numerical simulation has certain difficulties to reproduce real hydrodynamic conditions. In this particular study, the orifice is preferred to be at the upstream boundary (so that velocity can be prescribed correctly). However, the orifice should be located far from the boundary to avoid the effect of this boundary condition. Currently the author cannot specify “open” boundaries with predefined turbulent characteristics; so this problem would require further study in future.

8.2 Recommendations

This study deals with idealised cases of mutually-impinging submerged jet flows. The velocity distribution at the nozzles are uniform, and buoyancy is not included. The effect of the seabed boundary is neglected. These are far from realistic. However, such simple cases allow convenient comparison between outputs of the numerical solver with experimental data in the literature. The findings in this study also serves as a basis for further investigation on dynamics of interacting shear layers, which might be of interest in contemporary fluid mechanics. For example, the existence of a pair of

velocity maxima in each cross-section of the compound jet (see Section 6.2.2) suggests certain analogy with the counter-rotating vortex pair observed in jets issuing to a crossflow, which has gained interests in research (Fric and Roshko, 1994; Kelso et al., 1996; Cortelezzi and Karagozian, 2001).

The numerical model is readily extended to include practical applications in hydraulic engineering. The effects of fluid density can be verified through simulations with buoyant jets in a stratified ambient waterbody. Various ambient flow conditions (uniform co-flow, cross-flow, oscillating wave motion) can be realised by simulation through changing certain values and formulations. Recently there have been laboratory experiments for jets in oscillating flow (Tam and Li, 2008; Hsiao et al., 2011) as well as in confined waterbodies (Shinneeb et al., 2011).

A contemporary trend in wastewater outfall research is to couple the near field integral jet models with far field hydrodynamic models. An implementation is the CorLink software module (Bleninger, 2006) to connect between the CorMix model and coastal circulation models such as Delft3D. It should then be more natural to combine a numerical submodel to a larger numerical model, as data can be exchanged more conveniently with less need for human interference.

This study deals with hydrodynamic aspects of flow; the results on spreading of conservative tracers are only cursory. An analysis of the Schmidt number derived from simulation output should be compared to known values in the literature, e.g. (Tominaga and Stathopoulos, 2007). In turbulent jet flows, the Schmidt number is typically less than 1, which explains why the spreading extent of concentration is larger than that of momentum. Determining the Schmidt number is difficult though, since contaminants in water usually have very different chemical properties.

Computational technique and numerical simulation to date has certain difficulties in reproducing the actual hydrodynamic conditions at the boundaries. In this particular study, the orifice is preferred and set at the upstream boundary (so that velocity can be prescribed correctly). However, the orifice should also be located far from the boundary to avoid the effect of this boundary condition. As there is no proven and reliable technique in incorporating an “open” boundary with predefined turbulent characteristics; this issue of turbulent open boundary has to be left unresolved.

Regarding the numerical method used in this study, LES proves to be an expensive simulation technique. It doubles the computational time, as in this study, yet the model is not complete. An improvement using dynamic SGS would add complexity, while offering some insights to the distribution of turbulent viscosity in jet flows. Regardless of the SGS technique used, the grid resolution must be fine enough to describe the coherent structures of these jet flows. This is a promising approach, but also a demand

in computing power of the hardware used in research.

A shortcoming of the simulations conducted herein are the short time periods for integration. Physical experiments, e.g. (Papanicolaou and List, 1988) typically last for a few minutes with a sampling rate of 50 Hz. On the contrary, the sampling output are performed at the same timesteps during numerical simulation, which is of order of 10^{-4} s or 10000 Hz, and a shorter integration time, ~ 10 s. This might have affected the quality of output results, as too short a time period would not produce stable values for time averaging. Furthermore, the distribution of a flow quantity would not be smooth in space. (This is related to the ergodic condition.) Therefore, further study should be conducted with longer simulation time to reproduce turbulent flow fields with more realistic statistical properties. This problem, together with the problem of model resolution above, gives rise to the need for larger computing infrastructure.

References

- Abramovich, G. N., 1963. *The Theory of Turbulent Jets*. MIT Press.
- Adamczyk, A. A., Rimai, L., 1988. 2-dimensional particle tracking velocimetry (PTV): Technique and image processing algorithms. *Exp. Fluids* **6**, 373–380.
- Adrian, R. J., 1991. Particle-imaging techniques for experimental fluid mechanics. *Annu. Rev. Fluid Mech.* **23**, 261–304.
- Agbaglah, G., Delaux, S., Fuster, D., Hoepffner, J., Josserand, C., Popinet, S., Ray, P., Scardovelli, R., Zaleski, S., 2011. Parallel simulation of multiphase flows using octree adaptivity and the volume-of-fluid method. *CR Mecanique* **339** (2–3), 194–207.
- Albertson, M., Dai, Y., Jensen, R., Rouse, H., 1950. Diffusion of submerged jets. *T. Am. Soc. Civ. Eng.* **115**, 639–697.
- Ashgriz, N., 2011. Impinging jet atomization. In: Ashgriz, N. (Ed.), *Handbook of Atomization and Sprays*. Springer, pp. 685–707.
- Babu, P., Mahesh, K., 2004. Upstream entrainment in numerical simulations of spatially evolving round jets. *Phys. Fluids* **16** (10), 3699–3705.
- Babu, P., Mahesh, K., 2005. Direct numerical simulation of passive scalar mixing in spatially evolving turbulent round jets. In: 43rd AIAA Aerospace Sciences Meeting, Reno, NV.
- Baines, W., Chu, V., 1996. Jets and plumes. In: Singh, V., Hager, W. (Eds.), *Environmental Hydraulics*. Kluwer Academic, pp. 7–61.
- Bell, J., Colella, P., Glaz, H., 1989. A second-order projection method for the incompressible Navier-Stokes equations. *J. Comput. Phys.* **85** (2), 257–283.
- Beltaos, S., 1976. Oblique impingement of circular turbulent jets. *J. Hydraul. Res.* **14** (1), 17–36.

- Besbes, S., Mhiri, H., Le Palec, G., Bournot, P., 2003. Numerical and experimental study of two turbulent opposed plane jets. *Heat Mass Transfer* **39**, 675–686,
- Bleninger, T., 2006. Coupled 3D hydrodynamic models for submarine outfalls: Environmental hydraulic design and control of multiport diffusers. Ph.D. thesis, University Karlsruhe.
- Boris, J., Grinstein, F., Oran, E., Kolbe, R., 1992. New insights into Large Eddy Simulation. *Fluid Dyn. Res.* **10**, 199–228.
- Borthwick, A. G. L., Marchant, R. D., Copeland, G. J. M., 2000. Adaptive hierarchical grid model of water-borne pollutant dispersion. *Adv. Water Resour.* **23**, 849–865.
- Bush, J., Hasha, A., 2004. On the collision of laminar jets: fluid chains and fishbones. *J. Fluid Mech.* **511**, 285–310.
- Canuto, V., Cheng, Y., 1997. Determination of the Smagorinsky-Lilly constant C_s . *Phys. Fluids* **9** (5), 1368.
- Cederwall, K., 1971. Buoyant slot jets into stagnant or flowing environments, California Institute of Technology.
- Cho, Y.-H., Lee, S.-W., Yoon, W.-S., 2004. Orifice diameter ratio effect on the mixing performances for split triplet injectors. *J. Propul. Power* **20** (1), 69–75.
- Choo, Y., Kang, B., 2003. A study on the velocity characteristics of the liquid elements produced by two impinging jets. *Exp. Fluids* **34**, 655–661.
- Chorin, A., 1968. Numerical solution of the navier-stokes equations. *Math. Comput.* **22**, 745–762.
- Cooper, D., Jackson, D., Launder, B., Liao, G., 1993. Impinging jet studies for turbulence model assessment:—I. Flow-field experiments. *Int. J. Heat Mass Transfer* **36** (10), 2675–2684.
- Copeland, G., Monteiro, T., Couch, S., Borthwick, A., 2003. Water quality in Sepetiba Bay, Brazil. *Mar. Environ. Res.* **55**, 385–408.
- Corrsin, S., 1943. Investigation of flow in an axially symmetrical heated jet of air. Tech. Rep. 3L23, National Advisory Committee for Aeronautics.
- Corrsin, S., 1974. Limitation of gradient transport models in random walks and in turbulence. In: Frenkiel, F., Munn, R. (Eds.), *Turbulent Diffusion in Environmental Pollution*. Academic Press, pp. 25–60.

- Corrsin, S., Kistler, A., 1953. The free-stream boundaries of turbulent flows. Technical Note 3133, John Hopkins University.
- Cortelezzi, L., Karagozian, A., 2001. On the formation of the counter-rotating vortex pair in transverse jets. *J. Fluid Mech.* **446**, 347–373.
- Davidson, P., 2004. *Turbulence: An Introduction for Scientists and Engineers*. Oxford University Press.
- Deardorff, J. W., 1970. A numerical study of three-dimensional turbulent channel flow at large Reynolds numbers. *J. Fluid Mech.* **41**, 453–480.
- Donaldson, C. d., Snedeker, R. S., Margolis, D., 1971. A study of free jet impingement. Part 2. Free jet turbulent structure and impingement heat transfer. *J. Fluid Mech.* **45**, 477–512.
- Durbin, P., Medic, G., 2007. *Fluid Dynamics with a Computational Perspective*. Cambridge University Press.
- Ferziger, J., Perić, M., 2001. *Computational Methods for Fluid Dynamics*, 3rd Edition. Springer.
- Fischer, H., List, J., Koh, C., Imberger, J., Brooks, N., 1979. *Mixing in Inland and Coastal Waters*. Academic Press.
- Fric, T., Roshko, A., 1994. Vortical structure in the wake of a transverse jet. *J. Fluid Mech.* **279**, 1–74.
- Fuster, D., Hoepffner, J., Popinet, S., Zaleski, S., 2011. Parallel simulation of multi-phase flows using the volume-of-fluid method. In: Proceedings of ASME-JSME-KSME Joint Fluid Conference 2011, July 24–29, 2011, Hamamatsu, Japan.
- Gáspár, C., 1996. Flow modelling using quadtrees and multigrid techniques. In: Proc. 9th Int. GAMM Workshop on Parallel Multigrid Methods.
- Gáspár, C., Józsa, J., 1992. Fast adaptive approximation of wind-induced horizontal flow patterns in shallow lakes using quadtree-based multigrid method. In: *Proc. IX Int. Conf. Computational Methods in Water Resources*.
- Germano, M., Piomelli, U., Moin, P., Cabot, W., July 1991. A dynamic subgrid-scale eddy viscosity model. *Phys. Fluids A-Fluid* **3** (7), 1760–1765.

- Gill, G. S., Nurick, W. H., 1976. Liquid rocket engine injectors. Tech. Rep. NASA-SP-8089, NASA Glenn Research Center.
- Goldschmidt, V., Moallemi, M., Oler, J., 1983. Structures and flow reversal in turbulent plane jets. *Phys. Fluids* **26** (2), 428–432.
- Grant, H. L., 1958. The large eddies of turbulent motion. *J. Fluid Mech.* **4** (02), 149–190.
- Greaves, D. M., Borthwick, A. G. L., 1998. On the use of adaptive hierarchical meshes for numerical simulation of separated flows. *Int. J. Numer. Meth. Fl.* **26**, 303–322.
- Guo, Z.-R., Sharp, J., 1996. Characteristics of radial jets and mixing under buoyant conditions. *J. Hydraul. Eng.-ASCE* **122** (9), 495–502.
- Gutmark, E., Wygnanski, I., 1976. The planar turbulent jet. *J. Fluid Mech.* **73** (3), 465–495.
- Hall, C., Maxwell, W., Snorrason, A., 1981. Measurements in merging flow. Tech. Rep. A-103-ILL, University of Illinois, Water Resources Center.
- Hargrave, G., Williams, T., Anandarajah, K., Halliwell, N., 2006. The 3D velocity field of an impacting turbulent jet. *J. Phys. Conf. Ser.* **45**, 162–172.
- Hodgson, J., Moawad, A., Rajaratnam, N., 1999. Concentration field of multiple circular turbulent jets. *J. Hydraul. Res.* **37** (2), 249–256.
- Hsiao, S., Hsu, T., Lin, J., Chang, K., 2011. Mean and turbulence properties of a neutrally buoyant round jet in a wave environment. *J. Waterw. Port C. – ASCE* **137** (3), 109–122.
- Ilyushin, B., Krasinsky, D., 2006. Large eddy simulation of the turbulent round jet dynamics. *Thermophys. Aeromech.* **13** (1), 43–54.
- Jarrin, N., Benhamadouche, S., Laurence, D., Prosser, R., 2006. A synthetic-eddy-method for generating inflow conditions for large-eddy simulations. *Int. J. Heat Fluid Fl.* **27** (4), 585–593.
- Jirka, G., Doneker, R., Barnwell, T., 1991. Cormix: An expert system for mixing zone analysis. *Water Sci. Technol.* **24** (6), 267–274.
- Kang, B., Poulidakos, D., 1996. Holography experiments in a dense high-speed impinging jet spray. *J. Propul. Power* **12** (2), 341–348.

- Kaye, N., Linden, P., 2004. Coalescing axisymmetric turbulent plumes. *J. Fluid Mech.* **502**, 41–63.
- Kelso, R., Lim, T., Perry, A., 1996. An experimental study of round jets in cross-flow. *J. Fluid Mech.* **306**, 111–144.
- Khokhlov, A., 1998. Fully threaded tree algorithms for adaptive refinement fluid dynamics simulations. *J. Comput. Phys.* **143** (2), 519–543.
- Kondo, K., Muramaki, S., Mochida, A., 1997. Generation of velocity fluctuations for inflow boundary condition for LES. *J. Wind Eng. Ind. Aerod.* **67** & **68**, 51–64.
- Kotsovinos, N., 1975. A study of the entrainment and turbulence in a plane buoyant jet. Ph.D. thesis, California Institute of Technology.
- Kraichnan, R. H., 1970. Diffusion by a random velocity field. *Phys. Fluids* **13** (1), 22–31.
- Launder, B., Spalding, D., 1974. The numerical computation of turbulent flows. *Comput. Method Appl. M.* **3** (2), 269–289.
- Lee, J., Chen, G., Kuang, C., 2002. Mixing of a turbulent jet in crossflow—the advected line puff. In: Shen, H., Cheng, A., Wang, K., Teng, M., Liu, C. (Eds.), *Environmental Fluid Mechanics: Theories and Applications*. ASCE Press, pp. 47–84.
- Lee, J., Chu, V., 2003. *Turbulent Jets and Plumes: A Lagrangian Approach*. Kluwer.
- Liang, Q., Borthwick, A. G. L., Stelling, G., 2004. Simulation of dam- and dyke-break hydrodynamics on dynamically adaptive quadtree grids. *Int. J. Numer. Meth. Fl.* **46** (2), 127–162.
- Liang, Q., Du, G., Hall, J. W., Borthwick, A. G. L., 2008. Flood inundation modeling with an adaptive quadtree grid shallow water equation solver. *J. Hydraul. Eng.-ASCE* **134** (11), 1–14.
- Lilly, D., 1966. On the application of the eddy viscosity concept in the inertial sub-range of turbulence. Tech. rep., University Corporation for Atmospheric Research.
- Lin, P., Philip, L.-F. L., 1998. A numerical study of breaking waves in the surf zone. *J. Fluid Mech.* **359**, 239–264.
- Liseth, P., 1976. Wastewater disposal by submerged manifolds. *J. Hydr. Eng. Div.-ASCE* **102** (HY1), 1–14.

- Mallinson, G. D., de Vahl Davis, G., 1972. False diffusion in numerical fluid mechanics. Tech. rep., University of New South Wales.
- Mi, J., Nobes, D., Nathan, G., 2001. Influence of jet exit conditions on the passive scalar field of an axisymmetric free jet. *J. Fluid Mech.* **432**, 91–125.
- Moin, P., Mahesh, K., 1998. Direct numerical simulation: a tool in turbulence research. *Annu. Rev. Fluid Mech.* **30**, 539–578.
- Muppidi, S., Mahesh, K., 2007. Direct numerical simulation of round turbulent jets in crossflow. *J. Fluid Mech.* **574**, 59–84.
- Nielsen, P., 1992. *Coastal Bottom Boundary Layers and Sediment Transport*. World Scientific.
- Ninomiya, N., Kasagi, N., 1993. Turbulence statistics in the self-preserving region of an axisymmetric free jet. *T. JSME Ser. B* **59**, 1532–1538.
- O'Donovan, T., 2005. Fluid flow and heat transfer of an impinging air jet. Ph.D. thesis, University of Dublin.
- Orszag, S., Pao, Y.-H., 1974. Numerical computation of turbulent shear flows. In: Frenkiel, F., Munn, R. (Eds.), *Turbulent Transport in Environmental Pollution*. Academic Press, pp. 225–236.
- Panchapakesan, N. R., Lumley, J. L., 1993. Turbulence measurements in axisymmetric jets of air and helium. Part 1. Air jet. *J. Fluid Mech.* **246**, 197–223.
- Papanicolaou, P., 1984. Mass and momentum transport in a turbulent buoyant vertical axisymmetric jet. Tech. rep., W. M. Keck Laboratory of Hydraulics and Water Resources.
- Papanicolaou, P., List, E., 1988. Investigations of round vertical turbulent buoyant jets. *J. Fluid Mech.* **195**, 341–391.
- Park, K.-Y., Borthwick, A. G. L., 2001. Quadtree grid numerical model of nearshore wave-current interaction. *Coast. Eng.* **42**, 219–239.
- Pera, L., Gebhardt, B., 1975. Laminar plume interactions. *J. Fluid Mech.* **68** (2), 259–271.
- Pope, S., 2000. *Turbulent Flows*. Cambridge University Press.

- Popinet, S., 2003. Gerris: a tree-based adaptive solver for the incompressible Euler equations in complex geometries. *J. Comput. Phys.* **190** (2), 572–600.
- Popinet, S., 2012. Adaptive modelling of long-distance wave propagation and fine-scale flooding during the Tohoku tsunami. *Nat. Hazard. Earth Sys.* **12** (4), 1213–1227.
- Popinet, S., Rickard, G., 2006. A tree-based solver for adaptive ocean engineering. *Ocean Model.* **16** (3–4), 224–249.
- Popinet, S., Smith, M., Stevens, C., 2004. Experimental and numerical study of the turbulence characteristics of air flow around a research vessel. *J. Atmos. Ocean. Tech.* **21** (10), 1574–1589.
- Rajaratnam, N., 1976. *Turbulent Jets*. Elsevier.
- Reichardt, H., 1943. On a new theory of free turbulence. *J. R. Aeronaut. Soc.* **47** (1), 167–176.
- Rider, W. J., 1995. Approximate projection methods for incompressible flow: Implementation, variants and robustness. Tech. rep., LANL Unclassified Report LA-UR-94-2000, Los Alamos National Laboratory.
- Roberts, P., 1979. Line plume and ocean outfall dispersion. *J. Hydr. Eng. Div.-ASCE* **105** (HY4), 313–332.
- Roberts, P., Snyder, W., Baumgartner, D. J., January 1989. Ocean outfalls. I: Submerged wastefield formation. *J. Hydraul. Eng.-ASCE* **115** (1), 1–25.
- Rodi, W., 1993. *Turbulence Models and Their Application in Hydraulics*, 3rd Edition. IAHR.
- Rouse, H., Yih, S., Humphreys, H., 1952. Gravitational convection from a boundary source. *Tellus A* **4** (3), 201–210.
- Rupe, J., 1953. The liquid-phase mixing of a pair of impinging streams. Progress report 20-195, Jet Propulsion Laboratory, Caltech.
- Rupe, J., 1956. A correlation between the dynamic properties of a pair of impinging streams and the uniformity of mixture-ratio distribution in the resulting spray. Progress report 20-209, Jet Propulsion Laboratory, Caltech.

- Samet, H., 1984. The quadtree and related hierarchical data structures. *ACM Comput. Surv.* **16** (2), 187–260.
- Schlichting, H., 1960. *Boundary layer theory*. McGraw-Hill.
- Schwarz, W., 1963. The radial free jet. *Chem. Eng. Sci.* **18**, 779–786.
- Shinneeb, A., Balanchandar, R., Bugg, J., 2011. Confinement effects in shallow-water jets. *J. Hydr. Eng. – ASCE* **137** (3), 300–314.
- Smagorinsky, J., 1963. General circulation experiments with the primitive equations. *Mon. Weather Rev.* **91**, 99–164.
- Smagorinsky, J., 2010. Some historical remarks on the use of nonlinear viscosities. In: B., G., Orszag, S. (Eds.), *Large Eddy Simulation of Complex Engineering and Geophysical Flows*. Cambridge University Press, pp. 3–36.
- Smirnov, A., Shi, S., Celik, I., 2001. Random flow generation technique for large eddy simulations and particle-dynamics modeling. *J. Fluid Eng.-T. ASME* **123** (2), 359–371.
- Srinarayana, N., McBain, G., Armfield, S., Lin, W., 2008. Height and stability of laminar plane fountains in a homogeneous fluid. *Int. J. Heat Mass Transfer* **51**, 4717–4727.
- Staron, L., Lagrée, P.-Y., Popinet, S., 2012. The granular silo as a continuum plastic flow: the hourglass vs the clepsydra. *Phys. Fluids* **24**, 103301.
- Suresh, P. R., Srinivasan, K., Sundararajan, T., Das, S. K., 2008. Reynolds number dependence of plane jet development in the transitional regime. *Phys. Fluids* **20** (4), 044105.
- Tam, B. Y.-F., Li, C.-W., 2008. Flow induced by a turbulent jet under random waves. *J. Hydraul. Res.* **46** (6), 820–829.
- Taylor, G., 1953. Dispersion of soluble matter in a solvent flowing slowly through a tube. *P. Roy. Soc. Lond. A Mat.* **219**, 186–203.
- Tennekes, H., Lumley, J., 1972. *A First Course in Turbulence*. MIT Press.
- Tölke, J., Freudiger, S., Krafczyk, M., 2006. An adaptive scheme using hierarchical grids for lattice Boltzmann multi-phase flow simulations. *Comput. Fluids* **35** (8–9), 820–830.

- Tomar, G., Fuster, D., Zaleski, S., Popinet, S., 2010. Multiscale simulations of primary atomization. *Comput. Fluids* **39** (10), 1864–1874.
- Tominaga, Y., Stathopoulos, T., 2007. Turbulent Schmidt numbers for CFD analysis with various types of flowfield. *Atmos. Environ.* **41**, 8091–8099.
- Townsend, A., 1980. *The Structure of Turbulent Shear Flow*, 2nd Edition. Cambridge University Press.
- Tryggvason, G., Scardovelli, R., Zaleski, S., 2011. *Direct Numerical Simulations of Gas-Liquid Multiphase Flow*. Cambridge University Press.
- Tsai, C.-C., Hou, T.-H., Popinet, S., Chao, Y., 2013. Prediction of waves generated by tropical cyclones with a quadtree-adaptive model. *Coast. Eng.* **77**, 108–119.
- USEPA, 1994. *Water Quality Standards Handbook*, 2nd Edition. US Government, updated March 2012.
- Wang, H., 2000. Investigations of buoyant jet discharges using digital particle image velocimetry (DPIV) and planar laser induced fluorescence (PLIF). Ph.D. thesis, Nanyang Technological University.
- Wang, H., Law, A.-K., 2002. Second-order integral model for a round turbulent buoyant jet. *J. Fluid Mech.* **459**, 397–428.
- Weersink, R., 2010. Experimental study of the oscillatory interaction between two free opposed turbulent round jets. Master's thesis, Siracuse University, NY, USA.
- Weisgraber, T., Liepmann, D., 1998. Turbulent structure during transition to self-similarity in a round jet. *Exp. Fluids* **24**, 210–224.
- Witze, P., Dwyer, H., 1976. The turbulent radial jet. *J. Fluid Mech.* **75** (3), 401–417.
- Wood, I., May 1993. Asymptotic solutions and behavior of outfall plumes. *J. Hydraul. Eng.-ASCE* **119** (5), 555–580.
- Wood, I. R., Bell, R. G., Wilkinson, D. L., 1993. *Ocean Disposal of Wastewater*. World Scientific.
- Wynanski, I., Fiedler, H., 1969. Some measurements in the self-preserving jet. *J. Fluid Mech.* **38** (3), 577–612.
- Yang, T., Shy, S., 2003. The settling velocity of heavy particles in an aqueous near-isotropic turbulence. *Phys. Fluids* **15** (4), 868–880.

- Yiu, K. F. C., Greaves, D. M., Cruz, S., Saalehi, A., Borthwick, A. G. L., 1996. Quadtree grid generation: Information handling, boundary fitting and CFD applications. *Comput. Fluids* **25** (8), 759–769.
- Zheng, J., 2013. Hydraulic characteristics of co-planar jets. Ph.D. thesis, Nanyang Technological University.

Appendix A

Running Gerris

A.1 Syntax of input file

The Gerris computer program reads input from a text file, which contains a series of instructions to specify the geometry, boundary and initial conditions, as well as the requirements for output data. The syntax for these instructions is a mixture between three following formats:

- Command statements, often each starts with a keyword followed by argument values (or expressions), or a collection of parameters in the form {param1=... param2=...}. For example, the first line below sets a parameter specifying the ending time of simulation, while the second line sets an expression for a Dirichlet-type boundary condition:

```
Time {end = 60}

BcDirichlet Uo * cos(alpha)
```

- Assignments of parameters and mathematical expressions written in C programming language syntax, in which the result must be returned:

```
#define VISC 1.0E-6

return (Cs*Cs * cellside*cellside * 1.414*2 * (dudx*d2udxdx
+ 0.25*(dudy+dvdxdx)*(d2udydy+d2vdxdy)
+ 0.25*(dudz+dwdx)*(d2udzdz+d2wdxz)));
```

- Unix shell commands to execute system-related tasks.

In this appendix, only the part specifying computational aspects within the input file are presented; the details relating to output requirements are omitted. Since Gerris input is described using a text file and without a graphical interface, we have to declare:

- Physical quantities of the fluid or bulk quantities of the flow condition: viscosity, diffusivity, and density. For example:

```
SourceViscosity (Uo * d / Re)
```

- Dimension and topology of the flow domain. Topology is relevant in case the domain includes multiple cubes. In our case, for example, two cubic shapes aligning side-by-side represents two control volumes with a common surface, or a link between them. The topology is represented such as: `1 2 right`, meaning that Cube number 1 has Cube number 2 on the right connected to it.
- Boundary conditions of either Dirichlet or Neumann type. Boundary conditions can be prescribed as time-dependant functions.
- Initial condition, which in Gerris includes the condition given at the beginning of a certain time step. For example, since derivative quantities are not intrinsic variables in Gerris, it is necessary to define them at every time step.

A.2 Scenarios

A.2.1 Single, horizontal jet

A simple input file which serves as basis for all jet simulations is listed below; some of these elements are taken from the example case “Atomisation of a liquid jet”.¹ Note that, unlike the modelling convention used in this study, Gerris sets the coordinate origin at the centre of the cube number 1. Therefore the centre of the middle orifice is located at $(-0.5, 0, 0)$ in the numerical model.

¹<http://gerris.dalembert.upmc.fr/gerris/examples/examples/atomisation.html>

```

2 1 GfsSimulation GfsBox GfsGEdge {} {
  Global {
    #define radius 3./512.
    #define Re 10900
    #define Uo 1.0
    #define VISC (2*Uo*radius / Re)
    #define R2(y,z) ((y)*(y) + (z)*(z))
  }
  Time { iend = 10000 }

  Refine (x<-0.4875 && R2(y,z)<2.*radius*radius ? 9 : 7)

  VariableTracer T
  SourceViscosity VISC

  AdaptFunction { istart = 1 istep = 1000 } {
    minlevel = 0    maxlevel = 7
  } (-0.5+12*radius<=x && x<1 && R2(y,z)<0.04*(x+0.5)*(x+0.5))

  AdaptFunction { istart = 1 istep = 1000 } {
    minlevel = 0    maxlevel = 9
  } (x < -0.5+12*radius && 0.2*Uo < U)

  ## Defining cumulative variables, to be mentioned later

  EventBalance { istep = 1 } 0.1
  OutputSimulation {istart=1000 istep=1000} save-%ld {}
  OutputSimulation {start = end} hotstart-%ld { }
}

GfsBox { pid = 0
  top = Boundary
  bottom = Boundary
  back = Boundary
  front = Boundary
  left = Boundary {
    BcDirichlet U (y*y+z*z < radius*radius ? Uo : 0.0 )
    BcDirichlet T (y*y+z*z < radius*radius ? 1.0 : 0.0 )
    BcDirichlet V 0
    BcDirichlet W 0
  }
}

GfsBox { pid = 1
  top = Boundary

```

In the preliminary stage of simulation, a value for viscosity can be chosen based on the decrease of kinetic energy and total vorticity of the system, cf. Eq. (3.2). To track these two flow quantities, it is necessary to include the following lines in the input files:

```
OutputScalarSum {istep=10} kineticLEVEL {v = Velocity2}
OutputScalarSum {istep=10} vorticityLEVEL {v = Vorticity
* Vorticity}
```

Consequently, two files named `kineticLEVEL` and `vorticityLEVEL` are created, which contain time series of $|\mathbf{U}|^2$ and ω^2 , respectively. To calculate the corresponding viscosity, a spreadsheet file is prepared:

	C	D	E	G	H	I	J
1	t	Δt	$ \mathbf{U} ^2$	$0.5 \mathbf{U} ^2 - E_{\text{inp}}$	ω^2	Avg. ω^2	ν_{num}
2	13.0220		1.662650e-3		2.2566e+1		
3	13.0253	0.0033	1.664298e-3	-0.000000824	2.6217e+1	24.3912	-1.0237E-5
4	13.0284	0.0031	1.666251e-3	0.000002923	2.7226e+1	26.7215	3.5287E-5

wherein C3, E3, H3 = obtained from `kineticLEVEL` and `vorticityLEVEL` files

predefined values: U_0 and d

$$E_{\text{inp}} = 0.5 * U_0^2 * \text{PI}() * d^2 * U_0$$

$$D3 = C3 - C2$$

$$G3 = -0.5 * E3 + E_{\text{inp}} * D3$$

$$I3 = 0.5 * (H3 + H2)$$

$$J3 = G3 / (D3 * I3)$$

Turbulence modelling with simple Smagorinsky model can be performed with the addition of the following code to a Gerris input file:

```

# Compute subgrid-scale stresses
Init { istep = 1 } {
  cellside = dL
  dudx = dx("U")
  dudy = dy("U")
  dudz = dz("U")
  dvdx = dx("V")
  dvdy = dy("V")
  dvdz = dz("V")
  dwdx = dx("W")
  dwdy = dy("W")
  dwdz = dz("W")
  d2udxdx = dx("dudx")
  d2udydy = dy("dudy")
  d2udzdz = dz("dudz")
  d2udxdy = dy("dudx")
  d2udxdz = dz("dudx")
  d2vdx dx = dx("dvdx")
  d2vdydy = dy("dvdy")
  d2vdzdz = dz("dvdz")
  d2vdx dy = dy("dvdx")
  d2vdydz = dz("dvdy")
  d2wdx dx = dx("dwdx")
  d2wdydy = dy("dwdy")
  d2wdzdz = dz("dwdz")
  d2wdx dz = dz("dwdx")
  d2wdydz = dz("dwdy")
}

Source U {
  return (Cs * Cs * cellside * cellside * 1.414 * 2 *
    (
      abs(dudx) * d2udxdx
    + 0.25 * abs(dudy + dvdx) * (d2udydy + d2vdx dy)
    + 0.25 * abs(dudz + dwdx) * (d2udzdz + d2wdx dz) ));
}

Source V {
  return (Cs * Cs * cellside * cellside * 1.414 * 2 *
    (
      abs(dvdy) * d2vdydy
    + 0.25 * abs(dudy + dvdx) * (d2udxdy + d2vdx dx)
    + 0.25 * abs(dvdz + dwdy) * (d2vdzdz + d2wdydz) ));
}

Source W {
  return (Cs * Cs * cellside * cellside * 1.414 * 2 *

```


Time integration and averaging are important techniques in turbulence analysis. To achieve this, a common practice is to use cumulative variables to obtain statistical flow quantities in a certain simulation time period. In the following lines of input file, the names SU, SV, SW, ST correspond to $\sum u$, $\sum v$, $\sum w$, $\sum c$ and SU2, SV2, SW2— $\sum u^2$, $\sum v^2$, $\sum w^2$. There can also be sums of products between variables, e.g. SUV, SUW, SVW, SUT for $\sum uv$, $\sum uw$, $\sum vw$, $\sum uc$.

```

EventSum {istart = 8000   istep = 1} U SU
EventSum {istart = 8000   istep = 1} V SV
EventSum {istart = 8000   istep = 1} W SW
EventSum {istart = 8000   istep = 1} U*U SU2
EventSum {istart = 8000   istep = 1} V*V SV2
EventSum {istart = 8000   istep = 1} W*W SW2
EventSum {istart = 8000   istep = 1} U*V SUV
EventSum {istart = 8000   istep = 1} U*W SUW
EventSum {istart = 8000   istep = 1} V*W SVW
EventSum {istart = 8000   istep = 1} T ST

```

A.2.2 Multiple, inclined jets

A simulation with multiple jets would require modification in both the inflow boundary conditions and adaptive meshing specifications. Here a factor k^2 is introduced ($k^2 = \cos^2 \alpha$) to address the modification in geometric shape of the two sideways jet orifices. The elliptical shape at the boundary condition of is given by: $k^2(y - y_o)^2 + z^2 = \frac{1}{4}d^2$, where $y_o = \pm s$ is the location of two orifice centroids.

```

2 1 GfsSimulation GfsBox GfsGEdge {} {
  Global {
    ...
    #define sin_a 0.5
    #define cos_a 0.866
    #define k2 (cos_a * cos_a)
    #define pos1 (-5*radius)
    #define pos2 (5*radius)
    #define R2pos1(y,z) ((k2)*((y)-pos1)*((y)-pos1) + (z)*(z))
    #define R2pos2(y,z) ((k2)*((y)-pos2)*((y)-pos2) + (z)*(z))
    #define R2pos3(y,z) ((y)*(y) + (z)*(z))
    #define SGN(expr) ((expr) > 0 ? 1 : -1 )
  }
  ...
  Refine (x < -0.5 + length && \
    (R2pos1(y,z) < 2.*radius*radius || \
    R2pos2(y,z) < 2.*radius*radius) ? 10 : 7)

  VariableTracer T1
  VariableTracer T2
  VariableTracer T3
  ...
}
GfsBox { pid = 0
  ...
  left = Boundary {
    BcDirichlet U ((radius*radius-R2pos1(y,z)) * \
      (radius*radius-R2pos2(y,z)) < 0 ? \
      Uo * cos_a : ( R2pos3(y,z) < radius*radius ? Uo : 0.0 ) )
    BcDirichlet T1 (R2pos1(y,z) < radius*radius ? 1.0 : 0.0 )
    BcDirichlet T2 (R2pos2(y,z) < radius*radius ? 1.0 : 0.0 )
    BcDirichlet T3 (R2pos3(y,z) < radius*radius ? 1.0 : 0.0 )
    BcDirichlet V (R2pos2(y,z) < radius*radius ? Uo*(-sin_a) : (\
      R2pos1(y,z) < radius*radius ? Uo * sin_a : 0.0 ) )
    BcDirichlet W 0
  }
}
...

```

The radially-aligned sample points on an arbitrary cross-section (with centroid at (X_o, Y_o, Z_o) and normal vector (n_i, n_j, n_k)) can be generated by a script file as follows. In most case a cross section perpendicular to the x -axis has a centroid $(X_o, 0, 0)$ and a normal vector $(1, 0, 0)$. The following computer code is written in Python², a programming language convenient for generating text (ASCII) text files, as well as extracting data from them.

²<http://python.org>

```

# Generate xyz file for points at various cross-sections
# at x/d = varied, as specified in the list x_over_d
# jet centreline is on the plane xy,
# making an angle (ang) with x-axis;
# jet origin is at (x=-0.5 , y=0 , z=0).

import math, sys

x_over_d = range(6,17,2)      # 6, 8, 10, 12, 14, 16
ang = 30                      # input angle, in degrees
sin45 = cos45 = math.cos(math.radians(45))
cos_a = math.cos(math.radians(ang))
sin_a = math.sin(math.radians(ang))
tan_a = math.tan(math.radians(ang))

f = open('output_loca.pts', 'w')

d = 6.0 / 512
yo = -5 * d / 2
dl = 0.005 # output spacing on each axis
num = 7    # number of points on each ray

xcls = [x_ * d * cos_a - 0.5      for x_ in x_over_d ]
ycls = [yo + x_ * d * sin_a      for x_ in x_over_d ]

for (xcl,ycl) in zip(xcls, ycls):
    for y in range(-num, num+1):
        f.write(str(xcl-y*dl*sin_a) + " " +
                 str(ycl+y*dl*cos_a) + " 0\n")
    for z in range(-num, num+1):
        f.write(str(xcl) + " " + str(ycl) + " " +
                 str(z*dl) + "\n")
    for i in range(-num, num+1):
        f.write(str(xcl-i*dl*sin_a*cos45)+" " +
                 str(ycl+i*dl*cos_a*cos45)+" " +
                 str(i*dl*cos_a*sin45)+"\n")
    for j in range(-num, num+1):
        f.write(str(xcl+j*dl*sin_a*cos45)+" " +
                 str(ycl-j*dl*cos_a*cos45)+" " +
                 str(j*dl*cos_a*sin45)+"\n")
f.close()

```

Figure A.1: Comparison in axial velocities between a jet emitted at an angle of 30° (upper half) and a horizontal jet (lower half). (a) Plan view, and (b) cross-sectional view. Contours denote values starting from $0.05U_o$ with increment $0.05U_o$.

A.2.3 A check on the diagonal flow near the orifices

A case with $Re = 1.1 \times 10^4$ is considered. For this study, only the behaviour of the jet part near the orifice was examined. The reasons are two-fold; firstly, the diagonally-aligned jet only exists some distance before impinging to another jet. Secondly, by focusing on this part of the jet, we were able to use uniform, fine grid resolution, so that the false diffusivity can be evaluated: the mean velocity there is just axially-oriented.

The result shows that U_1 , the diagonal velocity component, has less spreading rate compared to U in the case of horizontally-aligned jets. Also, the jet potential core in the first case is longer (Figure A.1). However, the angle of jet centreline approximates the predefined angle of velocity alignment at the inflow boundary. The total discharge at a specific cross-section, however, is similar regardless the value of α . Thus diagonally-aligned jets are acceptable alternatives for normally-aligned jets in that they can produce similar flow conditions.

Appendix B

Postprocessing

B.1 Extraction and transformation of Gerris output data

The data produced by Gerris solver is visualised using a specific program named GfsView. The program, however, does not provide numerical data extraction. One method to complete this task is to write scripts, or short computer program files, to generate a collection of point coordinates in the form of a text file.

Next, a Gerris hotstart file is invoked, and this text file is used to obtain the hydrodynamic variables at corresponding points. Finally the whole output in the form of a tab-separated value (TSV) file (see below) can be imported as a data table in a spreadsheet program.

Sample output file of Gerris displaying flow variables at five sampling points

```
# 1:t 2:x 3:y 4:z 5:P 6:Pmac 7:U ...
38.6625 -0.3125 -0.5 0 0.00662981 -0.102732 0.416865 ...
38.6625 -0.3125 -0.498 0 0.0049301 -0.107194 0.342884 ...
38.6625 -0.3125 -0.494 0 0.00105531 -0.13083 0.18389 ...
38.6625 -0.3125 -0.49 0 -0.000619935 -0.139071 0.069095 ...
38.6625 -0.3125 -0.486 0 0.000358032 -0.111437 0.00553285 ...
```

Each row in this data file contains all flow variables at a specific time instance and at a point in space. A single header row indicates which column is associated with which flow variable. Data filtering and transformation can be carried out using a simple text processing software tool such as Awk.¹ For example, to calculate and print out V_r for all output points on the cross-sectional plane located at $x_* = 50$,

```
awk 'BEGIN {Xo=-0.5; d=6/512.; eps=1E-5}
($2==Xo+50*d) {r = sqrt($3*$3+$4*$4); Vr = ($3*$8+$4*$9)/(r+eps);
print $1,$2,$3,$4,r/d,Vr}'
```

¹Robbins, A. (1996) *Awk Language Programming*, <http://snap.nlc.dcccd.edu/reference/awkref/>

B.1.1 Line integral of flow variables

The integration of a flow variable along the ϕ direction is performed using the Pivot Table feature of the LibreOffice Calc² spreadsheet software.

Input data table:

	A	B	C	D	E	F	G	H
1	x	r_*	SU	SU2	U_*	u_{rms}/U_{CL}	v_{rms}/U_{CL}	$\overline{u'v'}/U_{CL}^2$
2	0.32	0	0.19859	0.0138934	0.06238113	0.12071179	0.22276811	0.00246064
3	0.32	0.85	0.18053	0.0116306	0.06029751	0.14206694	0.21798376	0.00304932
4	0.32	1.71	0.17450	0.0109049	0.06064444	0.14474768	0.24342727	0.01594028

wherein A2, B2, C2, D2 = obtained from filtering the output data file,

E2, F2, G2, H2 = calculated using the formulae presented in Section 3.1.3.

The configuration for pivotting data is

Page Fields		
Column Fields		
r/d	Mean - U*	
	Mean - urms/UCL	
	Mean - vrms/UCL	
	Mean - u'v'/UCL2	
Row Fields	Data Fields	

from which the resulting Pivot Table is

	A	B	C	D	E
1	r/d	Average - U	Average - urms/UCL	Average - vrms/UCL	Average - u'v'/UCL2
2	0	0.0654995681	0.1207117892	0.2227681101	0.0024606376
3	0.85	0.0668025657	0.1294826009	0.2117816061	0.0001235855
4	1.71	0.0685622408	0.144516499	0.2134425663	0.0036826839

B.1.2 Data extraction to determine characteristics of a double shear layer

The characteristic radial velocity, $\mathcal{U}$, and thickness h of a band (Section 4.3.3), can be found from simulation output file. Given an xyz-file named `dsl.pts` which indicates locations of points within the double shear layer (here chosen to conform to the following specification) [total $m \times N$ entries],

²LibreOffice Calc software <http://www.libreoffice.org/features/calc/>, retrieved 15-Jan-2013.

```

r1  y1  0
r1  y2  0
...
r1  ym  0
r2  y1  0
...
rN  y1  0
...
rN  ym  0

```

The time-integrated velocity, again $m \times N$ entries, are obtained from Gerris simulation output file, and listed in a file `U.txt`. The following MATLAB³ code performs interpolation and yield h , thickness of the double shear layer.

```

% reads the radial U values from a text file
% into a vector of length N*m
U = dlmread('U.txt');

% distances to the symmetry plane (normalised with d)
y = [0  0.1667:0.33333:3.5];

m = length(y);      % number of sample points per profile
N = length(U) / m;  % number of comput. point or profiles
Tsim = 5.9786;      % time of integration ( = simulation )
Uo = 1.0;           % inflow velocity

fprintf('CL.Vel.   Rep.Vel.   Thickness \n\n')

for i = 1: m : m*N
    Urep = U(i) ./ sqrt(2);
    b = interp1(U(i:i+m-1), y, Urep, 'cubic');
    h = b * sqrt(pi/2);
    Urep_normalized = Urep / Tsim / Uo;
    Ucl = U(i) / Tsim / Uo;
    fprintf ('%f  %f  %f\n', Ucl, Urep_normalized, h)
end

```

³MATLAB is a registered trademark of The MathWorks, Inc.

B.2 Extraction of data produced by experiments using PIV

The following computer code is intended for extracting PIV experiment data of flow (vector) field, e.g. the dataset in Section ?? . Each vector data file, produced by DaVis software of LaVision BioTec GmbH, contains a header line following by a dataset in four columns, representing the x and y coordinates of the sample grid, and two velocity components in x and y directions.

Format of DaVis data file (100×75 lines of data in total)

```
TITLE = "B00001"
VARIABLES = "x", "y", "Vx", "Vy"
ZONE T="Frame 0", I=100, J=75
-82.312181 64.731484 -0.059466 0.006708
-80.618549 64.731484 -0.060504 0.007565
-78.924917 64.731484 -0.061346 0.007058
-77.231285 64.731484 -0.059764 0.006322
-75.537653 64.731484 -0.058474 0.004317
...
```

```
flowdata = dlmread('B00001.txt', '\t', 1, 0);
x = flowdata(:,1);
y = flowdata(:,2);
u = flowdata(:,3);
v = flowdata(:,4);

quiver(x,y,u,v,80); axis equal

xgrids = linspace(-114.773, 153.557, 100);
ygrids = linspace(-113.861, 86.7091, 75);
[xx yy] = meshgrid(xgrids, ygrids);

mag = sqrt(u.*u + v.*v);
mag = reshape(mag, 75, 100);
figure;
pcolor(xx,yy,mag); shading interp;
```

Appendix C

Experimental data of single jet

This dataset is an adapted version of Papanicolaou (1984) where physical quantities are normalised, for easier comparison with simulation output. Here $U_* = U/U_o$ and similarly for $\max U$, $\min U$. On the other hand, $u_{\text{rms}*} = u_{\text{rms}}/U_{CL}$, $v_{\text{rms}*} = v_{\text{rms}}/U_{CL}$, $\overline{u'v'}_* = \overline{u'v'}/U_{CL}^2$ and $y_* = y/d$. In Papanicolaou's experiments, the probes were arranged on a single transect; thus y is used instead of r .

PAP16						
y_*	U_*	$\max U_*$	$\min U_*$	$u_{\text{rms}*}$	$v_{\text{rms}*}$	$\overline{u'v'}_*$
-5.33	0.008	0.035	-0.028	0.045	0.032	0.0001
-4.	0.028	0.055	-0.028	0.086	0.058	0.0007
-2.67	0.082	0.125	-0.016	0.19	0.111	0.0041
-1.33	0.161	0.166	0.013	0.229	0.138	0.0021
0.	0.215	0.160	0.015	0.255	0.148	0.0023
1.33	0.191	0.154	0.012	0.248	0.134	-0.0018
2.67	0.101	0.108	-0.018	0.196	0.113	-0.0013
4.	0.042	0.069	-0.026	0.132	0.071	-0.0008
5.33	0.019	0.048	-0.024	0.074	0.051	-0.0003

PAP17						
y_*	U_*	$\max U_*$	$\min U_*$	$u_{\text{rms}*}$	$v_{\text{rms}*}$	$\overline{u'v'}_*$
-10.67	0.004	0.027	-0.026	0.029	0.028	0.0001
-8.	0.014	0.089	-0.023	0.105	0.064	-0.0011
-5.33	0.055	0.172	-0.018	0.237	0.121	-0.0066
-2.67	0.097	0.197	0.012	0.245	0.156	-0.009
-1.33	0.119	0.228	0.02	0.247	0.16	-0.0016
0.	0.121	0.213	0.02	0.273	0.167	0.005

1.33	0.114	0.211	0.012	0.255	0.159	0.0091
2.67	0.09	0.187	0.007	0.247	0.147	0.0105
4.	0.059	0.145	−0.009	0.218	0.148	0.0097
6.67	0.021	0.092	−0.037	0.139	0.092	0.0031
8.	0.01	0.078	−0.032	0.08	0.052	0.0012
10.67	0.003	0.032	−0.034	0.032	0.039	−0.0001
12.	0.004	0.037	−0.035	0.025	0.016	0.

PAP20						
y_*	U_*	$\max U_*$	$\min U_*$	$u_{\text{rms}*}$	$v_{\text{rms}*}$	$\overline{u'v'}_*$
−10.67	0.022	0.074	−0.009	0.204	0.131	0.0122
−6.67	0.043	0.095	−0.002	0.253	0.164	0.015
−2.67	0.061	0.113	0.011	0.255	0.174	0.0086
0	0.061	0.116	0.014	0.242	0.174	0.0021
2.67	0.06	0.108	0.008	0.253	0.165	−0.0025
6.67	0.039	0.088	−0.008	0.228	0.158	−0.0071
10.67	0.022	0.071	−0.006	0.196	0.106	−0.0049
14.67	0.014	0.052	−0.014	0.159	0.064	0.0008
18.67	0.004	0.025	−0.015	0.06	0.064	0.0008
22.67	0.002	0.019	−0.015	0.033	0.026	0.

PAP21						
y_*	U_*	$\max U_*$	$\min U_*$	$u_{\text{rms}*}$	$v_{\text{rms}*}$	$\overline{u'v'}_*$
14.67	0.003	0.034	−0.025	0.044	0.029	0.
10.67	0.013	0.081	−0.025	0.131	0.095	0.0036
6.67	0.036	0.123	−0.008	0.211	0.129	0.0063
2.67	0.075	0.143	−0.015	0.254	0.162	0.0098
0	0.086	0.175	0.027	0.233	0.173	−0.0005
−2.67	0.073	0.147	−0.004	0.255	0.168	−0.0042
−5.33	0.046	0.126	−0.028	0.226	0.127	−0.0072
−9.33	0.015	0.075	−0.031	0.16	0.083	−0.0027
−13.33	0.008	0.065	−0.029	0.086	0.051	−0.0012

PAP30						
y_*	U_*	$\max U_*$	$\min U_*$	$u_{\text{rms}*}$	$v_{\text{rms}*}$	$\overline{u'v'}_*$
17.33	0.001	0.008	−0.014	0.03	0.038	0.
13.33	0.013	0.055	−0.006	0.106	0.071	0.0021

9.33	0.032	0.104	−0.009	0.18	0.117	0.007
5.33	0.06	0.123	0.014	0.217	0.144	0.0098
2.67	0.073	0.151	0.018	0.232	0.147	0.0086
0	0.075	0.132	0.026	0.229	0.144	−0.001
−2.67	0.068	0.121	0.017	0.249	0.135	−0.0074
−6.67	0.04	0.105	0.005	0.207	0.131	−0.0091
−10.67	0.02	0.074	−0.003	0.143	0.088	−0.0047
−14.67	0.008	0.045	−0.012	0.098	0.072	−0.0018

PAP32						
y_*	U_*	$\max U_*$	$\min U_*$	$u_{\text{rms}*}$	$v_{\text{rms}*}$	$\overline{u'v'}_*$
9.33	0.001	0.033	−0.061	0.036	0.039	0.0000
6.67	0.015	0.099	−0.039	0.125	0.087	0.0038
4.00	0.053	0.161	−0.016	0.185	0.116	0.0069
1.33	0.101	0.184	0.014	0.207	0.147	0.0072
0.00	0.123	0.221	0.035	0.207	0.15	0.0052
−1.33	0.126	0.218	0.042	0.213	0.152	0.0003
−2.67	0.100	0.217	0.011	0.239	0.136	−0.0064
−5.33	0.054	0.153	−0.002	0.188	0.116	−0.0060
−8.00	0.017	0.080	−0.015	0.101	0.068	−0.0014
−10.67	0.004	0.053	−0.013	0.041	0.038	0.0000

Appendix D

Miscellaneous problems

D.1 Determining the boundary condition for a double shear layer flow

The velocity distribution at the upstream boundary of the double shear layer is found from simulation result with the help of Gnuplot in curve-fitting. The procedure for finding the distribution of the layer thickness and radius of curvature along the edge of the zone of impact, i.e. boundary, is as follows. As a first approximation, both h_i and R_i are assumed constant.

Substitution into the equations for conservation of mass, momentum and energy:

$$2h_i R_i \mathcal{U}_{i,0} \int_0^{\phi_{\text{limit}}} f(\phi) d\phi = \frac{1}{4} \pi d^2 U_o \quad (\text{D.1})$$

$$2h_i R_i \mathcal{U}_{i,0}^2 \int_0^{\phi_{\text{limit}}} f^2(\phi) d\phi = \frac{1}{4} \pi d^2 U_o^2 \quad (\text{D.2})$$

$$2h_i R_i \cdot \frac{1}{2} \mathcal{U}_{i,0}^3 \int_0^{\phi_{\text{limit}}} f^3(\phi) d\phi = \frac{1}{8} \pi d^2 U_o^3 \quad (\text{D.3})$$

where $f(\phi) = \exp(-\phi^2/0.062474)$. Numeric values for the above integrals are:

$$\int_0^{\phi_{\text{limit}}} f(\phi) d\phi = 0.22151, \quad \int_0^{\phi_{\text{limit}}} f^2(\phi) d\phi = 0.15663, \quad \int_0^{\phi_{\text{limit}}} f^3(\phi) d\phi = 0.12789$$

It can be verified that Eqs. (D.1), (D.2), and (D.3) are inconsistent: dividing Eq. (D.2) by (D.1), or (D.3) by (D.2) an inequality $0.2246 \neq 1$ is obtained. The reason for this discrepancy is, aside from that $\mathcal{U}_i$ was approximated, h_i and R_i are assumed to be constants and are therefore moved out of the integrals. With $\mathcal{U}_{i,0}^* = 0.3177$, it is possible to deduce: $h_i R_i = (5.580 \rightarrow 95.76) d^2$. This numerical inconsistency can

be eliminated using a more accurate treatment where h_i and R_i are approximated with polynomials:

$$R_i = (6.6488 + 0.3870\phi)d = g(\phi)d \quad (\text{D.4})$$

$$h_i = (0.710 + 0.1505\phi^2)d = \mathfrak{h}(\phi)d \quad (\text{D.5})$$

upon which the equations for mass, momentum and energy conservation can be simplified to:

$$2\mathcal{U}_{i,0}^* \int_0^{\phi_{\text{limit}}} f(\phi)g(\phi)\mathfrak{h}(\phi)d\phi = \frac{1}{4}\pi \quad (\text{D.6})$$

$$2\mathcal{U}_{i,0}^{*2} \int_0^{\phi_{\text{limit}}} f^2(\phi)g^2(\phi)\mathfrak{h}^2(\phi)d\phi = \frac{1}{4}\pi \quad (\text{D.7})$$

$$2 \cdot \frac{1}{2}\mathcal{U}_{i,0}^{*3} \int_0^{\phi_{\text{limit}}} f^3(\phi)g^3(\phi)\mathfrak{h}^3(\phi)d\phi = \frac{1}{8}\pi \quad (\text{D.8})$$

Now the integrals, calculated by Simpson's formula, are:

$$\int_0^\pi f g \mathfrak{h} d\phi = 1.0636, \quad \int_0^\pi (f g \mathfrak{h})^2 d\phi = 3.5627, \quad \int_0^\pi (f g \mathfrak{h})^3 d\phi = 13.769$$

Substituting these integrals into Eqs. (D.1), (D.2) and (D.3) leads to different inequalities, $0.6758 \neq 0.7854$, $0.7192 \neq 0.7854$, and $0.8830 \neq 0.7854$ respectively. While still presents, the level of inconsistency has decreased. Such discrepancies are due to various mechanisms in transformation of mass, momentum and energy from a round jet to a plane jet, which takes place in the zone of impact. Also, the constraint in angle $-\phi_{\text{limit}} \leq \phi \leq \phi_{\text{limit}}$ leads to the omission of a certain part of the radial flow.

Rewriting Eqs. (4.12), (D.4), and (D.5) in nondimensional form:

$$R_i^* = 6.6488 + 0.3870\phi$$

$$h_i^* = 0.710 + 0.1505\phi^2$$

$$\mathcal{U}_i^* = 0.3177 \exp(-\phi^2/0.06274).$$

D.2 Implementation of the double shear layer model

The following C code yields radial velocity and thickness of the double shear layer at regular interval along the layer. Initial variables required include: number of bands, `jMAX=13`; number of nodes along each band, `iMAX=100`; interval for output display, `iPRN=20`. The bands are approximately 0.03π apart, `parts_of_pi` is a collection of

ϕ/π values for all bands. U_{plus} and h_{plus} are velocity and thickness values to be calculated, i.e. U_{i+1} and h_{i+1} . The other variable names are self-explanatory.

```

for (int j=0; j<jMAX; j++) {
    /* edge of the zone of impact */
    phi[j] = PI * parts_of_pi[j];
    r[j] = 6.6488 + 0.3078*phi[j];
    U[j] = 0.3177*exp(-phi[j]*phi[j]/0.06274);
    h[j] = 0.710 + 0.1505*phi[j]*phi[j];
}
for (int i=0; i<iMAX; i++) {
    for (int j=0; j<jMAX; j++) {
        r[j] += Delta_r;
        Uplus[j] = U[j] + C2*r[j]*U[j]*Delta_r/(h[j]*h[j]);
        hplus[j] = h[j]*(2 - Uplus[j]/U[j]) + C1*Delta_r;
        U[j] = Uplus[j];
        h[j] = hplus[j];
    }
    if (i % iPRN == 0) {
        printf("%d %7.4f %7.4f\n", i, U[0], h[0]);
    }
    /* for phi=0 */
}

```

D.3 Finding the distribution of the velocity peak on a cross-section

All points in each data set shown in Figure 6.8 are sorted ascending and, together with their cumulative frequencies, comprise an empirical cumulative distribution function (CDF). The fitting process was done manually; each CDF (model) was fitted to a corresponding empirical one. The quality of fitness was evaluated through a sum of square errors (SSE). First, a pair of values for the mean and standard deviation was chosen. The mean value is then adjusted so that the midpoint of the model CDF comes close to that of the empirical CDF, to obtain a small value of SSE. Finally the standard deviation was adjusted so that the slope of model CDF approximates that of the empirical CDF (Figure D.1). The values of mean and standard deviation to minimise the SSE were selected.

Figure D.1: Cumulative distribution and fitting normal probability curves to the data representing displacement of axial velocity peak from centroid of cross-section

Table D.1: Sum of square errors in fitting CDF to normal distributions. Values of mean and standard deviation are also shown in Figure 6.9.

α	x_*	Mean r	SD	SSE
1-jet	7.	0.93	0.54	2.17
	12.	1.18	0.57	1.45
	17.	1.79	0.75	1.72
15°	5.7	1.23	0.15	15.1
	9.61	1.67	0.61	2.28
	14.61	2.0	0.8	3.59
30°	7.5	1.1	0.46	0.50
	12.5	1.35	0.64	1.11
	17.5	1.47	0.72	1.09
	22.5	1.92	0.70	1.00
45°	8.56	1.24	0.53	1.57
	13.56	2.12	0.59	2.30
	18.56	2.6	0.67	11.8
	23.56	3.2	0.55	18.7

Figure D.2: (Left) Shaded region in which the maximum is found, in case the known sample maximum, $U_{i,j}$, is located at the centre node. (Right) To locate the maximum value along the j^{th} -grid line

D.4 Interpolating the maximum on a grid of samples

The maximum point is located based on an array of 16×16 sampled points with a spacing of $\frac{1}{128}$ or $\frac{2}{3}d$. The largest value of U found on this array by sequential searching algorithm provides an approximation of the location of the velocity peak. Or formally, on the (y, z) plane, the $\arg \max_{y,z} U(y, z)$ is sought. The process can be summarised as follows:

- If the largest scanned value is located at the edge of the sampling array then the maximum must also lie on the edge, otherwise
- The largest scanned value is located at position (i, j) in the array; then the maximum must locate within the square (with side of $2h$) formed by four grid points $(i-1, j-1)$, $(i-1, j+1)$, $(i+1, j-1)$, $(i+1, j+1)$ (Figure D.2, left). Assume that the location of maximum in one dimension (e.g. along the j^{th} -grid line) is determined by the difference between the largest value found at i and the values of its two neighbours ($i-1$ and $i+1$), specifically (see Figure D.2, right):

$$\frac{a}{b} = \frac{U_i - U_{i-1}}{U_i - U_{i+1}}.$$

Through manipulation the algebraic value for true displacement of the maximum from location i is:

$$d_i = \frac{h}{2} \left(1 - \frac{U_i - U_{i-1}}{U_i - \frac{1}{2}(U_{i-1} + U_{i+1})} \right), \quad (\text{D.9})$$

and similar displacement from location j along the i^{th} -grid line:

$$d_j = \frac{h}{2} \left(1 - \frac{U_j - U_{j-1}}{U_j - \frac{1}{2}(U_{j-1} + U_{j+1})} \right). \quad (\text{D.10})$$

The formulae (D.9) and (D.10) were implemented in a script file written in Python programming language to locate the velocity peak in Section 6.2.3.

```

f1 = open("infile.csv","r")
f2 = open("outfile.csv", "w")

h = 1. / 128          # grid size
ny = nz = 17          # 17x17 grid points
Uo = 2.221

for block in range(4000):
    max = 0
    ym, zm = 0,0
    U = {}             # store matrix of velocity
    for i in range(ny*nz):
        line = f1.readline()
        record = line.split()
        y = float(record[0])
        z = float(record[1])
        U[y,z] = float(record[2])

        if U[y,z] > max:
            ym, zm, max = y, z, U[y,z]

    if -0.0625 < ym < 0.0625 and -0.0625 < zm < 0.0625:
        # max not on the boundary of interrogation window
        Uy1 = U[ym-h, zm]
        Uy2 = U[ym+h, zm]
        Uz1 = U[ym, zm-h]
        Uz2 = U[ym, zm+h]
        # calculate eccentricities
        ey = h * 0.5 * (Uy1 - Uy2) / (2*max - Uy1 - Uy2)
        ez = h * 0.5 * (Uz1 - Uz2) / (2*max - Uz1 - Uz2)
        ym += ey
        zm += ez

    f2.write("%f %f %f\n" % (ym, zm , max/Uo))

f1.close()
f2.close()

```

D.5 Evaluation of intermixing between three jets

Evaluate the mixing capability of three jets (Section 7.4). The script file is written in Python programming language.

```
# This script file calculates Imix

# f : file contains distribution of tracer concentration
#   on the cross-section
# t : time period of simulation
# all sums are initiated as zero

for line in f:
    if line[0] == '#':
        continue
    line = line.split()
    x,y,z = line[1], line[2], line[3]
    c1,c2,c3 = float(line[15])/t, float(line[16])/t, \
               float(line[17])/t

    # No negative concentrations!
    c1 = max((c1,0))
    c2 = max((c2,0))
    c3 = max((c3,0))

    prod = c1*c2*c3

    geom_av = prod**(1./3)
    sum_av += geom_av
    sum1 += c1
    sum2 += c2
    sum3 += c3

print "Imix = ", sum_av**3 / (sum1*sum2*sum3)
```