

CRANFIELD UNIVERSITY

APOSTOLOS SPANELIS

ENGINE EXHAUST PLUME MIXING SIMULATIONS FOR MINIMIZING
ENVIRONMENTAL EMISSIONS IMPACT

SCHOOL OF ENGINEERING
PHD THESIS

PHD IN AEROSPACE ENGINEERING
ACADEMIC YEAR: 2010 - 2013

SUPERVISOR: PROF. MARK SAVILL
OCTOBER, 2013

CRANFIELD UNIVERSITY

SCHOOL OF ENGINEERING

PHD THESIS

PHD IN AEROSPACE ENGINEERING

ACADEMIC YEAR: 2010 - 2013

APOSTOLOS SPANELIS

ENGINE EXHAUST PLUME MIXING SIMULATIONS FOR MINIMIZING
ENVIRONMENTAL EMISSIONS IMPACT

SUPERVISOR: PROF. MARK SAVILL

OCTOBER, 2013

THIS THESIS IS SUBMITTED IN PARTIAL FULFILMENT OF THE REQUIREMENTS
FOR THE DEGREE OF PHD

© CRANFIELD UNIVERSITY 2013. ALL RIGHTS RESERVED. NO PART OF THIS
PUBLICATION MAY BE REPRODUCED WITHOUT THE WRITTEN PERMISSION OF
THE COPYRIGHT OWNER.

Abstract

Local air quality is one of several issues constraining the development of airports. Ongoing research shows that takeoff contributes considerably to the level of near airport pollution. Installation of a wind-brake system behind the runway threshold has been recently proposed as a way to accelerate natural lift-off of the exhaust plumes. This thesis aims to validate large eddy simulations (*LES*) for the numerical investigation of this problem, and contribute to a joint effort towards the installation of baffles in major airports. Ways to limit the enormous cost of such simulations are suggested, and methods to reduce the problem complexity are established by means of parametric analyses and staged cross-validations. In particular, wall-jet simulations are performed alongside wind-tunnel experiments. Results reveal that *LES* accurately predicts the baffle induced drag force, as well as scalar dispersion. One source of weakness in this study is the level of accuracy of the numerical representation of the wind tunnel boundary layer. Lift-off of the plume was not observed within the investigated range (380 nozzle diameters), neither in the absence of baffles, nor in the presence of the basic baffle arrangement. Certain geometrical modifications, however, have shifted the onset of lift-off upstream enough, so that it is identified by the streamwise evolution of mean flow characteristics. An additional achievement of this work was to devise and validate a dynamic response algebraic model for the representation of the baffles in the flow. The established numerical approach, together with the baffle representation model is promising for the numerical investigation of the real scale situation at longer distances, up to 2000 nozzle diameters. This thesis sets a strong basis for the continuation of the research, adding to a growing body of literature regarding local air quality around airports.

To my beloved grandmother Vasiliki

"You will always live inside me"

Acknowledgements

Foremost, I would like to express my sincere gratitude to my supervisor Prof. Mark Savill for the continuous support during my PhD study and research, for his patience, motivation, enthusiasm, and immense knowledge. His guidance helped me in all the time of research and writing of this thesis. I could not have imagined having a better advisor and mentor for my PhD study.

I would also like to thank Prof. Kevin Garry from the department of Aerospace Engineering of Cranfield University, both for the provision of experimental data of the free-round jet and his support during the experiments in the atmospheric boundary layer wind-tunnel, as well as Stefan Velikov for the great cooperation we had during the experiments.

I am indebted to Prof. Paul Tucker from Cambridge University for providing the geometrical details and experimental data of the coaxial jet model, supporting the validation tests of this work.

I would like to express my deepest appreciation to my friends Pamela and Christoforos. Their help has been invaluable in the most critical stages of this PhD.

These acknowledgements would be incomplete without sincere thanks to my mother Nefeli for her support and faith in me during the hardship of my studies, and to my brother Aris and sister Mirsini who encouraged me to achieve the best.

The research project has received funding from the European Union's Seventh Framework Program (FP7/2007-2013) for the Clean Sky Joint Technology Initiative under grant agreement NO CJSU-GAMSGO-2008-001.

Contents

List of Figures	vii
List of Tables	xi
List of Equations	xiv
Nomenclature	xv
1 Introduction - Literature Review	1
1.1 ENVIRONMENTAL CONSIDERATIONS	3
1.1.1 Near Airport Pollution and Effects to Human Health	3
1.1.2 Dispersion of the Pollution	6
1.2 RESEARCH APPROACH	9
1.3 INTRODUCING TURBULENCE	10
1.4 TURBULENT JETS	12
1.4.1 Transition to Turbulence	12
1.4.2 Fully-Developed Unbounded Round Jet	19
1.4.3 Turbulent Wall-Jet	22
1.5 BUOYANT PLUMES AND WIND BRAKES	29
1.5.1 Accounting for Buoyancy	29
1.5.2 Wind Brakes (Baffles)	32
1.6 NUMERICAL SIMULATIONS	34
1.6.1 LES Challenges and Applications	35
1.6.2 Pollutant Dispersion and Mixing Simulations	37
1.7 LITERATURE REVIEW OUTCOME	39
2 Methodology	41
2.1 CENTRAL OBJECTIVE AND APPROACH	42
2.1.1 Modelling the Problem	43
2.1.2 Theoretical Predictions	44
2.1.3 Representation of the Pollution in the Simulation	45
2.1.4 Identification of Lift-off of the Plume	45
2.2 POST-PROCESSING METHODS	46

2.2.1	Axisymmetric Jets	46
2.2.2	Three-Dimensional Offset Jets	46
2.2.3	Final Results	47
2.3	VALIDATION OF THE NUMERICAL APPROACH	48
2.4	HARDWARE-SOFTWARE-SCRIPTING	49
3	Solver Description	51
3.1	SUMMARY OF THE SOLUTION STRATEGY	52
3.2	DISCRETIZATION SCHEMES	54
3.3	FILTERING THE NAVIER-STOKES EQUATIONS	56
3.4	SUB-GRID SCALE STRESS SGS MODELLING	57
3.4.1	Standard Smagorinsky Model (SSM)	57
3.4.2	Dynamic Smagorinsky Model (DSM)	58
3.4.3	Wall-Adapting Local Eddy-Viscosity (WALE)	58
3.5	LES NEAR-WALL TREATMENT	59
3.6	PASSIVE SCALAR EQUATIONS	60
3.7	INLET RANDOM FLOW GENERATION	60
4	Numerical Setup	63
4.1	BOUNDARY AND INITIAL CONDITIONS	64
4.2	GRID GENERATION	67
4.2.1	Axisymmetric Jet Simulation Meshing	67
4.2.2	3DWJ Simulation Meshing	71
4.2.3	Mesh Parameters	74
4.3	MODELLING THE BAFFLES	79
4.3.1	Summary of the Immersed Boundary Method	80
4.3.2	Present Work Model Approximations	82
4.3.3	Present Work Model Implementation	86
5	Theoretical Analysis	93
5.1	MEAN FLOW CHARACTERISTICS	94
5.2	BUOYANT PLUME THEORETICAL PREDICTIONS	97
5.2.1	Natural Lift-off of the Plume	98
5.2.2	Plume Lift-off Under the Baffle Influence	100
5.3	SIMULATED TIME REQUIREMENT (STR)	101
5.3.1	Establishment of a Statistically Steady Flow	102
5.3.2	Statistical Convergence	102
5.3.3	Evaluation of the Statistical Uncertainty	103
5.4	COMPUTATIONAL COST CONSIDERATIONS	104
5.4.1	Typical Factors Influencing Computational cost of LES	104
5.4.2	STR in Computational Cost of Jet LES	105

6	Description of Investigated Setups	111
6.1	SETUPS FOR THE VALIDATION TESTS	112
6.1.1	Single Free Circular Jet in Strong Co-flow	112
6.1.2	High Speed Free Coaxial Jet (Turbofan Model)	114
6.1.3	3DOJ In Strong Co Flow	115
6.2	ABLWT EXPERIMENTS	116
6.2.1	1/100 Scale (<i>Drag Force Measurements</i>)	118
6.2.2	1/200 Scale (<i>Cross Wind Pollution Measurements</i>)	120
6.2.3	1/200 Scale (<i>Final Setup for FID and HWA Measurements</i>)	122
6.2.4	Experimental Equipment	125
7	Validation Tests	
	Part 1: <i>Free-Shear Jets</i>	127
7.1	SINGLE FREE AXISYMMETRIC JET IN CO-FLOW	128
7.1.1	Meshing Approach	128
7.1.2	Convergence Criteria	129
7.1.3	Timestep Size	131
7.1.4	Inlet Turbulence Intensity and Shear layer Resolution	132
7.1.5	Under-Relaxation Factors	134
7.1.6	Temporal Convergence	134
7.1.7	Grid Convergence	135
7.1.8	Pressure Coupling and SGS Modelling	136
7.1.9	Mesh Isotropy in the Far-Field	137
7.1.10	Comparison with previous LES	141
7.1.11	Comparison with Theoretical Trends	142
7.2	HIGH SPEED COAXIAL JET	143
7.2.1	An Advanced Technique to Initialize the Solution	143
7.2.2	Modelling the Nozzle	146
8	Validation Tests	
	Part 2: <i>Wall Bounded Jets</i>	153
8.1	POST PROCESSING METHODS	155
8.1.1	Conventional Post-Processing	155
8.1.2	Advanced Post-Processing	157
8.2	ANALYSIS OF THE RESULTS	163
8.2.1	Study Nr.1: SGS Modelling (H-Type)	163
8.2.2	Study Nr.2: Convergence Criteria	164
8.2.3	Study Nr.3: Non-Dimensional Wall Distance (y^+)	166
8.2.4	Study Nr.4: Temporal Resolution	170
8.2.5	Study Nr.5: Inlet Ground Boundary Layer (H-Type)	170
8.2.6	Study Nr.6: H-Type Mesh Density	172

8.2.7	Study Nr.7: H-Type Vs O-Type Meshing	173
8.2.8	Study Nr.8: O-Type Mesh Density	176
8.2.9	Study Nr.9: Inlet Ground Boundary Layer (O-Type)	177
8.2.10	Study Nr.10: SGS Modelling (O-Type)	178
8.2.11	Studies Nr.11-13: Physical Effects	178
9	Baffle Validation	185
9.1	FIRST SETUP: 1/100 Scale	188
9.2	SECOND SETUP: 1/200 Scale	193
9.3	LIFT-OFF OF THE PLUME	194
10	Results and Analysis	197
10.1	SIMULATION VALIDITY TESTS	199
10.1.1	Temporal Convergence Study	199
10.1.2	Grid Convergence Study	200
10.1.3	Incorporation of Buoyancy to the Numerical Simulation	201
10.1.4	Spectral Analysis of Velocity Time Series	203
10.2	CROSS-VALIDATIONS	205
10.2.1	Unresisted Hot 3DOJ	205
10.2.2	Original Baffle Arrangement	208
10.2.3	Large Baffle Arrangement	210
10.2.4	Threshold Large Baffle Arrangement	212
10.3	EXTENDED NUMERICAL ANALYSIS	214
10.3.1	Evaluation of the Different Baffle Arrangements	214
10.3.2	Varying Wind (co-flow) Velocity on the THRESHOLD Baffle Arrangement	220
10.4	BAFFLE REPRESENTATION METHOD	224
11	Conclusions and Recommendations for Further Work	227
11.1	TOWARDS THE COMPLETE SIMULATIONS	229
11.2	LIMITATION OF CROSS-VALIDATIONS	234
11.3	CURRENT ASSESSMENT AND FUTURE RECOMMENDATIONS	236
	References	239
	APPENDICES	255
A	Free Shear Validation Tests	A-1
B	Wall Bounded Validation Tests: Conventional Post-Processing	B-1
C	Wall Bounded Validation Tests: Advanced Post Processing	C-1

D Final Results:

Streamwise Evolution Plots	D-1
D.1 Temporal Convergence Study	D-2
D.2 Grid Convergence Study	D-6
D.3 Buoyancy Effect Study	D-10
D.4 Hot Unresisted 3DOJ	D-14
D.5 Original Baffle Arrangement	D-18
D.6 Large Baffle Arrangement	D-22
D.7 Large Threshold Baffle Arrangement	D-26
D.8 Co-Flow Effect Study	D-30
D.9 Baffle Arrangement	D-34

E Final Results:

Vertical Profiles	E-1
E.1 Hot Unresisted 3DOJ	E-3
E.2 Original Baffle Arrangement	E-5
E.3 Large Baffle Arrangement	E-7
E.4 Threshold-Large B. Arrangement	E-9

F Final Results:

Axisymmetric-Mirrored Comparisons	F-1
F.1 Designed Vs Modelled Baffles	F-1

G Code and Scripting

	G-1
G.1 FLUENT User Defined Functions	G-1
G.2 Tecplot Post-Processing Script	G-17

List of Figures

1.1	Picture of Aircraft Taxiing.	5
1.2	Heathrow Airport contribution to NO_x concentrations	5
1.3	Early takeoff picture.	7
1.4	Late takeoff picture.	8
1.5	Picture of the trailing vortices of a civil airliner.	8
1.6	Initial instability of the shear layer, and the roll-up mechanism . .	16
1.7	Shadowgraphs of a plane-mixing layer	16
1.8	The vortex pairing mechanism	16
1.9	The leapfrogging phenomenon	17
1.10	Vortex rings leading to alternated pairing	17
1.11	Streamwise structure on a side-view of a jet at $Re = 5500$	17
1.12	Braid region and side jets	18
1.13	Coaxial jet schematic and visualization	18
1.14	Longitudinal cross-section of a round-jet	19
1.15	Plane wall-jet velocity profile	23
1.16	3DWJ velocity profiles	25
1.17	Coherent structure model proposed by (Namgyal & Hall, 2013) .	27
1.18	Sketch of the vertical cross-section of a $3DOJ$	28
1.19	Wind brake concept for enhancing dispersion of engine exhaust at take-off.	32
1.20	Picture of the real scale baffle array.	33
3.1	Iterative Time Advancement Solution Method	53
4.1	spectral synthesizer method	65
4.2	Domain for the axisymmetric jet simulations	66
4.3	Domain for high co-flow velocity $3DWJ$ simulations	66
4.4	Domain boundaries of the $3DWJ$ simulation	66
4.5	Key dimensions of the O-Type mesh cross-section	68
4.6	O-Type mesh improvement	69
4.7	Single circular nozzle O-Type meshing.	70
4.8	Coaxial nozzle O-Type meshing.	70
4.9	Near field mesh simplification.	70
4.10	Pure H-Type meshing for the $3DWJ$ simulation.	73
4.11	Modified O-Type meshing for the $3DWJ$ simulation	73

4.12	Hybrid H-Type/O-Type meshing for the final <i>3DOJ</i> simulations . .	76
4.13	Mesh Dimensions Top View	77
4.14	Mesh Dimensions Side View	77
4.15	Near-Baffle Meshing	78
4.16	DRAMBI Concept	83
4.17	DRAMBI baffle function	84
4.18	DRAMBI FD Concept	85
4.19	<i>DRAMBI</i> logical diagram	90
4.20	Operations of the Dynamic Response Algorithm (<i>DRA</i>).	91
5.1	Theoretical streamwise distribution of U_m	95
5.2	Theoretical estimation of the jet length scales	96
5.3	Hanna's lift-off parameter plotted	99
5.4	Variation of the individual time intervals of <i>STR</i> with <i>FLI</i>	107
5.5	Variation of <i>STR</i> with <i>FLI</i> and variation of the ratio of $T_{11}a_1 + T_{11}a_2$ to the <i>STR</i> with <i>FLI</i>	108
6.1	Free-jet computational domain dimensions and boundary conditions	113
6.2	The short cowl coaxial nozzle (WARJEN)	114
6.3	Schematic of the single circular offset jet in co-flow	115
6.4	Baffle Arrangement on the Balance Platform	116
6.5	Schematic representation of the baffle setup. The length and width of the balance platform are not scalable and relate to the 1/100 scale setup.	117
6.6	Wind profile numerical fit	119
6.7	1/200 scale experimental setup	121
6.8	Cross flow domain modification	121
6.9	Final experimental setup	122
6.10	Calculation of heights and spacing between the additional baffles.	123
7.1	Volume cut of the two meshes of study <i>Far-field Mesh Isotropy</i> . .	139
7.2	Results of farfield mesh isotropy and grid sensitivity	140
7.3	Comparison of theoretical mean flow characteristics with <i>LES</i> . .	143
7.4	Initializing instantaneous velocity fields	145
7.5	Results of study " <i>Modelling the Nozzle</i> "	148
8.1	Conventional post-processing entities	156
8.2	Specification of Φ at the nozzle exit	156
8.3	Visual representation of the advanced post-processing method. . .	158
8.4	Advanced post-processing output explained 1	160
8.5	Advanced post-processing output explained 2	161
8.6	Advanced post-processing output explained 3	162
8.7	Streamwise variation of Courant-number	165

8.8	y^+ distribution	168
8.9	Wall-shear distribution	169
8.10	BL at inlet H-Type	171
8.11	Cross-sectional comparison of four different meshes	174
8.12	BL at inlet O-Type	177
8.13	Effect of physical parameters on iso surfaces of $\Phi = 6.5\%$	183
9.1	Time-series of data of <i>DRAMBI</i>	186
9.2	Control volumes of <i>DRAMBI</i>	187
9.3	Contours of $\bar{U}$ on plane $Y = 0$	189
9.4	Contours of U_{rms} on the plane $Y = 0$	189
9.5	Contours of $\bar{\Phi}$ on the plane $Y = 0$	190
9.6	Contours of Φ_{rms} on the plane $Y = 0$	190
9.7	Iso-surfaces of mean passive scalar concentration $\bar{\Phi} = 5.2\%$. . .	192
9.8	Streamwise evolution of the velocity half-width $z_{1/2}$	196
10.1	Analysis of velocity spectra	204
10.2	<i>UNRESISTED</i> jet, LES VS ABLWT, $\bar{U}$ and $\bar{\Phi}$ at $X/D = 40$	206
10.3	<i>UNRESISTED</i> jet, LES VS ABLWT, $\bar{U}$ and $\bar{\Phi}$ at $X/D = 170$. . .	207
10.4	<i>ORIGINAL</i> baffles, LES VS ABLWT, $\bar{U}$ and $\bar{\Phi}$ at $X/D = 170$. . .	209
10.5	<i>LARGE</i> baffles, LES VS ABLWT, $\bar{U}$ and $\bar{\Phi}$ at $X/D = 170$	211
10.6	<i>THRESH-LARGE</i> baffles, LES VS ABLWT, $\bar{U}$ and $\bar{\Phi}$ at $X/D = 170$	213
10.7	Mirrored comparisons, <i>ORIGINAL</i> VS <i>ADDITIONAL</i> , U and $\bar{U}$. .	218
10.8	Mirrored comparisons, <i>ORIGINAL</i> VS <i>ADDITIONAL</i> , Φ and $\bar{\Phi}$. .	219
10.9	Mirrored comparisons, $U_r = 0.5$ VS $U_r = 0.25$, U and $\bar{U}$	222
10.10	Mirrored comparisons, $U_r = 0.5$ VS $U_r = 0.25$, Φ and $\bar{\Phi}$	223
A.1	Profiles of the study “ <i>Meshing Approach</i> ”.	A-3
A.2	Profiles of the study “ <i>Convergence Criteria</i> ”.	A-5
A.3	Profiles of the study “ <i>Timestep Size</i> ”.	A-7
A.4	Profiles of the study “ <i>Inlet Turbulence Intensity</i> ”.	A-9
A.5	Profiles of the study “ <i>Shear Layer Resolution</i> ”.	A-11
A.6	Profiles of the study “ <i>Temporal Convergence</i> ”.	A-13
A.7	Profiles of the study “ <i>Grid Convergence</i> ”.	A-15
A.8	Profiles of the study “ <i>Pressure Coupling Algorithm</i> ”.	A-17
A.9	Profiles of the study “ <i>SGS Modelling</i> ”.	A-19
A.10	Profiles of the study “ <i>Far-field Mesh-Anisotropy</i> ”.	A-21
A.11	Profiles of the study “ <i>Comparison with Previous LES</i> ”.	A-23
A.12	Profiles of the study “ <i>Modelling the Nozzle</i> ”.	A-25
B.1	Vertical Profiles of study “ <i>H-Type Mesh Density</i> ”	B-3
B.2	Horizontal Profiles of study “ <i>H-Type Mesh Density</i> ”	B-5
B.3	Wall Profiles of study “ <i>H-Type Mesh Density</i> ”	C-7

C.1	Streamwise plots of study “ <i>SGS Modelling (H-Type)</i> ”	C-5
C.2	Streamwise plots of study “ <i>Convergence Criteria</i> ”	C-9
C.3	Streamwise plots of study “ <i>Non-Dimensional Wall Distance (Y^+)</i> ”	C-13
C.4	Streamwise plots of study “ <i>Temporal Resolution</i> ”	C-17
C.5	Streamwise plots of study “ <i>Inlet Ground Boundary Layer (H-Type)</i> ”	C-21
C.6	Streamwise plots of study “ <i>H-Type Mesh Density</i> ”	C-25
C.7	Streamwise plots of study “ <i>H-Type Vs O-Type</i> ”	C-29
C.8	Streamwise plots of study “ <i>O-Type Mesh Density</i> ”	C-33
C.9	Streamwise plots of study “ <i>Inlet Ground Boundary Layer (O-Type)</i> ”	C-37
C.10	Streamwise plots of study “ <i>SGS Modelling (O-Type)</i> ”	C-41
C.11	Streamwise plots of study “ <i>Co-Flow Effect</i> ”	C-45
C.12	Streamwise plots of study “ <i>Wall Effect</i> ”	C-49
C.13	Streamwise plots of study “ <i>Three-Dimensionality Effect</i> ”	C-52
D.1	Streamwise plots of the final results “ <i>temporal convergence study</i> ”	D-5
D.2	Streamwise plots of the final results “ <i>grid convergence</i> ” study.	D-9
D.3	Streamwise plots of the “ <i>buoyancy effect</i> ” study.	D-13
D.4	Streamwise plots of the unresisted jet simulation.	D-17
D.5	Streamwise plots of the “ <i>ORIGINAL</i> ” baffle arrangement.	D-21
D.6	Streamwise plots of the “ <i>LARGE</i> ” baffle arrangement.	D-25
D.7	Streamwise plots of the “ <i>THRESHOLD-LARGE</i> ” baffle arrangement.	D-29
D.8	Streamwise plots of the co-flow effect study.	D-33
D.9	Streamwise plot comparison between different baffle arrangements.	D-37
E.1	Vertical profiles <i>UNRESISTED</i> jet	E-3
E.2	Vertical profiles <i>ORIGINAL</i> baffles designed	E-5
E.3	Vertical profiles <i>ORIGINAL</i> baffles modelled	E-5
E.4	Vertical profiles <i>LARGE</i> baffles designed	E-7
E.5	Vertical profiles <i>LARGE</i> baffles modelled	E-7
E.6	Vertical profiles <i>THRESHOLD-LARGE</i> baffles designed	E-9
E.7	Vertical profiles <i>THRESHOLD-LARGE</i> baffles modelled	E-9
F.1	Mirror, <i>designed</i> VS <i>modelled</i> , <i>ORIGINAL</i> baffles, U and $\overline{U}$	F-2
F.2	Mirror, <i>designed</i> VS <i>modelled</i> , <i>ORIGINAL</i> baffles, Φ and $\overline{\Phi}$	F-3
F.3	Mirror, <i>designed</i> VS <i>modelled</i> , <i>LARGE</i> baffles, U and $\overline{U}$	F-4
F.4	Mirror, <i>designed</i> VS <i>modelled</i> , <i>LARGE</i> baffles, Φ and $\overline{\Phi}$	F-5
F.5	Mirror, <i>designed</i> VS <i>modelled</i> , <i>THRESH-LARGE</i> baffles, U and $\overline{U}$	F-6
F.6	Mirror, <i>designed</i> VS <i>modelled</i> , <i>THRESH-LARGE</i> baffles, Φ and $\overline{\Phi}$	F-7

List of Tables

2.1	Astral node configuration details.	50
6.1	Details of the 1/100 scale baffle arrangement	119
6.2	Geometrical details of the <i>ORIGINAL</i> baffle arrangement	124
6.3	Geometrical details of the <i>LARGE</i> baffle arrangement.	124
6.4	Geometrical details of the <i>THRESHOLD</i> baffle arrangements.	124
7.1	Simulations of Study “ <i>Convergence Criteria</i> ”	130
7.2	Simulations of Study “ <i>Timestep Size</i> ”	131
7.3	Simulations of Study “ <i>Inlet Turbulence Intensity</i> ”	133
7.4	Simulations of Study “ <i>Modelling the Nozzle</i> ”	146
8.1	Simulations-studies of the wall bounded validation tests	154
8.2	Parametrization of the cross sectional cell distribution	175
9.1	Simulations of the baffle validation	187
9.2	Drag forces Experiment Vs <i>LES</i>	188
9.3	Comparison of maximum local concentrations	193
10.1	Summary of the setup and history of the final simulations.	198

List of Equations

1.1	20
1.1	20
1.2	21
1.3	21
1.4	24
1.5	30
1.6	30
1.7	30
1.8	31
1.9	31
1.10	31
1.11	32
1.12	32
3.1	54
3.2	54
3.3	54
3.4	55
3.5	55
3.6	55
3.7	56
3.8	56
3.9	56
3.10	56
3.11	56
3.12	57
3.13	57
3.14	57
3.15	57
3.16	57
3.17	59
3.18	59
3.19	60
3.20	60
4.1	81

4.2	81
4.3	82
4.4	86
4.5	87
4.6	87
4.7	87
4.8	87
4.9	88
4.10	88
4.11	88
4.12	88
4.13	88
4.14	89
5.1	94
5.2	94
5.3	95
5.4	96
5.5	101
5.6	102
5.7	103
5.8	103
5.9	104
6.1	118
8.1	166
8.1	171
9.1	189

Nomenclature

Acronyms

Symbol	Description
<i>3DWJ</i>	Three Dimensional Wall Jet
<i>3DOJ</i>	Three Dimensional Offset Jet
<i>ABL</i>	Atmospheric Boundary Layer
<i>ABLWT</i>	Atmospheric Boundary Layer Wind Tunnel
<i>DRA</i>	Dynamic Response Algorithm
<i>DRAMBI</i>	Dynamic Response Algebraic Model for Baffle Imitation
<i>DSM</i>	Dynamic Smagorinsky Model
<i>FAAM</i>	Facility for Airborne Atmospheric Measurements
<i>FID</i>	Flame Ionization Detector
<i>FLI</i>	Far-field Location of Interest
<i>HWA</i>	Hot Wire Anemometer
<i>IATA</i>	International Air Transport Association
<i>ICAO</i>	International Civil Aviation Organisation
<i>DKEM</i>	Dynamic Kinetic Energy Model
<i>LDA</i>	Laser Doppler Anemometry
<i>LES</i>	Large Eddy Simulation
<i>LIDAR</i>	Light Detection And Ranging
<i>PIV</i>	Particle Image Velocimetry
<i>SGS</i>	SubGrid Scale
<i>SSM</i>	Standard Smagorinsky Model
<i>STR</i>	Simulated Time Requirement
<i>WALE</i>	Wall-Adapting Local Eddy-viscosity

Lattin Letters

Symbol	Units	Description
a_i	—	Cell's Weighting Factor for Momentum
b_i	—	Cell's Weighting Factor for Baffle Shape
h	m	Nozzle Offset and Baffle Height
i	—	Spatial Index
n	—	Temporal Index
t	s	Time
Δt	s	Time Step Size
$x, X \ y, Y \ z, Z$	m	Cartesian Coordinates
$y_{1/2}$	m	Horizontal Velocity Half-Width (calculated at the offset $Z = z_{1/2}$)
$y_{1/2\Phi}$	m	Horizontal Concentration Half-Width (calculated at the offset $Z = z_{1/2}$)
$z_{1/2}$	m	Vertical Velocity Half-Width
$z_{1/2\Phi}$	m	Vertical Concentration Half-Width
AR_{2D}	—	Two Dimensional Aspect Ratio
D	m	Nozzle Diameter
F_B	N/s	Buoyancy Flux
F_D	N	Baffle Induced Drag Force
F_L	N	Baffle Induced Lift Force
F_S	N	Baffle Induced Side Force
F_M	N	Engine Thrust
ΔF	N	DRA Force-Step Size
ΔF_o	N	DRA Base Force-Step Size
I	—	Turbulence Intensity
L_{11}	m	Longitudinal Integral Length Scale
L_2	m	Lateral Integral Length Scale
L_T	m	Threshold Distance

Symbol	Units	Description
R	deg	Radius of a Baffle Array
R, r	m	Radial Coordinate
R'	—	Baffle Non-Dimensional Radial Coordinate
Re_j	—	Jet Exit Reynolds Number
S_D	N/m^3	X-Momentum Source Term for Drag
S_L	N/m^3	Z-Momentum Source Term for Lift
S_S	N/m^3	Y-Momentum Source Term for Side Force
T_{11}	m	Longitudinal Integral Time Scale
T_a	K	Ambient Flow Temperature
T_j	K	Jet Temperature at the Nozzle Exit
$\bar{U}$	m/s	X Component of the Mean Velocity Vector
$U_{0.5}$	m/s	Half mixing Velocity = $U_m/2$
U_b	m/s	Jet Bulk Velocity
U_f	m/s	Co-flow Velocity
U_m, U_{max}	m/s	Maximum value of $\bar{U}$ at Constant X
U_{mAVG}	m/s	Average Centreline or Maximum Velocity on Selected Range
U_r	m/s	Wind Profile Control Velocity
U_{rms}	m/s	X Component of the <i>RMS</i> Velocity Vector
V_A	m^3	DRAMBI Active Volume
V_C	m^3	DRAMBI Actual Control Volume
V_{C_0}	m^3	DRAMBI Ideal Control Volume
V_i	m^3	Cell Volume
X_L	m	Lift-off Onset
X'_L	m	Lift-off Onset Under the Baffle Influence
Z'	—	Baffle Non-Dimensional Vertical Coordinate
Z_r	m	Wind Profile Control height

Greek Letters

Symbol	Units	Description
α	<i>deg</i>	Baffle Angle of Incidence
$\alpha_{\Delta F}$	—	DRA Asymptotic Function
λ	—	DRA Slope of $\alpha_{\Delta F}$
ρ_a	<i>kg/m³</i>	Ambient Flow Density
ρ_j	<i>kg/m³</i>	Jet Density at the Nozzle Exit
σ_s	—	DRA Down-Slope Smoother
τ	<i>s</i>	Plume Travel Time
τ'	<i>s</i>	Plume Travel Time Under the Baffle Influence
ϕ	<i>deg</i>	DRAMBI Azimuthal Coordinate
$\overline{\Phi}, \Phi_{mean}$	—	Mean Passive Scalar Concentration
$\Phi_{0.5}$	—	Half Mixing Concentration = $U_m/2$
Φ	—	Local Mean Passive Scalar Concentration
Φ_m, Φ_{max}	—	Maximum Value of $\overline{\Phi}$ at Constant X
Φ_{rms}	—	<i>RMS</i> Passive Scalar Concentration
Ψ	—	DRA Active Volume Ratio
Ψ_{ID}	—	DRA Slope Identifier of Ψ

Chapter 1

Introduction - Literature Review

Contemporary civil aviation has become a main competitor in civil transportation. More than 2.4 billion passengers travelled by aircraft in 2010 while this number is expected to increase up to 16 billion until 2050 according to IATA (Giovanni, 2011). This gradual increase in air travel demand necessitates maximization of the utility of airports within their existing energy and land usage. However, the use of the turbofan engine, which is the prime mover in modern civil aviation, is linked with the generation of certain chemical products related to human health and welfare impacts (Waitz *et al.*, 2004). It is, therefore, important to improve our understanding of the physics governing the dispersion of the pollution produced by the aircraft engine in operations which take place in proximity to central airports.

Growth of interest in this direction dates back to 2001 when *ACARE* (the Advisory Council for Aeronautics Research in Europe) set a strategic research agenda to give a roadmap for aeronautical technology developments leading to the establishment of large initiatives for air traffic management research, such as the Clean Sky project which funds this research. A precursor of this work has been OMEGA (MMU, 2013), an academic program of 40 technology projects that aim to see how the aviation system and the environment are linked. In this respect, (Poll *et al.*, 2009) explained the role played by the evolving jet engine efflux in the dispersion of pollutants, and (Dinesh *et al.*, 2009) carried out relevant *LES* investigation using an in-house solver. A certain simulation setup of

Dinesh, which is supported by experimental data from (Garry & Holt, 2008), is repeated in this work during the solver assessment studies, allowing for multiple cross-validations. During the same period, numerical investigations into the dispersion of aircraft jets in co-flowing takeoff conditions were being carried out by (Aloysius *et al.*, 2007; Aloysius & Wrobel, 2009b; Aloysius & Wrobel, 2009a), the latter performing *LES* of a complete Boeing 737 geometry during takeoff, investigating emission dispersion. Finally, field-trial investigation of a method reducing the local air quality impact of the exhaust jets is in progress (Bennett *et al.*, 2013), supported by experiments in Cranfield University (Velikov, 2013). Large part of the present research is carried out alongside, in a complementary manner with the experiments of the latter.

The current chapter includes a detailed presentation of the scope and objectives of the research, as well as the relevant literature review. In particular, the problem motivating this research and its sources are presented in section 1.1. The selected approach to investigate this problem is then justified in section 1.2. Then, a major component of the problem, turbulence, is introduced in section 1.3. In the following, section 1.4 explains the turbulent jet dynamics through an extensive literature review expanding at the various stages of development of different types of jets. Section 1.5 introduces the concept of buoyancy, and explains the philosophy behind the installation of a wind brake system with respect to the acceleration of lift-off of the plume. Section 1.6 summarizes modern *CFD* techniques available to model turbulence, also referring to previous numerical investigations which have focused to the jet dynamics and mixing. Finally, the last section includes a concluding summary of the literature review carried out in this chapter, explaining how the collected information is used in the present investigation.

1.1 ENVIRONMENTAL CONSIDERATIONS

The main target of this work is the improvement of local air quality in proximity to central airports. The present section discusses the sources of the problem and explains the effects to human health. Furthermore, the dispersion of the pollution as it occurs in different stages of the flight is discussed.

1.1.1 Near Airport Pollution and Effects to Human Health

There are two major causes for the deterioration of air quality in proximity to central airports. One is the increasing level of the automotive traffic and of subsequent vehicle generated pollution in the surrounding area, and the other is the pollution produced in the airport (Brogaard *et al.*, 2012). Regulators have long been aware of the vehicle generated pollution and countless legislative policies have been designed to reduce harmful emissions from these sources (Auffhammer & Kellogg, 2011). However, aircraft and airport emissions have only recently become the subject of legislation, although little has been done to reduce or manage emissions generated by airports and air travel. Some effort has been put to curtail the substantial CO_2 emissions generated by aircraft, but there has been relatively little effort to control or contain some of the more pernicious air pollutants generated by jet engines (Schlenker & Walker, 2011).

The turbofan engine is the prime mover for commercial aircraft operations carried out in the civil airline industry. This device follows a high work cycle that begins with a large amount of air intake which undertakes successive compression, heat by the consumption of fuel, expansion in turbines, and is finally released back into the atmosphere. The injection and burning of fuel implies that several chemical reactions take place in the engine cycle. Because of this procedure, the chemical composition of the air that returns to the environment is different from the incoming ambient air, including chemical products such as Nitrogen Oxides (NO_x), Carbon Monoxide (CO), Sulphur Dioxide (SO_2), Unburned Hydrocarbons (UHC), Particulate Matter (PM), and more.

As explained by the UK department for environment food and rural affairs,

exposure at elevated levels and/or long term exposure to air pollution can lead to serious symptoms and conditions affecting human health. For instance:

SO_2 and NO_2 irritate the airways of the lungs, increasing the symptoms of those suffering from lung diseases.

Fine particles can be carried deep into the lungs where they can cause inflammation and a worsening of heart and lung diseases.

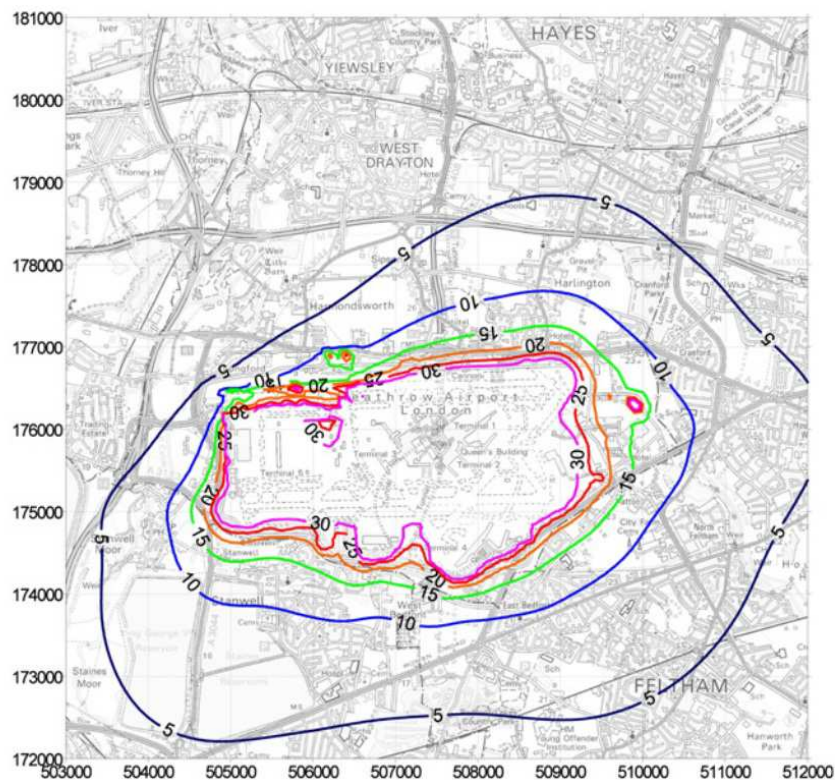
CO prevents the uptake of oxygen by the blood which can possibly lead to a significant reduction in the supply of oxygen to the heart, particularly in people suffering from heart disease

Additionally, (Schlenker & Walker, 2011) suggested that airport-driven CO exposure increases hospitalization rates for asthma, respiratory, and heart related emergency room admissions that are an order of magnitude larger than conventional CO dose-response estimates.

Current EU limit for NO_2 and for coarse particles is $40 \mu g/m^3$, and the limit value for SO_2 is $125 \mu g/m^3$ on a 24 hour average. From 2015 a limit for fine particles is set to $25 \mu g/m^3$, while there is no limit set yet for ultra fine and nano-particles (Brogaard *et al.*, 2012). Measurements made during the Project for the Sustainable Development of Heathrow (DFT, 2006), showed that this area already breaches the current EU limit on annual NO_2 concentrations. The situation would be worse, however, were it not for the natural rise of the aircraft emissions due to buoyancy. Simulations for air quality around Schiphol, showed that “for a mean emission height of 15m, most (59%) of the local NO_2 impact arises from the takeoff mode. Decreasing this height to 5m more than doubles the impact; increasing it to 40m reduces the impact by more than a factor of 3” as discussed and further extended by (Bennett *et al.*, 2013). (Fig.1.1) shows a typical taxiing queue, which reflects the increasing demand of airports, and figure (Fig.1.2) illustrates contours of 2008-2009 period-mean NO_X concentrations at Heathrow airport, taken from recent studies on “Heathrow Air Quality” (HAQ, 2011).



Figure 1.1: Picture of Aircraft Taxiing. (Online, 2013)



Reproduced from Ordnance Survey digital map data © Crown copyright 2009. All rights reserved.

Figure 1.2: Heathrow Airport contribution to 2008/9 period-mean NO_x concentrations (contours shown for $5 \mu g/m^3$ to $30 \mu g/m^3$). (HAQ, 2011)

1.1.2 Dispersion of the Pollution

The pollution produced in the GT engine is released to the atmosphere in the form of a high speed jet. The physics governing such a flow cause the jet to mix with the ambient fluid, and its initially high pollution concentration to continuously drop “*disperse*”. It is important to introduce the term “*plume*” which refers to the condition of the jet where the gas physical properties vary smoothly with increasing distance from the jet centreline and where the bulk flow has established growth characteristics that exhibit self-similarity when normalized with the appropriate variables (Poll *et al.*, 2009). In the context of pollutant dispersion, the flow conditions wherein the plume evolves are of paramount importance and these conditions depend highly on the phase of the flight. The three phases of flight to be discussed in this section are the *cruise*, *take-off*, and *landing*.

DURING THE CRUISE, the jet efflux passes from two different flow regions. The first region extends from the exit of the nozzle to a distance of approximately two wingspans downstream. Here, the jet flow is closely aligned with the direction of flight and the mixing rate is mainly determined by the temperature and velocity differences across the shear layers. Along the second region, the plume is captured in the dynamics of the two wing-tip vortices trailing behind the aircraft. As a result, the exhaust plume is fed directly into the core of the vortices and further development of the jet follows a completely different pattern than its freely decaying counterpart (Ganier *et al.*, 1997a). There is also a physical chemistry importance within this flow region and the procedure of ice crystal and aerosol formation needs to be better understood.

EARLY TAKEOFF is a dynamic phase of the flight since the relative velocity between the aeroplane, the ground, and the plume varies continuously. During this stage, the engines are at almost full power and pollution is, therefore, generated at the maximum rate. A good understanding of the plume behaviour is important in this phase. At the very beginning of takeoff, the aircraft is stationary and the engines produce maximum thrust; consequently, the engine efflux develops in stationary air and advances a considerable distance downstream. It



Figure 1.3: A civil airliner lines up on the runway before taking-off, at Blagnac International Airport in Toulouse, France. (AIRBUS, 2014).

is subject to the atmospheric conditions when the buoyant forces will break the jet away from the surface and start raising it together with the pollution. (Fig.1.3) shows a civil airliner while it lines up on the runway before taking-off. As the aircraft gradually gains forward speed, the relative velocity between the jet and the ground decreases and the mixing rate of the jet is gradually reduced because of the continuous drop of the velocity difference that creates the shear layer.

LAST STAGES OF TAKEOFF AND LANDING, the velocity of the aircraft is adequately high so that strong wing tip vortices form behind the aircraft. Depending on the aircraft geometry these vortices may capture the plumes and as they descend they take the pollution down to the ground with them. (Fig.1.4) shows a civil airliner during the last takeoff stage, and (Fig.1.5) illustrates a real picture of the vortices captured at a low level flight of a civil air transport aircraft. The strength of the wing-tip vortices determines the amount of pollution driven back to the ground. It is anticipated that accurate prediction of the trajectories that these vortical structures follow is possible (Ganier *et al.*, 1997b). (Poll *et al.*, 2009) suggest that such prediction would be valuable in the context of planning aircraft move-

ments at busy airports. Additionally, it could further become part of a strategy to comply with future regulations limiting air-transport-generated substances with harmful consequences to human health.



Figure 1.4: Last stage of take off of an AIRBUS A380, at Blagnac International Airport in Toulouse, France. (AIRBUS, 2014).



Figure 1.5: Picture of the trailing vortices of a civil airliner during landing. (Online, 2011).

1.2 RESEARCH APPROACH

Applying measurements in the wakes of jet engines at operational airports is a very difficult task to undertake. However, noteworthy is the effort of (Bennett *et al.*, 2013) who use Light Detection And Ranging (*LIDAR*) instrumentation while gathering a large body of data of real scale measurements.. Due to the high complexity and the low level of controllability of the environments where *LIDAR* measurements are applied, it is very difficult to use such data to develop detailed models and to validate global theories.

Alternatively, a scaled version of the physical situations of interest can be artificially set-up in an experimental environment. In this way, the problem variables can be changed independently of one another and detailed and accurate measurements can be made. An important variable of our investigations is the natural wind affecting the plume development. This condition can be reproduced in an atmospheric boundary layer wind tunnel (*ABLWT*). Such a facility exists in Cranfield University and experiments to investigate the “*lift off*” of the plume under the influence of a wind-brake system mounted at some distance behind the engine are performed by (Velikov, 2013). A comprehensive study of the situation under investigation requires certain variables to be changed independently. The fulfilment of these objectives requires a large number of realizations for every experiment, which is of course, followed by an increase in the total cost of the investigation. Additionally, the space and time precision, and the level of accuracy of the experimental measurements, bring about an exponential increase in the cost of the experiment itself.

On the other hand, accuracy in such measurements is important, especially concerning future legislation to be applied to the airports. The solution comes with the application of (*CFD*), reducing the investigation cost and providing with information that would be difficult/expensive, or even impossible otherwise. The main challenge of the *CFD* approach is turbulence modelling. In particular, Reynolds Averaged Navier Stokes (*RANS*) and its unsteady counterpart (*URANS*), have been found inappropriate for simulations where turbulent mixing and pollutant

dispersion is of high importance (Salim *et al.*, 2011). On the other hand, direct calculation of the full set of the non-linear terms of the momentum equations in both space and time, the so called Direct Numerical Simulations (*DNS*), requires enormous computational power and storage memory. However, a special *CFD* approach named Large Eddy Simulation (*LES*) has made it possible to separate the turbulent motions into those highly responsible for mixing (large scales easy to calculate), and those with very small contribution to it (small scales with high computational demand). Although *LES* meets the research requirements, also being afforded by the available computational resources, its predictions depend in a range of factors such as the quality of the computational mesh, the boundary conditions, the solution convergence, and more. Therefore, wind-tunnel measurements are required for the parametric validation of the *CFD* solver.

1.3 INTRODUCING TURBULENCE

The physical situation motivating this work has been presented in section 1.1. The leading element of the problem is “*turbulence*” which causes the engine efflux to mix with the ambient fluid. For this reason, the current section aims to introduce the subject of turbulence and its basic properties.

Turbulence is one of the most challenging subjects of the classical mechanics. Numerous studies and experiments have been carried out during the last century aiming to help us improve our understanding of turbulence; however, as (Davidson, 2004) stresses, “*even though the governing equations describing turbulence have been known since 1845, there is still surprisingly little we can predict with relative certainty*”. More specifically, the non-linearity of the Navier-Stokes equations (Foias *et al.*, 2001) results in a chaotic behaviour, which makes the analytical definition of even relatively simple real-world turbulent flows, impossible to be achieved. Alternatively, turbulence can be accurately predicted in terms of its statistical properties at most engineering applications. Numerous studies have been carried out based on that; while, this is the basis for most turbulence models utilized in modern *CFD* simulations.

Either way someone decides to observe turbulence, the beginning of this chaotic procedure strongly relies on the boundary conditions of the flow considered. According to (Lesieur, 2007), “turbulence can only develop in rotational flows”. The existence of shear is a necessary condition for the infinitesimal perturbations, which always exist in real flows, to magnify and finally degenerate into coherent vortices. These vortices are not only the starting point towards turbulence, but also the largest motions encountered in a turbulent spectrum. Since the time and space scales of these motions highly rely on boundary conditions, the assumptions of isotropy and universality are not valid to describe them, and this is a major issue when attempting to model turbulence in mixing layers.

However, under a series of non-linear interactions between the primary eddies, named the “*vortex stretching*”, new vortical motions develop. These new-born motions are smaller than their predecessors in size, and higher in frequency. This procedure continues for successively smaller scales; while it happens, the universality and isotropy of these scales increases. A very enlightening example of vortex stretching and the small-scale isotropy is presented by (Bradshaw, 1975) in his book “*An Introduction to Turbulence and its Measurement*”. According to Kolmogorov, there is a certain eddy size for every flow, at which vortex stretching cannot further occur, reason being the balance between viscous and inertial forces acting on the fluid. Therefore, the Reynolds number of the flow under consideration is deterministic for the scale at which the cascade of the energy-decay will stop and thermal energy will be generated instead. Detailed presentation of the scales of turbulence and the Kolmogorov hypotheses is given by (Pope, 2000).

1.4 TURBULENT JETS

Amongst a wide range of engineering applications, turbulent jets represent one of the most important components of turbo-machinery propulsion systems. In order to interpret the results of this thesis, it is important to have a good understanding of the dynamics governing the jet flow. The current section highlights the fluid dynamics which occur at different stages of the jet development.

1.4.1 Transition to Turbulence

To start with, we recall the necessity of shear for the generation of any coherent structure, as highlighted in chapter 1.3. Shear-flows can be classified into two main families; the wall bounded flows, and the free-shear flows. Free jets are representatives of the later family. The presentation of the theory is made through progressively more complex problems, extending from the relatively simple “*plain mixing layer*”, to its three dimensional counterpart “*round jet*”, and finally the more complicated “*coaxial jet*”.

PLAIN MIXING LAYER: In free-shear flows, viscosity plays a minor role in the development of the primary instabilities; characterized for this reason as “inviscid instabilities”, while they are also much more vigorous than the viscous ones (Chiang, 2007; Schmid & Henningson, 2001). The linear stability theory is found to be extremely difficult to succeed predicting turbulent flows. This is because in real flows there is no steady velocity field upon which a small perturbation may be superimposed (Cohen & Wygnanski, 1987). However, considerable success was met by (Gaster *et al.*, 1985) who assumed that the timescale of the mean velocity fluctuations is short comparing to a period representative of the large coherent structures and neglected possible exchanges of momentum between those structures.

The instability modes are highly sensitive to inflow conditions such as the transverse curvature, flow divergence, inhomogeneous inflow conditions and the detailed shape of the mean velocity profile as shown by (Cohen & Wygnanski, 1987). Additionally, the latter work showed that the number of the unstable modes

of the shear layer is subjected to its relative thickness.

(Michalke, 1965) predicted the existence of a roll-up mechanism of the traveling instability waves which results into a periodic array of growing vortices. His work was followed by several experiments that visualized the pairing process, amongst which are the (Freymuth, 1966) and (Winant, 1974).

The general picture of the shear layer was initially given by the fundamental works of (Crow & Champagne, 1971; Brown & Roshko, 1971) and (Winant, 1974) who discovered the existence of large coherent structures developing in shear layers. It was realized that the evolution of the flow could be highly connected to the generation and development of those structures. (Fig.1.6) presents the amplification of the initial instability and the subsequent roll-up into discrete vortices (Winant, 1974), and (Fig.1.7) shows a shadowgraph of a plane-mixing layer as it was experimentally seen by (Brown & Roshko, 1971).

(Fig.1.8.a) schematically presents the pairing process and (Fig.1.8.b) is a photograph extracted by a photo-sequence from the experiment of (Winant, 1974) which also illustrates the pairing process on a plain shear layer. Conclusively, it can be said that pairing would infinitely carry on at a plane shear layer; at least for as long as the boundary conditions would not influence the pairing process.

ROUND JETS can be fairly considered as an axisymmetric shear layer in the near-field, as long as the momentum thickness is small in comparison to the diameter of the orifice. For round jets, the initial Kelvin-Helmholtz instability and subsequent roll-up occurs exactly in the same way as for plane shear layers. However, due to its geometrical shape, the primary coherent structures of a round jet develop into vortex rings. Downstream the initially thin shear layer becomes gradually thicker and each side of a ring begins to interact with the opposite side. Hence, the pairing process stops and the frequency of passage of the rings remains approximately the same up to the end of the potential core; therefore, it is termed the preferred mode of the jet. In their paper (Gutmark & Ho, 1983) suggest that the development of the preferred mode can be facility dependent due to the sensitivity of the shear layer instabilities on the background noise.

Even after the formation of the preferred mode, the tendency for further pair-

ing between consecutive rings remains. It is important to realize that the process of mixing between two rings involves alternative expansions and contractions of these structures. However, after the preferred mode has formed, the thickness of the rings is very large in comparison to the local jet diameter and it is no longer possible for a ring to pass through the core of another. A better understanding of this process, called the leapfrogging phenomenon, can be obtained by observing the numerical results of (Kudela & Pawel, 2004) (Fig.1.9). Alternatively, secondary instabilities are getting amplified causing new modes to reveal. As presented by (Urbin & Métais, 1997), in this stage the sub-harmonic mode is amplified causing the rings to deviate from the jet centreline and undergo “*alternate*”, or else, “*helical*” pairing. (Fig.1.10) demonstrates the transitional procedure towards the helical pairing flow pattern as sketched by (Urbin & Métais, 1997).

Following the helical pairing process, a new structure appears at the end of the potential core, (Fig.1.14 for a schematic representation of the potential core). These streamwise structures, or “*braids*”, evolve and amplify in pairs in the braid region between consecutive vortex-rings. Additionally, the streamwise vortex system is deterministic for the rate at which entrainment of ambient fluid occurs in the near-field region $X/D < 10$ as stressed by (Liepmann & Gharib, 1992). (Fig.1.11) shows a side view of a jet at $Re = 5500$, revealing the existence of hairpin vortices (observe arrow). (Fig.1.12.a) presents a cross section of the same jet in two consecutive instants giving a better view of the unsteady development of these structures. Additionally, the formation of pairs of hairpin vortices is followed by the generation of side-jets in the braid region as shown by (Brancher *et al.*, 1994). (Fig.1.12.b) schematically presents the mechanism leading to the formation of side jets. Only a single pair is represented (hatched structures) while two consecutive vortex rings are sketched (white structures). The radial ejection of fluid is symbolized by the two black arrows emerging from the stream-wise pair (Brancher *et al.*, 1994).

COAXIAL JETS are encountered in a variety of engineering applications such as aeroacoustic, propulsion and combustion. Aviation propulsion today is mainly achieved by the use of the turbofan engine. Several experiments have been con-

ducted in the past in order to improve our understanding of coaxial jets. The first experimental investigation of coaxial jets was carried out by (Forstall & Shapiro, 1950). Amongst several other outcomes they reported that self-similarity of a compound jet is obtained at $X/D \geq 8$. Further studies of the near-nozzle region of coaxial jets were implemented by numerous researchers such as the (Chigier & Beer, 1964; Williams *et al.*, 1969; Champagne & Wygnanski, 1971; Durao & Whitelaw, 1973; Hammersley, 1969) and more, in order to build a stronger basis on the coaxial jet physics. In particular, (Durao & Whitelaw, 1973) investigated the developing region of coaxial jets at downstream distances up to $X/D = 17$ for various velocity ratios. Their results showed that coaxial jets tend to reach a self-preserving state much more rapidly than axisymmetric single jets. (Champagne & Wygnanski, 1971) examined different area ratios between the inner and the outer jet nozzles as well as different velocities issuing from each nozzle.

In recent studies, the interest has been turned towards low velocity ratio coaxial jets; those being representative of the flow resulting from the turbofan engine. In this way, (Dahm *et al.*, 1992) examined the dynamics of the initial stage of low-velocity ratio in coaxial jets. (Fig.1.13.a) schematically illustrates the influence of the velocity ratio to the generation of coherent structures. Dahms findings were enhanced by the presentation of enlightening visualizations showing the interactions occurring between different structures. Several vortex patterns were identified this way. (Fig.1.13.b) shows a representative visualization of Dahm's experiment illustrating the development of double trains of vortices on the coaxial jet double shear layer.

Additional studies of varying velocity ratios were carried out by (Sadr & Klewicki, 2003). For a constant inner to outer diameter ratio $D_{in}/D_{out} = 0.39$, they observed that there is a second train of vortices emerging from the near nozzle region, when the velocity ratio was $\eta < 0.18$. They also mentioned that the inner layer structures are highly anisotropic, highlighting that the levels of anisotropy are a relatively strong function of velocity ratios.

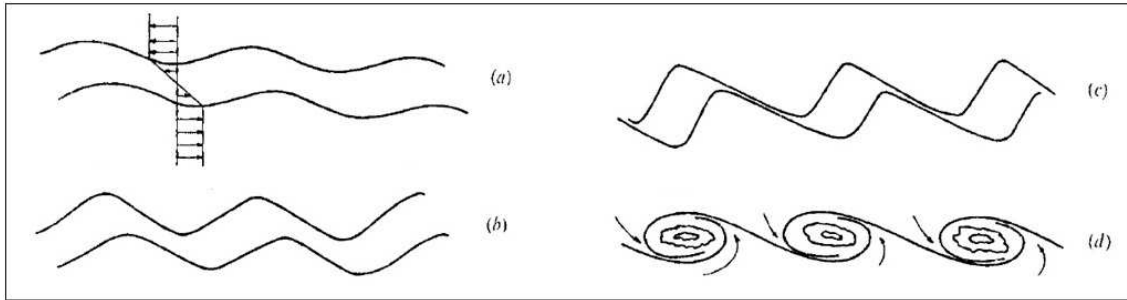


Figure 1.6: Schematic diagram of the initial instability of the shear layer, and the roll-up into discrete vortices (Winant, 1974).



Figure 1.7: Example of the shadowgraphs of a plane-mixing layer (Brown & Roshko, 1971).

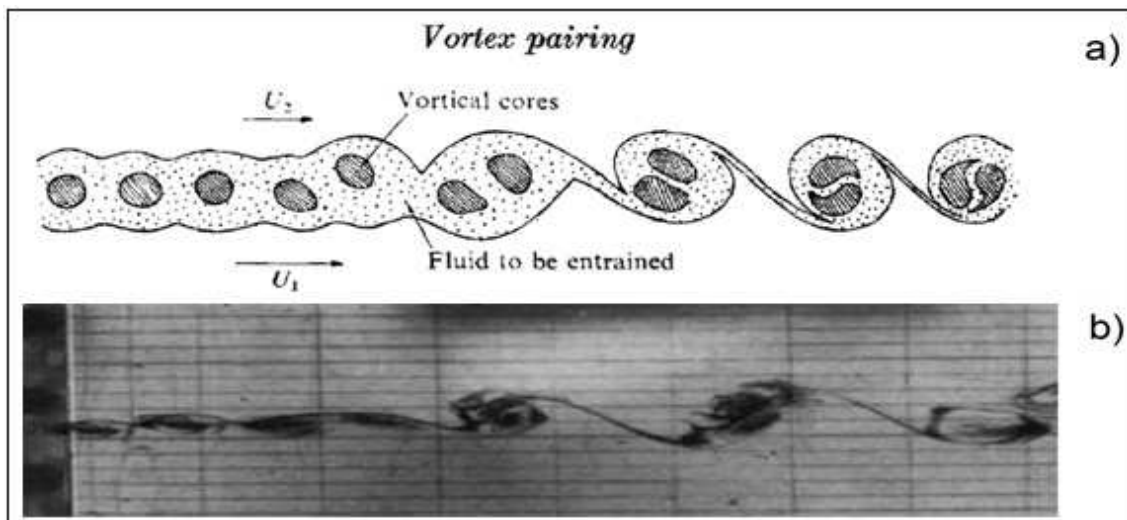


Figure 1.8: The vortex pairing mechanism. a) Schematic representation and b) part of the photograph sequence obtained by the experiment of (Winant, 1974).

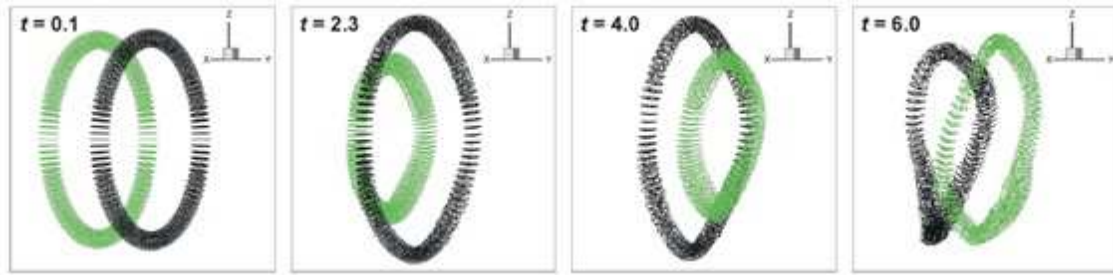


Figure 1.9: The sequence of the time position of the vortex particles for the leapfrogging phenomenon in the inviscid flow (Kudela & Pawel, 2004).

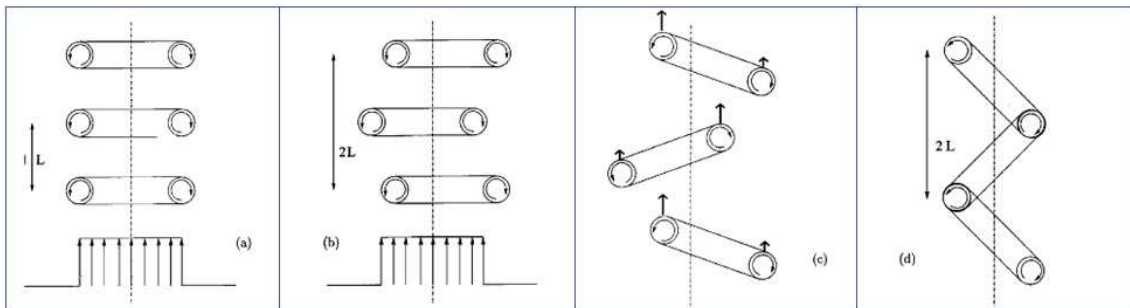


Figure 1.10: Sketch of the vortex rings arrangement leading to the occurrence of alternated pairing events (Urbin & Métais, 1997).

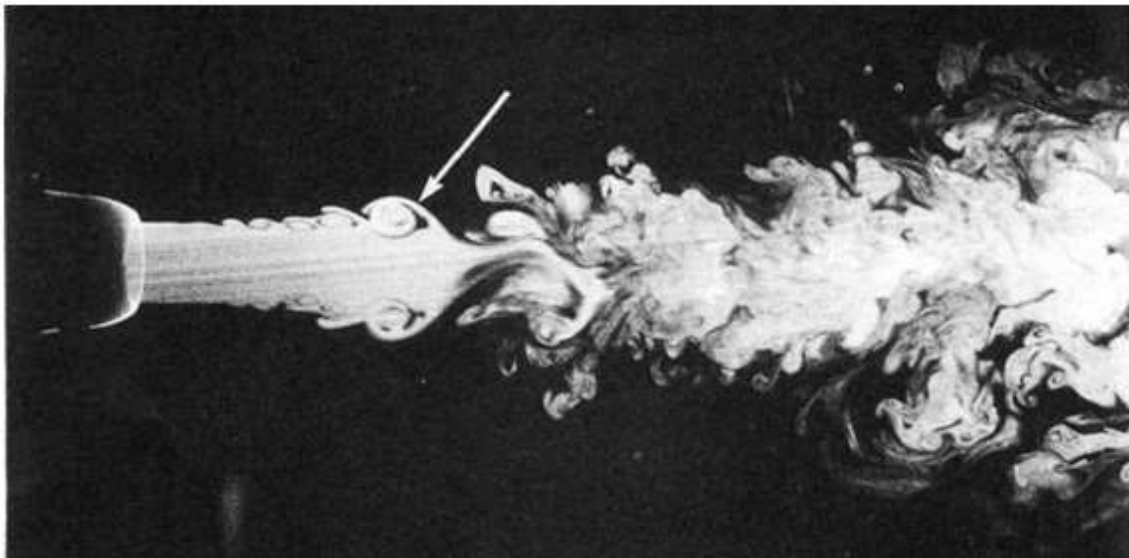


Figure 1.11: Laser-Induced-Fluorescence (*LIF*) side-view of a jet at $Re = 5500$. The arrow indicates a streamwise structure (Liepmann & Gharib, 1992).

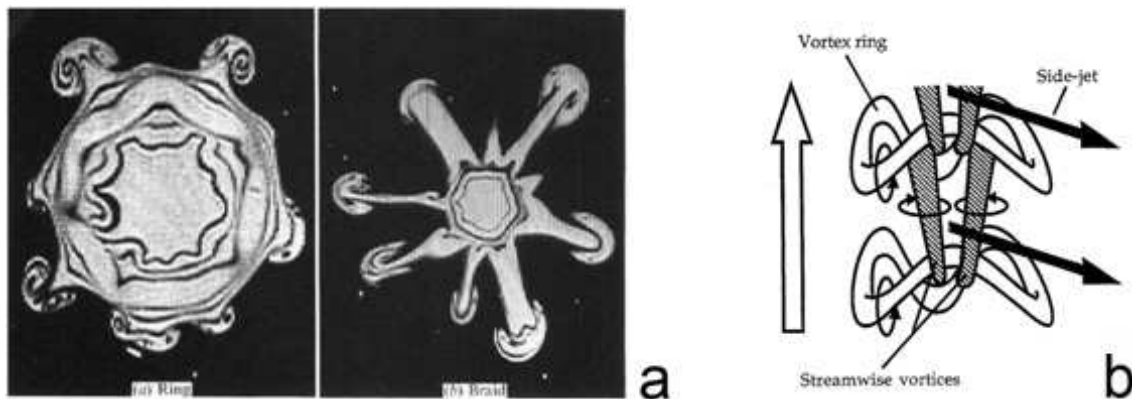


Figure 1.12: a) Cross section from a video sequence of the experiment of (Liepmann & Gharib, 1992). Two different instances of a cross-section in the braid region of the jet are presented b) Schematic presentation of the mechanism leading to the formation of side jets. (Brancher *et al.*, 1994).

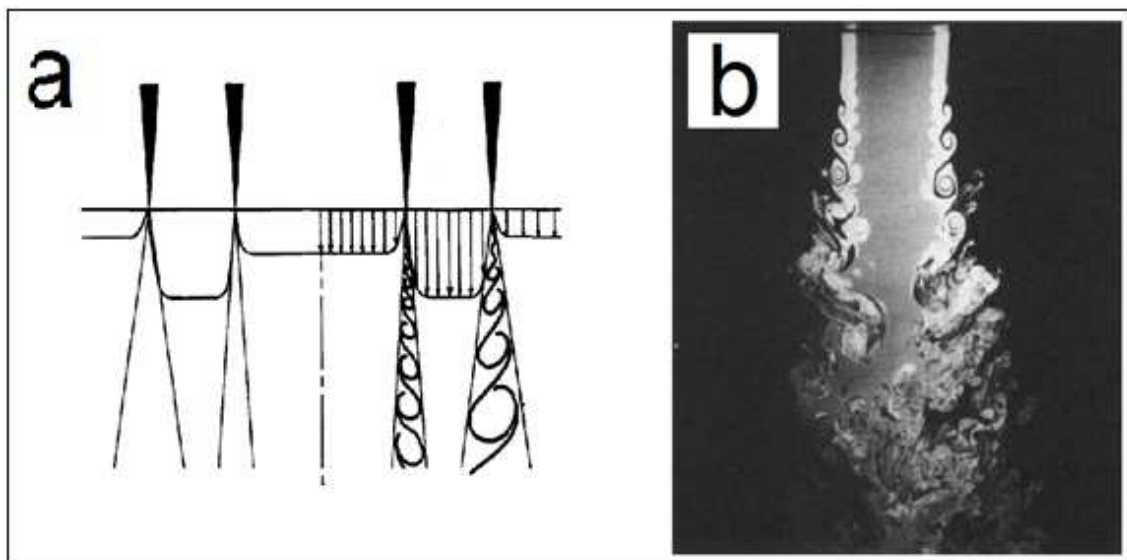


Figure 1.13: a) Schematic representation the velocity ratio influence on the development of the primary coherent structures, and b) an example of a coaxial jet vortex pattern visualization (Dahm *et al.*, 1992).

1.4.2 Fully-Developed Unbounded Round Jet

The mechanisms participating in the transition to turbulence of jets have been explained in section 1.4.1. As shown therein, the “*braid region*”, which is located at the end of the potential core signifies the end of the transition from laminar to turbulent jet. It is the same region where another transition, this time towards the “*self similar*” or “*self preserving*” jet, begins. This transitional zone is characterised by the penetration of the mixing layer across the centreline and by the continuing enlargement of the turbulent fluid radially outwards (Lipari & Stansby, 2011). These first two zones together are usually called “*near-field*” or “*zone of flow establishment/development*”. The final zone, which from now on will

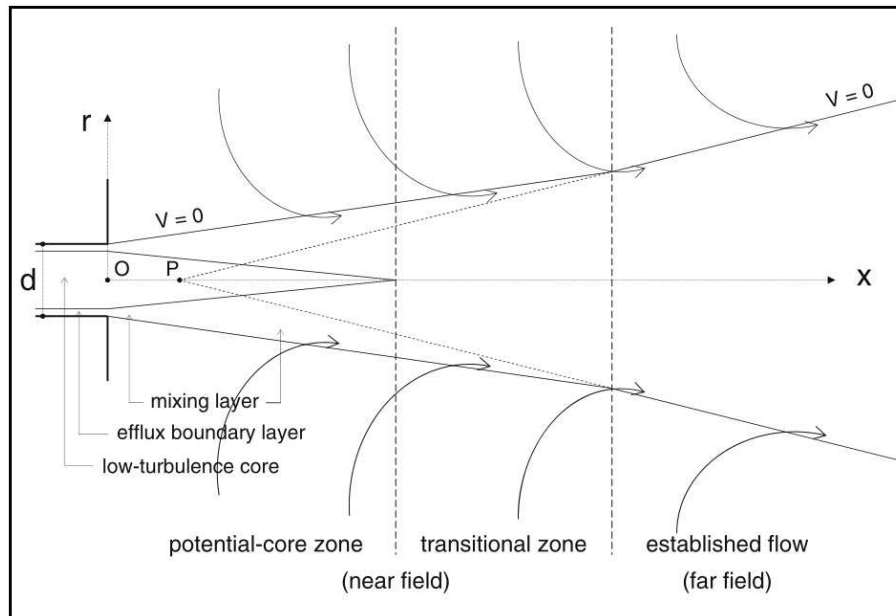


Figure 1.14: Longitudinal cross-section of a round-jet, as sketched by (Lipari & Stansby, 2011). The nominal border here is the surface with zero mean transverse velocity. The highlighted virtual origin P is associated with the conical surface in the farfield. The curved arrows indicate flow entrainment.

be referred to as “*self similar jet region*”, or “*jet farfield*” or simply, “*plume region*”, extends further downstream and is characterized by the established self-similarity of the mean-velocity field. The three subdivisions of the round jet development are presented in (Fig.1.14), which illustrates a customary sketch provided by (Lipari & Stansby, 2011).

(Wyganski & Fiedler, 1969) showed that higher-order velocity moments do

not obtain self-similarity until further downstream. Additionally, (Weisgraber & Liepmann, 1998) mentioned that the normal stresses obtain self-similarity later than the shear stress, however, quantitative information about the downstream distance where it occurs is not available in the reviewed literature. Further to the mean-velocity-based self-similarity, (Mi *et al.*, 2001) suggested an alternative way to define the beginning of the self-similar region; at the location where the distribution of a passive scalar obeys self-similarity.

Deterministic parameters of self-similarity in the farfield of unbounded round jets include the rate of the centreline velocity decay dU / dx , the self similarity of the cross sectional mean velocity distribution $\bar{U}$, and the growth rate of the velocity half-width $dz_{1/2} / dx$ ¹. Various review papers in the literature have compared results of multiple experiments of round jet, such as the fundamental works of (Rodi, 1975) and (List, 1982), and the more recent review of (Lipari & Stansby, 2011).

In round jets, the centreline velocity decay is inversely proportional to the transverse length-scale growth, following the relation:

$$\frac{U_m}{U_b} = B \frac{D}{X - X_B} \quad (1.1)$$

where U_m is the velocity along the jet centreline (maximum) and U_b is the bulk velocity at the jet exit. The value of B has been estimated by several experiments in the past, including the (Wynagnanski & Fiedler, 1969), (Panchapakesan & Lumley, 1993), (Hussein *et al.*, 1994) and (Xu & Antonia, 2002), and was found to vary in the range $5.7 \leq B \leq 6.48$. However, more recent experiments have reported lower values as presented in the recent publications of (Sivakumar *et al.*, 2012) and (Or *et al.*, 2011). X_B is the momentum virtual origin, which signifies the location along the X direction “*at which a pure momentum jet would be located to give the flow equivalent to any jet in question*” (Fischer *et al.*, 1979). Recently measured values of X_B lie in the range $2 \leq X_B \leq 3$ (Sivakumar *et al.*, 2012).

¹In wall jets $z_{1/2}$ refers to the vertical half-width; however, the axisymmetry of round jets allows the use of $z_{1/2}$ for all transverse directions.

The growth rate of the velocity half-width $y_{1/2}$ was estimated by experimental data from (Panchapakesan & Lumley, 1993), (Hussein *et al.*, 1994), and (Xu & Antonia, 2002). From these experiments it was found that growth rate follows a linear relationship as:

$$\frac{dz_{1/2}}{dx} = 0.095 \pm 0.001 \quad (1.2)$$

Slightly different is the suggestion of (Shiri *et al.*, 2008), which calculated the value $dy_{1/2}/dx = 0.0093$. This difference has been attributed to the smaller chamber size of their experiment.

The integral length scales grow linearly with X , independent from Reynolds number when its value is greater than a few thousands as stressed by (Adel-Rahman, 2010). In particular, (Pope, 2000) reports that in the farfield of axisymmetric jets the longitudinal integral scale grows as $L_{11} = 0.7z_{1/2}$ and the lateral integral scale as $L_{22} = 0.3z_{1/2}$. This trend is also observed at the experimental measurements of (Burattini *et al.*, 2005).

The proper orthogonal decomposition (*POD*) investigation of (Gamard *et al.*, 2004) revealed that the timescale of the dominant most energetic mode apparent in the farfield of axisymmetric turbulent jets is the azimuthal mode-1, or “*helical mode*”. This mode is suggested to be a convected instability, the frequency of which was found to correspond to a time-scale T_{11} such as:

$$T_{11} \approx \frac{z_{1/2}}{U_m} \quad (1.3)$$

The passive scalar concentration half-width $z_{\Phi 1/2}$, defined by the radial location at which $\bar{\Phi} = 1/2\Phi_m$, has been found to spread faster than $z_{1/2}$. As reported by (Lee & Chu, 2003), who have considered the velocity and passive scalar half-widths at the locations where $\bar{U} = e^{-1}U_m$ and $\bar{\Phi} = e^{-1}\Phi_m$ respectively, the concentration spreads faster than the velocity by a factor of 1.2. It is expected that the ratio will not change significantly because of the small dislocation of the half-widths ($1/2 > e^{-1}$).

1.4.3 Turbulent Wall-Jet

The presence of a solid surface in proximity to the path of development of a turbulent jet causes major changes to the fluid dynamics involved. The axis-symmetric shape of free-round jets no longer exists in a wall-jet, which can fairly be considered of higher complexity than its unbounded counterpart. There are various types of wall-jets encountered in engineering applications. A number of parameters such as the shape of the initial jet efflux, the number of the ejected streams, the density differences between different streams and the ambient environment, compose a wide range of different types of wall-jets. Three different wall-jet types are discussed in the following paragraphs. In the first paragraph, the relatively simple case of an infinite width “*two-dimensional*” turbulent plane jet is presented in order to highlight common features of wall-jets. The second paragraph refers to the more complex three-dimensional wall-jet (*3DWJ*), which is very similar to the type of flow that forms behind the aeroplane in takeoff conditions and a vast amount of literature is available to discuss. Finally, the third paragraph is dedicated to a sub-category of *3DWJs*, the Three Dimensional Offset Jets *3DOJs*. *3DOJs* are even closer to the type of flow that forms behind the aeroplane in takeoff conditions; however, the literature resources are limited.

Plane Turbulent Wall-Jet

A plane wall-jet is formed when fluid is injected parallel to a wall in such a way that the jet velocity is greater, at some distance above the wall, than that of the ambient flow. Two different types of canonical shear layer are present in this flow. The shear layer which forms between the wall and the fluid, also called “*inner layer*”, has the features of a boundary layer and extends from the wall-surface up to the position of maximum velocity. The shear layer which forms between the jet and the ambient fluid, also called “*outer layer*”, resembles a free shear layer. (Fig.1.15) illustrates a typical vertical mean-velocity distribution of a developed plane wall jet. The important velocity scales include the maximum velocity and its half, U_m and $U_m/2$ respectively. The important length scales include the vertical

distance above the ground at which U_m occurs, z_m , as well as the vertical distance above the ground at which $U_m/2$ occurs, $z_{1/2}$, alternatively called “velocity half-width”.

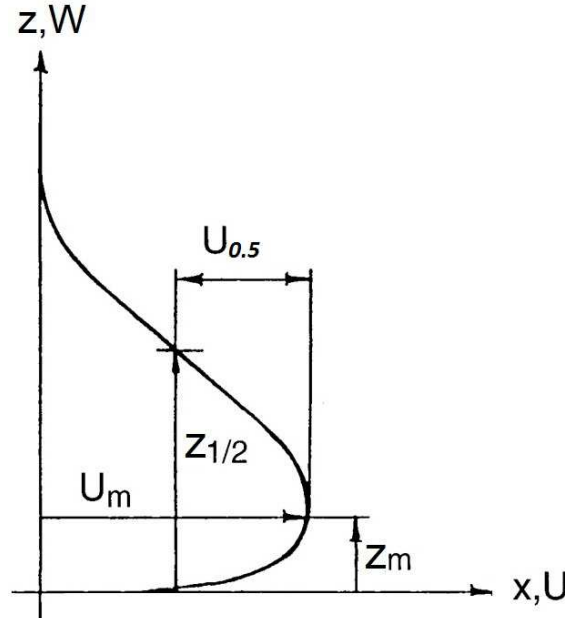


Figure 1.15: Sketch of a typical vertical profile of mean streamwise velocity of the developed region of a plane wall-jet. Modified picture found at (Eriksson *et al.*, 1998)

Early theoretical calculations of laminar and turbulent wall jets were implemented by (Glauert, 1956). The first experimental investigation of a plane turbulent wall-jet was carried out by (Förthmann, 1936), who used the term “*partially open jet*” for the description of this flow. Förthman’s experiment was followed by several other experimental investigations such as the work of (Schwarz & Cosart, 1961), who used hot wire anemometer (*HWA*) to measure the mean velocity distribution of the incompressible plane wall jet. This experiment evaluated the Reynolds shear stress, the Boussinesq exchange coefficient, and the shear stress at the wall.

Turbulent wall jets have also been the subject for major experimental studies in recent years, such as the works of (Wyganski *et al.*, 1992) and (Abrahamsson *et al.*, 1994) who used *HWA* to investigate the wall-jet in stagnant environment, as well as the experiment of (Schneider & Goldstein, 1994) who performed laser doppler velocimetry *LDV*.

(Launder & Rodi, 1983) proposed that plane wall-jets spread linearly, following the relation:

$$\frac{dz_{1/2}}{dx} = 0.073 \pm 0.002 \quad (1.4)$$

As (Launder & Rodi, 1983) stated, “*This growth rate is more than 30% below the consensus value for the plane free jet*”. They suggest that this difference arises by the damping of turbulent velocity fluctuations in the direction normal to the wall due to turbulent pressure reflections from the wall itself.

Three-Dimensional Wall Jet (3DWJ)

A fundamental feature of a *3DWJ*, differentiating it from a plane turbulent wall jet, is that the former issues from a nozzle of finite width. This feature increases the complexity of the physics involved at the jet development. In particular, further to the vertical variation of the mean-velocity of a plane wall-jet (Fig.1.15) it also involves span-wise variation of a highly inhomogeneous three-dimensional character. Here, the free-shear layer (which occurs at the outer mixing layer for plane wall jets) intersects with the wall-boundary layer. In (Fig.1.16), a vertical and a horizontal velocity profile at a station of the fully developed region of a three-dimensional wall jet is drawn, based on the observations made at the simulations of this thesis. The horizontal profile is taken at an offset z_m from the ground. Further to the important velocity and length scales of the vertical profile, which are similar to the plane wall-jet, here also appears the horizontal “*velocity half-width*” $y_{1/2}$. The boundary of the jet is highlighted, helping to visualise the intersection between the free-shear layer and the wall-boundary layer.

3DWJs have been the subject of experimental investigation for more than four decades now; however, knowledge of this high complexity flow is yet to become robust. Early experiments of *3DWJ* have been carried out by (Sforza & Herbst, 1967), (Newman *et al.*, 1972), (McLean & Herring, 1974), (Davis & Winarto, 1980), and (Koso & Ohashi, 1982), followed by the review article of (Launder & Rodi, 1981), later summarized in (Launder & Rodi, 1983). In the

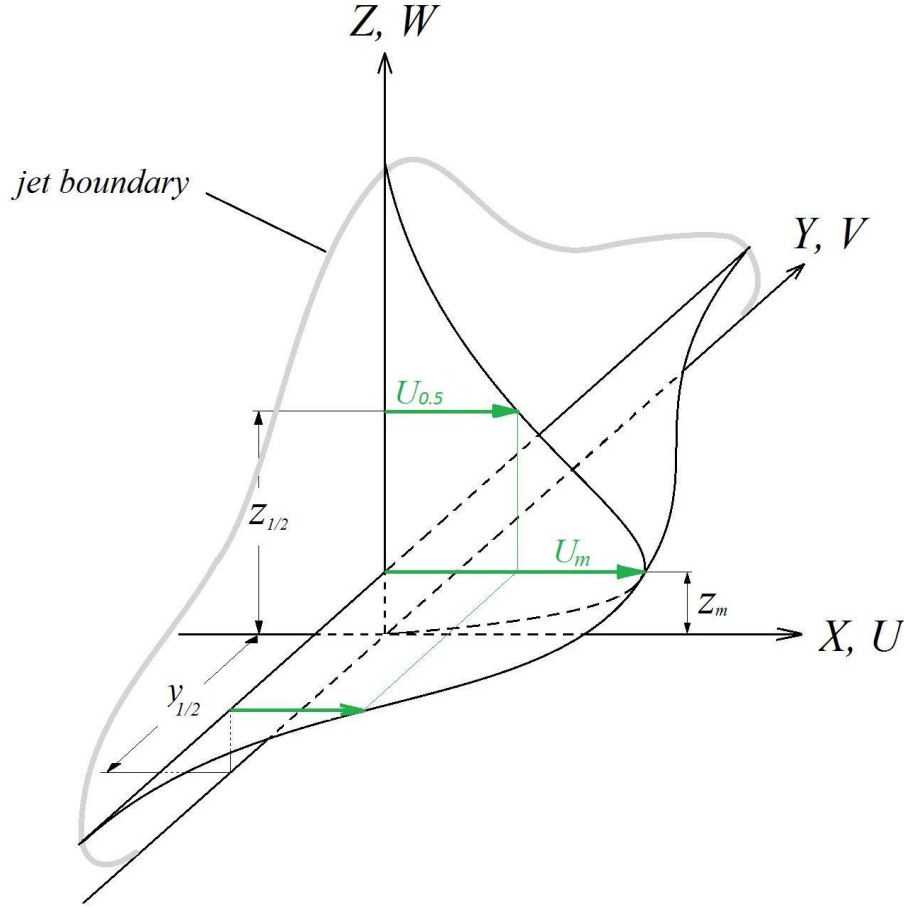


Figure 1.16: Sketch of vertical and horizontal velocity profiles of the fully developed region of a three-dimensional wall jet. The boundary of the jet is highlighted.

nineties, important experimental work was carried out by (Matsuda *et al.*, 1990), (Padmanabham & Gowda, 1991) and (Abrahamsson *et al.*, 1996), while the latest recent experiments include the ones of (Sun & Ewing, 2002), (Law, 2002), (Hall & Ewing, 2007), and (Hall & Ewing, 2010).

The maximum velocity U_m of 3DWJs was found to decay inversely proportional to X . In particular, the work of (Padmanabham & Gowda, 1991) suggests a decay as $X^{-1.15}$, while (Abrahamsson *et al.*, 1996) suggests $X^{-1.29}$.

(Launder & Rodi, 1981) recommended that the spread rate of the vertical and horizontal velocity half-widths is linear, and suggested the values $dz_{1/2}/dx = 0.048$ and $dy_{1/2}/dx = 0.26$ respectively; this is a ratio of spreading rates of about 5.5 : 1. A larger ratio was measured by (Davis & Winarto, 1980), where at large distances from the nozzle it approached the value of 8.5 : 1. The experiment of (Abrahamsson *et al.*, 1996) showed larger spreading rates $dz_{1/2}/dx = 0.065$ and

$dy_{1/2} / dx = 0.32$, which corresponds to a ratio of about 5 : 1. The *PIV* measurements of (Law, 2002) on a circular wall jet showed that induced turbulence was still evolving at $X/D = 50$ and the jet had not achieved self-similarity. This is a very important finding because most of the available experimental data in the literature do not extend in much more downstream locations than that; hence self-similarity of *3DWJs* has not been extensively investigated yet.

The explanation given by (Davis & Winarto, 1980) regarding the large spreading rate ratios in *3DWJs* is that the enhanced spreading is attributed to augmented turbulent diffusion parallel to the wall. However, (Launder & Rodi, 1981) suggested that it is caused by the presence of strong mean secondary flows in the jet. Similar was the explanation given by the early work of (Newman *et al.*, 1972), which suggested that the large spreading rate ratio arises from a large secondary flow causing a substantial lateral outflow parallel to the wall with a strong entrainment velocity being induced normal to the surface. This suggestion was later confirmed experimentally by (Abrahamsson *et al.*, 1996), and then from the analytical computations of (Craft & Launder, 2001). However, the link between the mean secondary flow (the mean streamwise vorticity) and the unsteady streamwise vorticity associated with coherent structures in the jet is not clear (Namgyal & Hall, 2013). For this reason, a considerable effort is recently being put in order to interpret the turbulent mechanisms involved to the evolution of a *3DWJ*.

(Matsuda *et al.*, 1990) investigated the flow structure at the near-field of a *3DWJ*, using the *HWA* technique. Their results revealed “*the presence of large-scale horseshoe-like structures, whose legs are inclined and stretched to form the streamwise vortices in the mixing region of the jet*”. The works of (Ewing & Pollard, 1997), (Sun & Ewing, 2002), (Hall & Ewing, 2007), (Hall & Ewing, 2010), and (Namgyal & Hall, 2013) which followed, validated the mechanism suggested by Matsuda, and gave a more detailed description of it having used different measuring and post-processing techniques. In particular (Namgyal & Hall, 2013) used stereoscopic *PIV* in the near-field region $X/D \leq 5$ and applied proper orthogonal decomposition *POD* to all three components of the velocity field to investigate

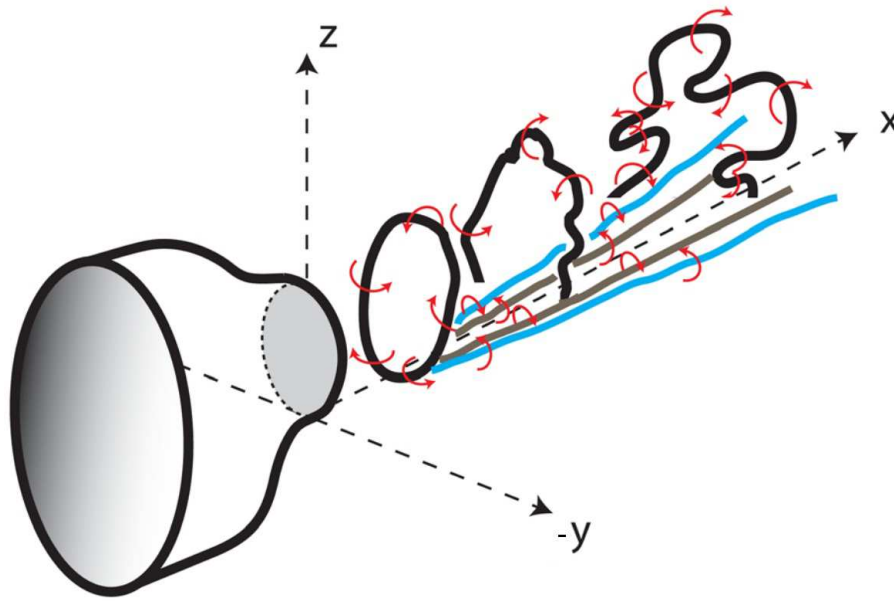


Figure 1.17: Proposed coherent structure model in the near-field of the turbulent $3DWJ$ exiting from a contoured nozzle. (Namgyal & Hall, 2013).

the underlying coherent structures in the flow. The proposed model of (Namgyal & Hall, 2013) for the coherent structures in the near-field of the three-dimensional wall jet is illustrated in (Fig.1.17). The orientation of the Cartesian coordinate system of (Fig.1.17) has been modified to agree with the notation used in this thesis.

Three-Dimensional Offset Jet (3DOJ)

The discussions about $3DWJ$ dynamics presented above are limited to the case where the nozzle is attached to the solid surface. When the nozzle is displaced by some distance h above the solid surface, then the initial development of the jet resembles that of an unbounded jet. However, the continuous growth of the three-dimensional mixing layer, together with the attraction forces which form between the jet and the solid surface (Coandă effect (Reba, 1966)), results for the jet to descent, and eventually attach on that surface. After the jet has completely attached, it begins to develop into a self-similar $3DWJ$ following the physics described in the previous paragraph. (Fig.1.18) illustrates a sketch of the vertical cross-section at the symmetry of a $3DOJ$. The different regions of its development and important dimensions are highlighted. The paths of development of

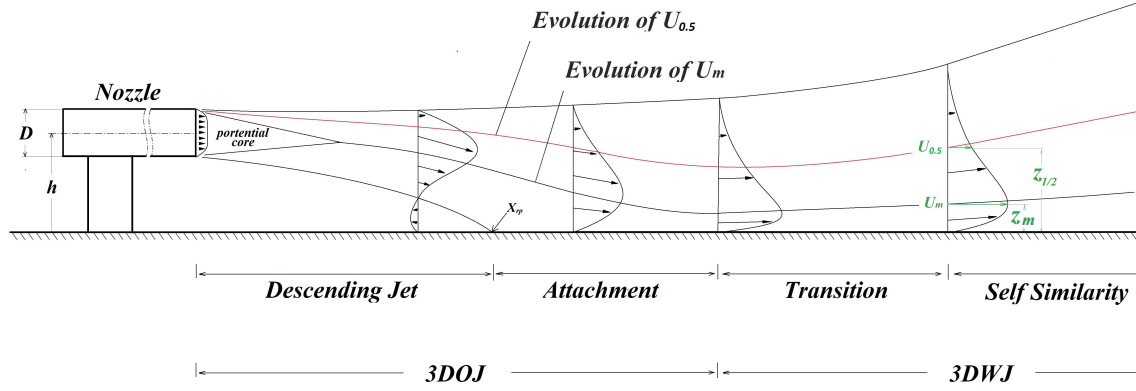


Figure 1.18: Sketch of the vertical cross-section of a 3DOJ. The different regions of its development, as well as important dimensions are drawn.

U_m and $U_m/2$, as well as the point of reattachment X_{rp} , are also included on the graphic. The black arrows point in the direction of the local velocity vector, while the green ones represent the streamwise component of the velocity vector, thus, pointing in the streamwise direction.

A considerable amount of experimental work related to offset jets has been reported in the open literature, however, the majority of these experiments are dedicated to two dimensional offset jets. Such work can be found in (Pelfrey & Liburdy, 1986b; Pelfrey & Liburdy, 1986a; Nasr & Lai, 1997; Nasr & Lai, 1998; Gao & Ewing, 2007), and (Gao & Ewing, 2008). Several techniques of measurement have been used to measure flow variables in those experiments, including Pitot tubes, *HWA*, *LDA*, and *PIV*.

Unfortunately, the experimental work directly related to the 3DOJ is limited. The first comprehensive study of 3DOJs was carried out by (Davis & Winarto, 1980). In this experiment *HWA* was used to measure the mean velocity and streamwise turbulent intensity within the region $X_{rp} \leq X \leq 64$. This experiment mainly focussed on the effect of the nozzle offset h on the wall jet region, and covered the range $0.5 \leq h/D \leq 4$. The main effect of the nozzle displacement from the wall appeared to be the streamwise elongation of the “*descending*”, “*attachment*” and “*transition*” regions with increasing h . The spreading rates observed at the developed region, for offset $h/D = 2$, was $dy_{1/2}/dx = 0.29$, and $dz_{1/2}/dx = 0.039$ corresponding to a lateral to normal spreading rate ratio of

about 7.5.

Recently, (Agelin-Chaab & Tachie, 2011) extended the work of (Davis & Winarto, 1980), using the *PIV* technique. The measurements of Agelin-Chaab were applied at much longer downstream distances, covering the range $0 \leq X/D \leq 120$. The effect of the jet exit Reynolds number $(Re_j)^2$ to the velocity decay, spread rates, and reattachment length X_{rp} , was tested and found to be very small. In particular, for nozzle offset $h/D = 2$ the decay rate of $dU/dx = 1.17$ and spread rates of $dy_{1/2}/dx = 0.250$, and $dz_{1/2}/dx = 0.056$ were reported. According to these values, the lateral to normal spreading rate ratio is about 4.5. A linear development of y_m was observed in the self similar region. Finally, profiles of mean velocity, turbulence intensity, and Reynolds shears stresses were found to be nearly independent of Re_j in the self-similar region.

The main disagreement between the findings of (Davis & Winarto, 1980) and (Agelin-Chaab & Tachie, 2011) is on the ratio of the lateral to normal spread rates. More specifically, for the same range of nozzle offsets $1 \leq h/D \leq 4$ the ratio of Davis and Winarto varied in the range 3.7 to 8.6 while for Agelin-Chaab, the range is narrower, only varying from 4.5 to 4.7. However, no clear answer has been given yet to justify this disagreement.

1.5 BUOYANT PLUMES AND WIND BRAKES

The importance of the early take-off mode contribution to local air quality in proximity to major airports was discussed in section 1.1.1. The present section introduces the concept of buoyancy in the plume development, and explains the philosophy followed for the design of wind brakes (baffles), which is an investigated approach for the improvement of local air quality.

1.5.1 Accounting for Buoyancy

Typical commercial aero-engines produce thrust by accelerating gases backwards, parallel to the ground. As explained in section 1.4.3, such a configuration can

²Reynolds number varied by changing the jet exit velocity.

be described by a circular jet which reattaches on the proximate solid surface (ground), and is followed by the formation of a three dimensional wall jet (*3DWJ*). So far we have described the farfield of a *3DWJ* as a self-similar flow. However, the situation would be different if buoyancy, resulting from the difference in density between the jet and the ambient flow, becomes a significant parameter.

In the present problem, buoyancy is a source of vertical momentum, with upwards positive direction. Broadly, the vertical momentum of the material emitted per unit time increases at a rate F_B , the “buoyancy flux” (eq. 1.5), while its horizontal momentum F_M (eq. 1.6), i.e. the engine thrust, remains approximately constant with streamwise development of the jet.

$$F_B = \rho_a g A U_b \frac{\rho_a - \rho_j}{\rho_a} \quad (1.5)$$

$$F_M = \rho_a A U_b^2 \quad (1.6)$$

Here, A is the cross-sectional area of the nozzle, ρ_a and ρ_j are the ambient and the jet flow densities respectively, U_b is the jet bulk velocity, and g is the gravitational acceleration.

Near the nozzle, buoyancy is very small comparing to thrust, therefore, its effects to the plume development are negligible. However, moving downstream, the two will eventually become equally effective after a travel time:

$$\tau = \frac{F_M}{F_B} \quad (1.7)$$

For a modern high-bypass aero-engine, this scale time is of order $80s$ (Graham *et al.*, 2008).

There is a variety of theoretical models in the literature for ground based buoyant plumes as reviewed by (Ramsdale & Tickle, 2001). A more relevant, to the present study, model was later suggested by (Bennett *et al.*, 2013). According to their approach the wall bounded plume is considered as a half cone with ex-

pansion rate corresponding to an axisymmetric jet, while co-flow (wind) is not taken into account. Bennett *et al.* suggest that the reattachment length can be approximated by:

$$X_{rp} \approx \frac{2h}{\beta\sqrt{5}} \quad (1.8)$$

where h is the nozzle vertical offset from the ground, and β is the free-jet spreading parameter, selected as $\beta = 1/9$, slightly larger than the round jet expansion rate seen in (eq.1.2). For a typical offset of the nozzle above the ground, i.e. $h = 2$, the reattachment length can be calculated by (eq.1.8) $X_{rp} = 16$.

Once the centreline of the jet has reached the surface, the downward momentum flux from the entrainment of air from above tends to keep it there (Coandă effect), while any buoyancy in the jet provides a countervailing upward momentum. Accordingly, the model of Bennett *et al.* suggests that the two balance at a travel distance of:

$$X_L \approx \left(\frac{\tau}{\pi}\right)^{1/2} \left(\frac{2F_M}{\pi\rho_a}\right)^{1/4} \quad (1.9)$$

which by definition suggests that X_L is proportional to $(F_M)^{3/4}$.

An alternative lift-off criterion for ground based plumes has been suggested by (Hanna *et al.*, 1998). In particular, taking into account that shear in the surface boundary layer will disrupt a plume, spreading the emission horizontally faster than it can rise, Hanna *et al.* suggested that a suitable criterion for whether a buoyant surface emission will rise or not is that:

$$\frac{U^3 W \pi \rho}{F_B} < 100 \quad (1.10)$$

where U is the ambient wind speed and W the local plume width. (Bennett *et al.*, 2013) reports that the criterion of Hanna *et al.* gives a comparable result to their formula for X_L if the wind velocity is substituted by the jet effective velocity.

1.5.2 Wind Brakes (Baffles)

Part of the central objective of this work is to enhance plume lift-off through the installation of an array of inclined baffles on the airfield behind the starting aircraft. This idea has been initiated and primarily investigated in real scale by (Bennett *et al.*, 2013). In the present section we summarize their concept which sets a base point for the present work.

If, overall, a wind brake system imposes a drag F_D and a lift F_L on the exhaust jet, then the scale time to buoyant lift-off, τ' , is that at which $F_B\tau' + F_L = F_M - F_D$, that is:

$$\tau' = \tau \left(1 - \frac{F_D + F_L}{F_M} \right) \quad (1.11)$$

τ' is thus minimized by maximizing $F_D + F_L$. Drag alone would reduce the travel distance before the jet escapes the Coandă effect such as:

$$X'_L \approx X_L \left(1 - \frac{F_D}{F_M} \right)^{3/4} \quad (1.12)$$

Figure 1.19 illustrates a sketch of the wind brake concept for enhancing dispersion of engine exhaust at take-off.

The design of the baffles has to maximize its efficiency and in the same time

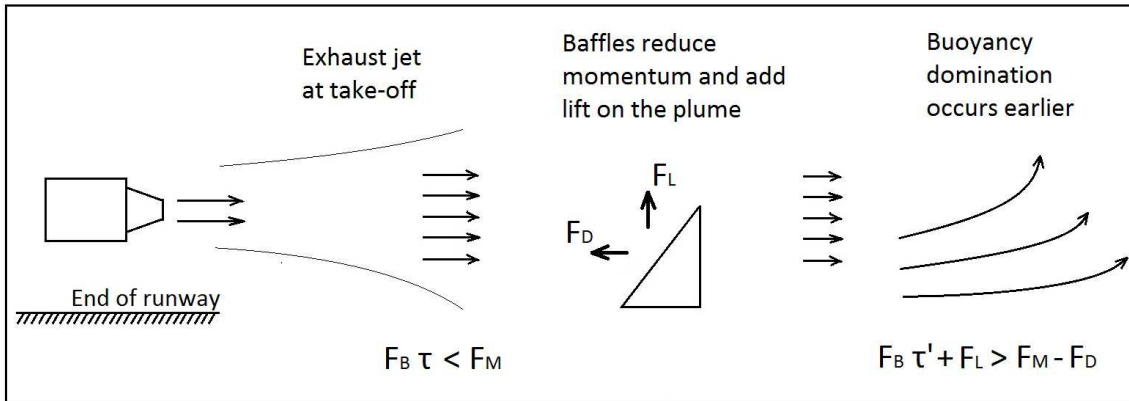


Figure 1.19: Wind brake concept for enhancing dispersion of engine exhaust at take-off (schematic; not to scale).



Figure 1.20: FAAMs BAe 146-301 Atmospheric Research Aircraft in field trials at Cranfield University and the tested baffle array (final baffle design) (Bennett *et al.*, 2013).

conform with airport regulations. Detailed information of the procedure followed in order to reach the final design of the baffles is available at (Bennett *et al.*, 2013). The final design of the baffles is illustrated in (Fig.1.20) together with the FAAMs BAe 146-301 atmospheric research aircraft in the field trials which took place at Cranfield Airport over the period 14-21 September 2011. A modelled version of the final design has been tested in the wind tunnel, also being numerically simulated in the present work. In particular, the induced drag of this baffle arrangement was measured and found to absorb $> 30\%$ of the initial jet thrust (refer to chapter 9).

The theoretical formulae of the present section are applied to the problem setup of this work (section 5.2) providing a-priori estimates for the plume lift-off onset, both when a wind brake system is installed or not.

1.6 NUMERICAL SIMULATIONS

Experimental investigation of turbulent jets is linked with high financial cost and limited analysis capabilities. These two important research parameters, in combination with the continuously increasing power of modern supercomputers have lead to the computational simulation of turbulent jets. This section summarizes modern *CFD* techniques available to model turbulence, also referring to previous numerical investigations that focused to the jet development and mixing.

Reynolds-Averaged Navier-Stokes (*RANS*) is a technique that models turbulence based on Reynolds averaging of the governing equations (Randall, 2006). For this reason, *RANS* simulations do not require any transient calculations, while its formulation is simplified by using the Boussinesq hypothesis which postulates that the momentum transfer caused by turbulent eddies can be modelled with an eddy viscosity. Although simple and computationally inexpensive, *RANS* modelling is non acceptable here due to the intermittent phenomena highly responsible for mixing in turbulent jets (Salim *et al.*, 2011).

Direct Numerical Simulation of turbulence (*DNS*), is an alternative approach; though, a much more expensive one. With *DNS* all the length and time-scales of turbulence are directly resolved; hence, no turbulence models and subsequent approximations are required. Indeed, *DNS* is the most accurate approach for the numerical simulation of turbulent problems, however, it is restricted to moderate Reynolds numbers limiting the area of applicability in engineering applications.

The main advantages of *RANS* and *DNS* are combined in *LES*. The principle of *LES* is that the large and anisotropic turbulent scales are directly resolved similar to *DNS*, while the smaller and “*universally isotropic*” scales are modelled to reduce the computational cost. This is achieved by filtering the $N-S$ equations in space and time. Hence, when the modelled turbulent motions are isotropic, the solution obtained by *LES* is of similar accuracy to *DNS*. This is easier to achieve in free-shear flows, where turbulence isotropy occurs at larger scales comparing to wall bounded flows.

1.6.1 LES Challenges and Applications

There are doubts about the small-scale isotropy assumption and the wavelength at which isotropy occurs, as well as certain issues in the “communication” between the resolved and the modelled scales in the implementation of *LES* (Meneveau *et al.*, 1999). Sub-Grid Scale (*SGS*) modelling has been the subject of several parametric studies of recent investigations, such as the works of (Ilyushin & Krasinsky, 2006), (Liu *et al.*, 2008), and (Zemtsop *et al.*, 2009). Jet *LES* was carried out by (Volkov, 2008) with the use of the Reynolds-averaged, space-filtered Navier-Stokes equations, closed by the $k - \epsilon$ model of turbulence and the subgrid *RNG* model of eddy viscosity. Their results agreed with experimental data. Fundamental issues of *SGS* models were also identified and discussed by (Chen *et al.*, 2005). Through this study it is revealed that *SGS* models need more research to further improve and validate their application in highly unsteady flows. Critical issues involved in *LES* relating to the treatment of unresolved sub-grid scale flow features are discussed by (Grinstein, 2009). Three different *SGS* models available in FLUENT-*LES* solver, were tested by (Zemtsop *et al.*, 2009). In particular, a parametric study comparing the Standard Smagorinsky Model (*SSM*), the Dynamic Smagorinsky Model (*DSM*), and the Dynamic Sub-grid Kinetic Energy Transport Model (*DKEM*) revealed the superiority of *DSM* and *DKEM* comparing to *SSM*.

The most challenging jet-flow region to apply filtering is the near-nozzle three dimensional shear layer. This is due to the fact that the “long-wave” and the “dissipative” intervals of the turbulent spectrum, intersect in this region (Pope, 2000). In this way, (Ilyushin & Krasinsky, 2006) investigated the influence of the C_s parameter of the Standard Smagorinsky Model (*SSM*) in the transition of round jets and found out that the optimum value is $C_s = 0.17$.

(Meneveau, 2012) makes generalized interpretation of the Germano identity, stating that *the sum of resolved and modelled contributions to basic quantities of intrinsic physical interest must be independent of filter scale*. (Burton, 2006) performed *LES* of round turbulent jets using the Inertial Large Eddy Simulation

(*ILES*) method with multi-fractal sub-grid-scale modelling and accurately recovered the large scale structures, showing that *ILES* is an alternative approach to model the sub-grid scale motions in jet simulations. (Ilyushin & Krasinsky, 2006) highlighted the necessity of adequately large computational domains in *LES* of turbulent unbounded jets. In particular, their results showed that the boundary conditions can be even more responsible for the *LES* solution than the *SGS* modelling itself. Critical issues involved in *LES* relating to initial and boundary conditions are also discussed by (Grinstein, 2009).

A considerable effort has been put in finding approaches of generating artificial velocity data representative of the nozzle exit plane including the works of (Smirnov & Shi, 2001) and (Klein *et al.*, 2003). Additionally, in the recent study of (Zemtsop *et al.*, 2009), FLUENT *LES* computations were implemented in an attempt to investigate the influence of the inlet conditions on jet mixing. In this way, the outlet conditions (nozzle exit plane) of a real jet engine were being systematically extracted through *RANS* computations, and successively being used as inlet profiles for the *LES*. This method was validated by means of experimental data and found to be consistent. Although the turbulent field of a round jet consists of organized structures (Cittriniti & George, 2000), it is extremely difficult to compose realistic artificial unsteady boundaries to imitate the fully developed farfield jet behaviour.

In section 1.4.1 it was shown that after the breakdown of the primary coherent-vortices, streamwise braids dominate the flow. The importance of swirling in the rate of mixing of round jets was investigated through *LES* by (McIlwain & Pollard, 2002). They highlighted the deterministic effects of swirl in the breakdown of the primary ring-vortices. They also showed that the stronger the streamwise vortices in the braid region, the higher the growth rate of the jet in the farfield. The spatial evolution of the turbulent flow in a confined square coaxial jet was investigated numerically using in-house *LES* code by (Xu *et al.*, 2003). Their high resolution *LES* include detailed inlet conditions in order to investigate turbulence nature in the near-field $X/D \leq 15$. Comparison of the *LES* predictions met reasonable agreement with experimental data in terms of 1st and 2nd order statistics.

Large eddy simulations have been used by previous researchers in coaxial jet investigations. Further to the results of (Dinesh *et al.*, 2010) and (Xu *et al.*, 2003) that have been mentioned above, (Apte *et al.*, 2003) performed coaxial jet *LES* solving the incompressible, spatially filtered Navier-Stokes equations on unstructured grids. Mean and turbulent quantities were compared to experimental data and good agreement was observed. The *LES* results were found significantly more accurate than the *RANS* predictions of the same problem. In the later work of (Fröhlich *et al.*, 2007), the co-annular jet mixing was investigated by means of *LES*. Geometrical modifications of the double nozzle exit were found to affect flow structure in the near-field and downstream mixing.

1.6.2 Pollutant Dispersion and Mixing Simulations

The majority of the studies of turbulent jet *LES* available in the literature limit their computational domains in the near-field $X/D \leq 20$. This is a consequence of the enormous increase in computational cost with increasing the downstream location of interest.³ However, this thesis focuses at the very farfield, where the aircraft engine exhaust plume mixes with the ambient flow. Therefore, numerical investigations having considered jet mixing at higher distances at the farfield $X/D \geq 20$ are summarized in this paragraph.

LES does not represent viscous dissipation, but only captures the large scale convective motions. However, (Mahesh *et al.*, 2007) explain that as long as the energy containing convective motions and the approximate thickness of the scalar fronts are captured, a good prediction of the scalar field will be achieved by *LES*. The analyses made in (Mahesh *et al.*, 2007) also reveals the superiority of *LES* comparing to *RANS* in scalar mixing simulations. Additionally, external intermittency is deterministic for passive scalar mixing and makes the application of *RANS* modelling inappropriate. (Dinesh *et al.*, 2010) published *LES* results investigating mixing and intermittency of a coaxial turbulent jet discharging into an unconfined domain, and they showed that intermittency is very significant close to the outer edge of the flow. External intermittency was also investigated by

³A more analytical explanation of this statement is provided in section 5.4.

(Gilliland *et al.*, 2012), who compared their *LES* results of a plane jet, with *DNS* and experimental data, and found good agreement. Similar level of success was also met by (Ribault & Sarkar, 2001), who simulated the passive scalar mixing at a plane jet. These works, in combination with the findings of (Mahesh *et al.*, 2007) mentioned above, have motivated the use of *LES* in this investigation.

(Boersma *et al.*, 1998) have performed one of the early *DNS* to study the farfield self-similarity and its response on variations of the inflow conditions. Extending this work (Lubbers *et al.*, 2001) investigated the self-similarity of a passive scalar field in the turbulent jet. In a later work, (Babu & Mahesh, 2004) performed *LES* investigating the effects of the inlet entrainment on the self-similar region of the jet suggesting that inflow entrainment is important for the simulation of turbulent jets, especially for studies where near-field behaviour is important. More recently, (Dinesh *et al.*, 2012) performed *LES* of isothermal co-axial jets and they observed the farfield dynamics. The introduction of a passive scalar in the jet cores helped to better elucidate the mixing process. Additionally, several flow parameters, such as the velocity ratio, inlet swirling and wall effect were observed and a comprehensive qualitative analysis followed.

Numerical investigation of pollutants dispersion behind a B737-500 type aircraft during idling operations was performed by (Testa *et al.*, 2012). Their results improve the understanding of the jet blast behaviour and represent an initial assessment for the use of *CFD* techniques in the investigation of pollutants dispersion. Aircraft pollutant dispersion during takeoff was simulated using *CFD* with *RANS* closure by (Koutsourakis *et al.*, 2006). It was followed by the work of (Aloysius & Wrobel, 2009b) who performed *LES* of buoyant free and wall-jets in order to obtain an initial understanding of the plume dynamics created by an aircraft engine in the takeoff phase. Experimental data and theoretical velocity profiles were used for the validation of their *LES* results. In a later report (Aloysius & Wrobel, 2009a) modelled the complete geometry of a Boeing 737 and performed *LES* investigating emission dispersion during take-off. The predictions of the plume-rise agreed with *LIDAR* data. Additionally, their data show that the wing-tip vortices of the aircraft in a head wind configuration, contribute to the plume natural rise.

1.7 LITERATURE REVIEW OUTCOME

Understanding the purpose of the research, as well as familiarisation with the problems to be encountered in this work, have been attempted by this introductory chapter. In particular, section 1.1 introduced the environmental issues related to civil aviation operations, and discussed the dispersion mechanisms which take place at different stages of the flight, in this way broadening our understanding regarding the source of the problem. Then, section 1.2 justified the selection of a numerical approach for the implementation of this research, and compared this approach to the real scale investigation and the wind tunnel modelling.

One of the most significant parameters of the investigated problem is turbulence. In this way the concept of turbulence and its properties have been introduced in section 1.3, followed by an extensive literature review of jets in section 1.4. In the latter, the mechanisms leading to jet transition to turbulence, as well as the different stages from which a jet passes on its downstream development, have been thoroughly discussed. As a result understanding of the core elements of the physical problem, which is attempted to numerically model and simulate, has been improved. Theoretical and empirical models describing certain features of the jet flow, including the mean jet characteristics, as well as the structure of the most energetic turbulent scales, necessary for the numerical and theoretical analyses of this work, have been identified.

A crucial parameter of the problem, buoyancy, has been separately reviewed in section 1.5. The reviewed literature suggests that after an adequate travel distance, buoyancy dominates the plume development, leading to its natural lift-off. Two different lift-off criteria have been selected from the literature which, together with the formulae of the mean jet characteristics, are applied to the theoretical analysis of this work in chapter 5. Furthermore, the philosophy behind the installation of a wind brake system with respect to the acceleration of the lift-off of the plume has been explained in detail in section 1.5.

Finally, section 1.6 summarized modern *CFD* techniques available to model turbulence, by reviewing the literature of previous numerical investigations which

have focused in jet dynamics and mixing. This review has provided information which justifies the selection of *LES* for the numerical simulations of this work, highlighted weaknesses of *LES*, and provided the experience of previous work to be used as a starting point in this research.

Chapter 2

Methodology

The environmental issues motivating this research, and the physical phenomena encountered at the early take-off stage of the flight, have been presented in chapter 1. The present chapter explains the methodology followed by this research in order to achieve its goals. In particular, section 2.1 explains in detail the central objective of this work, explains why *LES* has been selected for the numerical investigation. The same section discusses the modelling approach of the real situation, the level of applicability of theoretical analysis, the representation of the pollution in the simulation, and the selected method to identify plume lift-off in the simulation. In the following, section 2.2 describes the post-processing methods used in this work. Then, section 2.3 separately explains the solver validation process, which has been a major part of this work. Finally, section 2.4 provides the details of the computational platform upon which our simulations are performed, names the software used for the various stages of the *CFD* study, and summarises the scripting codes written to support this work.

2.1 CENTRAL OBJECTIVE AND APPROACH

The central objective of this work is to improve understanding of the flow dynamics of a buoyant three-dimensional offset jet (*3DOJ*), representative of the jet engine exhaust plume of typical airliners during the early takeoff stage, while a wind-brake system is installed on the path of its development. Parallel studies in the same direction are carried out by (Bennett *et al.*, 2013) and (Velikov, 2013), the former performing field-trials supported by *LIDAR* measurements, and the latter performing wind-tunnel experiments scaling down the real situation. Although the most straightforward approach is the field-trial investigation, financial cost is a deterministic constraint for extensive tests to be carried out. The experimental investigation reduces the financial cost, but not to a satisfactory level so that extensive parametric tests are feasible. An alternative approach to the problem is the numerical simulation using the science of computational fluid dynamics (*CFD*). A significant component of the physics governing this problem is “*anisotropic turbulent mixing*”, which relies on the most challenging task of a *CFD* algorithm, “*turbulence closure*”. A commonly used *CFD* technique for turbulence closure is *RANS* modelling. This technique has been preferred by the majority of past *CFD* research due to its advantages in terms of computational cost. However, the modelling approximations of *RANS* make its predictions weak for anisotropic turbulent mixing.

Although turbulence consists of a variety of scales, the direct simulation of which is very expensive, or even impossible in most of engineering applications, the evolution of super-computing abilities in modern science has made possible their explicit simulation. Large eddy simulation (*LES*) has attracted a lot of interest in recent *CFD* research because it is highly accurate for *anisotropic turbulent mixing*, combined by a moderate computational cost. In particular, (Mahesh *et al.*, 2007) stated that “*as long as the energy-containing convective motions and the approximate thickness of the scalar fronts are captured, a good prediction of the scalar field will be achieved by LES*”. The *LES* algorithm embedded in the commercial software ANSYS FLUENT is selected for the simulations of this thesis.

Details about the algorithm, as well as information referring to the numerical setup are separately presented in chapter 3. Specific details of the numerical setup of the large eddy simulations of this work are separately presented in chapter 4 including the geometrical details of the computational volumes, as well as the initial and boundary conditions selected for different problems. An alternative approach for the representation of the baffles in the numerical simulation is also introduced in the same chapter.

2.1.1 Modelling the Problem

Accurate numerical representation of the buoyant exhaust plumes of an aircraft at the early takeoff stage would have to include multiple exhaust plumes (usually two or four) which soon merge into a combined *3DWJ*. Then, the initial forward acceleration of the aircraft, as well as the varying thrust power of the jet-engines at this stage of the flight are significant conditions, necessary for an ideal replication of the real situation.

Replication of the varying aircraft position has been attempted in the *3D – URANS* simulations of (Koutsourakis *et al.*, 2006). However, turbulence closure of this work was of low level, therefore, the level of uncertainty of the predictions is high. A more relevant to the approach of this work has been used by (Aloysius *et al.*, 2007; Aloysius & Wrobel, 2009a). The latter report has the title “*Large eddy simulation of plume dispersion behind an aircraft in the takeoff phase*”. In that work a real scale complete geometry of a Boeing 737 was modelled and the plumes were allowed to naturally develop for $42s$, which is the duration of a typical takeoff process according to ICAO time-in mode, with the computational domain extending about 2000 diameters downstream, in a simulation performed by FLUENT-*LES*. Their analysis has been based on instantaneous *LES* passive scalar concentrations, and results were compared with *LIDAR* measurements of relevant field-trials.

The nature of *LES* is such that the flowfield varies randomly in time; hence, *LES*-Experiment cross-validation in terms of instantaneous flow quantities include a high level of uncertainty. For this reason, accurate quantitative analysis is

only possible by means of statistical quantities. In particular, this work considers two levels of statistics, 1st order statistics representing the time averaged value of certain flow quantities in every grid point, and 2nd order statistics representing the root mean square of the same quantities. Long time intervals are required to be simulated in order to obtain these statistical quantities, depending on the type of the problem and the acceptable level of uncertainty. The theoretical analysis of this work, presented analytically in chapter 5, highlights the difficulty in achieving statistical solutions at long farfield locations, therefore, the computational volumes are limited to a maximum of $X/D = 380D$. A-priori definition of the time intervals is possible. The method applied herein is based on theoretical estimation of the mean flow characteristics of jets, and it is analytically presented in section 5.3.

2.1.2 Theoretical Predictions

The literature review of the previous chapter has provided adequate information in order to carry out theoretical analysis and obtain a-priori estimates of what is expected to be encountered in the numerical simulations. In this way, chapter 5 utilises the theoretical formulae describing the mean jet characteristics in the absence of co-flow, and uses certain approximations in order to predict the ones in a co-flowing environment. This forms the basis for the implementation of a comprehensive computational cost analysis, necessary for the simulation setup, time management, and cost evaluation of this work. Additionally, theoretical estimations of the lift-off onset of the plume is obtained by two different theoretical models; the model of (Bennett *et al.*, 2013) which only requires jet inflow conditions and omits the co-flow (wind) velocity is especially useful for the estimation the lift-off onset under the baffle influence. On the other hand, the model of (Hanna *et al.*, 1998), which uses local values of the flow, allows for the co-flow to be included on the theoretical analysis. The high complexity of the problem required significant approximations in order to obtain our theoretical predictions, which for this reason are not suggested to be taken too literally. However, these theoretical predictions provide a-priori estimates of the examined flows, therefore, reducing the time required to setup the numerical investigation.

2.1.3 Representation of the Pollution in the Simulation

The present work does not aim to investigate the environmental impact of the use of the turbofan engine on its source. In other words, the chemical composition of the exhaust plume is not explicitly taken into account. Instead, a passive scalar equation is introduced to the simulation, with an initial value for the scalar $\Phi = 1$ at the nozzle exit, which drops as the jet develops and mixes with the ambient air. The actual concentration of specific combustion products can be later obtained using the local concentrations of Φ and the nozzle exit chemical composition. Of course, the chemical reactions taking place in the initial turbulent region of the jet might be significant in some cases, i.e. the after-nozzle evolution of CO and NO discussed by (Borghi, 1975), and black carbon aerosol discussed by (Petzold *et al.*, 1999). However, such reactions are considerable only near the nozzle where the jet temperature is high; therefore, the pollutant concentrations can be later corrected during the final air quality evaluation.

2.1.4 Identification of Lift-off of the Plume

As explained in section 1.5, lift-off onset is the distance from the nozzle where the buoyant forces equal the Coandă effect and their contribution is significant to the flow dynamics. From the literature review and the theoretical analysis of the buoyant plume it can be understood that the high complexity of the present problem does not allow for a robust lift-off criterion to be defined without significant approximations. For this reason, it has been decided that it is more reliable to observe the plume mean flow characteristics, i.e. the shape of the plume, at different streamwise locations, and from this to identify whether the plume behaves similar to a cold $3DWJ$ or if buoyancy has changed this behaviour. This is achieved by comparing the hot jet simulation results to the ones of a, identical otherwise, cold jet. Similarly, the effect of the baffles to the plume lift-off is observed by comparing the results when the baffles are present or not. An advanced post processing method has been used in order to facilitate the comparison between results of multiple simulations (refer to section 2.2).

2.2 POST-PROCESSING METHODS

Depending on the requirements of the investigated setups, various methods are being employed for the post-processing of the *LES* results. General description of these methods is made in the following paragraphs, while expanded details are presented together with the description of each investigated setup in chapter 6.

2.2.1 Axisymmetric Jets

The statistical quantities of the axisymmetric jet simulations are relatively simple to analyse as the problem is effectively two dimensional. This means that all radial profiles of a specific station are identical to each other. A conventional method is, therefore, selected for the axisymmetric jet post-processing. According to this method, cross-sectional profiles of the investigated statistical quantities are plotted at specified streamwise locations. The streamwise locations agree with the ones of the relevant experiments. Although the majority of the free-jet results have been post-processed and cross-validated based on this approach, this method presented in the following paragraph has been also applied for the comparison with theoretical values.

2.2.2 Three-Dimensional Offset Jets

In *3DOJ* simulations axisymmetry no longer exists, therefore, post-processing is more complicated. The conventional post-processing approach is extended in two directions with the resulted number of profiles being larger than in the axisymmetric case (refer to the discussion of section 8.1.1). This makes the analysis and discussion of the results difficult and vague. Furthermore, the analysis of the conventional approach is limited to a small number of streamwise locations and no information is analysed in between those stations. On the other hand, the mean flow characteristics of a *3DWJ* are well described by certain local values of velocity and concentration, as well as their locations upon a specific cross-section. Additionally, a *3DOJ* involves multiple transitions, which in order to be observed

require analysis of a large number of streamwise stations. Achieving such a post-processing detail in the streamwise direction is also essential for the results of the final simulations, where obstacles (baffles) lie at various farfield locations, causing additional transitions to take place. For these reasons, *3DOJ* post-processing is based on a more sophisticated method utilizing the TecPlot scripting language, as described in section 8.1.2 (code is available in Appendix G.2). The streamwise evolution of mean-velocity characteristics has been commonly used in the analyses of various *3DWJ* experimental investigations in the past, including the ones of (McLean & Herring, 1974), (Davis & Winarto, 1980), and the most recent one of (Agelin-Chaab & Tachie, 2011). Recent *LES* have also used the same analysis method, such as the plane jet *LES* of (Dejoan & Leschziner, 2005) and the *3DWJ LES* of (Zhi-wei *et al.*, 2012). Unlike present work, none of these studies have extended the streamwise post-processing method to the mean concentration characteristics.

2.2.3 Final Results

Analysis of the final results falls in two categories. The first category refers to cross-validations of the numerical approach, and is carried out by direct comparison of the *LES* predictions with the experimental data in terms of vertical profiles of mean streamwise velocity and mean scalar concentration at various streamwise locations, as well as cross-sectional contours of the same statistical quantities before and after the baffle region. The second category involves comparisons between results of different simulations, aiming to parametrically investigate the *LES* predictions. Different investigated parameters include variations of the geometrical characteristics of the wind-brake system, as well as variation of numerical and physical parameters. Further to the methods used for the cross-validations, post-processing of the second category is extended in two-dimensional contours of mean and instantaneous streamwise velocity and passive scalar concentration on vertical and horizontal interior planes, capturing the streamwise evolution of the *3DOJ*.

2.3 VALIDATION OF THE NUMERICAL APPROACH

Validation of the numerical approach is a necessary step of a *CFD* investigation. Although FLUENT-*LES* has been validated by numerous researchers in the past, the problem of the main thesis objective is highly complicated considering the multiple parameters composing it. For this reason, validation of the solver is required to be carried out in a sectionalized pattern. In particular, progressive parametric validations have been scheduled, each one gradually increasing in setup complexity.

An essential part of the initial stages of this work has been the establishment of an efficient meshing approach (refer to chapter 4.2). Additionally, a robust numerical setup is selected, including initial and boundary conditions, as well as numerical schemes for the discretization of the governing equations (refer to chapter 4). Further to these basic numerical parameters, the numerical solution relies on a number of secondary parameters, such as the convergence criteria, timestep size, jet inlet turbulence intensity, shear layer resolution, under relaxation factors, temporal convergence, mesh density, pressure coupling, *SGS* modelling, and mesh isotropy in the farfield. All these factors are being parametrically investigated on a relatively simple case consisting of a low speed axisymmetric cold jet in strong co-flow in chapter 7, with the aid of experimental data available from (Garry & Holt, 2008), also compared with previous *LES* carried out by (Dinesh *et al.*, 2009). Within the same chapter, the validation is extended to the setup of a high speed co-axial jet in stagnant environment. The ability of the numerical approach to reproduce experimental data and previous high resolution *LES* results is available from (Eastwood *et al.*, 2008), is being assessed. Furthermore, the importance of accurate representation of the complex turbofan nozzle geometry in the simulation is also evaluated.

An additional parameter of the situation describing the main thesis objective is added to the numerical setup in chapter 8. In this way, the low speed free-jet investigated setup is repeated with the only modification being the addition of a solid surface at a certain offset parallel to the jet centreline. A new paramet-

ric study is carried out retesting certain numerical parameters of the free-shear validation tests. Moreover, this set of tests includes the non-dimensional wall-distance (y^+), and the inlet ground boundary layer of the co-flow. Further to the numerical parameters, the wall-bounded validation tests also include investigation of physical parameters, such as the co-flow velocity variation, the presence or absence of the wall, and the three dimensionality of the jet.

A strong basis for the credibility of the results of this thesis *LES* is set by the free-shear and the wall-bounded validation tests of chapters 7 and 8. There are two additional parameters requiring validation before performing the final simulations corresponding to the main objective of the thesis. One parameter is the representation of the baffles in the numerical simulation. In chapter 9 two different methods are tested using drag measurements on a previously optimized baffle arrangement, available from wind tunnel experiments contacted by (Velikov, 2013). Apart from the drag measurements, the baffle validations also include a small set of experimental data of passive scalar concentrations; however, the major part of the passive scalar cross-validations is found at the final results chapter. Finally, the buoyancy effect on the turbulent mixing process is only investigated in the absence of baffles, and the analysis is presented in section 10.1.3.

2.4 HARDWARE-SOFTWARE-SCRIPTING

Simulations are performed on ASTRAL-cluster platform 3000 DL140G3 of Cranfield University. The cluster consists of 80 nodes. Table 2.1 summarizes the details of Astral nodes. The number of nodes used per job vary between 1 and 8, which corresponds to 16-128 parallel processors, depending on the size of the simulation. The actual speed of the computations vary between simulations due to I/O contention or memory conflicts, as the cluster is used by multiple users. Other machines used for low demand processes include a 64-bit windows machine, Quad-CPU, Q9550@2.83GHz; and a 32-bit windows machine, dual-core, T8300@2.4GHz.

Design and meshing of the computational volumes are performed by the com-

Table 2.1: Astral node configuration details.

2 x Intel E5-2660 2.20GHz/ GHz/20M/8.0GT/s 95W eight core Xeon processor
8 x 8GB DDR3 Registered dual rank DIMM, 1600MHz (total 64GB)
500GB 7200 RPM 3Gbs 3.5" SATA DISK
2 x On-Board GbE NICs
On-board Infiniband Mellanox Connect-X

mercial software GRIDGEN. Simulations are performed with the *LES* algorithm embedded in the commercial software ANSYS FLUENT. Post-processing is performed with the commercial platforms Tecplot360 and Tecplot Chorus.

A variety of short codes have been written during this thesis. One type of code was to develop a fully automatic numerical setup able to execute transient operations in the simulation when running in batch mode, without having to stop and resubmit the same simulation several times. An additional code-executed-task, was the definition of the boundary conditions. Furthermore, sample collection and averaging of the passive scalar field had to be manually implemented through C-programming based user defined functions (*UDF*). Moreover, the dynamic response algebraic model for baffle imitation (*DRAMBI*) has been also written on a C-based *UDF*. Finally, a scripting-based program has been written to perform advanced post-processing operations, consisting by multiple data extractions, manipulation, and plotting. The most significant programming and scripting material is provided in Appendix G.

Chapter 3

Solver Description

The three dimensional large eddy simulations of this thesis are performed with the *LES* algorithm available at the commercial solver FLUENT 12.1. The current chapter provides important information about this solver, adequate to allow future replication of the simulations of this work using a different *CFD* package. Section 3.1 provides an overall summary of the solution strategy. In the following, section 3.2 presents the numerical schemes selected for the discretization of the governing equations in space and time. Then, section 3.3 presents the filtering operation performed by the *LES* solver, followed by a summary of the subgrid scale models used in this study in section 3.4. Section 3.5 presents the near wall modelling, followed by section 3.6 which presents the equations for the passive scalar transport. Finally, section 3.7 presents the selected inlet velocity perturbation method.

3.1 SUMMARY OF THE SOLUTION STRATEGY

ANSYS FLUENT offers a variety of algorithms in order to support a wide range of problems. The selected solution strategy for the simulations of the present work is summarised in this section.

The Mach number of the flows simulated in the present work is considerably low, i.e. $Ma \ll 0.3$, thus the density field can be unlinked from the pressure field¹, without affecting the final predictions. In this way, the pressure-based approach is selected. The pressure-based solver of FLUENT employs an algorithm which belongs to a general class of methods called the projection method.

Mass conservation is achieved by solving a pressure correction equation. The pressure equation is derived from the continuity and the momentum equations in such a way that the velocity field, corrected by the pressure, satisfies the continuity. Two different pressure velocity coupling algorithms are tested in section 7.1.8, which justifies the final selection of the Semi-Implicit Method for Pressure Linked Equations (*SIMPLE*); (Versteeg & Malalasekera, 1995) give a detailed description of this method.

By nature, *LES* is a time dependent computational approach. In this work the solution process is sub-divided to a series of smaller problems separated by each other by time-step size Δt . All the equations are solved iteratively, for a given time-step, until the convergence criteria are met. Advancement of the solutions by one time-step requires a number of outer iterations (5 – 10 as suggested by FLUENT). An overview of the selected iterative time advancement method is presented in (Fig.3.1). Non-linearity of the individual equations and inter-equation couplings are fully accounted for in the iterative approach, thus eliminating the splitting error apparent in the non-iterative one.

Since the governing equations are non-linear and coupled to one another, the solution process involves iterations wherein the entire set of governing equations is solved repeatedly until the solution converges. This is achieved by an Algebraic Multi-Grid (*AMG*) algorithm which solves the system of discretized equations,

¹The variation of density due to temperature is calculated by the incompressible ideal gas law.

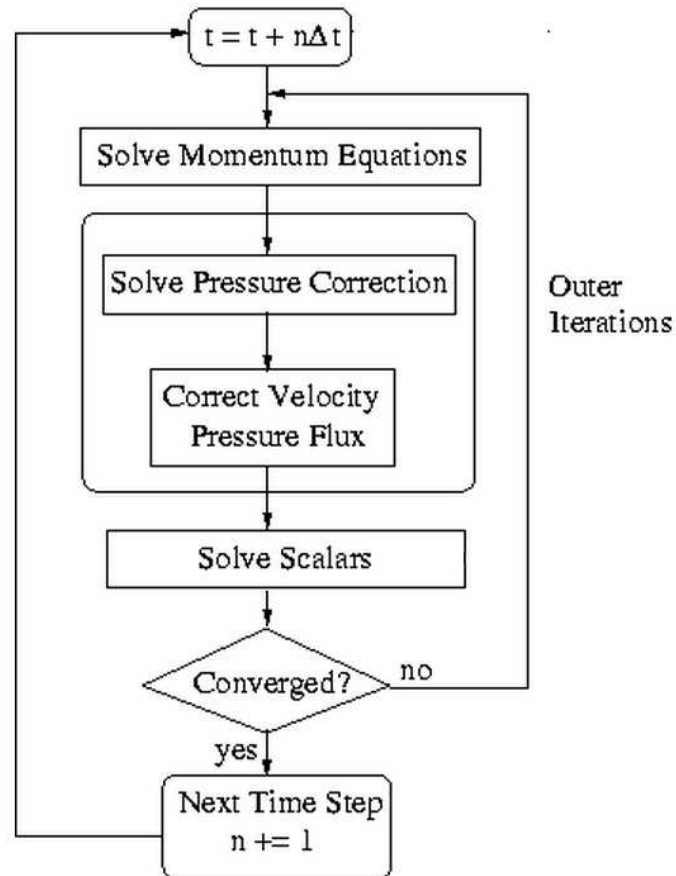


Figure 3.1: Overview of the iterative time advancement solution method (Fluent, 2012).

while the Point Gauss-Seidel relaxation is employed as a smoother (Kim, 2004).

Various schemes are used for the discretization of the governing equations; these are separately presented and discussed in the following section.

3.2 DISCRETIZATION SCHEMES

Considering the unsteady conservation equation for transport of a scalar quantity ϕ in integral form for an arbitrary control volume V we have:

$$\int_V \frac{\partial \rho \phi}{\partial t} dV + \oint \rho \phi \vec{v} \cdot d\vec{A} = \oint \Gamma_\phi \nabla \phi \cdot d\vec{A} + \int_V S_\phi dV \quad (3.1)$$

where ρ is the density, $\vec{v}$ is the local velocity vector, $\vec{A}$ is the surface area vector, Γ_ϕ is the diffusion coefficient for ϕ , $\nabla \phi$ is the gradient of ϕ , and S_ϕ is the source of ϕ per unit volume.

Discretization of (eq.3.1) on a given cell yields:

$$\frac{\partial \rho \phi}{\partial t} V + \sum_f^{N_{\text{faces}}} \rho_f \vec{v}_f \phi_f \cdot \vec{A}_f = \sum_f^{N_{\text{faces}}} \Gamma_\phi \nabla \phi_f \cdot \vec{A}_f + S_\phi V \quad (3.2)$$

where N_{faces} is the number of faces enclosing a computational cell ($N_{\text{faces}} = 6$ for the structured meshes used in this work), ϕ_f is the value of ϕ convected through a face, $\rho_f \vec{v}_f \cdot \vec{A}_f$ is the mass flux through the face, $\vec{A}_f$ is the area of the face, $\nabla \phi_f$ is the gradient of ϕ at the face, and V is the cell's volume.

TEMPORAL DISCRETIZATION is applied at the first term of (eq.3.2) by an implicit second order backwards differencing scheme, such as:

$$\frac{3\phi^{n+1} - 4\phi^n + \phi^{n-1}}{2\Delta t} = F(\phi) \quad (3.3)$$

where $n + 1$ represents the value at the next time level, $t + \Delta t$, n is the value at the current time level, t , and $n - 1$ is the value at the previous time level, $t - \Delta t$. The function $F(\phi)$ incorporates any spatial discretization. In order to have unconditionally stable convergence with respect to the time step size Δt , $F(\phi)$ is evaluated at the future time level using an implicit integration method.

SPATIAL DISCRETIZATION is required for the evaluation of the face values ϕ_f of the convection terms in (eq.3.2).

The momentum equations are obtained by setting in (eq.3.2) $\phi = u, v, w$, for the x , y , and z -momentum equations respectively. A second-order central-differencing scheme (eq.3.4) is applied to calculate the face value for a variable (ϕ_f), which, as suggested by (Fluent, 2012), provides improved accuracy for *LES*.

$$\phi_f = \frac{1}{2} (\phi_0 + \phi_1) + \frac{1}{2} (\nabla \phi_0 \cdot \vec{r}_0 + \nabla \phi_1 \cdot \vec{r}_1) \quad (3.4)$$

The indices 0 and 1 refer to the cells that share face f . $\nabla \phi_{r,0}$ and $\nabla \phi_{r,1}$ are the reconstructed gradients at cells 0 and 1, respectively, and $\vec{r}$ is the vector directed from the cell centroid toward the face centroid.

All other transport equations are discretized by a second order upwind scheme, according to which the face value ϕ_f is computed using the following expression:

$$\phi_{f,SOU} = \phi + \nabla \phi \cdot \vec{r} \quad (3.5)$$

where ϕ and $\nabla \phi$ are the cell-centred value and its gradient in the upstream cell respectively, and $\vec{r}$ is the displacement vector from the upstream cell centroid to the face centroid. Finally, the gradient $\nabla \phi$ is evaluated based on equation (eq.3.6), and is limited so that no new maxima or minima are introduced.

GRADIENTS AND DERIVATIVES of the scalar ϕ at the cell centre $c0$ are evaluated based on the Green-Gauss theorem, in the following discrete form:

$$(\nabla \phi)_{c0} = \frac{1}{V} \sum_f \bar{\phi}_f \vec{A}_f \quad (3.6)$$

where ϕ_f is computed by the “*least squares cell-based technique*”, which, together with the “*Green-Gauss node-based*”, is the most accurate technique available in FLUENT.

DIFFUSION TERMS are central-differenced and are second-order accurate.

3.3 FILTERING THE NAVIER-STOKES EQUATIONS

The application of *LES* requires filtering of the $N - S$ equations in either Fourier space or in physical space. In this way, eddies smaller than the filter width, are filtered out. The dynamics of the large eddies are therefore governed by the resulted equations. A filtered variable is defined by:

$$\bar{\Phi}(x) = \int_D \Phi(y) G(x, y) dy \quad (3.7)$$

where D stands for the computational domain, $y \in V$ and G is the filter function which determines the minimum size of the scales to be resolved. The spatial discretization is implemented with a finite volume scheme. Therefore the discrete solution of $\bar{\Phi}_{(x)}$ is defined as:

$$\bar{\Phi}(x) = \frac{1}{V} \int_D \Phi(y) dy \quad y \in V \quad (3.8)$$

where V is the cell volume. equation 3.8 represents a filtering operation. Therefore, the implied filter function $G_{(x,y)}$ is a top-hat filter given by:

$$G(x, y) = \begin{cases} \frac{1}{V} & \text{for } |x - y| \in V \\ 0 & \text{otherwise} \end{cases} \quad (3.9)$$

For the incompressible $N - S$ employed in our simulations, we can write:

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} + \frac{\partial}{\partial x_j} \left(\nu \frac{\partial \bar{u}_i}{\partial x_j} \right) \quad (3.10)$$

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (3.11)$$

where τ_{ij} is the subgrid-scale stress (*SGS*) defined by:

$$\tau_{ij} \equiv \overline{u_i u_j} - \bar{u}_i \bar{u}_j \quad (3.12)$$

3.4 SUB-GRID SCALE STRESS SGS MODELLING

As shown in the previous section, the filtering operation results in *SGS* turbulent stresses, which require closure. The model proposed by (Smagorinsky, 1963), is based on isotropic eddy viscosity. The *SGS* stresses are therefore computed by:

$$\tau_{ij} - \frac{1}{3}\tau_{kk}\delta_{ij} = 2\nu_t S_{ij} \quad (3.13)$$

where ν_t is the *SGS* eddy viscosity, and S_{ij} is the resolved rate of strain tensor. The closure problem is therefore limited in determining the value of ν_t .

3.4.1 Standard Smagorinsky Model (SSM)

Here the *SGS* eddy-viscosity is calculated by:

$$\nu_t = C_S \bar{\Delta}^2 |\bar{S}| \quad (3.14)$$

where C_S is a model constant, $|\bar{S}| \equiv \sqrt{2\bar{S}_{ij}\bar{S}_{ij}}$ and $\bar{\Delta}$ is the grid filter length defined by:

$$\bar{\Delta} = V^{\frac{1}{3}} \quad (3.15)$$

The *SGS* stresses can now be computed by:

$$\tau_{ij} - \frac{1}{3}\tau_{kk}\delta_{ij} = 2C_S \bar{\Delta}^2 |\bar{S}| S_{ij} \quad (3.16)$$

SSM is a simple, economic, and robust model; however, its major drawback is that there is no single value of C_S to be valid throughout the whole computational domain; therefore, a dynamic calculation of the model constant C_S , throughout the computational domain, is required.

3.4.2 Dynamic Smagorinsky Model (DSM)

In order to solve the issue of C_S described above, (Germano *et al.*, 1991) and later (Lilly, 1992) proposed a procedure at which the model constant C_S can be dynamically calculated throughout the computational domain by using information provided by the resolved scales of motion. This dynamic procedure performs an additional filtering operation utilizing a test filter $\tilde{\Delta}$ to identify the small coherent structures of the resolved field. Therefore, the obtained model constant C_S varies in space and time, and it is possible to take both positive and negative values. The possibility of negative values of C_S is a desirable attribute of dynamic models, since it can be interpreted as flow of energy from the small-unresolved scales to the large-resolved structures; a procedure referred to as “*back-scatter*”. However, as stated by (Kim, 2004), “*large negative values of eddy viscosity cause numerical instability and may result in excessive level of numerical noise or even divergence of the solution*”. Therefore, within fluent the minimum value of C_S is clipped at zero.

3.4.3 Wall-Adapting Local Eddy-Viscosity (WALE)

WALE was proposed by (Nicoud, 1999) for *LES* in complex geometries. This model is based on the square of the velocity gradient tensor. A main advantage of *WALE*, comparing to the classical Smagorinsky formulation, is that it is able to detect all the turbulence structures relevant for the kinetic energy dissipation rate. This is achieved through the mixing of both the local strain and rotation rates in the construction of the spatial operator. Another advantage is that the eddy-viscosity goes naturally to zero close to walls, and therefore there is no need for a dynamic constant or a damping function in this flow regime. Finally, *WALE* produces zero eddy-viscosity in a case of pure shear, such as a free-shear transitional jet where no turbulent kinetic energy exists near the cut-off (Ducros *et al.*, 1998). Therefore, transition to turbulence is reproduced through the growth of linear unstable modes.

3.5 LES NEAR-WALL TREATMENT

The no slip condition applies near solid surfaces. A common outcome of a large number of experiments is that the region near a solid surface can be largely subdivided into three layers. The first layer, adjacent to the solid surface, is called the viscous sublayer. Within this region, the flow is almost laminar, and the (molecular) viscosity plays a dominant role in momentum and heat or mass transfer. The next layer is the fully turbulent region; here turbulence dominates the flow. Between these two regions there is an interim where both the effects of viscosity and turbulence are important. This is called the buffer region.

In FLUENT-LES algorithm, when the resolution of the mesh near the wall is adequately high to resolve the laminar sublayer the wall shear stress is obtained from the laminar stress-strain relationship:

$$\frac{\bar{u}}{u_\tau} = \frac{\rho u_\tau y}{\mu} \quad (3.17)$$

In the alternative case where the resolution is not adequately dense to resolve the laminar sublayer, then fluent assumes that the centroid of the wall-adjacent cell falls within the logarithmic region of the boundary layer, and the law-of-the-wall is employed:

$$\frac{\bar{u}}{u_\tau} = \frac{1}{\kappa} \ln E \left(\frac{\rho u_\tau y}{\mu} \right) \quad (3.18)$$

where κ is the von Kármán constant and $E = 9.793$. If the mesh is such that the first near wall point is within the buffer region ($3 \leq y^+ \leq 10$), then the two above laws are blended in accordance with the function suggested by (Kader, 1981).

3.6 PASSIVE SCALAR EQUATIONS

The passive scalar field is calculated from equation:

$$\frac{\partial \rho \Phi}{\partial t} + \frac{\partial}{\partial x_i} \left(\rho u_i \Phi - \Gamma \frac{\partial \Phi}{\partial x_i} \right) = 0 \quad (3.19)$$

where Φ is the passive scalar, Γ the scalar diffusion coefficient and ρ the density.

The subgrid-scale turbulent flux of a scalar, is modelled using a subgrid-scale turbulent Prandtl number by:

$$q_j = \frac{\mu_t}{\sigma_t} \frac{\partial \Phi}{\partial x_j} \quad (3.20)$$

In the dynamic models, the subgrid-scale turbulent Prandtl number is obtained by applying the dynamic procedure, described above, to the subgrid-scale flux.

3.7 INLET RANDOM FLOW GENERATION

The inlet boundary of present work simulations represents a jet emerging from a circular nozzle. In the literature review of jets in section 1.4 it was explained that when a jet exits an orifice undergoes transition to turbulence under a series of events. Ancestor of all those transitional events is the existence of infinitesimal perturbations at the jet as (Lesieur, 2007) explains. Such perturbations disrupt the axisymmetry of the jet, therefore, allowing for asymmetric instabilities to develop, leading to a natural transitional process. In real flows such perturbations always exist, but in the numerical environment this is not the case. For this reason it is essential to transiently disturb the mean velocity profile at the inlet.

FLUENT provides two different algorithms to do that: “*Vortex Method*” and “*Spectral Synthesizer*”. After initial tests it was found that both methods respond similarly to this problem, but the former affects the convergence speed. For this reason Spectral Synthesizer is selected to disturb the jet at the inlet boundary. Spectral Synthesizer is a method of generating fluctuating velocity components. It

is based on the random flow generation technique originally proposed by (Kraichnan, 1970), later modified by (Smirnov & Shi, 2001). In this method, fluctuating velocity components are computed by synthesizing a divergence-free velocity-vector field from the summation of Fourier harmonics. In the implementation in FLUENT, the number of Fourier harmonics is fixed to 100.

In the present work the method is applied to the streamwise velocity component, U , with an intensity $I = 5\%$ selected after initial testing (refer to section 7.1.4 for this test).

Chapter 4

Numerical Setup

The present chapter presents specific details of the numerical setup of the large eddy simulations of this work. In particular, section 4.1 presents the philosophy followed for shaping the computational volumes, as well as the initial and boundary conditions selected for different investigated setups. Then, section 4.2 describes the different meshing approaches used in this work, explains the improvements applied in order to accelerate the solution convergence, and provides analytical details of the final mesh, necessary for its future replication. Section 4.3 introduces an alternative concept for the representation of the baffles in the numerical simulation. A review of relevant methods in the literature is carried out, and a conceptual discussion leading to a model able to satisfy the needs of the present investigation is made. The last paragraph of section 4.3 provides a detailed description of the baffle representation model designed for the present work and highlights its advantages comparing to other models and to the conventional baffle representation method.

4.1 BOUNDARY AND INITIAL CONDITIONS

The solution of *CFD* problems depends on the information specified at the boundaries of the computational domain. In particular, for the simulations of this work the selected boundary conditions include velocity inlets for the jet flow and co-flow (wind), and an outflow at the exit plane. The lateral boundaries are set to a cylindrical symmetry when the co-flow velocity is large as the jet expansion rate is low so that the inlet and the outflow planes is possible to be designed with identical spatial dimensions (Fig.4.2) and (Fig.4.3). Otherwise, in the case of low co-flow velocity the jet expansion rate is high; therefore, the outflow plane needs to be much larger than the inlet. In this case, symmetry condition is not appropriate for the lateral boundaries, and the velocity inlet covers the lateral boundary all the way to the outflow plain (Fig.4.4). Finally, no-slip condition is set to the horizontal solid surface, representing the ground, in the *3DWJ* simulations.

The lateral boundaries of the computational domain are set at adequate distance from the jet/plume flow, so that there are no significant interactions between them. Similarly, the outflow plane is set at $[X \geq FLI]$ (refer to section 5.3 for the definition of *FLI*).

Profiles of velocity, passive scalar and temperature are set to the inlet boundary by means of user defined functions (refer to Appendix G.1). In order to imitate the nozzle exit (jet-inlet) flow unsteadiness, necessary for the jets natural transition to turbulence, the “*spectral synthesizer*” method available in FLUENT (refer to section 3.7) is being applied to the streamwise velocity profile (Fig.4.1).

Initialisation is required for flow quantities, in each computational cell, in order for the solution convergence to begin. Depending on the type of the simulation, the initialization method varies. For instance, In the relatively simple case of the axisymmetric jet in strong co-flow, constant initial values are set throughout the domain letting the information to propagate from the inlet plane towards the outflow. The strong co-flow considerably accelerates this process (first term of equation 5.5). Simulation of the high Reynolds number co-axial jet, developing in stagnant fluid, brings about solver instability issues, especially during the early

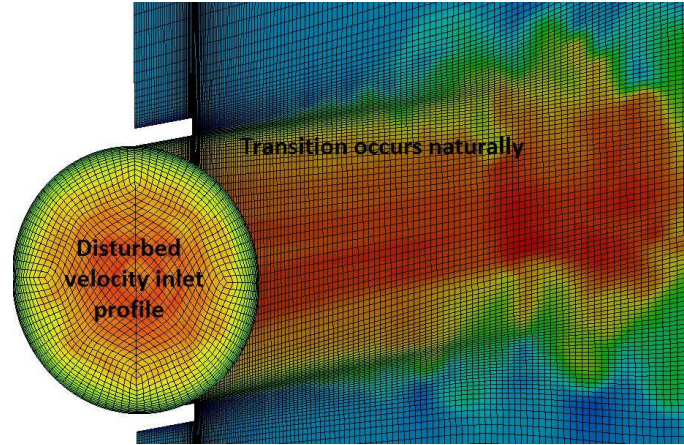


Figure 4.1: Application of the spectral synthesizer method, available in FLUENT, disturbing the streamwise velocity component of the profile set at the jet velocity inlet. Contours represent streamwise velocity

instance when the jet is still laminar but highly unstable to remain in this state; therefore, undergoing a massive transition. However, the time averaged results of a steady *RANS* simulation spatially disturbed to form a realistic *LES* environment, form more realistic initial conditions for the *LES*; hence solving this stability issue (refer to section 7.2 for detailed presentation of the application of this method). The latter method was found inappropriate for the simulation of *3DWJs*, especially in the case of low-co-flow speeds. The artificial spatial disturbance of the *RANS* field causes velocity and passive scalar gusts in the co-flowing environment. These flow disturbances take long time periods to be “*washed out*” from the outflow plane due to the small wind speed, highly increasing the first term of equation (5.5). In this case, it was found more efficient to sweep the co-flow velocity profile (boundary layer at the inlet) through the computational domain and then allow a natural development of the jet inside this environment.

When the simulation is part of a parametric study and the results of a previous simulation with similar conditions are available, interpolation of the flow variables of the previous simulation is applied to the computational mesh of the new simulation. This is clearly the most efficient method of initialization, considerably reducing the first term of equation (5.5). Finally, the timestep size Δt is selected so that the number of iterations per timestep is on the range 6 – 8, as suggested by (Fluent, 2012).

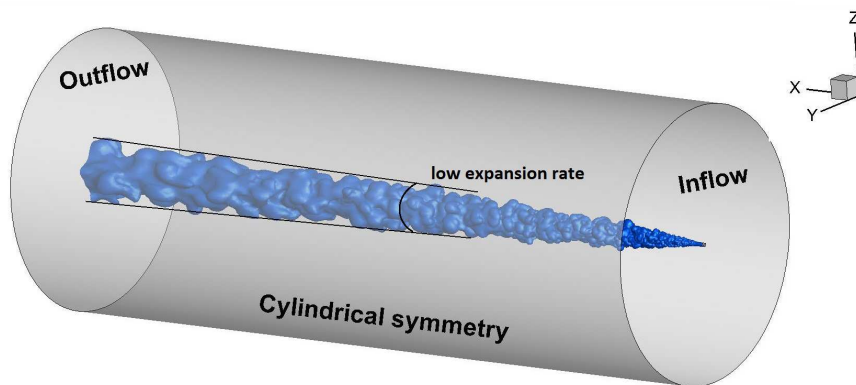


Figure 4.2: Configuration of the domain boundaries of a high co-flow velocity axisymmetric jet simulation.

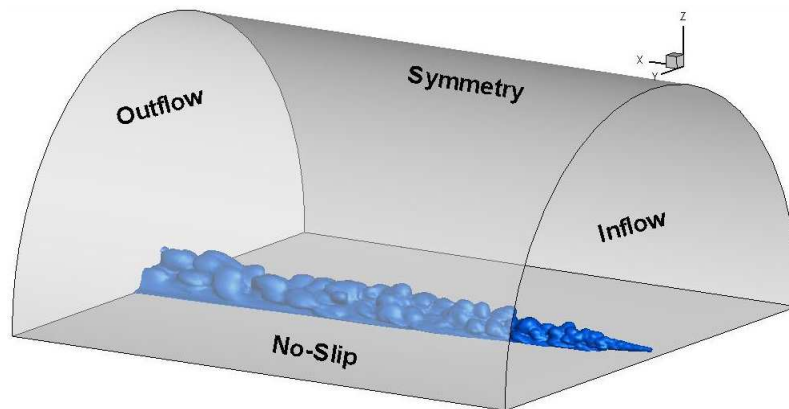


Figure 4.3: Configuration of the domain boundaries of a high co-flow velocity 3DWJ simulation.

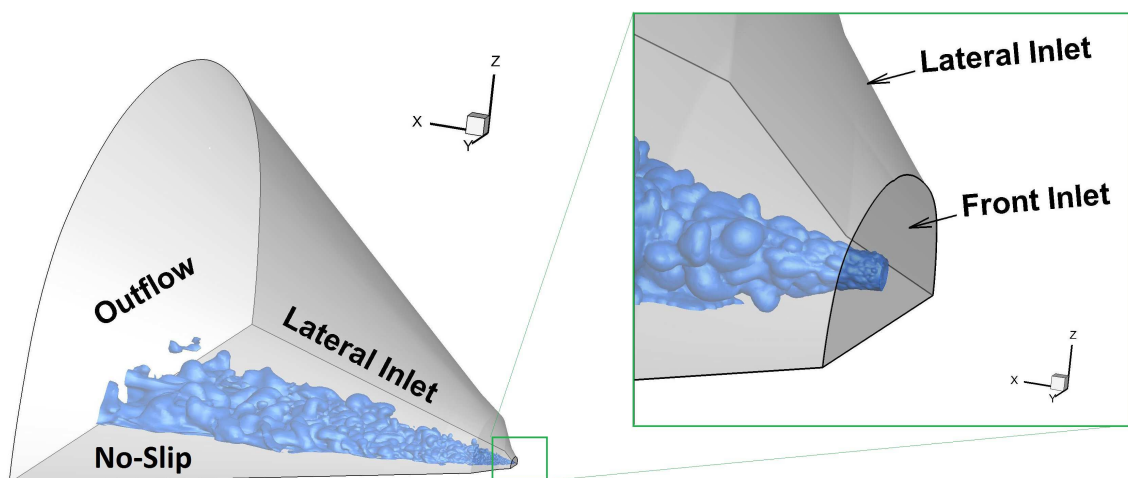


Figure 4.4: Configuration of the domain boundaries of a low co-flow velocity 3DWJ simulation.

4.2 GRID GENERATION

It is attempted to develop a meshing approach for the simulation of *axisymmetric jets* and *three dimensional wall jets (3DWJ)*. The great majority of the numerical investigations reported in the reviewed literature have employed the structured meshing approach in both cases. In particular, different techniques of structured meshing such as Regular Mesh, O-Type Mesh, and Cylindrical Mesh (with singularity points along the axis of symmetry) have been reported; all found to be appropriate for the large eddy simulation of jets. In *LES*, it is important choosing appropriate resolution in different regions of the flow; however, this depends on the imminent solver and subsequent *SGS* modelling, while it forms an important part of a necessary parametric study.

Different meshing techniques have been employed and tested, in both the free-shear axisymmetric jet and *3DWJ* simulations, in this thesis. Description of these techniques is provided in the following sub-sections.

4.2.1 Axisymmetric Jet Simulation Meshing

Having performed initial tests with unstructured meshing, hybrid structured / unstructured meshing, cylindrical meshing, H-Type meshing and O-Type meshing, it was quick to find out that the latter combines mesh quality, high solution convergence rates, and reduced computational cost. For this reason, O-Type meshing is the selected approach for the free-shear (axisymmetric) jet simulations.

The main features of the O-Type meshing used here are: the H-Type regular core which also defines the azimuthal resolution, and the number of layers of radially distributed cells, expanding isotropically in the lateral directions (Fig.4.5).

The main challenge of this cross-sectional cell distribution was the high skewness of the first layer of radial cells neighbouring with the rectangular H-Type core. This is a consequence of the imbalanced angle distribution, especially at the cells close to the four corners of the square core. This problem has been overcome by modifying the square shape of the core, balancing the angles, and with the application of smoothing functions available in the commercial software GRIDGEN in

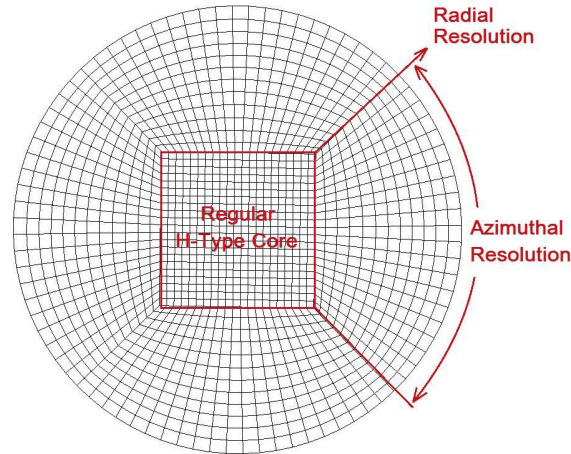


Figure 4.5: Key dimensions of the cross-section of the axisymmetric jet O-Type mesh.

order to universally minimize cell skewness (Fig.4.6).

The O-Type cross sectional cell distribution allows for straightforward description of the “*circular*” nozzle geometries used in this work. Additionally, it facilitates the local increase of resolution at the initial mixing layer region close to the nozzle (Fig.4.7), also being able to apply in coaxial nozzles representative of turbofan engines (Fig.4.8).

Preliminary simulations of the “*Free-Shear Validation Tests*” (refer to chapter 7) have shown that highly resolved computations of the near-field region only favour the near-field results, with the farfield solution being nearly independent of near-field inaccuracies. Therefore, reduction of the computational cost can be achieved by completely omitting the solid nozzle geometry and the high mesh resolution across the initial mixing layer as illustrated in (Fig.4.9). On the left-hand side of this figure it is shown that the H-Type core of the simplified mesh covers the full jet region in the near-field. This allows us to further increase the azimuthal resolution (Fig.4.5), keeping the mesh size at low level. It can also be observed that the resolution is similar at both the core and the mixing layer regions. This brings about an additional advantage, being the considerable reduction of the streamwise mesh resolution in the near-field, while in the same time the value of the cells’ aspect ratio is kept at low level. Therefore, the simplified meshing

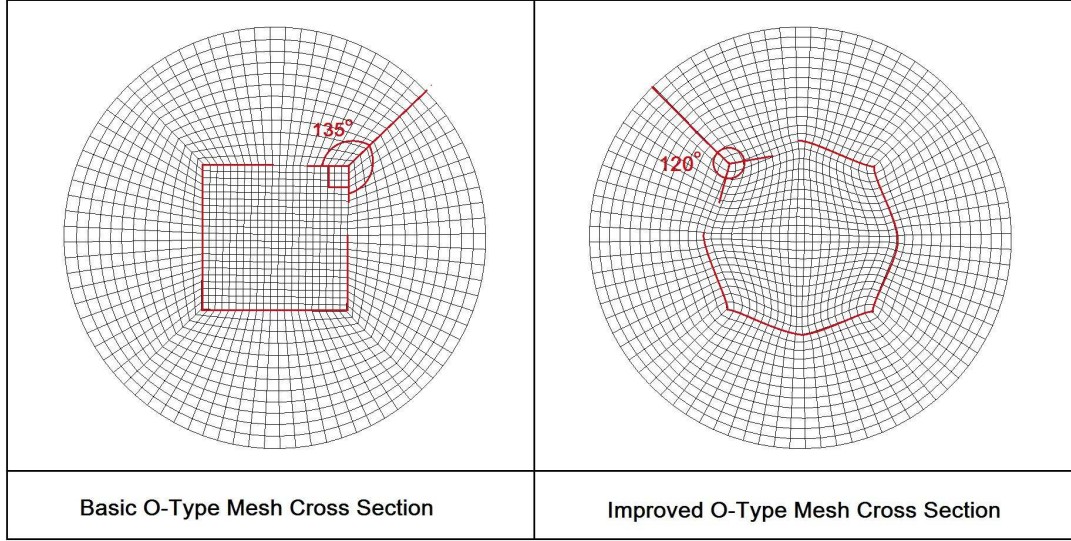


Figure 4.6: Improvement of the cross sectional cell distribution of the O-Type mesh.

approach allows closely isotropic¹ cells throughout the jet development.

In the self-similar region, the core of the O-Type mesh expands in the lateral directions with downstream distance according to the axisymmetric jet expansion rate which is described by equation (5.4). This means that the core jet flow is discretized by the H-Type core of the O-Type mesh, and the outer mixing layer by the cylindrical surrounding mesh.

A common strategy in this thesis has been to set a control section at $X/D = 15$, where transition is complete, with dimensions defined by observation of the unsteady results of a test-simulation. It is not reasonable to use theoretical functions in order to estimate the dimensions of section $X/D = 15$ because the *LES* solution in this region is still affected by the transitional errors following the mesh simplification described above.

Reduction of the streamwise resolution of the mesh is limited by the aspect ratio of the cells lying inside the mixing layer regions, as revealed by the results of the “*Free-Shear Validation Tests*” (refer to section 7.1.9).

¹the term *isotropic cell* stands for a cell with three dimensional aspect ratio (AR_{3D}) close to 1

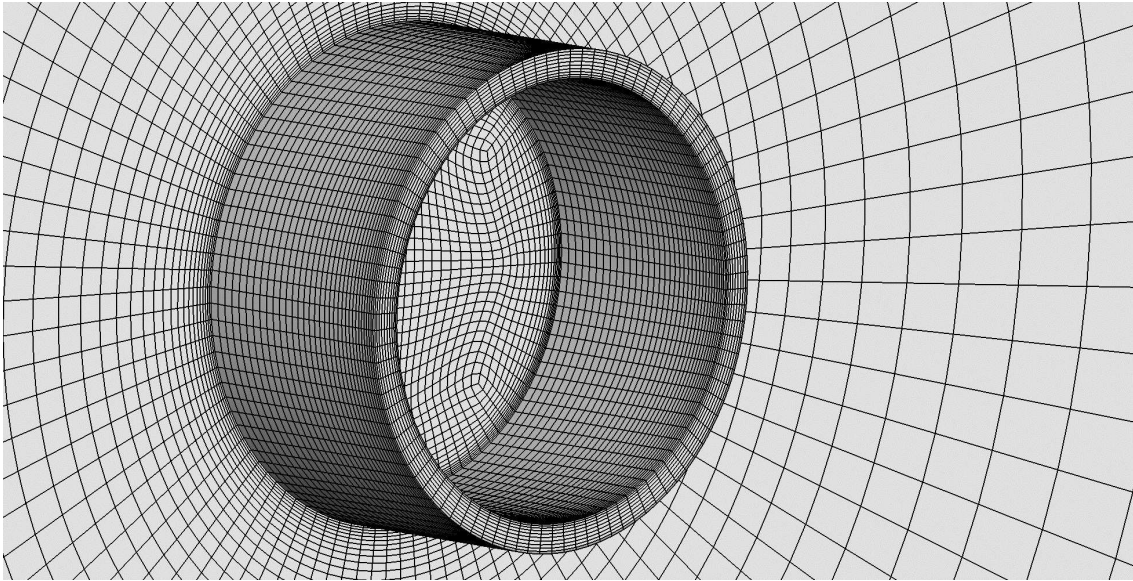


Figure 4.7: Single circular nozzle O-Type meshing.

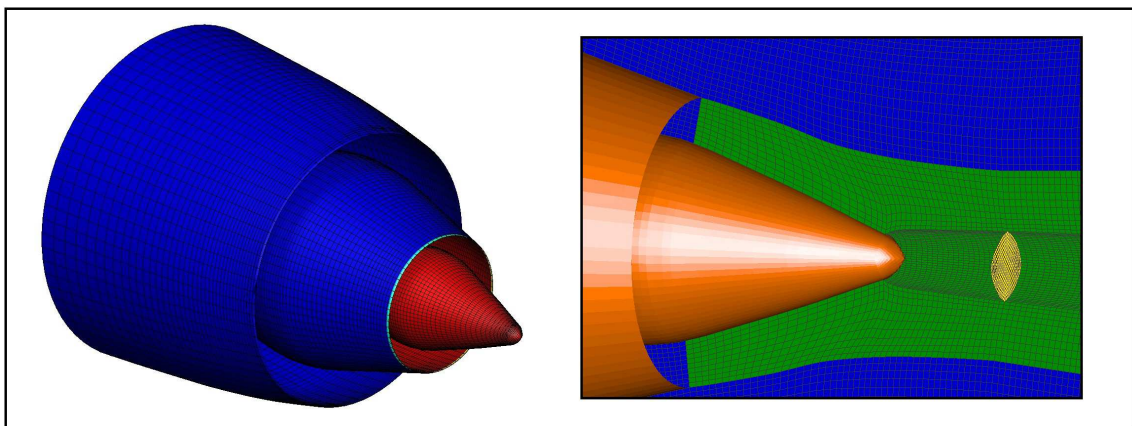


Figure 4.8: Coaxial nozzle O-Type meshing.

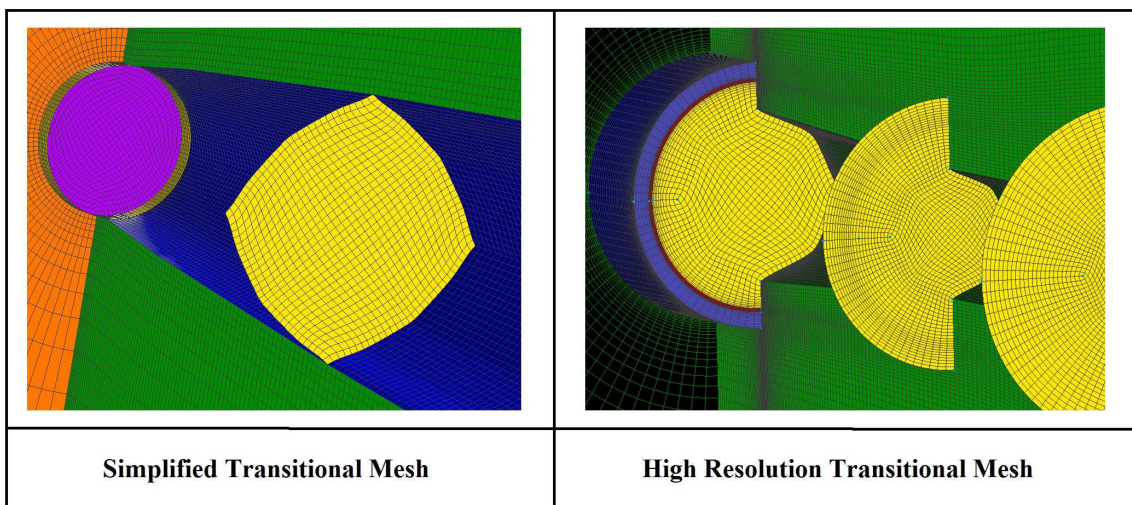


Figure 4.9: Near field mesh simplification.

4.2.2 3DWJ Simulation Meshing

The presence of a solid surface in proximity to the jet development disturbs the axisymmetric evolution of the jet. Consequently, the axisymmetric O-Type meshing technique used for the free-shear case above, is no longer appropriate for *3DWJs*.

A straightforward approach to discretize the computational volume of a *3DWJ* is the H-Type meshing. This is due to its direct alignment with the flat nature of the *3DWJ* flow, which is caused by the high lateral to vertical expansion rates as presented in section 1.4.3.

The three dimensional configuration of the H-Type meshing approach is illustrated in (Fig.4.10). Contours of Φ are colouring the wall surface in order to highlight the position of the jet within the three dimensional volume. Furthermore, iso-surfaces of $\Phi = 20\%$ are also included to show the initial three dimensional jet development. The mesh lines drawn on the inlet boundary are cut-off at a radius $R \geq 2D$ for clarity purposes, while the zoom-in view on the left-hand side highlights the curvilinear modification of the mesh at the inlet boundary in order to accurately describe the round-shaped jet profiles of Φ and U . Volume mesh lines are also drawn on a cross-section at $X/D = 40$ representing the developing wall-jet region. The downstream expansion of the mesh is dictated by the lateral and normal growth rates of the *3DWJs*.

The results of the “*Wall Bounded Jet Validation Tests*” (refer to chapter 8) have revealed that the number of cells required for H-Type meshing is high, imposing a great impact to the computational cost of the simulation. In order to overcome this issue, a modified version of the O-Type mesh employed at the simulation of free-shear jets, has also been tested at the wall bounded jet validation tests. The results revealed the superiority of O-Type meshing in terms of computational cost, with the predictions of the two meshes being similar. (Fig.4.11) illustrates the three dimensional configuration of the modified O-Type meshing; the format is identical to (Fig.4.10).

The main weakness of the modified O-Type mesh is the difficulty to be ap-

plied at low co-flow speeds. For instance, considering a $3DWJ$ simulation with zero co-flow, the same number of cells initially being included “*inside the nozzle*”, downstream is required to canonically expand in a continuously elongated (in the lateral direction) cross sectional area, the dimensions of which are dictated by the anisotropic $3DWJ$ expansion. On the other hand, the complete confinement of the square core by the radial surrounding mesh imposes limits to the aspect ratio of the cross section; therefore, an alternative approach is required for the low co-flow speed $3DWJ$ simulations.

Based on the discussion above, the final $3DWJ$ meshing approach incorporates features of both the H-Type and O-Type approaches. In particular, this approach combines the canonical cell distribution of the H-Type mesh within the jet region, and the radial expansion of the O-Type mesh surrounding the jet region. The advantages of the former include: streamwise mesh-spreading flexibility, meshing simplicity, and increased dimensional² control. The main advantage of the latter is the considerable reduction of mesh density in regions where it is not needed.

(Fig.4.12) illustrates the three dimensional configuration of the final, hybrid H-Type/O-Type meshing. The format is similar to figures 4.11 and 4.10. In addition, (Fig.4.12) includes contours of Φ on the cross section $X/D = 40$, highlighting the mesh window through which the developing $3DWJ$ passes. It is noteworthy that the number of cells resolving the $3DWJ$ flow (H-Type core at $X/D = 40$) is much larger than the number of cells resolving the initial axisymmetric jet (curvilinear mesh surrounded by a yellow line at the zoom-in window).

From the above discussion it is concluded that the hybrid O-Type/H-Type meshing approach is appropriate for the discretization of $3DWJs$ independently of the magnitude of the co-flow velocity. Its relative simplicity makes it even more attractive, considering the increased mesh complexity at the baffle region which is located at the jet farfield.

²the term *dimensional* here stands for the resolution of mesh entities (connectors)

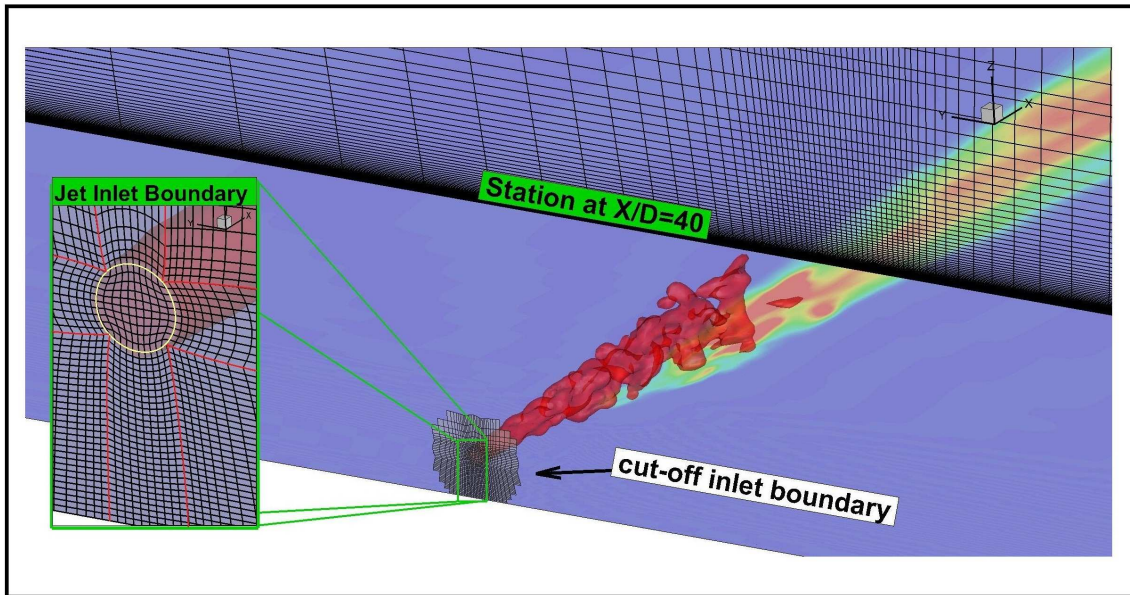


Figure 4.10: Pure H-Type meshing for the $3DWJ$ simulation.

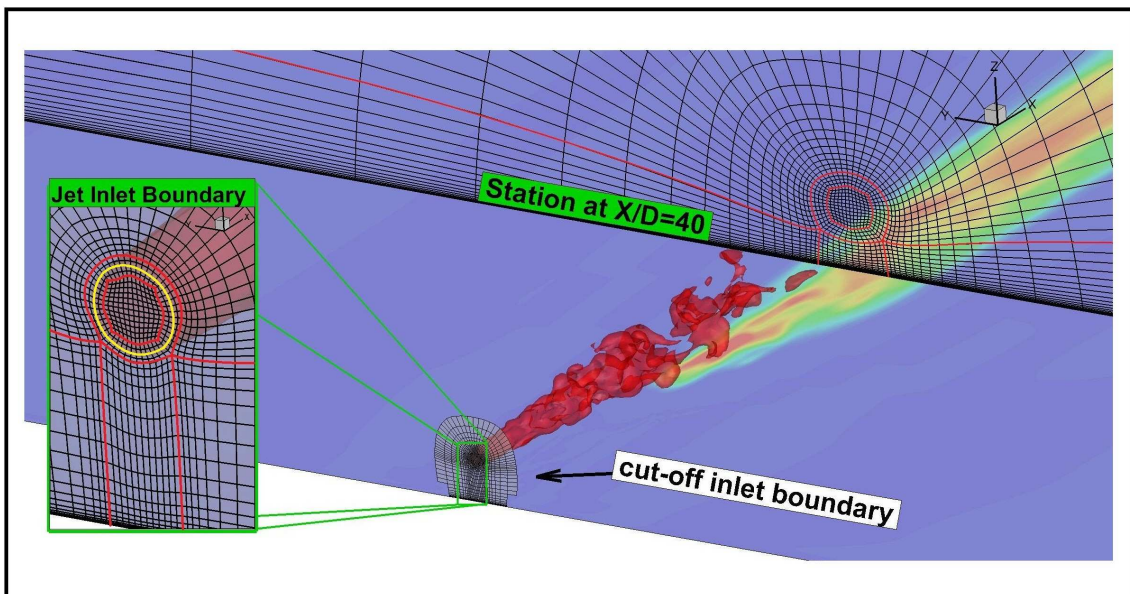


Figure 4.11: Modified O-Type meshing for the $3DWJ$ simulation. This meshing approach is restricted to large values of co-flow velocity.

4.2.3 Mesh Parameters

All simulations of the complete setup of this work have been performed with the hybrid mesh shown in (Fig.4.12). The present paragraph provides details of this mesh so that it can be accurately reproduced in future work.

The key dimensions of the cross section of the hybrid mesh are similar to the ones of the axisymmetric-jet mesh highlighted in (Fig.4.5), meaning that any streamwise cross section can be described by a regular H-type mesh inside the rectangular core, and a radial expansion outwards with azimuthal resolution defined by the dimensions of the core. The only difference here is that one part of the radial expansion (the bottom one) is omitted due to the presence of the solid surface, and the lateral ones spread parallel to this surface. It is already clear from (Fig.4.5) that the jet is discretised by the rectangular core alone, while the radial part of the mesh is used for the ambient flow development.

As explained in section 4.1 and illustrated in (Fig.4.4), the cross sectional area of the mesh increases with streamwise distance in order to accommodate the enormous expansion rates of a *3DWJ* in weak co-flow, which is the case for the complete simulations of this work (refer to section 6.2 for setup details). In the same time, the core-mesh is gradually stretched in the lateral direction in agreement with the lateral to vertical expansion rate ratio of the *3DWJ*, which in the presence of co-flow loses its linearity as explained in the theoretical analysis of section 5.1. The overall curvilinear topology of the mesh can be graphically extracted from (Fig.4.13) and (Fig.4.14) which illustrate a top view and a side view of the complete mesh topology respectively. It can be observed that the “jet-resolving” core consists of 79×64 cells, while the radial part consists of $(64 + 79 + 64) \times 25$ cells, which form a cross sectional resolution of approximately 10,000 cells in total. This number remains the same between different simulations, independent of the baffle arrangement installed. The cross section of the mesh at the nozzle exit is resolved by 30×30 cells (observe the yellow outline of the zoom-in window in figure 4.12); these cells are part of the H-Type core of the hybrid mesh, the full area of which is filled by the jet at $X/D = 15$

(reattachment region). The number of cells in the streamwise direction, however, varies depending on the baffle configuration. This is, 600 cells are adequate for the unresisted jet simulation (no-baffles), while 800 cells are needed when six arrays of baffles are included to the simulation (refer to table 10.1). The baffles on the final simulations have been designed as thin plates (Fig.4.15) with thickness $1mm = 1/6 D$, always resolved by four cells. In the case where the baffles are modelled by source terms (refer to section 4.3), the mesh lines are simplified to be regular (vertical and horizontal), but the local mesh resolution is preserved for consistency of the comparisons.

The size of the first cell above the solid surface is adequately low at the $3DWJ$ region ($X/D > 60$) with respect to y^+ , while it is under-resolved at the inlet, where the jet is not attached to the ground. This is a compromise made for the sake of computational cost reduction. In this way initial value $\delta Z = 0.25D$ is set at $X/D = 0$, which linearly meets the value $\delta Z = 2 \times 10^{-5}D$ at $X/D = 60$. After this point it increases according to the mesh vertical expansion. An average value of $\delta Z = 5 \times 10^{-5}D$ is maintained throughout the wall region. It is important to mention that the value of the non-dimensional wall distance becomes lower than $Y^+ = 3$ before the jet reaches the ground, drops to $Y^+ = 1$ during the reattachment region, and its value remains within $0.2 \leq \delta Z \leq 1$ in the $3DWJ$ region.

Finally, due to the inhomogeneity of the mesh the cell size varies significantly between different regions of the computational volume. A vertical expansion ratio of 1.2 is selected above the solid surface, which drops to 1 in the mixing layer region and returns to 1.2 at the radial part of the mesh. The expansion ratio in the streamwise direction is restricted by the criterion that the cells lying inside the mixing layer are isotropic in all three directions (refer to section 7.1.9 “mesh-isotropy test”). Generally, the streamwise expansion ratio is below 1.1.

The following section discusses an alternative method to represent the baffles in the flowfield, aiming to reduce the meshing complexity of the numerical baffle investigation.

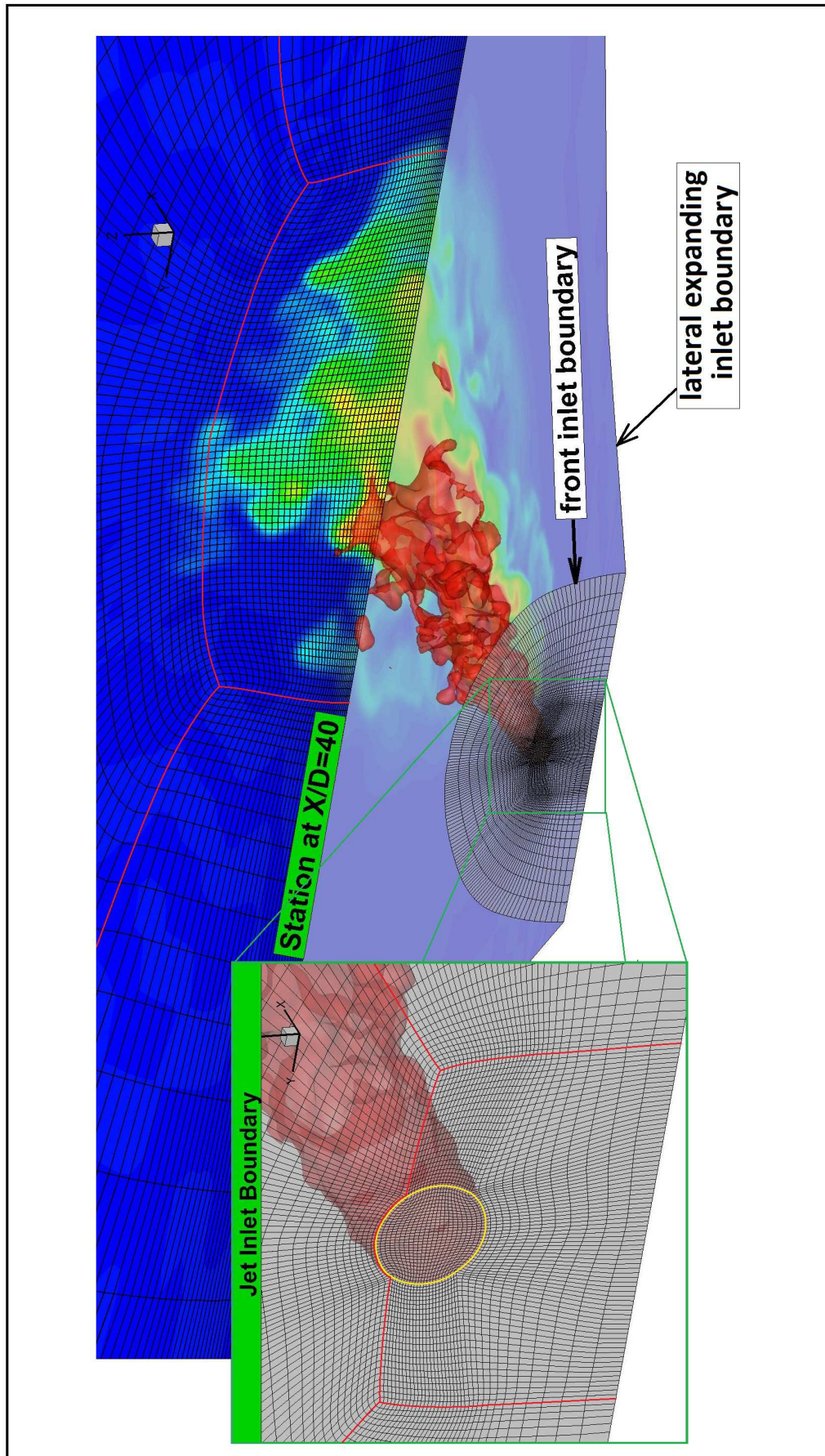


Figure 4.12: Hybrid H-Type/O-Type meshing for the final $3DWJ$ simulations. This meshing approach is appropriate for large values of co-flow velocity.

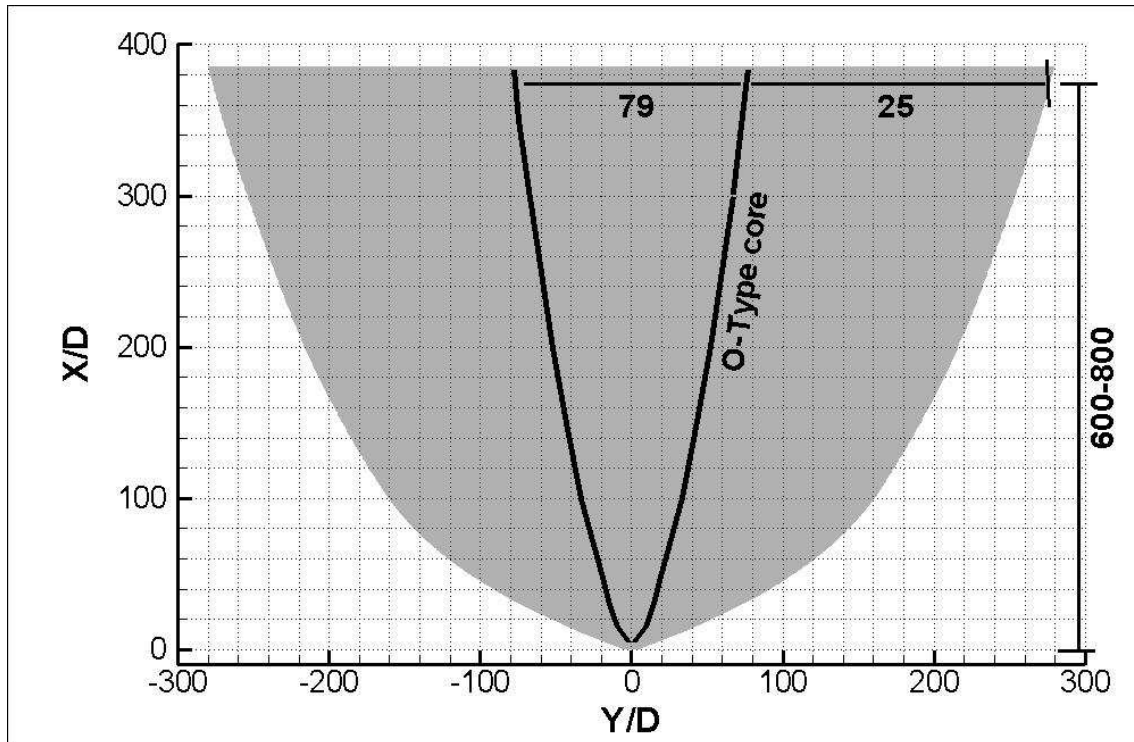


Figure 4.13: Top view of the complete mesh topology. The geometrical dimensions of the domain boundaries, as well as the O-Type core, are highlighted. Cartesian grid lines are drawn to aid data extraction. Resolution of different mesh entities is highlighted.

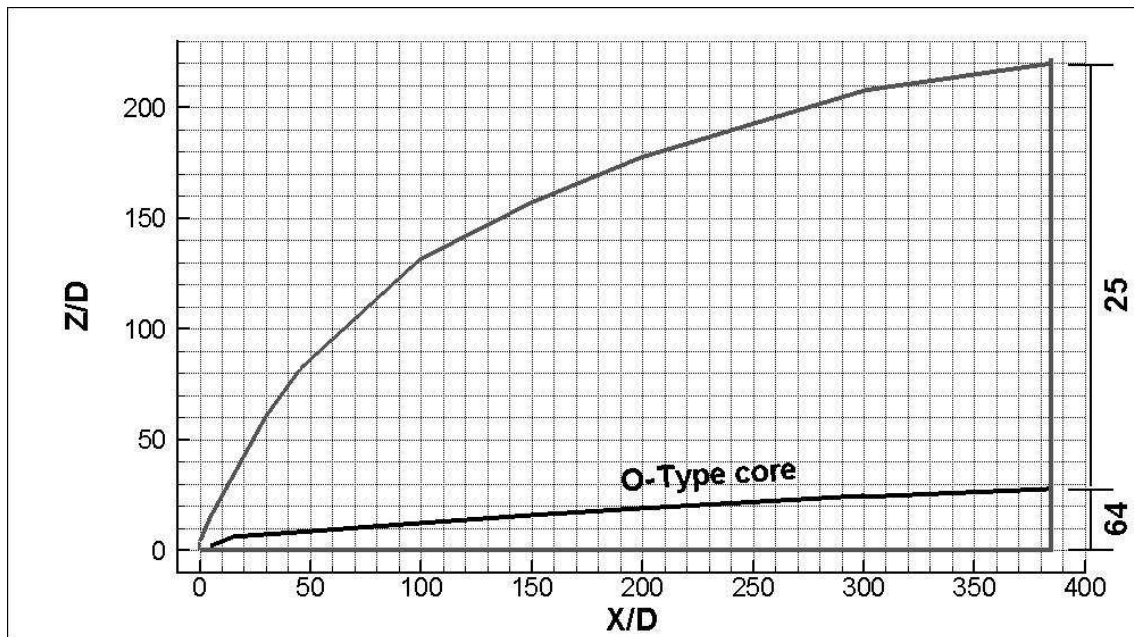


Figure 4.14: Side view of the complete mesh topology. The geometrical dimensions of the domain boundaries, as well as the O-Type core, are highlighted. Cartesian grid lines are drawn to aid data extraction. Resolution of different mesh entities is highlighted.

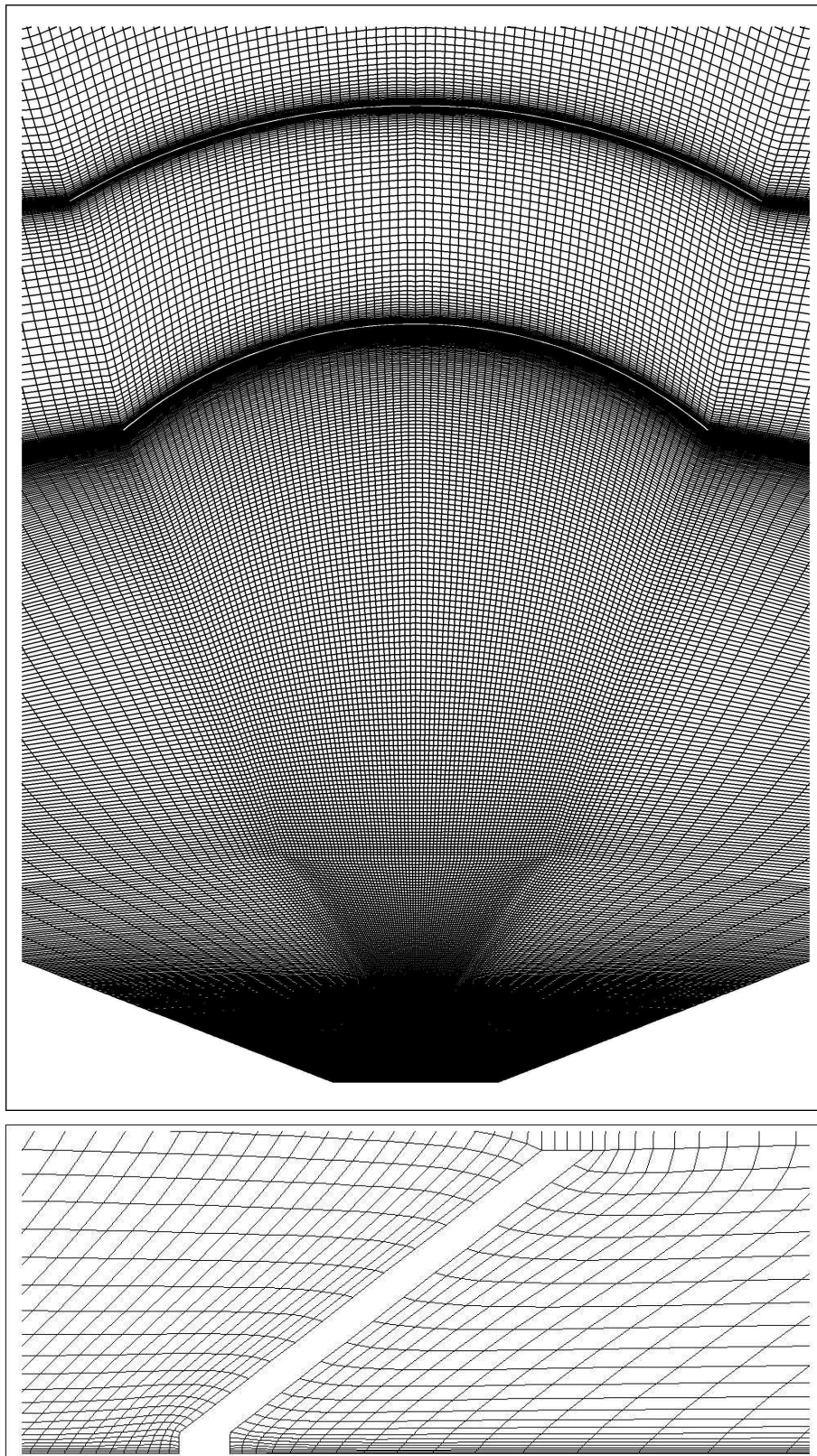


Figure 4.15: Near-baffle meshing example from the complete simulations. A top view of the near field is illustrated at the top image, and the cross section $Y = 0$ near the first baffle array is illustrated at the bottom.

4.3 MODELLING THE BAFFLES

The computational cost of this study's *LES* is susceptible to a variety of factors as described at section 5.4. Further to these factors, inclusion of the baffles to the numerical setup requires remarkable increase of the mesh resolution at the farfield region where they are located. Additionally, meshing complexity increases dramatically considering the multiple boundary layers apparent in such a flow configuration (Fig.4.15). This meshing complexity increases the effort required to prepare the mesh, imposing limitations to the investigation of the geometrical characteristics of the baffles arrays. It is also known that the wake of solid baffles generates high frequency fluctuations responsible for noise. Although noise is not taken into account in this study, simulation of these fluctuations affects the computational cost by requiring reduction of the timestep size Δt .

It is attempted to develop an alternative, more cost-efficient method, for the representation of the baffles in the simulation. Taking into account the objective of the wind brake system (refer to section 1.5.2), we can suggest a relatively simple way to model it. This is by distributing the baffle induced drag force inside a large part of the computational volume, within the baffle region, by incorporating source terms in the x-momentum equation. However, two issues arise in this concept. First, the drag force is not known, which would bring about additional modelling assumptions in order to estimate it pre-processing. Second, such an approach would not affect aerodynamically the plume, which means that apart from a smooth reduction of the plume momentum within the forced region, no deflections will occur in the flow. This is believed to be inappropriate considering the fact that when the plume is deflected by a real baffle, the high-concentration slow-moving fluid next to the surface is transferred closer to the outer mixing layer where velocity is higher and mixing with the ambient fluid is more intense.

Considering the above discussion the objective of the attempted model is to *“remove the baffles from the computational domain, and instead add source terms in the momentum equations, in an appropriate manner so that the flow responds aerodynamically as if the baffles were present”*. The present section is dedicated

to the development of an appropriate method for this study, while conducting a literature review of relevant existing models. Limitations of the existing models are discussed, assumptions and simplifications are suggested, and finally, a complete description of the application of the selected model in the *LES* algorithm is made.

4.3.1 Summary of the Immersed Boundary Method

A concept similar to the one attempted in this work was first introduced by (Peskin, 1972) who simulated cardiac mechanics and associated blood flow, and termed his new approach “*the immersed boundary method*”. In particular Peskin carried out the entire simulation on a Cartesian grid, which did not conform to the geometry of the heart, but “*immersed*” boundaries were used for the representation of the complex and moving heart tissues in the simulation. Since Peskin introduced this method, numerous modifications and refinements have been proposed and a number of variants of this approach now exists. A comprehensive review of such models has been published by (Mittal & Iaccarino, 2005).

To better understand the philosophy of the immersed boundary method, let us consider the two dimensional example of the “*virtual baffle surface*” as illustrated in figure 4.16. Assuming that this surface represents the immersed boundary of a simulation, it would have to be modelled by an additional set of points $X_k(t) \in \mathbb{R}^2$ (in two spatial dimensions) indicating the location of the boundary on a specific time t . Since the immersed boundary represents a solid surface, fluid should not flow across this boundary; therefore, the points X_k are shifted in each timestep according to the local fluid velocity $U(X_k, t)$. It is, therefore, understood that although the solver uses an Eulerian approach (mesh fixed in space) for the fluid, the immersed boundary is simulated in a Lagrangian form (mesh moving with the flow). Of course in the case of a stationary object the points X_k must not move, therefore, an appropriate forcing has to be applied as described below.

The effect of the immersed boundary on the fluid is not accounted for by the introduction of boundary conditions to the N-S equations, but rather by the inclusion of inhomogeneous forcing to the equations. The forcing is a discrete function with support only along the immersed boundary. For instance, in the discrete method

discussed in this example, the force would be of the form:

$$f(x, t) = \sum_k F_k(t) \delta(x - X_k(t)) \quad (4.1)$$

where $F_k \in \mathbb{R}^2$ is the force vector at a point X_k , and δ is the two dimensional Dirac delta function. This expression for F_k requires that the points X_k do coincide with the grid points. A grid independent form allows the force of each node X_k to be distributed on the surrounding grid points with the sharp delta function being essentially replaced by a smoother distribution function, denoted here by d , which is suitable for use on a discrete mesh. There is a variety of such functions used in different investigations in the past (Peskin, 1972; Beyer & LeVeque, 1992; Saiki & Biringen, 1996; Lai & Peskin, 2000). In this way, the momentum source terms to be set at a grid point, i.e. (i, j) , can be calculated by:

$$f_{ij} = \sum_k F_k(t) d(x_{ij} - X_k(t)) \quad (4.2)$$

So far, the fundamental operations of an immersed boundary algorithm have been described. A significant parameter of the algorithm, however, is the specification of the forces F_k exerted by the virtual surface. As suggested by (Beyer & LeVeque, 1992), depending on the nature of the problem, this parameter may increase the complexity of the algorithm, and raise the requirement for an implicit or at least semi-implicit method for the evaluation of F_k . For instance, the case of flow around a solid stationary body, which better describes the baffle problem, poses difficulties because the constitutive laws used for elastic boundaries are not generally well posed in the rigid limit.

As explained by (Mittal & Iaccarino, 2005), this problem could be circumvented by considering the body to be elastic but extremely stiff, or by considering the structure to be attached to an equilibrium location by a spring with a restoring force. The first approach requires an implicit scheme resulting on a large system of equations to be solved for F_k in order to ensure the no-velocity condition on the

virtual surface as discussed in the analysis of (Beyer & LeVeque, 1992). However, using the spring approximation, the second approach considerably reduces the computational requirement and has been successfully applied in the works of (Beyer & LeVeque, 1992) and (Lai & Peskin, 2000). Specifically, using a semi-implicit spring approach, for a given point of the virtual surface X_k , and a given time t_n , the force F_k^n is calculated by:

$$F_k^n = -\kappa(X_k^n - X_k^0) \quad (4.3)$$

where $\kappa \gg 1$ is the spring constant, and X_k^0 is the equilibrium position of point X_k .

The stiffness of the immersed boundary increases with increasing the value of κ . However, very large values result to a stiff system of equations which brings about stability constraints (Lai & Peskin, 2000; Stockie & Wetton, 1999). On the other hand, lower values of κ “*can lead to spurious elastic effects such as excessive deviation from the equilibrium location*” as stated in the review of (Beyer & LeVeque, 1992).

4.3.2 Present Work Model Approximations

The “*immersed boundary method*” described in the previous paragraph is a very promising approach for the accurate representation of the baffles in the computations of the present work. The immersed boundary method, available through an add-on module for ANSYS FLUENT 12.0 software (Wolfe, 2009) could be considered here. However, it is expected that the research acceleration due to reduced mesh-generation effort, would be eliminated by the large number of time-steps required for a complete simulation in the present investigation, i.e. $> 100,000$ time-steps (refer to Fig.5.5), combined with the multiple operations performed by this method in every timestep. Therefore, it is attempted to develop a simplified model, with lower computational requirement, which satisfies the original objective highlighted early in this section. This model is termed dynamic response algebraic model for baffle imitation (*DRAMBI*).

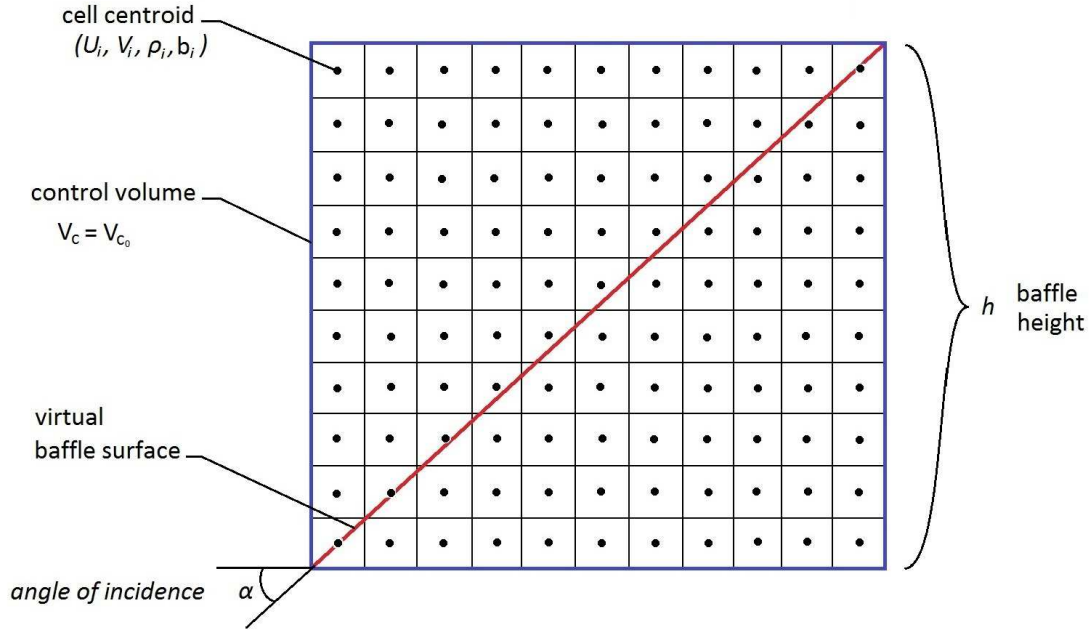


Figure 4.16: DRAMBI Concept.

The sketch illustrated in (Fig.4.16) highlights the core features of *DRAMBI*. In order to explain the philosophy of this approach, let us first consider a two dimensional version of the flow, i.e. on the symmetry plane $Y = 0$. The thickness of a baffle is very small, therefore, the virtual body can be described by an inclined flat plate (line in 2D). The dimensions of this flat plate explicitly define a Cartesian box (V_{C_0}). Assuming that the boundaries of this box coincide with the proximate cell faces as shown in (Fig.4.16), the area included is representative of the control volume (V_C) of the model. The dimensions of V_C can be scaled so that the model applies to different geometries, with X and Z becoming $[0 \leq X' \leq 1]$ and $[0 \leq Z' \leq 1]$, or $[0 \leq R' \leq 1]$ and $[0 \leq Z' \leq 1]$ on cylindrical coordinates (see next paragraph: Implementation of the Model).

In the absence of external forces, the streamwise velocity of all cells within this volume is positive³. It is easily understood that application of an upstream forcing F_D at the cells of V_C with positive velocity, would reduce the velocity of the approaching fluid, and above a critical value of F_D the fluid would undergo

³the effect of the no-slip condition on the ground is infinitesimal in the present analysis as the volume of the first cells above the wall is very small, i.e. $\delta Z = 5 \times 10^{-5} D$. Naturally, negative velocities may exist at the lateral sides of the mixing layer, but the presence of positive co-flow eliminates this problem.

a sudden deflection. Of course, although this change is very fast, it will take several timesteps to complete considering the fact that the timestep size is very small compared to the local conditions at the baffle region (farfield). This gives us the opportunity to dynamically evaluate F_D so that the flow within V_C remains in the transitional regime between a fully attached and a completely deflected flow. Naturally, the separation will begin from the bottom of V_C where the velocity is zero due to the no-slip condition, and will start growing into a discrete wake followed by a recirculation bubble. The surface of the wake which lies in V_C corresponds to the virtual surface of the baffle. In order to adapt to the flow inhomogeneity, and avoid a disproportional forcing of the flow, the force is distributed in V_C based on the momentum content of each cell. Additionally, enhancing the shape of the wake to better describe the modelled baffle is achieved by the application of a further weighting operation to the force distribution, this time based on the baffle function b (eq.4.6). The distribution of b in V_C is illustrated in (Fig.4.17).

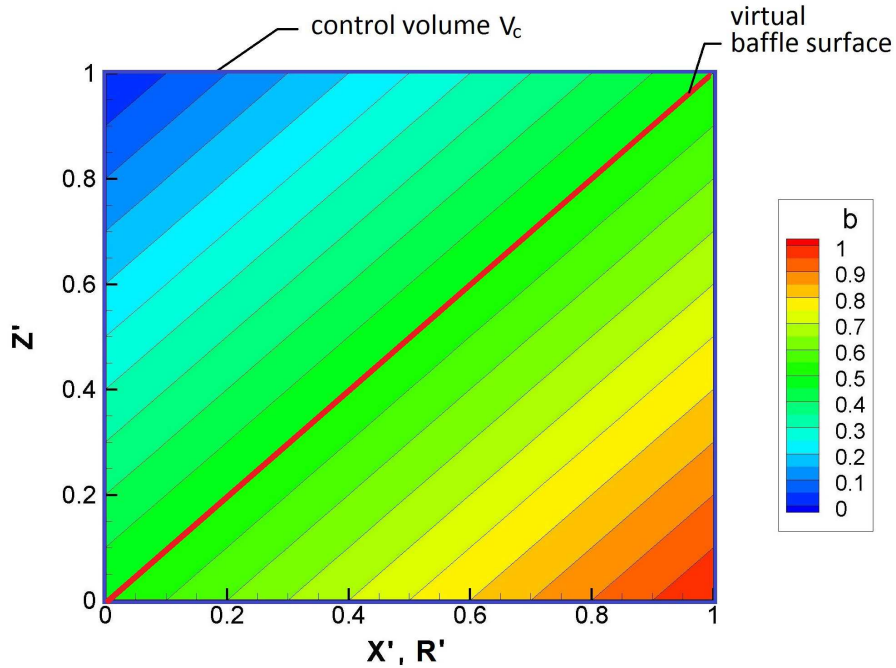


Figure 4.17: *DRAMBI* baffle function.

The idea of a spring with a restoring force (eq.4.3) is employed for the temporal control of the virtual baffle surface. In particular, a dynamic response algorithm (see *DRA* in Fig.4.20) has been devised, which attempts to predict an appropri-

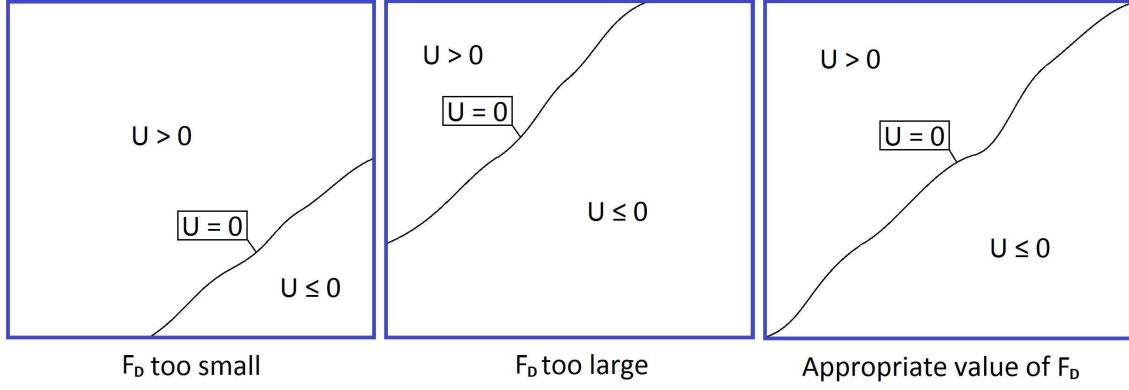


Figure 4.18: Graphical explanation of the criterion of *DRA* for the appropriate evaluation of F_D .

ate value of the drag force F_D for the next timestep. Because the present method does not perform cell-based operations, the evaluation of F_D is not based on the deviation of the actual surface points X_k from their equilibrium positions X_k^0 , but instead, an implicit criterion is selected. This criterion is given directly from the boundary conditions of the control volume V_C , which suggest that the appropriate position of the baffle forms when the portion of V_C with positive velocities equals the one with negative or zero velocities. The sketch of (Fig.4.18) graphically explains this concept. In order to overcome the stability issues discussed by (Lai & Peskin, 2000; Stockie & Wetton, 1999), *DRA* involves a transient evaluation of a step force ΔF based on which F_D changes. Depending on the deviation of the positive-velocity volume from $V_C/2$, ΔF increases or decreases dynamically by the use of a *tanh* function, with slope λ transiently adjusting to the flow conditions, thus, not requiring readjustment for the application in different baffle shapes and different flow conditions. The calculations of *DRA* are performed externally, and once for each timestep, while momentum weighting factors are calculated once for every outer iteration; therefore, the additional computations of this model are insignificant with respect to increasing the cost of the simulation, even when a large number of timesteps is involved.

So far the description of *DRAMBI* has been made based on a two dimensional version of the flow. Extending to three dimensions is straightforward considering the fact that the cross-sectional dimensions of V_C shown in (Fig.4.16), do not vary in the span-wise direction, and that the spatial velocity inhomogeneity

is already taken into account by the momentum weighting operation. Incorporation of the model to the selected solver is described in detail in the following paragraph.

4.3.3 Present Work Model Implementation

The dynamic response algebraic model for baffle imitation (*DRAMBI*), is based on a dynamic evaluation of the drag force F_D which is then distributed on the cells of a predefined control volume enclosing the modelled array of baffles. The model is operating in a coupled manner with FLUENT unsteady algorithm (Fig.4.19). The application is achieved through user defined functions *UDF* (refer to Appendix G.1). Preliminary validation of this method has been reported at (Spanelis *et al.*, 2013). In that report, modifications of the model were suggested, which have been taken into account for the final simulations presented in chapter 10.

The first step of *DRAMBI* is, once at the beginning of the simulation, to define the ideal control volume V_{C_0} within which the model applies. In the initial model presentation at (Spanelis *et al.*, 2013), V_{C_0} was defined as an orthogonal parallelepiped with dimensions in the streamwise X and vertical Z directions defined by the baffle's width and height respectively. However, the results of that work suggested that inclusion of the radial distribution of the baffle array (refer to Fig.6.5) is important. In this way, the coordinate system is changed into cylindrical, and the dimensions of V_{C_0} are given by R in the radial direction with centre at the runway threshold, ϕ in the azimuthal direction with zero angle along $Y = 0$, and Z in the direction normal to the ground with zero at the ground level.

The next step is to calculate the actual control volume V_C . If V_i is the cell's volume, and n the number of cells the centroid of which belongs to V_{C_0} , the actual control volume V_C is calculated by:

$$V_C = \sum_{i=1}^n V_i \quad (4.4)$$

In the following, scaling is applied to the coordinates of the cross section of

V_C , obtaining the scaled coordinates R' and Z' , for the radial and wall-normal directions respectively.

$$\left. \begin{array}{l} 0 \leq R' \leq 1 \\ 0 \leq Z' \leq 1 \end{array} \right\} \in V_C \quad (4.5)$$

The cell geometry-related weighting factor b_i of a cell (see Fig.4.17), is calculated by the geometrical function:

$$b = \left(\frac{R' - Z' + 1}{2} \right) \quad (4.6)$$

At the beginning of each timestep, the Dynamic Response Algorithm (*DRA*) is launched in order to dynamically evaluate the total instantaneous drag force F_D to be distributed in V_C . More specifically, *DRA* performs a series of logical and algebraic operations in order to predict a value of F_D that minimizes the absolute value of the active volume ratio Ψ defined as:

$$\Psi = \left(\frac{V_A}{V_C/2} \right) - 1 \quad (4.7)$$

where V_A is the active volume calculated by:

$$V_A = \sum_{i=1}^n V_i \quad \text{for } U \geq 0 \quad (4.8)$$

In order to minimize the variance of Ψ , *DRA* uses the asymptotic function $\alpha_{\Delta F} = \tanh(10\Psi) \lambda$. Stability is ensured through a dynamic procedure for the evaluation of the slope λ , and the smoother σ_s . The logical diagram of *DRA* is presented in (Fig.4.20).

The radial definition of V_{C_0} implies that the baffle resistance no longer aligns with the streamwise direction, but it is perpendicular to the “*surface of the baffle*”. Therefore, further to the X-directed component of the baffle resistance, which

is the drag force F_D predicted by *DRA*, there is also a Y-directed component describing the side forces calculated by:

$$F_S = F_D \tan(\phi) \quad (4.9)$$

In order to balance the non-solidness of the baffles, source terms are also added on the vertical z-momentum. The lift force F_L is calculated based on the value of the Ψ function, which is an indicator of the robustness of the baffle shape, by equation:

$$F_L = \Psi F_D \frac{\tan(\alpha)}{\cos(\phi)} \quad (4.10)$$

where α is the incidence angle of the baffle, which in the investigated setups of this work takes the values 40° , 50° , and 60° .

At the beginning of each outer iteration, *DRAMBI* calculates the cell weight-ing factors for momentum a_i , incorporating the geometrical function b_i . For n number of cells lying in V_A , a_i is calculated by:

$$a_i = \frac{\rho_i U_i V_i b_i}{\sum_{i=1}^n \rho_i U_i V_i b_i} \quad (4.11)$$

Finally, the calculated forces F_D , F_S and F_L are distributed in V_C based on the value of the weighting factor a_i of each cell. This is achieved with the application of volumetric source terms in the x, y and z-momentum equations. These are:

$$(S_D)_i = -\frac{a_i F_D}{V_i} \quad (4.12)$$

$$(S_S)_i = -\frac{a_i F_S}{V_i} \quad (4.13)$$

$$(S_L)_i = \frac{a_i F_L}{V_i} \quad (4.14)$$

The presented model is expected to reduce the computational cost by reducing the number of computational cells. However, the most significant advantage of this model is that it does not require designing a new mesh for every geometrical change applied on the baffles. In this way it is expected to accelerate the numerical investigation of the baffle-resisted plumes.

Comparing to other immersed boundary models the advantage of *DRAMBI* is that it does not require the solution of any additional system of equations, therefore, it does not delay the solution convergence. Additionally, the value of the drag force F_D , which as explained in section 1.5.2 is a major contributor for plume's lift-off, is explicitly calculated, therefore, no further effort/approximation is required to obtain it with this model. This feature will be especially useful in future real scale simulations for the evaluation of the wind brake system efficiency.

For increased accuracy of the predicted drag force F_D , it is suggested that the top side of the control volume coincides with the local grid points, so that the height of V_C coincides with the one of V_{C_0} . A similar adjustment could be made in the other two directions, but with less significant effect on the flow.

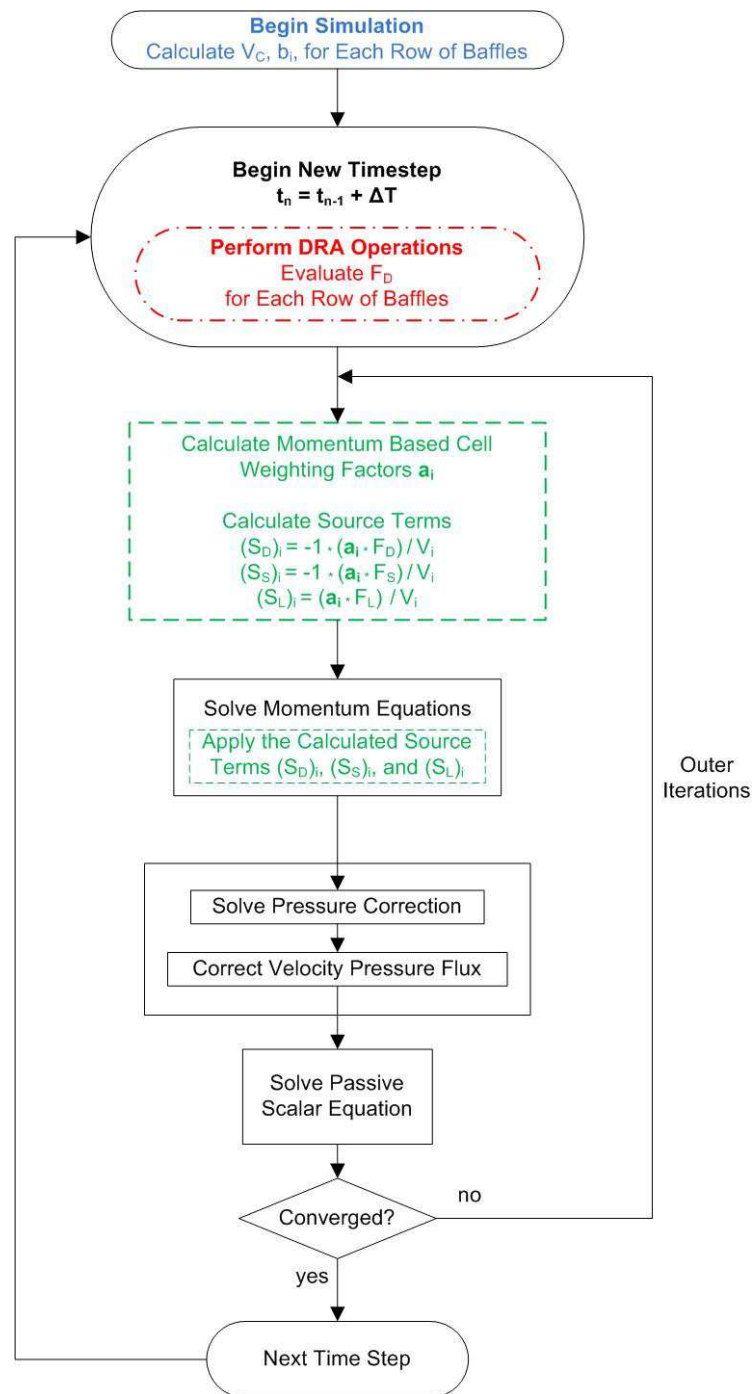


Figure 4.19: *DRAMBI* logical diagram. Default operations of FLUENT unsteady algorithm are coloured black. Blue colour is used for operations before simulation starts. Green and red colours are used for *DRAMBI* and *DRA* operations respectively.

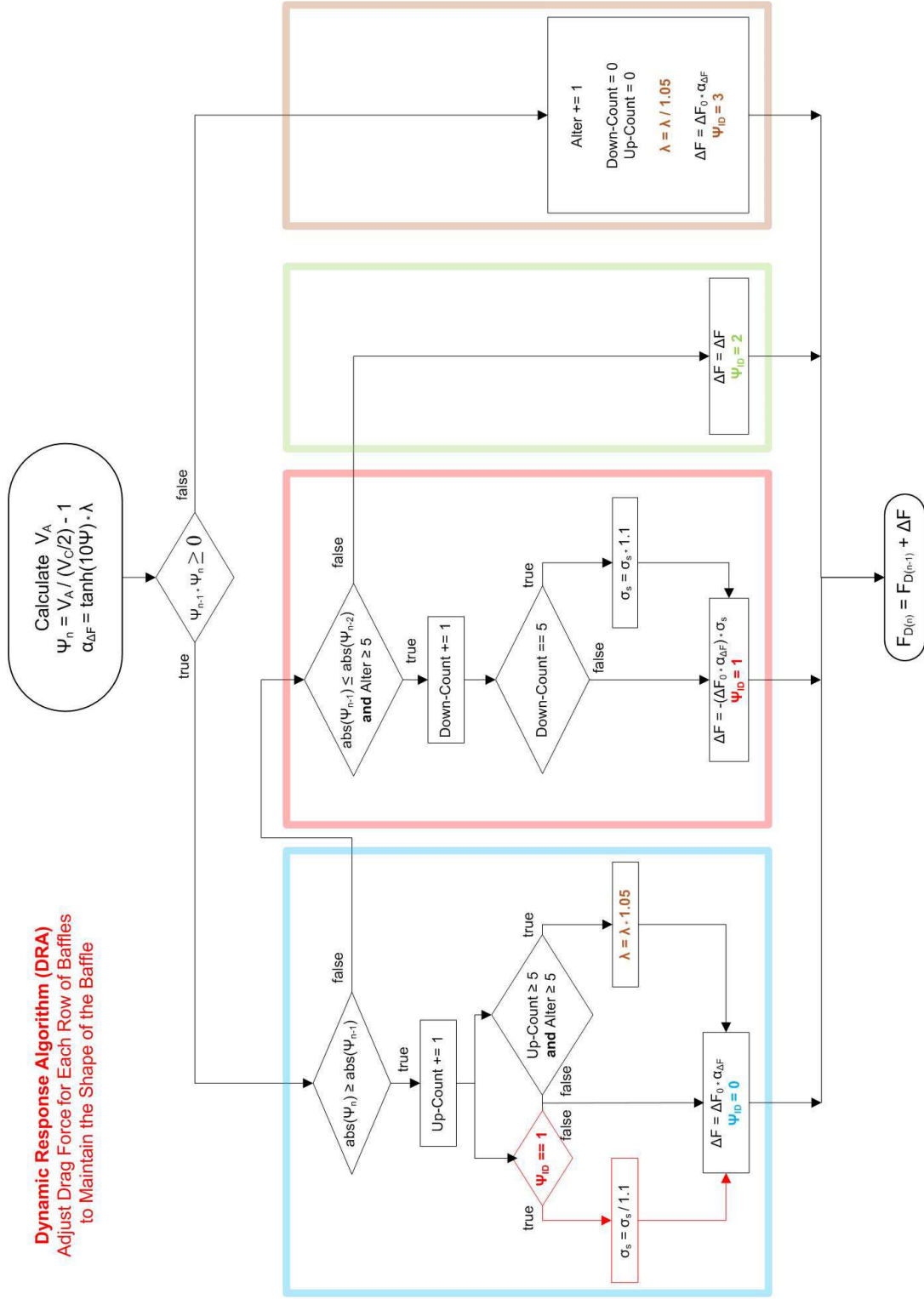


Figure 4.20: Operations of the Dynamic Response Algorithm (DRA).

Chapter 5

Theoretical Analysis

Before proceeding to the implementation of the multiple numerical simulations composing the solver validation and the actual investigation of this thesis, it is wise to first obtain a-priory estimates of the expected flow fields. In this way, section 5.1 modifies the theoretical formulae describing the mean jet characteristics in order to take into account the co-flowing wind present in our problem and presents theoretical streamwise evolution plots of both the axisymmetric jet and the *3DWJ* for various co-flow velocities. Then, section 5.2 presents theoretical estimations of the lift-off onset of the plume obtained by two different theoretical models; the model of (Bennett *et al.*, 2013) which only requires jet inflow conditions and omits the co-flow (wind) velocity, and the model of (Hanna *et al.*, 1998), which uses local values of the flow available from the results of section 5.1. In the following, section 5.3 introduces the concept of the Simulated Time Requirement (*STR*), and explains the different time intervals necessary to be simulated depending on the purpose of the simulation. Finally, a computational cost assessment of the simulations of this study is made in section 5.4, particularly discussing general sources of computational cost in *LES*, and specifically explaining the high importance of the Farfield Location of Interest (*FLI*) and of the co-flow velocity (U_f) in the present study.

5.1 MEAN FLOW CHARACTERISTICS

The streamwise distribution of the axisymmetric jet centreline velocity U_m is described by equation (1.1). Accounting for the co-flow velocity, equation (1.1) is modified to give the axisymmetric jet velocity decay as:

$$U_m = U_f + (U_b - U_f)B \frac{D}{X} \quad (5.1)$$

where U_f is the co-flow velocity, B is the velocity decay coefficient with value set in agreement with the findings of the experimental work summarised in section 1.4.2, $B = 6.1$.

In the case of a $3DWJ$ the velocity decays as X^a , with the value of the exponent selected as $a = -1.15$, in agreement with the findings of the experimental work reviewed. Again, including the effect of the co-flow we can estimate the velocity decay by:

$$U_m = U_f + (U_b - U_f)B(X/D)^{-1.15} \quad (5.2)$$

The analytical estimation of U_m for a $3DWJ$ in equation (5.2) includes assumptions such as a uniform distribution of the co-flow velocity around the jet mixing layer, and that the velocity ratio U_f/U_b has a linear effect on the jet characteristics. However, it is suggested that these assumptions are acceptable for the purpose of the current analysis. Additionally, the virtual origin X_B , which is included in the theoretical formulation of U_m is adjusted at the nozzle exit $X_B = 0$ instead of $X_B/D = 2$ for simplicity. However, the effect on this analysis is expected to be infinitesimal as it focusses at the very farfield $X/D > 200$.

Figure (5.1) is a double logarithmic plot illustrating the theoretical streamwise variation of U_m for different co-flow velocities, of both the axisymmetric jet and the $3DWJ$, as calculated by equations (5.1) and (5.2) respectively. The higher rate of decay of $3DWJs$ is clear. It can also be seen that the co-flow velocity U_f changes the logarithmic decay rates (appear linear in the log/log plot), which is “bended”

towards the asymptotic value $U_b = U_f$. This asymptotic value is met earlier with increasing the co-flow to bulk flow ratio U_f/U_b .

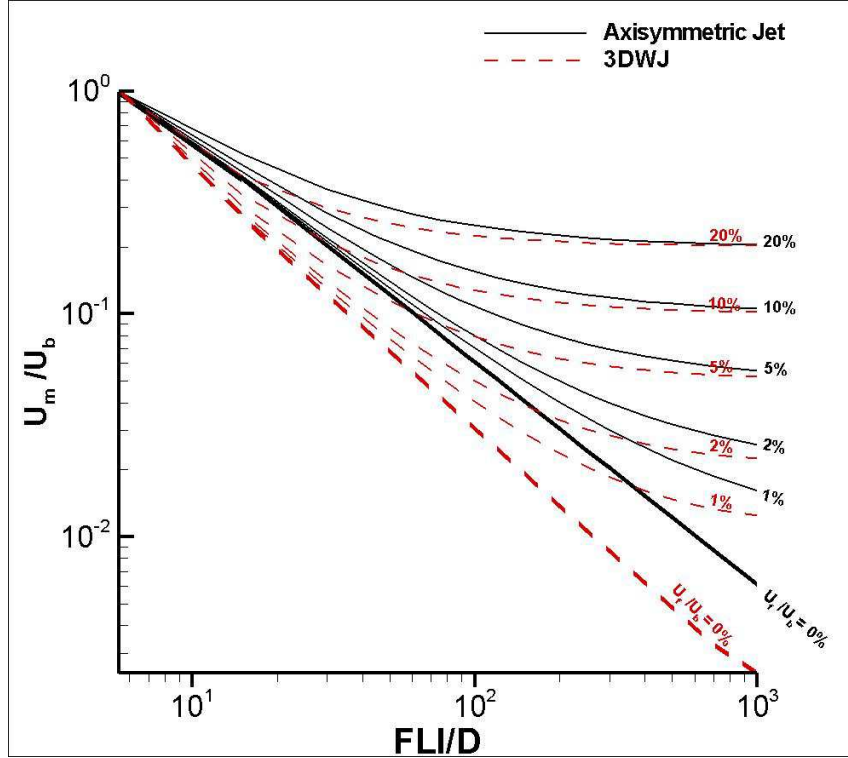


Figure 5.1: Theoretical streamwise distribution of U_m for the axisymmetric jet and 3DWJ. The labels on the plot represent the value of U_f/U_b corresponding to the proximate curve.

This analysis requires the estimation of the average value of U_m inside a range $[X_{min} \leq X \leq X_{max}]$. From now on this value is referred to as U_{mAVG} , calculated by:

$$U_{mAVG} = \frac{1}{X_{max} - X_{min}} \int_{X_{min}}^{X_{max}} U_m dX \quad (5.3)$$

In particular, U_{mAVG} is calculated by setting $X_{min} = L_{pc}$, because it is at the end of the potential core $X = L_{pc}$ where the jet velocity begins to decay, and $X_{max} = FLI$. FLI is the distance from the nozzle of the most remote location of interest of the simulation, or simply “*Far-field Location of Interest*”. The length of the potential core is selected at $L_{pc} = 5.5D$ complying with typical values for turbulent round jets.

Accordingly, the velocity half-widths $z_{1/2}$ and $y_{1/2}$ at the location $X = FLI$ are

calculated by:

$$L_{(z_{1/2}, y_{1/2})} \approx \frac{D}{2} + \frac{dL_{1/2}}{dx} (FLI - L_{pc}) \left(1 - \frac{U_f}{U_{m_{AVG}}} \right) \quad (5.4)$$

where $\frac{dL_{1/2}}{dx}$ takes the value 0.095 for the axisymmetric jet, and the values 0.048 and 0.3 for the 3DWJ in the vertical and lateral directions respectively. The last term of equation (5.4) is the correction applied to account for the co-flow velocity. In particular, for $U_f > 0$ the ratio $\frac{U_f}{U_{m_{AVG}}}$ varies dynamically with FLI , therefore, the linear growth rate of the jet dimensions no-longer applies. On the contrary, the plume is continuously being transferred at more downstream locations due to the co-flow (wind), not having the time to develop in size.

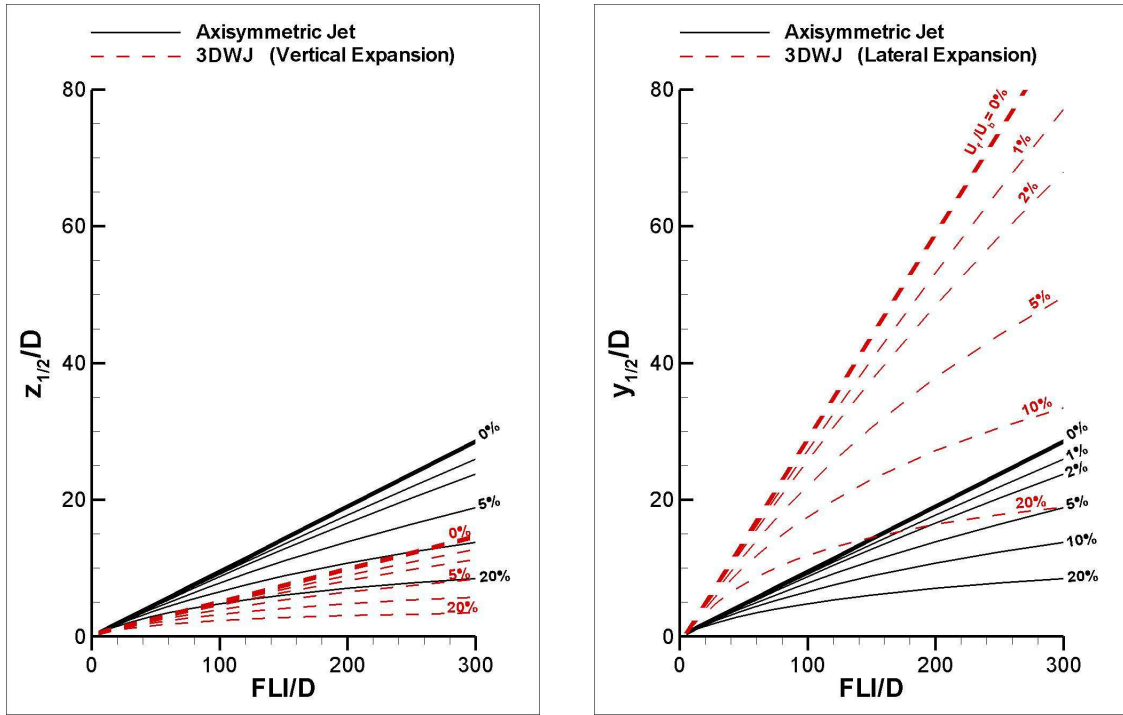


Figure 5.2: Theoretical estimation of the jet length scales with varying the farfield location of interest FLI . The labels on the plots represent the value of U_f/U_b corresponding to the proximate curve.

Based on equation (5.4) and the growth rate factors mentioned above, the curves of (Fig.5.2) are calculated. Particularly, the streamwise variation¹ of the axisymmetric jet velocity half-width is compared to the vertical one of a 3DWJ

¹In this plot FLI/D can be simply seen as X/D

for varying U_f (Fig.5.2, left), and to the lateral one (Fig.5.2, right). The extreme lateral to vertical expansion ratio of the $3DWJ$ can be clearly observed here. In particular, the vertical scale of the $3DWJ$ is broadly half the one of the axisymmetric jet (left plot), while its horizontal scale is more than double the one of the axisymmetric jet (right plot). It can also be observed that in both types of jet, the jet growth is significantly counteracted by the co flow velocity U_f .

5.2 BUOYANT PLUME THEORETICAL PREDICTIONS

Two different models for the theoretical estimation of the lift-off onset of ground based buoyant plumes were presented in the introduction chapter (section 1.5). These include the lift-off criterion of (Bennett *et al.*, 2013) and the one of (Hanna *et al.*, 1998). In the present section the theoretical formulas of these models are applied to the setup parameters of the $3DOJ$ experiment of (Velikov, 2013) (refer to sections 6.2 and 6.2.3 for detailed description), which is the central setup in the present work.

Briefly, Velikov's experiment is a $1/200$ scale of the field trials of (Bennett *et al.*, 2013), and consists of a jet of air-helium mixture ejected by a circular nozzle $D = 0.006m$ parallel to the wind tunnel surface at an offset $h = 0.012m = 2D$. The jet concentration in helium has been selected so that the densities of the jet mixture and the ambient flow (pure air) correspond to the temperatures $T_j = 412K$ and $T_a = 290.15K$ (field trial conditions). Consequently, the two densities are $\rho_j = 0.8472kg/m^3$ and $\rho_a = 1.2022kg/m^3$. Finally, the jet bulk velocity is $U_b = 25.4m/s$, and the co-flow velocity is $U_f = 0.5m/s$. It is important to mention that the co-flow is not uniform in this setup, but a boundary layer well described by the $1/7$ wind power law relationship is present. However, as mentioned at the beginning of section 5.1, the present theoretical analysis includes the assumption of a uniform co-flow.

From (eq.1.5) we can calculate the buoyancy flux $F_B = 0.000796N/s$, and from (eq.1.6) we can calculate the horizontal momentum (i.e. the engine thrust) $F_M = 0.02193N$. From these two values we can then calculate the travel time in

which the two become equal from (eq.1.7), which gives $\tau = 27.54s$.

The following sections aim to estimate the streamwise location at which the lift off onset occurs for a naturally developing plume (no-baffles disturbing its development), and for a baffle resisted plume, respectively.

5.2.1 Natural Lift-off of the Plume

The theoretical lift-off onset is now calculated based on the two reviewed models.

According to (eq.1.9), Bennett *et al.* suggest that the distance travelled by the plume, before the buoyant forces become equal to the Coandă effect, will be $X_L \approx \left(\frac{\tau}{\pi}\right)^{1/2} \left(\frac{2F_M}{\pi\rho_a}\right)^{1/4} = 0.972m = 162D$. As it is suggested in their report, this numerical value should not be taken too literally, as this is a very crude model. Two factors in particular should be borne in mind. Firstly, a single value for the jet spreading parameter has been assumed and secondly, the influence of ambient wind has been neglected. Additionally, this calculation does not take into account the co-flow velocity.

The lift-off onset is also calculated based on the criterion of Hanna *et al.*, which takes into account that shear in the surface boundary layer will disrupt a plume, spreading the emission horizontally faster than it can rise. Their criterion suggests that the plume will start rising when the value of $\frac{U^3 W \pi \rho}{F_B}$ is less than 100. In Hanna's model, U is the effective wind speed over the height of the plume; however, in the present setup it has to be replaced by U_m (calculated by eq.5.2), as the plume is not propagated only by the wind, but also due to the initial thrust of the jet. Similarly, W is the effective local lateral plume width which here can be replaced by $y_{1/2}$ (calculated by eq.5.4). In the absence of co-flow ($U_f/U_b = 0$) Hanna's criterion is met at the streamwise location $X_L/D = 174.5$, which is indeed very close to the prediction of Bennett's model ($X_L/D = 162$).

The fact that Hanna's model is based on the local characteristics of the plume allows its application in different wind speeds. In this way, following the procedure described in section 5.1, the streamwise evolution of U_b and $y_{1/2}$ are calculated for six different co-flow ratios. Accordingly, Hanna's parameter is recalculated, and the results are plotted in (Fig.5.3). It can be observed that increasing the co-flow

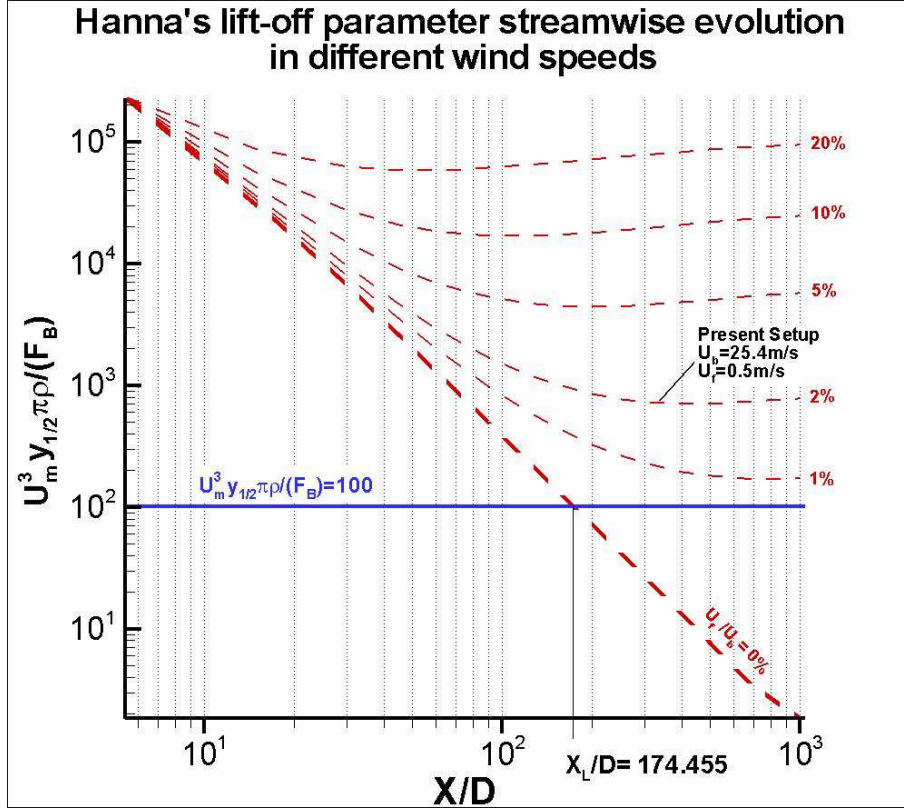


Figure 5.3: Hanna's lift-off parameter streamwise evolution in different wind speeds. Labels represent the value of U_f/U_b corresponding to the proximate curve.

velocity prevents the flow from meeting Hanna's criterion. Lift-off will occur for adequately low values of U_f , i.e. $U_f/U_b < 1\%$. This is justified to the fact that the most significant term opposing the domination of buoyancy in Hanna's model is U_m^3 , which in the presence of co-flow can not obtain a value lower than U_f^3 (see figure 5.1).

It is important to mention that our theoretical model for the estimation of the mean jet characteristics does not take into account two important parameters of the problem. First, the co-flow velocity varies with vertical distance (boundary layer), while the model assumes a constant value throughout the mixing layer. Second, the ratio of horizontal to vertical spreading rates of the *3DWJ* will gradually reduce with increasing the influence of buoyancy, which translates to lower values of $y_{1/2}$ (and larger values of $z_{1/2}$) when buoyancy becomes significant. Consequently, the curves of (Fig.5.3) are expected to shift downwards when approaching the value $\frac{U_m^3 y_{1/2} \pi \rho}{F_B} = 100$, which means that the lift-off onset X_L will

shift upwind for $0 \leq U_f/U_b \leq 0.8\%$, and lift-off will occur for even higher co-flow speeds, i.e. for $U_f/U_b > 0.8\%$, where lift-off has not been predicted by the present model.

5.2.2 Plume Lift-off Under the Baffle Influence

As explained in section 1.5.2, plume lift-off can be accelerated through the installation of an array of inclined baffles on the airfield behind the starting aircraft. The final baffle arrangement which has been tested in field trials by (Bennett *et al.*, 2013) and in wind tunnel experiments by (Velikov, 2013), in the present study called “*ORIGINAL baffle arrangement*”, consists of three radially distributed arrays of baffles (refer to section 6.2 for details). This setup is suggested to absorb $> 30\%$ of the initial jet thrust (Velikov, 2013). In the present section it is attempted to predict the new lift-off onset X'_L under the influence of the baffles.

The new lift-off onset X'_L is calculated from (eq.1.12), setting $X_L = 162D$ (calculated above), and $\frac{F_D}{F_M} \approx 30\%$ in agreement with the designer’s claim. We find out that the new lift-off onset is expected at $X'_L = 124D$, which is approximately 76.5% the travelled distance of the natural lift-off.

We can now assume a slightly different baffle arrangement, where the baffles are double in size, while the drag coefficient of the models remains the same. In the present work this setup is called “*LARGE baffle arrangement*”. In this case we expect the drag force to increase proportionally to the baffles’ frontal area; therefore, we expect $\frac{F_D}{F_M} \approx 60\%$. From (eq.1.12) it is now found out that the new lift-off onset is shifted further upstream at $X'_L = 81.5D$, which is approximately 50% the travelled distance of the natural lift-off.

Conclusion

The two reviewed theoretical models for buoyant plumes have been applied to the investigated setup of the present work in order to improve understanding of the problem. Although the model of (Bennett *et al.*, 2013), which only requires jet inflow conditions, does not take into account the co-flow (wind) velocity, it was found very useful for the estimation of the lift-off onset under the baffle influence.

On the other hand, the model of (Hanna *et al.*, 1998), which uses local values of the jet mean flow characteristics, allowed for the co-flow to be included on the theoretical analysis. The two models predicted similar travel distance for the natural plume lift-off, while this distance was calculated to reduce up to 22.5% with the original baffle arrangement of (Bennett *et al.*, 2013), and up to 50% with a twice as large a resistance.

This theoretical analysis indicates that plume lift-off highly depends on the magnitude of the co-flow (wind) velocity; however, more complex theoretical models are required for increased accuracy of the theoretical predictions of jet engine exhaust plumes in co-flowing winds. It is also suggested for future research that the local jet mean flow characteristics should be calculated properly, accounting for the smooth transition in the structure of the plume as its buoyancy gradually increases.

5.3 SIMULATED TIME REQUIREMENT (STR)

In transient jet simulations the solution propagates from the inlet-boundary, through which the jet enters the computational domain, towards the exit plane. The speed of propagation of the jet depends on the physical parameters defining the problem and it determines the time required for the jet to develop downstream. The jet evolution-time required to be simulated, so called, “*Simulated Time Requirement*” (*STR*), depends on the purpose of the simulation.

In particular, when statistical quantities of the jet *LES* are of interest, the following time intervals need to be simulated:

- the time required to establish a statistically steady flow pattern (T_0a_0)
- the time required to obtain 1st order statistics ($T_{11}a_1$)
- the time required to obtain 2nd order statistics ($T_{11}a_2$)
- the time required to obtain n^{th} order statistics ($T_{11}a_n$)

Therefore, the *STR* can be calculated by:

$$STR = T_0a_0 + \sum_{i=1}^n T_{11}a_i \quad (5.5)$$

Each time-interval of equation (5.5) can only begin after the previous time interval is complete. Before setting up a simulation it is crucial to identify the individual time intervals composing the *STR*. This would allow *a-priory* definition of the required number of timesteps at different stages of the simulation. In order to achieve this, the analytical expressions describing the mean jet characteristics, presented in section 5.1, are utilized in the following paragraphs.

5.3.1 Establishment of a Statistically Steady Flow

The first time interval is $T_0 a_0$ (eq.5.5), corresponding to the establishment of a statistically steady flow inside the computational domain. T_0 is the flow passage-time, defined as the time required for a fluid particle, which travels with the maximum local velocity U_m , to reach the farfield location of interest (*FLI*). Therefore, T_0 is calculated by:

$$T_0 = \frac{FLI}{U_{m_{AVG}}} \quad (5.6)$$

In order to achieve the goal of this time interval, T_0 has to be multiplied by the factor a_0 . When the initialization of the simulation uses constant values throughout the computational domain, the values $a_0 = 3$ (for axisymmetric jet), and $a_0 = 6$ (for *3DWJ*), are found appropriate. When the simulation is part of a parametric study the deviation of the new solution is not expected to vary significantly from the previous one; therefore, lower values are acceptable.

5.3.2 Statistical Convergence

The time required to obtain statistical convergence depends on the timescale of the most significant repetitive event, and the acceptable level of uncertainty of the statistical quantity examined.

The most significant event in a fully developed turbulent jet is the streamwise convection of the integral turbulent scale “*large eddy*”. (Pope, 2000) reports that the longitudinal integral length scale of self-similar axisymmetric turbulent jets is of the order of the velocity half-width. Similarly, the more recent *POD* investiga-

tion of (Gamard *et al.*, 2004) showed that the most energetic mode is mode-1 (helical mode) with Strouhal number $f z_{1/2}/U_m \approx 1$. Taking into account the co-flow velocity, the timescale of the streamwise convection of the integral turbulent structure lying at the centre of the mixing layer can be approximated by:

$$T_{11} \approx \frac{2L_{(z_{1/2}, y_{1/2})}}{U_m + U_f} \quad (5.7)$$

where the velocity half-width $L_{(z_{1/2}, y_{1/2})}$ is calculated by equation (5.4). It has to be noted that similarly to the velocity half-widths, in the axisymmetric jet there is a single time-scale which appears *axisymmetrically* in the mixing layer, while for the *3DWJ* there are different timescales in the vertical and the lateral directions. However, the estimation of T_{11} of the *3DWJ* is made based on the horizontal (largest) scale of the jet, $y_{1/2}$.

5.3.3 Evaluation of the Statistical Uncertainty

In order to evaluate the number of integral turbulent eddies required to pass through the *FLI* during averaging, it is necessary first to identify the averaging error.

In statistical terms, U_{rms} is the standard deviation of the velocity signal and a_1 is the time-averaging sample size. According to the analysis of (Ahn & Fessler, 2003), the standard error of the first order statistics (time average or mean), can be evaluated by:

$$\frac{\sigma_{\bar{U}}}{\bar{U}} = \frac{\frac{U_{RMS}}{\bar{U}}}{\sqrt{a_1}} \quad (5.8)$$

where $\frac{U_{RMS}}{\bar{U}} \approx 0.25$ at the centreline of self similar round jets (Pope, 2000).

The standard error of second order statistics (standard deviation or U_{RMS}) can be similarly estimated by:

$$\frac{\sigma_{U_{RMS}}}{U_{RMS}} = \frac{1}{\sqrt{2(a_2 - 1)}} \quad (5.9)$$

The values $a_1 = 35$ and $a_2 = 35$ are selected, while higher order statistics are not calculated in this investigation. The uncertainty of a cell's time-averaged streamwise velocity is $\sigma_{\bar{U}} = 0.04\bar{U}$ and the uncertainty of the *RMS* velocity is $\sigma_{U_{RMS}} = 0.12U_{RMS}$, based on the values of a_i mentioned above. Because the averaging process continues in parallel with the second order statistics process, the averaging sample finally used for the calculation of $\bar{U}$ includes both the values of a_1 and a_2 ; therefore, the uncertainty of $\bar{U}$ is $\sigma_{\bar{U}} = 0.03\bar{U}$.

These are the absolute, cell based, errors for cells lying on the jet centreline. However, in the analyses of this thesis the comparisons involve spatial interpolations of the averaged and *RMS* fields, implying much lower uncertainty; though, complicated to accurately quantify.

5.4 COMPUTATIONAL COST CONSIDERATIONS

The progress of this work depends on limited available computational resources; therefore, it is important to identify major factors which determine the computational cost of the simulation and accordingly implement efficient set-up. The present section highlights such sources of computational cost, and separately analyses its dependence on the farfield location of interest of a simulation (*FLI*), as well as the magnitude of the co-flow velocity U_f of the investigated setup.

5.4.1 Typical Factors Influencing Computational cost of LES

Although *Reynolds number* is a determining factor for the computational cost of *DNS*, it does not affect the computational cost of *LES* in free-shear flows; however, in wall-bounded flows the situation is slightly different with the computational cost increasing with Reynolds number as $Re^{1.8}$, that is the cost increases with a factor of 60 for every decade increase in Reynolds number (Pope, 2000). For this reason the Reynolds number of the *3DWJ* simulations performed in this study (i.e. $Re = 10,000$) is more than one decade lower than a real scale one (i.e. $Re = 300,000$), corresponding to more than two decades reduction of the compu-

tational cost.

Mesh density (or spatial resolution) and mesh *quality* are directly proportional to the computational cost. The former affects the duration of the simulation by influencing the time consumed for the implementation of the algorithmic operations, and the latter is responsible for the speed of convergence of the solution (i.e. the number of iterations per timestep). The *cell size* implicitly affects the computational cost; this is because the cell size is proportional to the *time-step size* according to the *CFL* condition. Since “*time*” can be seen as the fourth dimension of a transient three dimensional problem, time-resolution has a cost effect similar to the spatial ones. These factors require investigation before deciding on the optimum values.

5.4.2 STR in Computational Cost of Jet LES

Considering the theoretical jet-analysis presented at the previous sub-sections it is understood that the *STR* is a deterministic parameter of the computational cost of jet *LES*. It is already clear that *STR* depends on the order of statistics requested by the simulation (refer to section 5.3). Here, the theoretical equations describing the mean flow characteristics of jets, as well as the equations describing the individual time intervals of equation (5.5), are used to make quantitative predictions based on practical setups. It will be shown that the duration of each time-interval is highly affected by the most distant location of interest of a simulation, called “*Far-field Location of Interest*” (*FLI*), and by the strength of the co flow velocity (wind), here presented by the ratio U_f/U_b .

The parameters of the incompressible round jet of the experiment of (Garry & Holt, 2008), are used to give us a quantitative view of the theoretical model (refer to section 6.1.1 for the case description). This setup includes a nozzle of $D = 0.00635m$, data availability at $FLI = 200D$, jet bulk velocity $U_b = 23m/s$, and co-flow (wind) velocity $U_f = 5m/s$ corresponding to a ratio $U_f/U_b \approx 0.22$. Following the procedure described in previous sub-sections, the *STR* of this setup has been calculated $STR = 1.07s$, where the individual time intervals composing it are $T_0a_0 = 0.51s$, $T_{11}a_1 = 0.28s$ and $T_{11}a_2 = 0.28s$.

In a similar way, the parameters of the *3DOJ* experiment of (Velikov, 2013), are used (refer to sections 6.2 and 6.2.3 for detailed description). This setup includes a nozzle of $D = 0.006m$, data availability at $FLI = 200D$, jet bulk velocity $U_b = 25.4m/s$, and co-flow (wind) velocity $U_f = 0.5m/s$ corresponding to a ratio $U_f/U_b \approx 0.02$. The *STR* of this setup has been calculated $STR = 32.9s$, where the individual time intervals composing it are $T_0a_0 = 2.55s$, $T_{11}a_1 = 15.18s$ and $T_{11}a_2 = 15.18s$.

It is attempted to extend the theoretical predictions of *STR* examining its variation with FLI and U_f/U_b . In this way the individual time intervals of equation (5.5) have been calculated for six different velocity ratios in the range $0 \leq U_f/U_b \leq 0.2$ and for consecutive farfield locations of interest in the range $0 \leq FLI \leq 300$. The conditions of the experiment of (Velikov, 2013) (presented above) are selected for the calculations.

The variation of the individual time intervals of *STR* with U_f/U_b and FLI is presented in (Fig.5.4). In particular, the flow establishment time (T_0a_0) is plotted on the left. In this plot it can be seen that for both types of jet there is an exponential increase of T_0a_0 , with increasing the FLI , when there is no co-flow present $U_f/U_b = 0.0$ (thick lines). This is a result of the logarithmic decay of the maximum velocity U_m of the jet. In the case of a *3DWJ* the velocity decays faster (Fig.5.1), increasing the time requirement for the jet to travel towards the exit, however, the most significant factor here is the different value of a_1 , which for the axisymmetric jet is $a_1 = 3$, and for the *3DWJ* is $a_1 = 6$. The introduction of co-flow $U_f/U_b > 0.0$ has a significant “positive” effect in T_0a_0 for both types of jet. The cost appears to drop significantly with increasing U_f/U_b .

In a similar way, the required time for statistical convergence ($T_{11}a_i$) is plotted in (Fig.5.4, right). It can be seen that when there is no co-flow present, $U_f/U_b = 0.0$ (thick lines), this time interval grows exponentially with increasing the FLI . In particular, 30s of simulation are needed for collecting the sample for a single level of statistics at $FLI = 300$ in the axisymmetric jet, while this value jumps up to 300s for the *3DWJ*. In order to understand why these enormously large time requirements form in the absence of co-flow, one must, first of all, con-

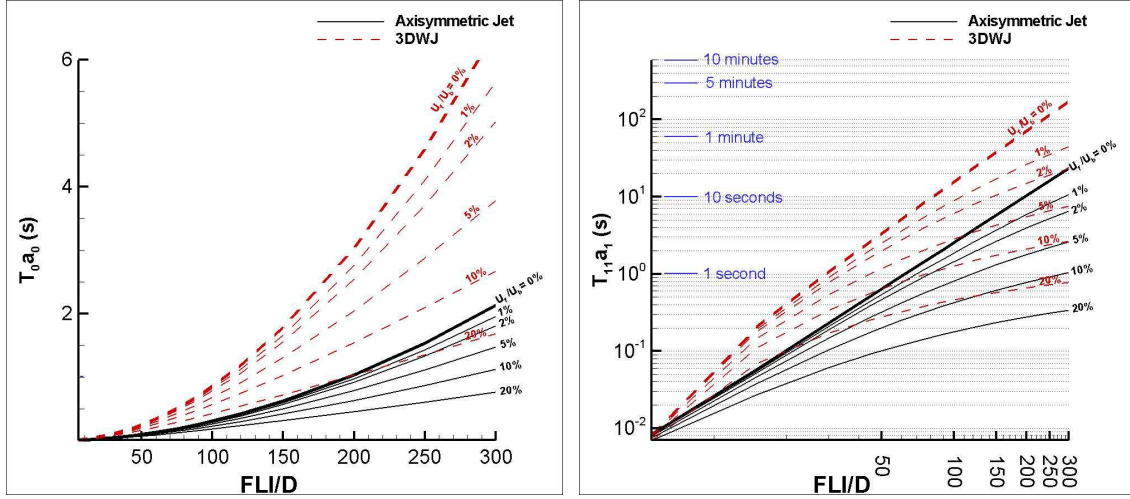


Figure 5.4: Variation of the individual time intervals of STR with FLI . The variation of the required time for the flow to establish a statistically steady behaviour $T_0 a_0$ is plotted on the left against FLI , and the required statistical convergence time $T_{11} a_i$ is plotted on the right. The labels on the plots represent the value of U_f/U_b corresponding to the proximate curve.

sider the linear increase with FLI of the integral timescale of jets equation (5.7), and that these turbulent structures are convected by the local jet velocity which approaches the zero value in a logarithmic decay rate. However, in the presence of co flow $U_f/U_b > 0.0$, the growth of the jet scales is delayed (Fig.5.2). In the same time the $a_i = 35$ eddies required to cross the averaging point (here calculated at FLI) are convected by the velocity at the centre of the mixing layer $U_f + \frac{U_m - U_f}{2}$, which tends asymptotically to the value of U_f (Fig.5.1). For *high* values of U_f the convective speed of these eddies is maintained at high level; therefore, the time interval $T_{11} a_i$ drops significantly.

The cumulative time requirement of the simulation $STR = T_0 a_0 + T_{11} a_1 + T_{11} a_2$ is plotted in (Fig.5.5, left). As expected from the above discussion, the STR follows trends of the statistical convergence time $T_{11} a_i$ at low co-flow speeds, and trends of the flow establishment time $T_0 a_0$ at high co-flow speeds. It is important to quantify the contribution of each time interval to STR for different co-flow velocities and for different types of jet. In this way, the ratio $\frac{T_{11} a_1}{T_0 a_0 + T_{11} a_1}$ is plotted in (Fig.5.5, right) against FLI for two types of jet and for six different co flows in the range $0 \leq U_f/U_b \leq 0.2$. It is observed that in the absence of co-flow $U_f/U_b = 0.0$ (thick lines) the contribution of the sample collection time for averaging $T_{11} a_1$ to the

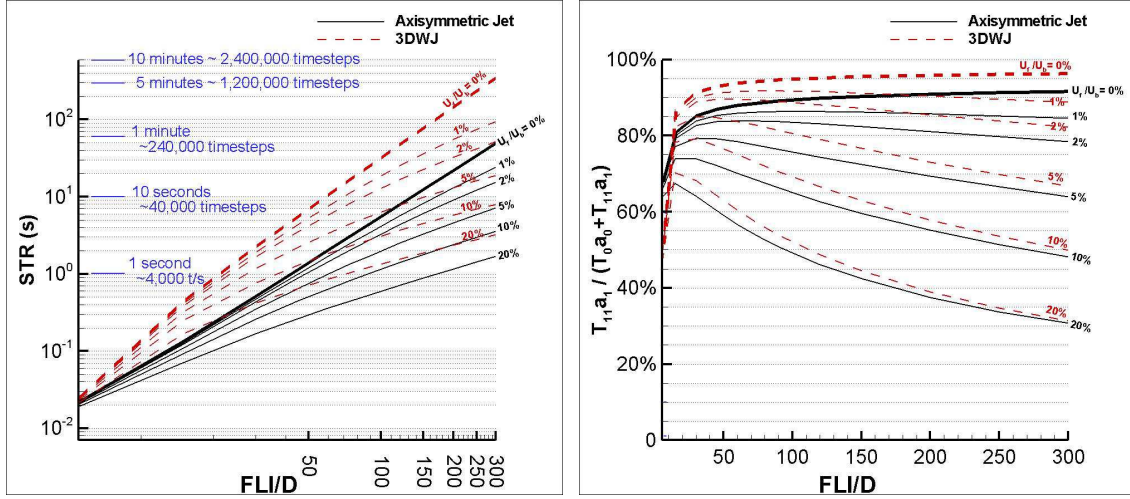


Figure 5.5: Variation of $STR = T_0a_0 + T_{11}a_1 + T_{11}a_2$ with FLI (left). And variation of the ratio $\frac{T_{11}a_1}{T_0a_0 + T_{11}a_1}$ with FLI (right). The labels represent the value of U_f/U_b corresponding to the proximate curve. The number of timestep requirement is calculated based on the typical timestep used $\Delta t = 0.00025s$.

simulated time (assuming the simulation focusses on averaging and not RMS) increases logarithmically with increasing the FLI , quickly dominating the simulated time. However, the introduction of co-flow significantly reduces the contribution of the averaging time $T_{11}a_1$, which at $X/D = 300$ becomes lower than the flow establishment time T_0a_0 . In the absence of co-flow the ratio $\frac{T_{11}a_1}{T_0a_0 + T_{11}a_1}$ increases with FLI while in the presence of co-flow it reduces proportional to U_f/U_b .

Conclusion

It is understood that the computational cost of jet LES , further to the typical factors influencing the cost of unsteady simulations, increases dramatically with extending the farfield location of interest (FLI) of the simulation. It has been also seen that in the absence of co-flow the simulated time requirement (STR) of a $3DWJ$ simulation becomes enormous at large FLI , increasing the number of timesteps, subsequently, the computational cost at prohibitive levels. However, the introduction of co-flow in the simulation was found to have a significant “positive” effect on STR , making the simulation of both the axisymmetric jet and the $3DWJ$ possible at large distances from the nozzle.

The presented theoretical model is being utilized during the simulation setup

in order to make a-priori definition of the number of timesteps of each stage of the simulation corresponding to the individual time-intervals of STR . In this way, consistency between different simulations is ensured.

Chapter 6

Description of Investigated Setups

The central aim of this work is the numerical investigation of the plume dispersion during the early takeoff stage. The high complexity of the flow under investigation requires simplifications to be applied on certain numerical parameters and boundary conditions, in addition to a necessary parametric validation of the selected solver. On the other hand, the theoretical analysis presented in chapter 5 clarifies that the cost of a $3DWJ$ in low speed co-flow, which is the situation at the early takeoff stage, is prohibitive even for small number of simulations. In this way, a series of different setups have been selected for the implementation of the validation tests before the final simulations were performed. Description of the experimental and/or numerical setups used throughout this thesis is included in this chapter.

6.1 SETUPS FOR THE VALIDATION TESTS

6.1.1 Single Free Circular Jet in Strong Co-flow

When performing jet *LES* at the very farfield, two flows of different scale have to be simultaneously resolved. Previous work dedicated in *LES* of jets shows that high mesh density is required in regions of the near-field, where the highly anisotropic transition to turbulence takes place $X/D < 20$ (Zhou & Luo, 2001; Salinas-Vazquez *et al.*, 2005; Zemtsop *et al.*, 2009); it is the same region where the highest velocities are encountered. Furthermore, in order to universally keep the *CFL* condition at reasonable levels, the timestep size Δt has to be kept low. Its value is determined by the near-field velocities and cell sizes according to $\Delta t = CFL \frac{\Delta x}{U}$, depending on the desired value of the *CFL* condition. On the other hand, motions on the farfield are slow and the scales of interest large, meaning that both the mesh spacing Δx and timestep size Δt required are larger than in the near-field. Additionally, computational cost also depends on numerical parameters such as the *convergence criteria* and the *under relaxation factors*, also determined by the near-field computations. Finally, the value of *turbulence intensity* superimposed at the velocity inlet, increases the difficulty of convergence. Accuracy of the computations in the near-field is not important for this study, because the area of interest is in the very farfield. However, near field can implicitly affect farfield predictions by downstream propagation of information. Therefore, investigation is required before any compromise or simplification is made.

Following the reasons clarified above, a series of sensitivity studies is required in order to establish a strong basis for the wall-jet simulations to follow. The experimental work of (Garry & Holt, 2008) is found convenient for this purpose, as it provides mean and *RMS* velocity profiles extending from the near-field $X/D = 5$ to the very farfield $X/D = 200$, and mean passive scalar profiles up to $X/D = 60$. In the same time, this setup includes a strong co-flow $U_f/U_b \approx 0.22$ that according to the theoretical analysis of section 5.4.2 minimizes the *STR*. The setup is described by a cold, single round air-jet to which a neutrally buoyant gas

is added. The jet has diameter $D = 6.35\text{mm}$ and bulk exit velocity $U_b = 23\text{m/s}$ and emerges to a co-flow of air with uniform velocity $U_f = 5\text{m/s}$.

O-Type meshing has been used for the discretization of the three dimensional volume. The dimensions of the computational domain and boundary conditions are illustrated in (Fig.6.1). In particular, a cylindrical domain is allowed here due to the low expansion rate resulting from the high co-flow. Furthermore, a cylindrical symmetry condition is used at the lateral boundaries in order to avoid mass fluxes from there. However, the domain extends $100D$ in the lateral directions, while the maximum jet diameter is less than $20D$ meaning that the boundary influence is negligible to the jet development. Discussion of the results of this study is made in chapter 7.

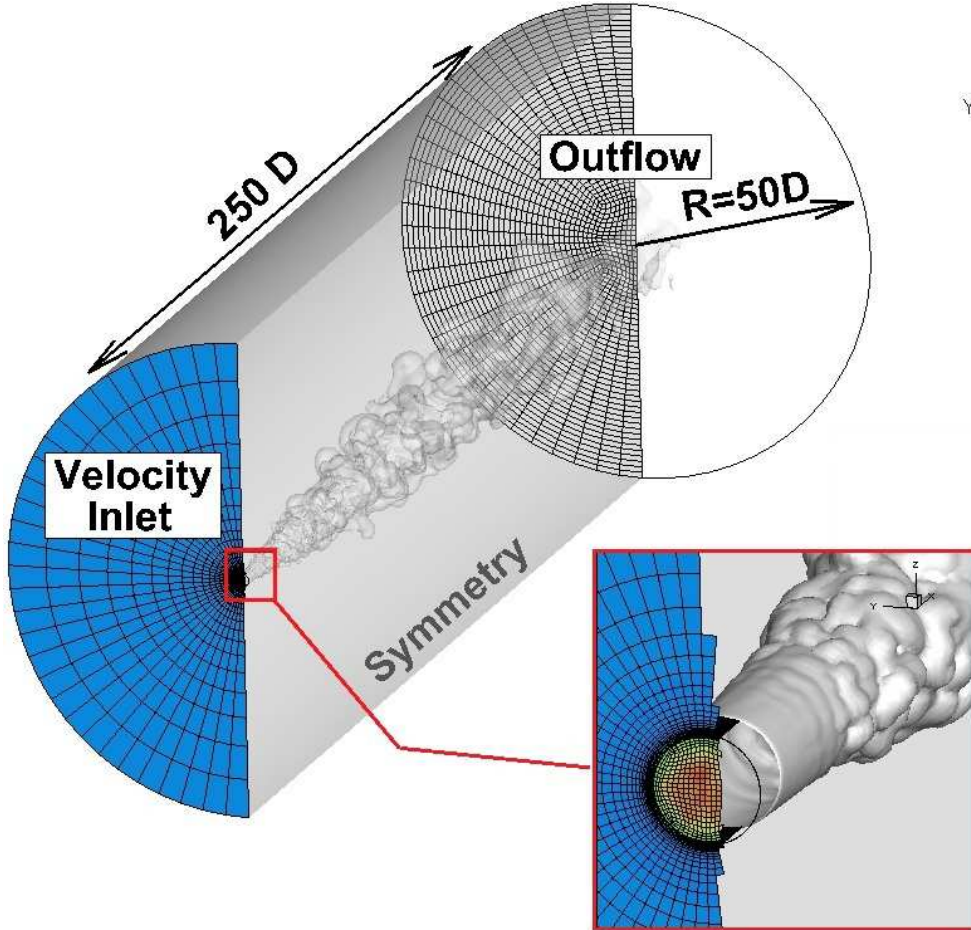


Figure 6.1: Free-jet computational domain dimensions and boundary conditions. Contours of streamwise velocity are colouring the inlet boundary. Iso-surfaces of constant velocity of the outer mixing layer are also included in order to highlight the position of the jet within the computational domain.

6.1.2 High Speed Free Coaxial Jet (Turbofan Model)

The vast majority of civil aeroplanes use the turbofan engine to achieve propulsion. The exhaust plume of a turbofan engine forms a two-stream coaxial jet. The outer stream is ejected by the by-pass duct and contains pure air in relatively low velocity and temperature. The inner stream is the one containing the high temperature gasses produced in the combustion chamber; therefore, containing the pollution of the engine.

The numerical modelling of a turbofan nozzle is highly complicated compared to the simple round jet, not only because of the increased complexity of the transitional region of the flow, but also because of the increased meshing complexity near the nozzle. It is suggested that the nozzle geometry is substituted by simple top-hat-like profiles for velocity and concentration at the inlet plane.



Figure 6.2: The short cowl coaxial nozzle (WARJEN) from the experiment of (Eastwood *et al.*, 2008).

Before carrying on with this substitution it is important to assess the effects of such a simplification. The experiment of (Eastwood *et al.*, 2008) is appropriate for this analysis, as it involves a scaled model of a turbofan engine, with available velocity measurements in the near-field taken through *LDA* and *PIV* techniques. The by-pass ratio of the nozzle is 8 : 1 and the by-pass duct exit diameter is 0.018m. The Reynolds number is matched to $Re = 300,000$ and the core and by pass exit velocities are set to 240m/s and 216m/s respectively. Finally, no buoyancy effects are taken into account in this study. The geometry of the ex-

perimental model, named *WARJEN*, is illustrated in (Fig.6.2). The geometrical profiles of *WARJEN* have been provided by prof. Paul Tucker (Cambridge University). The design and meshing of the nozzle and computational volume are implemented through the commercial software GRIDGEN. Discussion of the results this study is made in chapter 7.

6.1.3 3DOJ In Strong Co Flow

Introduction of a solid surface in proximity to the jet requires separate validation because of the different nature of the physics governing its development. In this way, an extensive parametric study is carried out repeating part of the free shear validation tests.

The strong co-flow axisymmetric jet of the experiment of (Garry & Holt, 2008) is also selected here by making a setup modification, being the addition of a solid surface at a distance 2.5 nozzle diameters offset from the nozzle. A schematic representation of the setup is illustrated in (Fig.6.3). The tests of this study are not accompanied by experimental data, however, there is a strong basis established by the axisymmetric jet tests described above. The meshing technique used is the modified O-Type mesh presented in (Fig.4.11). Discussion of the results of this study is made in chapter 8.

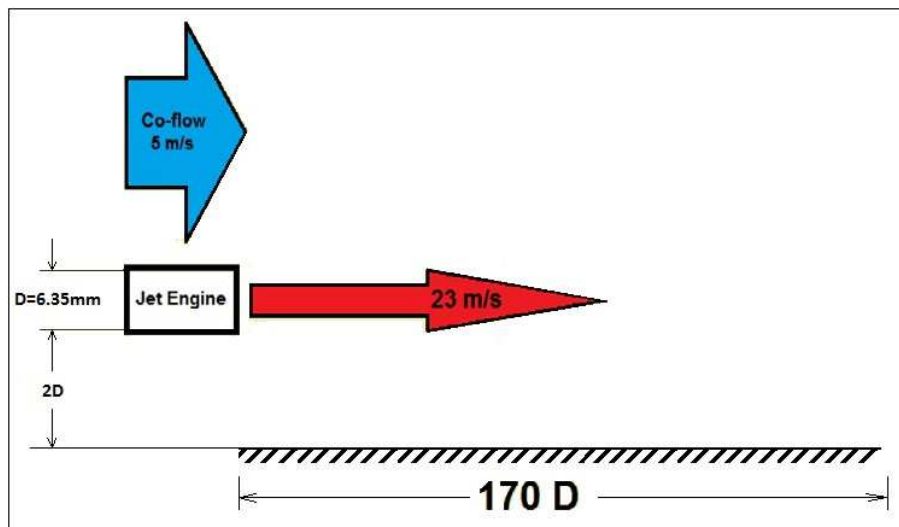
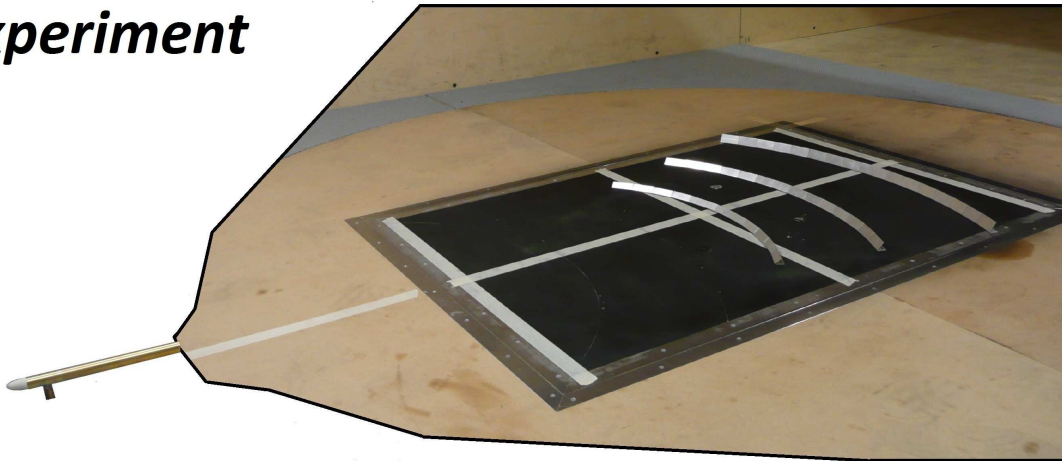


Figure 6.3: Schematic representation of the single circular offset jet in co-flow simulation. Modified setup of the experiment of (Garry & Holt, 2008).

6.2 ABLWT EXPERIMENTS

Wind tunnel experiments have been carried out in the Atmospheric Boundary Layer Wind Tunnel (*ABLWT*) of Cranfield University (Velikov, 2013). In the “*ORIGINAL*” setup, replicating the field-trials, three successive rows of baffle models with incidence angles of 40° , 50° and 60° respectively are arranged radially, centred at the runway-threshold as illustrated in (Fig.6.5). This baffle arrangement complies with the regulations of aerodrome licensing (CAA, 2011). Analytical discussion regarding the establishment of this baffle arrangement is available at (Bennett *et al.*, 2013).

Experiment



Simulation

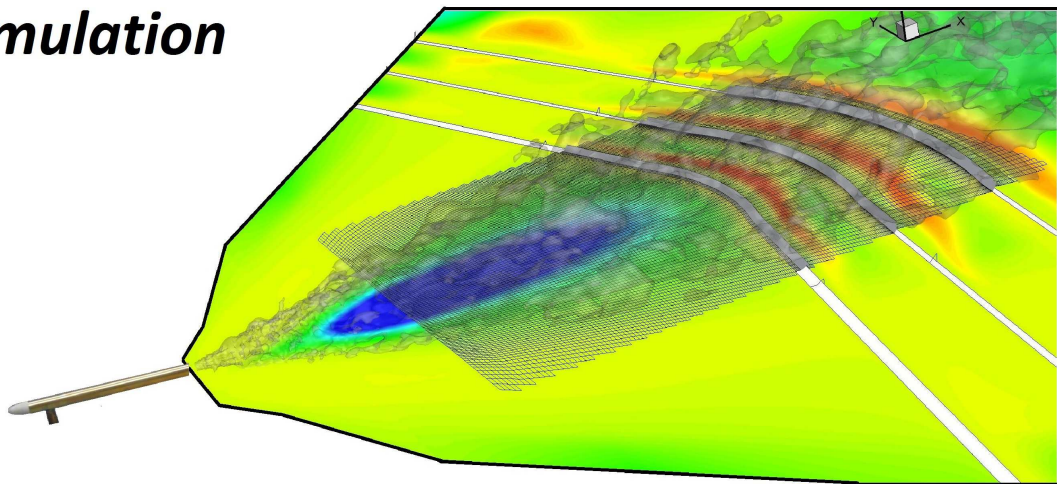


Figure 6.4: Baffle Arrangement on the Balance Platform. *ABLWT* setup of (Velikov, 2013) in juxtaposition with the numerical setup of the baffle testing simulation. The solid surface of *LES* is coloured with contours of x-wall shear. The balance platform (dark colour surface of the top figure) is highlighted in the *LES* figure (bottom) by drawing the mesh in the equivalent region.

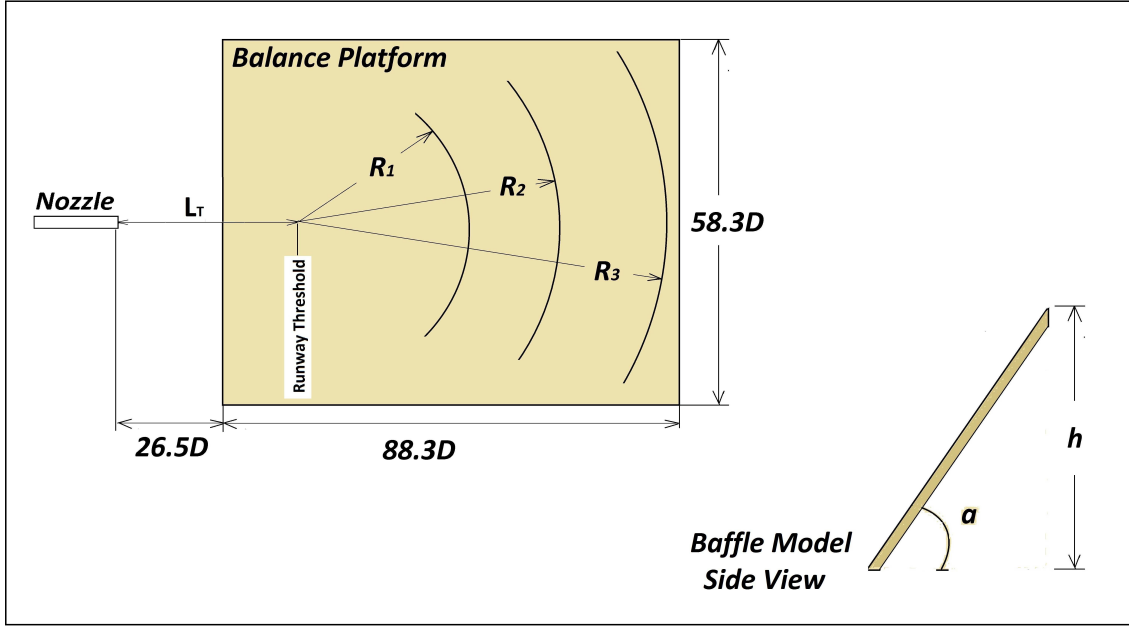


Figure 6.5: Schematic representation of the baffle setup. The length and width of the balance platform are not scalable and relate to the 1/100 scale setup.

The *ABLWT* experiments focus on three different types of measurements, including the aerodynamic drag experienced by the baffles, pollution concentration using a Flame Ionization Detector (*FID*), and mean velocity using the Hot Wire Anemometry (*HWA*) technique. (Fig.6.4) illustrates a perspective view of the baffles on the balance platform, in juxtaposition with an equivalent view of the *LES* domain. The contours of x-wall-shear in combination with the velocity iso-surfaces apparent in this figure help for the understanding of the problem setup.

Due to experimental restrictions it was impossible to perform all tests on the same scale. In particular, for a large scale, prohibitive amount of methane had to be released causing health and safety issues and large amount of helium would be required to achieve buoyancy of the plume, increasing the cost at a prohibitive level. On the other hand, for a small scale the error of the balance platform would be large (Velikov, 2013). Therefore, the experiments are employed in two different scales. The following paragraphs include basic information regarding the individual setups used for the *LES* simulations, the results of which are analysed and discussed in chapters 9, and 10.

6.2.1 1/100 Scale (*Drag Force Measurements*)

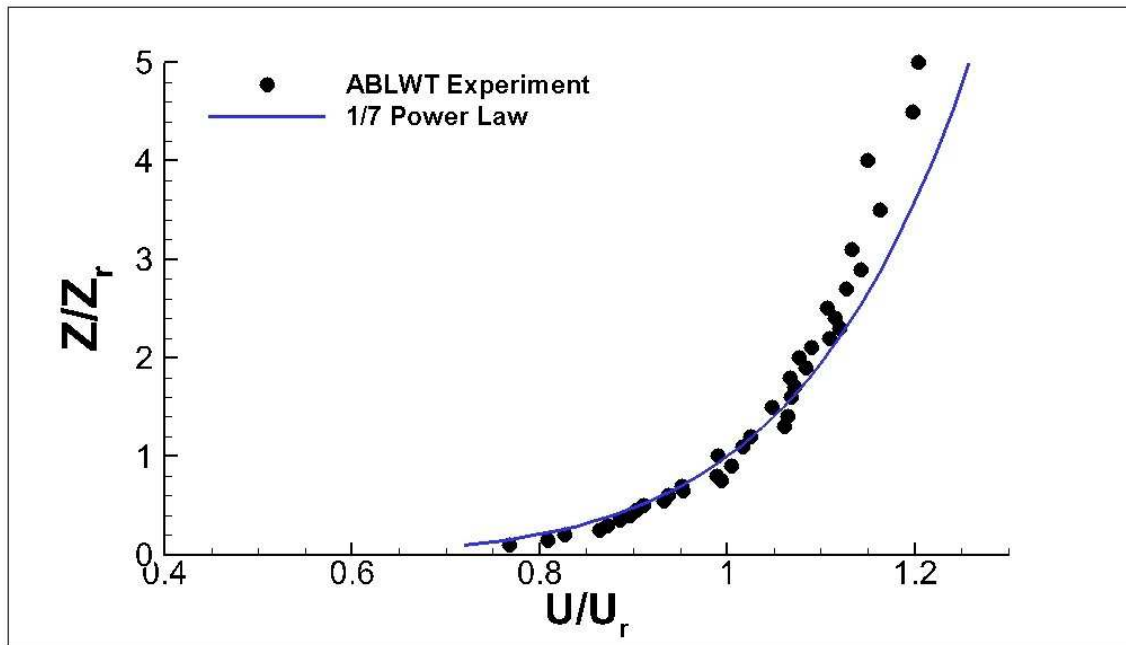
The objective of this setup is to compare the averaged drag force recorded by the balance platform with the one predicted numerically. This is especially useful for the evaluation of the Dynamic Response Algebraic Model for Baffle Imitation (*DRAMBI*). The first setup includes a cylindrical nozzle with diameter $D = 12mm$ located at an offset $h = 24mm = 2D$ above the wind tunnel floor. The jet bulk velocity is $U_b = 35.6m/s$ and the co-flow velocity forms an *ABL* velocity profile, flowing in the same direction with the jet, well described by the 1/7 wind profile power law relationship:

$$\frac{U_f}{U_r} = \left(\frac{Z}{Z_r} \right)^{(1/7)} \quad (6.1)$$

The control velocity U_r , and control height Z_r of equation (6.1) have been adjusted to the values of $0.7m/s$ and $0.08m$ respectively in order to fit the experimental profile as presented in (Fig.6.6). The free stream mean velocity measurements in the wind tunnel experiments include high uncertainty as stated by (Velikov, 2013). In particular it was reported that the low co-flow speed varied about 10% during different measurements due to the ambient conditions (refer to section 6.2.4). It can be observed in (Fig.6.6) that a lower exponent of the power law relationship, i.e. $1/9$, would better represent the experimental data, but the simulations have been run before the experiments were conducted; therefore, not allowing for such a refinement to be applied. Although the field-trials include a cross wind configuration (described on the second setup below), this parameter was found trivial with respect to the drag force measurements; therefore, was excluded from both the experimental and *LES* setups. Dimensions of the baffle arrangement are presented on table 6.1. The values are in correspondence of the graphical representations in (Fig.6.5). The results of the first setup are analysed in section 9.1.

Table 6.1: Geometrical details of the 1/100 scale baffle arrangement.

Baffle Testing Setup (Threshold Distance = 47.5 D)			
	1st Array	2nd Array	3rd Array
Centre Distance From Nozzle	77.5	91.9	109.2
Radius of Baffle Array	30	44.4	61.7
Baffle Height (h)	0.75	1.09	1.44
Baffle Angle (α)	40 deg	50 deg	60 deg
Lateral Extend	28.6	45.8	56
length dimensions are non-dimensionalized by the nozzle diameter (D)			

Figure 6.6: Wind profile as measured in the *ABLWT* experiment, in comparison with the 1/7 power law fit applied for the simulations of this thesis.

6.2.2 1/200 Scale (*Cross Wind Pollution Measurements*)

This setup is a 1/200 scale of the field-trials. In a separate, yet unpublished, work, Stefan Velikov (Velikov, 2013) compares the experimental measurements of this setup with the *LIDAR* measurements recorded in the field-trials. In the present work, data is used to make comparison of the maximum concentrations of pollution at different flow stations between *LES* and experiment. The spatial dimensions of the second setup are half the ones mentioned at the first setup above. According to the conditions recorded at the field-trials, the wind direction is not aligned with the jet, but it is angled at 35° to the jet centreline. The experimental setup is illustrated on (Fig.6.7). In order to accommodate the cross flow the *LES* computational domain is modified (stretched) (Fig.6.8).

The jet bulk velocity is adjusted, based on the Froude number, so that $U_b = 25.4m/s$. Accordingly, the control velocity of the wind profile is $U_r = 0.5m/s$. In order to achieve buoyancy of the plume in the wind tunnel, a mixture of air and helium was used for the jet flow. The analogy forms a density difference between the jet and the co-flowing wind, similar to the one between air at $412K^1$ and ambient air at $290.15K$. Additionally, in order to examine the pollution dispersion in the farfield, methane was also added to the gas mixture.

FID measurements of methane concentration have been implemented at four different farfield stations and offsets from the ground. Comparison of the *FID* data with the *LES* results is performed in terms of maximum² mean concentrations at the streamwise stations $X/D = 80.8, 114.2, 131.7$, and 148.3 . Measurements are available at the vertical offsets $Z/D = 4.2$ and 8.33 , while at the last station $X/D = 148.3$ measurements are also available at the offsets $Z/D = 12.5$ and 16.7 . The results of the second setup are analysed in section 9.2.

¹This temperature is calculated using the Steady Flow Energy Equation with considering the engine thrust, mass flow and fuel flow rates (Velikov, 2013)

²Maximum refers to the pick value of the mean concentrations recorded along a profile.

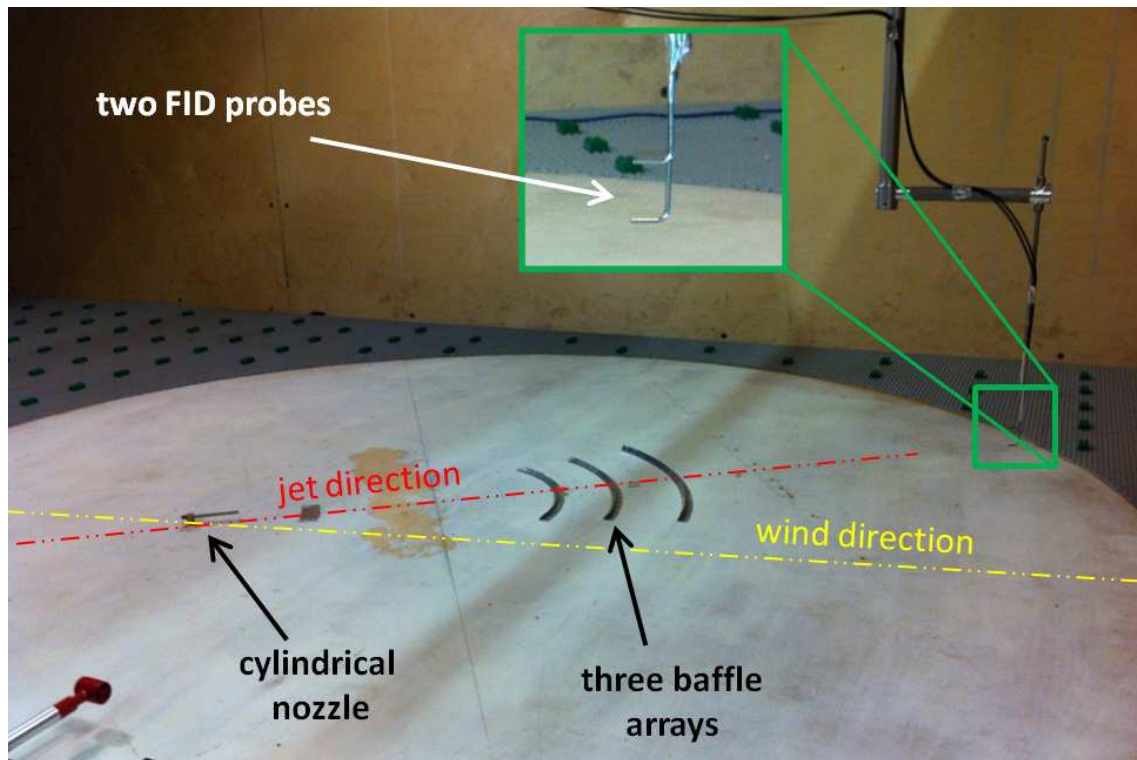


Figure 6.7: Picture from the *ABLWT* illustrating the 1/200 scale setup, with the *FID* probes in position $X/D = 170$.

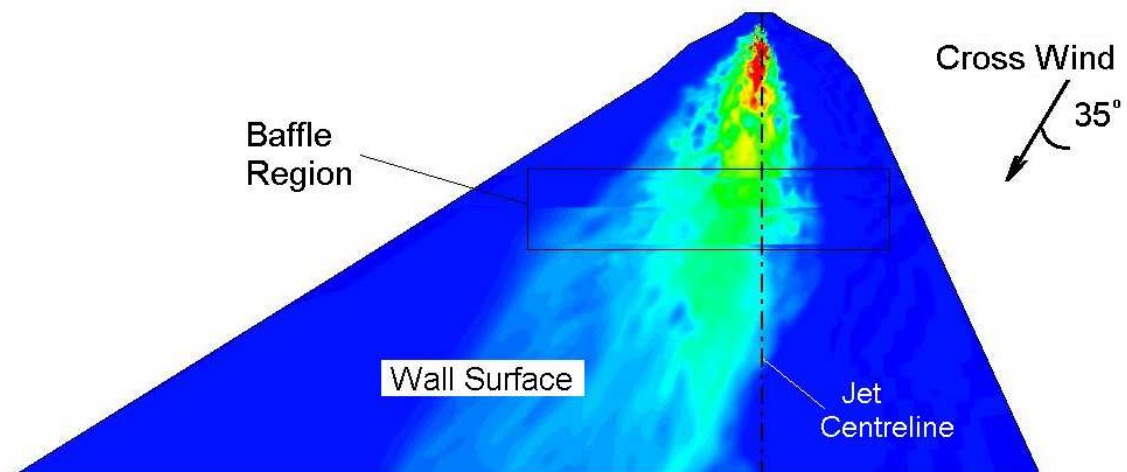


Figure 6.8: Modification of the shape of the computational domain in order to accommodate the cross flow simulation.

6.2.3 1/200 Scale (*Final Setup for FID and HWA Measurements*)

Further cross-validations between the *LES* results and the experimental measurements are essential to enhance understanding of the geometrical characteristics of the wind-brake system. A close cooperation of the authors with the experimental group mentioned above has lead to the formation of the final experimental setup, analytically described here. The methodology followed includes gathering a comprehensive dataset of both mean streamwise velocity ($\bar{U}$) and passive scalar concentration ($\bar{\Phi}$), while varying the geometrical characteristics of the wind-brake system.

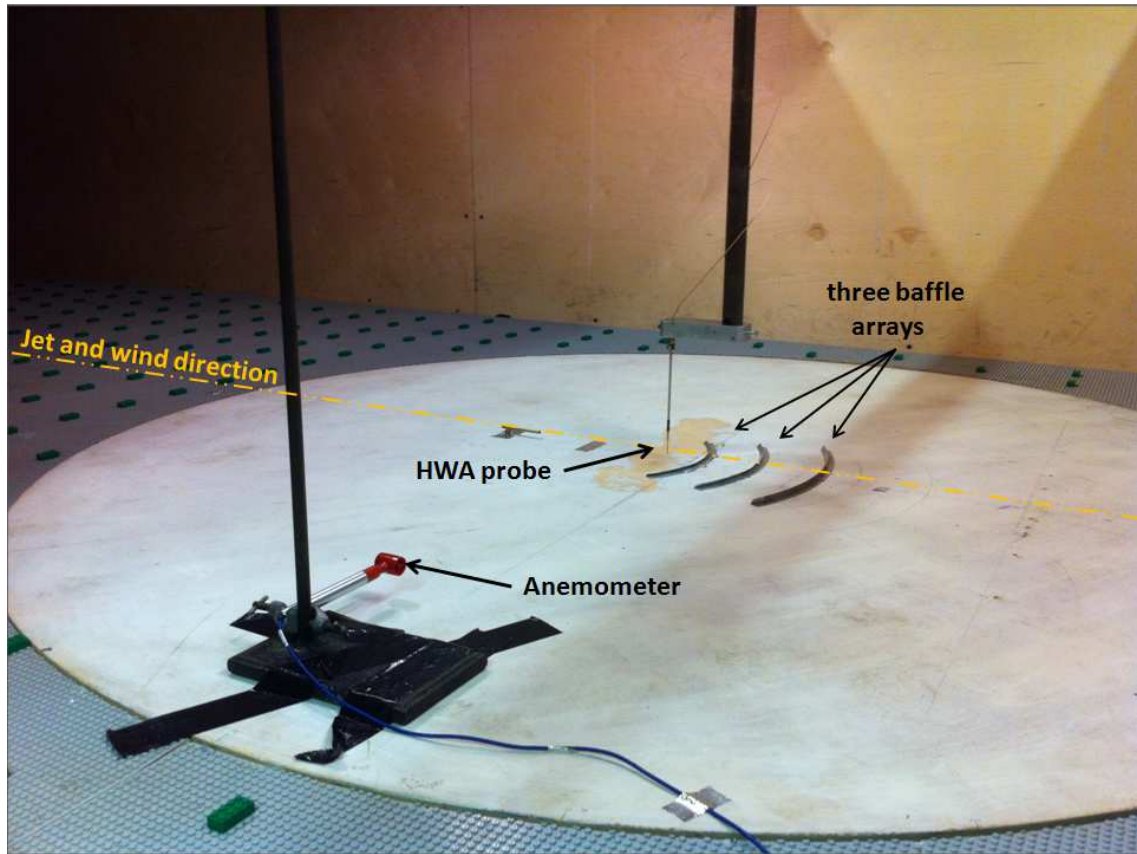


Figure 6.9: Picture from the *ABLWT* illustrating the final setup, with the original baffle arrangement installed and the *HWA* probe in position $X/D = 40$.

Considering the experimental limitations mentioned in section 6.2.2, the final experiments are also performed on a 1/200 scale of the field-trials. Due to dimensional wind tunnel limitations the maximum streamwise extent of the measurements is $X/D = 200$. Considering this constraint, the distance between the run-

way threshold and the nozzle, (L_T appearing in Fig.6.5), is reduced from $47,5D$ to $20D$. This threshold distance has been an alternative investigated scenario in the field-trials, therefore, this is a valid modification.

In order to reduce complexity of the numerical simulation, the wind direction is set parallel to the direction of the jet (head-wind configuration). Other setup parameters are identical to the second setup (refer to section 6.2.2).

There are three different baffle arrangements participating at the *LES*-Experiment cross-validations, the details of which are summarized on tables 6.2-6.4. The *ADDITIONAL* baffle arrangement is an extended version of the *ORIGINAL* baffle arrangement which includes three extra baffle arrays downstream. The height and spacing between the new baffle arrays has been calculated following trends of the already optimized by (Velikov, 2013) first three arrays, as illustrated in (Fig.6.10). The results of the final experiments are analysed in section 10.2.

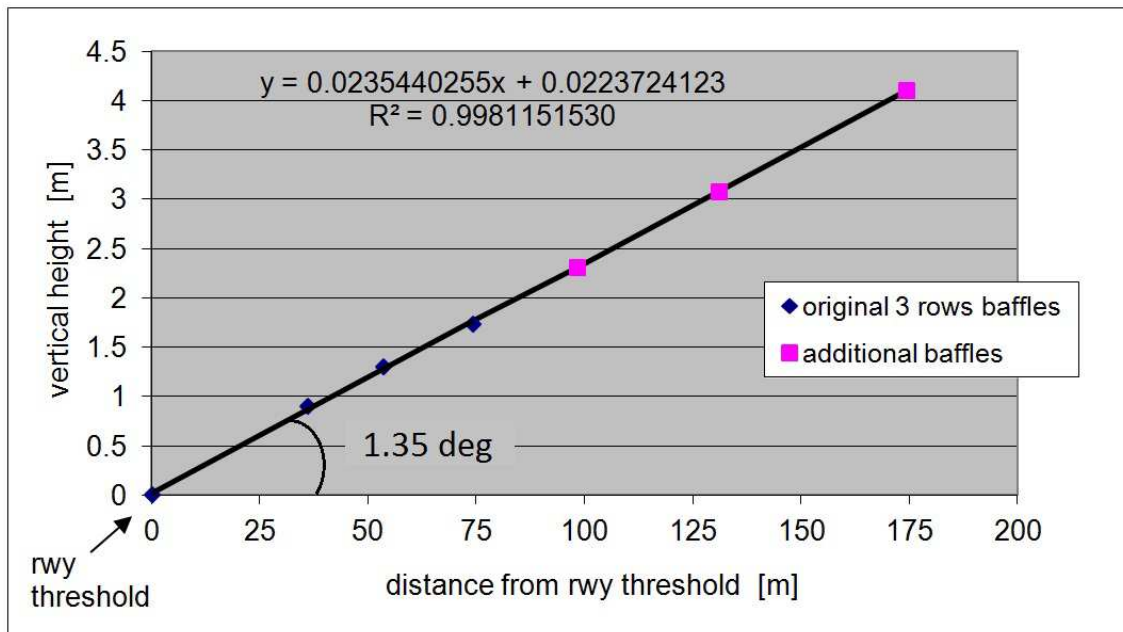


Figure 6.10: Calculation of heights and spacing between the additional baffles.

Table 6.2: Geometrical details of the *ORIGINAL* baffle arrangement. The Additional baffles are part of an extended numerical study and have not been part of the *ABLWT*.

Original Baffle Setup (Threshold Distance = 20D)				Additional Baffles		
	1st Array	2nd Array	3rd Array	4th Array	5th Array	6th Array
Centre Distance From Nozzle	50	64.4	81.7	101.2	129.2	165.4
Radius of Baffle Array	30	44.4	61.7	81.2	109.2	145.4
Baffle Height (h)	0.75	1.09	1.44	1.93	2.57	3.42
Baffle Angle (α)	40 deg	50 deg	60 deg	60 deg	60 deg	60 deg
Lateral Extend	28.6	45.8	56	64.2	74.5	89.1
length dimensions are non-dimensionalized by the nozzle diameter (D)						

Table 6.3: Geometrical details of the *LARGE* baffle arrangement.

Large Baffle Setup (Threshold Distance = 20D)			
	1st Array	2nd Array	3rd Array
Centre Distance From Nozzle	50	100	134.5
Radius of Baffle Array	30	44.4	61.7
Baffle Height (h)	1.50	2.17	2.89
Baffle Angle (α)	40 deg	50 deg	60 deg
Lateral Extend	28.6	45.8	56

Table 6.4: Geometrical details of the *THRESHOLD* baffle arrangements.

	Threshold	Threshold Large
Centre Distance From Nozzle	20	20
Radius of Baffle Array	inf	inf
Baffle Height (h)	1.44	2.89
Baffle Angle (α)	60 deg	60 deg
Lateral Extend	LES=21.5	EXP=30, LES=21.5

6.2.4 Experimental Equipment

Basic information regarding major experimental equipment used in the atmospheric boundary layer wind tunnel is summarised in the following paragraphs.

Balance Platform

Drag was measured using a one-component force balance modified at Cranfield University from an existing custom-built design intended for direct skin-friction measurements. *“It consists of a rectangular platform, measuring $1060 \times 700\text{mm}$, mounted on a circular floating plate. The floating plate is supported by flexures carrying four symmetrically arranged strain gauges, each in a Wheatstone bridge electrical circuit. A similar design is described by Winter and Gaudet (Winter & Gaudet, 1973). Loads applied on the platform cause distortion of the strain gauges, which in turn changes their electrical resistance and thus the voltage of the Wheatstone bridges. This voltage output was calibrated versus known applied forces prior to starting the tests and the balance precision was determined to be within $\pm 0.01\text{N}$ ”* (Velikov, 2013). However, the uncertainty of the measurement was evaluated $\pm 0.002\text{N}$ based on the repeatability recorded during experiment.

Flame-Ionisation-Detector

The Flame-Ionisation-Detector (*FID*) technique used a tracer gas (Methane) of known concentration in the nozzle jet, i.e. $20,000\text{ppm}$, to detect its concentration downstream of the nozzle within the developing plume. It is a standard method of measuring concentration of hydrocarbons by passing them through a hydrogen-air flame where they are burnt producing ions, and detecting the current proportional to the rate of ionisation. The uncertainty of the *FID* data is $\pm 0.2\%$ based on the repeatability of the measurements.

Hot-Wire-Anemometer

Velocities data are collected with a standard Hot-Wire-Anemometry (*HWA*) technique using a one-dimensional hot-wire probe to measure velocity in the direction

of the flow. The method uses a fine wire, positioned perpendicular to the flow direction, which is pre-heated before starting the measurements. When subjected to the flow the wire cools down changing its electrical resistance which is sensed and recorded as voltage output. Prior to starting the measurements, this voltage output was calibrated versus measured flow velocity within the range of expected velocities. The uncertainty of the *HWA* measurements was statistically calculated for several cases, and was found to lie within 5 – 6%; however, lower velocity cases, i.e. around $0.6 - 0.7 \text{ m/s}$ (as we move away from the plume in to the boundary layer of the free-stream), the uncertainty may grow up to 10 – 12%.

Chapter 7

Validation Tests

Part 1: *Free-Shear Jets*

The first part of the validation tests is dedicated to free-shear jet mixing. In this way, the ability of the selected numerical approach to predict free-shear jet mixing is assessed, the solution sensitivity to a variety of numerical and physical parameters is analysed, and reduction of the problem complexity is achieved where appropriate. Two different experimental setups are employed serving different targets of the validation. This chapter is accordingly separated in two sections; the first one investigating a free, circular, low Reynolds number jet in strong co-flow, and the second one a high-speed co-axial jet emerging in a static environment.

7.1 SINGLE FREE AXISYMMETRIC JET IN CO-FLOW

The experimental setup of (Garry & Holt, 2008) has been replicated numerically according to the information presented in section 6.1.1. Numerical and physical parameters of the selected approach are investigated in this section.

Comparisons of the *LES* predictions with the experimental data are made in terms of mean and *RMS* velocity profiles, in the range $5 \leq X/D \leq 200$, and mean concentrations at $X/D = 15, 30$ and 60 . The *FLI* of the simulations varies according to the nature of the parameter examined. Moreover, the dataset of mean concentrations $\bar{\Phi}$ was not available at the beginning of this study; therefore, comparisons of the first six tests are limited in $\bar{U}$ and U_{rms} profiles. The complete map of the comparison plots of this tests is included in Appendix A.

7.1.1 Meshing Approach

The first test to be carried out is “*Meshing Approach*”. Three different types of meshing have been tested at the intermediate field $FLI = 45$. In this way, the results of an H-Type mesh of 3.5×10^6 cells, a radial (revolution) mesh of 1.3×10^6 cells, and an O-Type mesh of 1.5×10^6 cells, are compared in (Fig.A.1), using the experimental results of (Garry & Holt, 2008) as reference. The domain size in the streamwise direction of the H-Type mesh is $55D$ while for the other two cases is $100D$. The numerical setup also includes: jet inlet turbulence intensity $I = 5\%$, timestep size $\Delta t = 5 \times 10^{-6}$, and *SGS* modelling by *DSM*.

At the first stations $X/D = 5$ and 15 the predictions of the H-Type mesh are superior to the other two approaches in terms of both the mean and the *RMS* velocity profiles. This is attributed to the higher density of the H-Type mesh in this regions, which reduces contribution of the *SGS* modelling. However, all three methods quickly converge to similar solutions downstream, according to the observations at $X/D = 45$. It has to be stressed here that the H-Type mesh statistical convergence time T_{11a_i} was smaller than that of the other cases, therefore, the *RMS* profile is less smooth in this case at the last station plotted, $FLI = X/D = 45$.

All three methods appear appropriate for the simulation of axisymmetric jets.

The H-Type mesh, however, is a much more expensive one, and the revolution mesh is difficult to apply in cases where the jet axis of symmetry no longer exists, i.e. *3DWJ*. For this reason the investigation continues with the O-Type meshing approach, also applying the improvements presented in section 4.2.1.

7.1.2 Convergence Criteria

Next step is to test the required level of accuracy of the computations. In a flow such as the free axisymmetric jet the difficulty of convergence decreases logarithmically with downstream distance. This is easy to understand considering the logarithmic decay of the jet centreline velocity and the closely linear expansion of the jet scales downstream. The time advancement of the solution is achieved by the progressive solution of discrete time intervals of the problem, determined by the selected timestep size Δt . Consider two different eddies of the flow, one being a Kelvin-Helmholtz ring at the initial shear layer, with velocity $\approx U_b/2$, and with streamwise length scale L_{11} comparable to the nozzle diameter D , and the second being an eddy responsible for the helical mode at the farfield of the jet with velocity close to U_f and streamwise length scale L_X a few decades larger than D . The flow conditions at a computational cell lying at the region of the former eddy vary significantly during time Δt , while the solution at the farfield, where the latter eddy develops requires larger times to achieve comparable flow variations. This means that the farfield computations are easier to converge, therefore, the local error is much lower there than at the near-field. On the other hand, FLUENT only allows for universal values of the convergence criterion to be specified for each solved equation. Therefore, the selection of the value of the *convergence criteria* requires testing. In this way, a 2.1×10^6 cells O-Type mesh has been used. The computational domain extends $55D$ in the streamwise direction. The rest of the numerical parameters are kept the same as in the previous test.

The simulation has been repeated three times, setting three different values for the *convergence criteria* 10^{-3} , 10^{-4} and 10^{-5} . The number of iterations per timestep required for each case varies (Table 7.1). The same table presents the simulated time of each case. It is important to mention here that due to

Table 7.1: Variables and History of the Simulations of Study “*Convergence Criteria*”.

	Con. Criterion	Iterations/Timestep	Simulated Time
Case A	10^{-5}	8	0.17s
Case B	10^{-4}	5	0.25s
Case C	10^{-3}	2	0.29s

lack of experience during the implementation of these initial tests the convergence time of the second order statistics has not been well controlled, resulting to increased uncertainty of the U_{rms} results, especially at the most remote region $X/D = FLI = 45$.

Furthermore, the value of the *convergence criterion* is being selected the same for all solved equations, including continuity, momentum and scalars (although the latter is not included at this test). However, it has been observed that continuity (pressure) is the slowest to converge; therefore, being the one to determine the number of iterations required for each timestep. For instance, when continuity reaches the residual error of 10^{-5} , the rest of the equations approach much lower values, in the order of 10^{-7} . However, for clarity and consistency of the study the discussion will continue with a unique error value for each simulation.

The results of this test are presented in (Fig.A.2). It is observed that reducing the requested level of accuracy has minor effect on the velocity statistics examined at both the laminar region $X/D = 5$ and the turbulent region $X/D > 5$. For the rest of this study we select the intermediate accuracy 10^{-4} . Lower accuracy than this causes the number of outer iterations per timestep to drop very low (Table 7.1), which is unacceptable according to (Fluent, 2012) suggesting 5 – 10 iterations per timestep. Any higher accuracy requested only increases the number of the outer iterations per timestep, while the *LES* predictions stay effectively unaffected.

7.1.3 Timestep Size

In order to investigate the effects of the timestep size the simulation is repeated for three different values of Δt . An attempt is made here to achieve higher values of Δt while keeping the number of outer iterations in the range suggested by (Fluent, 2012) (5-10 iterations/timestep). In this way, the near-field radial resolution of the mesh has been reduced having been found unnecessary (refer to section 7.1.4). This has allowed to increase the streamwise resolution also reducing the two dimensional aspect ratio AR_{2D} of the near-field cells. The resulted mesh consists of 2.6×10^6 cells. Here the convergence criteria are set to the value 10^{-4} for all three cases. Three different values of Δt have been investigated (Table 7.2). Increasing the value of Δt increases the difficulty of convergence, therefore, increases the number of outer iterations required to reach the requested accuracy of convergence. Finally, the increased timestep size has allowed increased simulated time, therefore, uncertainty of statistical convergence is reduced comparing to the previous test.

Table 7.2: Variables and History of the Simulations of Study “*Timestep Size*”.

	Δt	Iterations/Timestep	Simulated Time
Case A	1.0×10^{-5}	6	0.30s
Case B	1.5×10^{-5}	8	0.53s
Case C	2.0×10^{-5}	10	0.56s

The results of this test are presented in (Fig.A.3). The differences between the three different cases are infinitesimal in both the first and second order statistics of the streamwise velocity. The differences with the previous test, where an even smaller timestep had been used, are also very small. Considering the above, it is concluded that the timestep size is still small. Therefore, further coarsening of the near-field meshing is expected to be acceptable.

7.1.4 Inlet Turbulence Intensity and Shear layer Resolution

Before carrying on to further coarsening the mesh and increase Δt it is essential to investigate two parameters with impact to the quality and cost of the near-field computations.

Transition to turbulence of free jets is susceptible to a variety of parameters, both physical and numerical. A highly deterministic physical parameter is the momentum thickness of the jet efflux, which presently is defined by the $1/7$ power law equation, representative for fully developed pipe flows. Another physical parameter is the *turbulence intensity* of the velocity fluctuations at the nozzle exit (jet inlet). The former is not subjected to investigation in this section; however, the latter is found necessary to be parametrised and tested.

Inlet Turbulence Intensity

The experiment of (Garry & Holt, 2008) suggests that the flow exiting the round pipe is fully developed and the local turbulence intensity is $I = 10\%$. The artificial generation of such strong perturbations at the inlet streamwise velocity component, being achieved by the *spectral synthesizer* algorithm, has been observed to significantly delay solution convergence. For this reason, lower values of turbulence intensity are considered here. In a similar format with the previous studies, three different values of turbulent intensity $I = 0\%$, 5% , and 10% are investigated. The computational mesh of 2.1×10^6 cells is used. The variables and history of the three simulations are presented in (Fig.7.3). It is observed that the timestep size used in Case A ($I = 0\%$) is smaller than in the other two cases. Such a small timestep was necessary in this case in order to give time to the solver to form random anomalies based on the asymmetric numerical diffusion in order to avoid a symmetric (unrealistic) transition. However, as revealed from the discussion of paragraph 7.1.3, the *LES* solution is independent from timestep in the selected range; therefore, this setup-difference between the compared cases is acceptable.

The results of study “*Turbulence Intensity*” are illustrated in (Fig.A.4). Very

Table 7.3: Variables and History of the Simulations of Study “Inlet Turbulence Intensity”.

	I	Δt	Simulated Time
Case A	0%	5×10^{-6}	0.23s
Case B	5%	1.5×10^{-5}	0.40s
Case C	10%	1.5×10^{-5}	0.45s

close to the nozzle, at $X/D = 5$, specification of the value suggested by the experiment $I = 10\%$, meets the best agreement with the experimental data. As expected, lower value of the inlet turbulence level delays jet transition, therefore, the peaks of both the mean and RMS profiles in (Fig.A.4) are always higher for lower intensities in the turbulent region of the jet. In the same time it was observed that $I = 5\%$ has infinitesimal effect to the convergence of the solution, while $I = 10\%$ has a significant effect, delaying the solution by approximately 30%, as observed by the authors during the simulations.

Shear Layer Resolution

Except for the physical factors influencing the length of transition, it is also subjected to the numerical modelling. In particular, the SGS contribution increases with increasing the cell size. The most important cells in the potential core region $0 \leq X/D \leq 6$ are the ones resolving the mixing layer, wherein the primary instabilities grow and interact with each other (refer to section 1.4.1). Provided that the streamwise mesh resolution is kept identical between different simulations, a crucial parameter determining the length of transition is the mesh resolution across the initial shear layer. In order to investigate this numerical parameter, this study compares the results of four different simulations in which the mesh resolution across the initial shear layer varies. The values of the lateral spacing Δ_r at $X = 0$ (nozzle exit) tested are 0.18×10^{-4} , 0.4×10^{-4} , 0.7×10^{-4} , and 2.6×10^{-4} . The streamwise resolution of the mesh is kept similar between the four different cases. The largest value $\Delta_r = 2.6 \times 10^{-4}$ is formed when the cell size is kept closely homogeneous throughout the cross sectional area which defines the jet

(simplified mesh of Fig.4.9), while smaller values imply that the cell size at the jet core is larger than that at the shear layer region (transitional mesh of Fig.4.9).

Results of this study are presented in (Fig.A.5). It is observed that increasing the value of Δ_r accelerates jet transition. However, this acceleration is small enough so that the solutions at $X/D = 45$ are similar for all cases.

From the above discussion it can be seen that reducing the value of the turbulence intensity of the velocity fluctuations at the inlet delays jet transition, while in the same time it reduces the computational cost. It can also be seen that increasing the mesh spacing across the initial shear layer accelerates the jet transition. In the same time the mesh density, extensively, the computational cost, reduces. An additional advantage of using a homogeneous cross-sectional mesh distribution is the reduced meshing complexity which is very important considering the large number of meshes required for this work.

Therefore, it has been decided to use a lower turbulence intensity at the inlet, combined with a homogeneous mesh spacing throughout the jet cross-sectional area, the two having opposite effects on the jet transition, and both contributing to the reduction of the computational cost.

7.1.5 Under-Relaxation Factors

It is well known that increasing the values of the *under relaxation factors* accelerates the convergence of the computations. Attention is required because very high values of the *under relaxation factors* increase the danger of solution divergence. Improvement of the mesh quality by means of the smoothing functions available at the commercial meshing software *GRIDGEN V15* has allowed stable convergence to be achieved with the high values of 0.65, 0.7, and 1.0 for the *under relaxation factors* of pressure, momentum, and scalars respectively.

7.1.6 Temporal Convergence

The results of the simulations presented so far were found to require further statistical convergence time in order for the U_{rms} field to become smooth at *FLI*. In

this test it is attempted to examine the *LES* solution in three different periods of statistical convergence.

Based on the findings of section 7.1.4 the computational mesh has been simplified at the near-field, now having a homogeneous cross-sectional distribution. This has lead to a reduction of the mesh density throughout the computational domain, consequently, allowed to use a larger timestep $\Delta t = 4 \times 10^{-5}$. In this way we have managed to consider the full length computational domain $0 \leq X/D \leq 250$ at moderate computational cost. Other numerical parameters used in this study include: mesh density 3×10^6 cells, inlet turbulence intensity $I = 5\%$, convergence criteria 10^{-4} , and *SGS* modelling by *DSM*.

The results are plotted in (Fig.A.6). It is observed that simulated time $0.4s$ is adequate for an overall good statistical convergence up to $X/D = 90$. The *RMS* field has required longer simulation to converge. In particular, the statistical solutions of stations $X/D \geq 120$ vary considerably with extending the simulation at $0.9s$. Further extension of the simulation seems to be unnecessary as indicated by the results after $1.2s$. These findings are in agreement with the theoretical estimation of $STR = 0.79s$ which was made in section 5.4.2 using the parameters of the present setup.

7.1.7 Grid Convergence

A necessary step of *CFD* investigations is the execution of a “*Grid Convergence Study*”. Increasing the density of the mesh leads in reduction of the discretization error, which is an unavoidable drawback of discrete solutions. Particularly, in *LES* of small scale anisotropic turbulence, mesh density also affects the *SGS* modelling contribution to the predicted turbulent structure. The *SGS* models employed in this thesis assume isotropy of the scales smaller than the filter width; the size of the later being proportional to the volume of the computational cell (refer to chapter 3 for more details). Hence, mesh independence is harder to approach in *LES* than it is in *RANS* simulations.

The numerical setup used in section 7.1.6 is repeated at this test by homogeneously varying the density of the mesh. The three different mesh densities

examined are 1×10^6 , 2.5×10^6 and 5×10^6 . The simulated time was higher than $1s$ in all cases, in agreement with the findings of section 7.1.6; therefore, eliminating statistical errors.

The results are presented in (Fig.A.7). The highest deviations of the *LES* results from the experimental measurements lie at the near-field. This is expected considering the simplified (coarse in the near-field) meshing approach employed. Homogeneous densification of the mesh is observed to reduce the transitional errors (increased numerical dissipation) in both $\bar{U}$ and $\bar{\Phi}$ in the near-field $X/D \leq 30$. However, downstream the three solutions tend to equalize, also recovering the deviation from the experimental data, an observation agreeing with the discussion of section 7.1.2.

7.1.8 Pressure Coupling and SGS Modelling

FLUENT supports a variety of pressure coupling algorithms. *SIMPLE* is the algorithm selected for the simulations so far. Here, a modification of this algorithm, which affects the evaluation of the pressure correction term (Versteeg & Malalasekera, 1995) *SIMPLEC*, is investigated. This modification is expected to accelerate the solution as *SIMPLEC* does not require relaxation of the pressure correction equation; however, the simplification applied at the face flux correction (Fluent, 2012) requires testing.

Two different simulations are run using the 3×10^6 cells mesh, one using *SIMPLE* and one using *SIMPLEC*, with the rest of the computational parameters same as in the previous test. The results presented in (Fig.A.8) indicate that the two algorithms perform the same in the current problem. The fact that no gains are observed in terms of computational cost has lead to the decision to continue the study using *SIMPLE*.

The simulation is repeated by varying the *SGS* model. The models tested are the standard Smagorinsky model (*SSM*) with the Smagorinsky coefficient selected as $C_s = 0.17$, the dynamic Smagorinsky model (*DSM*) where C_s is dynamically evaluated, the wall-adapting local eddy-viscosity model (*WALE*) which is based on the square of the velocity gradient tensor and has a proper near

wall behaviour¹, and the dynamic kinetic energy model (*DKEM*) which accounts for the transport of the subgrid-scale turbulence kinetic energy. The results of (Fig.A.9) reveal that *DSM* and *WALE* produce similar solutions in the full length of the computational domain, also being in best agreement with the experimental data. *SSM* is weak in anisotropic turbulence. This is confirmed by the results in sections $X/D = 5$ and $X/D = 15$. In particular, the *SSM* profiles of $\bar{U}$ and $\bar{\Phi}$ at $X/D = 5$ disclose that the jet transition is on an earlier stage than for the other models. Within the range $5 \leq X/D \leq 15$ *SSM* causes the jet to suddenly grow much faster than the other models. Finally, although *DKEM* presents similar behaviour to *DSM* and *WALE* for $\bar{U}$ and U_{rms} , it fails to do so for $\bar{\Phi}$. In particular, all plotted stations reveal that Φ decays faster when turbulent mixing is modelled by *DKEM*; however, the information given by FLUENT about the model application, as well as its links with the *SGS* flux, is not adequate in order to justify this behaviour.

7.1.9 Mesh Isotropy in the Far-Field (*Streamwise Resolution*)

Conservation of isotropy of the mesh cells, when varying the mesh density, requires proportional variation of the individual resolutions in all three directions. In section 7.1.4 it was shown that local increase of the mesh resolution leads to high aspect ratio of the cells in the region of refinement (initial shear layer region). However, this is only locally occurring as the cell isotropy is recovered downstream. On the other hand, reducing the mesh resolution in the streamwise direction while keeping it the same in the lateral ones is a common and relatively simple way to reduce the mesh size, extensively, the computational cost. However, such a method increases the aspect ratio of the cells, which become elongated in the streamwise direction.

In this test, isotropy of the computational cells is being investigated with respect to its effect in turbulent mixing. In order to show this we have designed two meshes, similar to the ones used at the previous studies. The streamwise

¹Although there is no wall present in the flow, *WALE* is tested here as it is planned to be utilized later for the *3DWJ* simulations

resolution of the first mesh is considerably reduced so that the two dimensional aspect ratio is high $AR_{2D} \sim 4$, with final mesh density 1×10^6 cells. For the second mesh the streamwise resolution is thoroughly controlled so that $AR_{2D} \sim 1$, with final mesh density 3×10^6 cells. (Fig.7.1) illustrates volume slices of the two meshes. Other computational parameters are kept the same as in the previous test. The results are analytically presented in (Fig.A.10). It is observed that although the two meshes produce similar results for $\bar{U}$ and U_{rms} in the full range of the computational domain, the situation is different for $\bar{\Phi}$. In particular, at the end of transition $X/D = 15$ a considerably reduced maximum concentration is observed for the anisotropic mesh results. This deviation is not only conserved downstream, but it increases continuously.

In order to better investigate the effect of mesh anisotropy, results of $\bar{U}$, U_{rms} , and $\bar{\Phi}$, corresponding to a single farfield station, $X/D = 150$, are presented in (Fig.7.2) for two different studies. In particular (Fig.7.2,bottom) presents the results obtained at the grid convergence study, where approximately isotropic cells are ensured throughout the computational volume of all three meshes. The mesh density has merely caused any effect on $\bar{\Phi}$, with only a slight under-prediction for the coarsest mesh, attributed to the increased numerical diffusion. The top plots of the same figure contain the results of this study at the same station of the farfield. The U_{rms} plot, on the top-centre, shows that mesh anisotropy affects the turbulent activity of the streamwise velocity fluctuations. This change of turbulence activity has not significantly affected the jet growth, as revealed by the $\bar{U}$ plot on the top-left; however, there is a major effect on the mean passive scalar concentration $\bar{\Phi}$ as it can be observed at the plot on the top-right. It is therefore, essential to avoid high aspect ratio cells at free mixing layer regions in order to ensure realistic simulation of turbulent mixing.

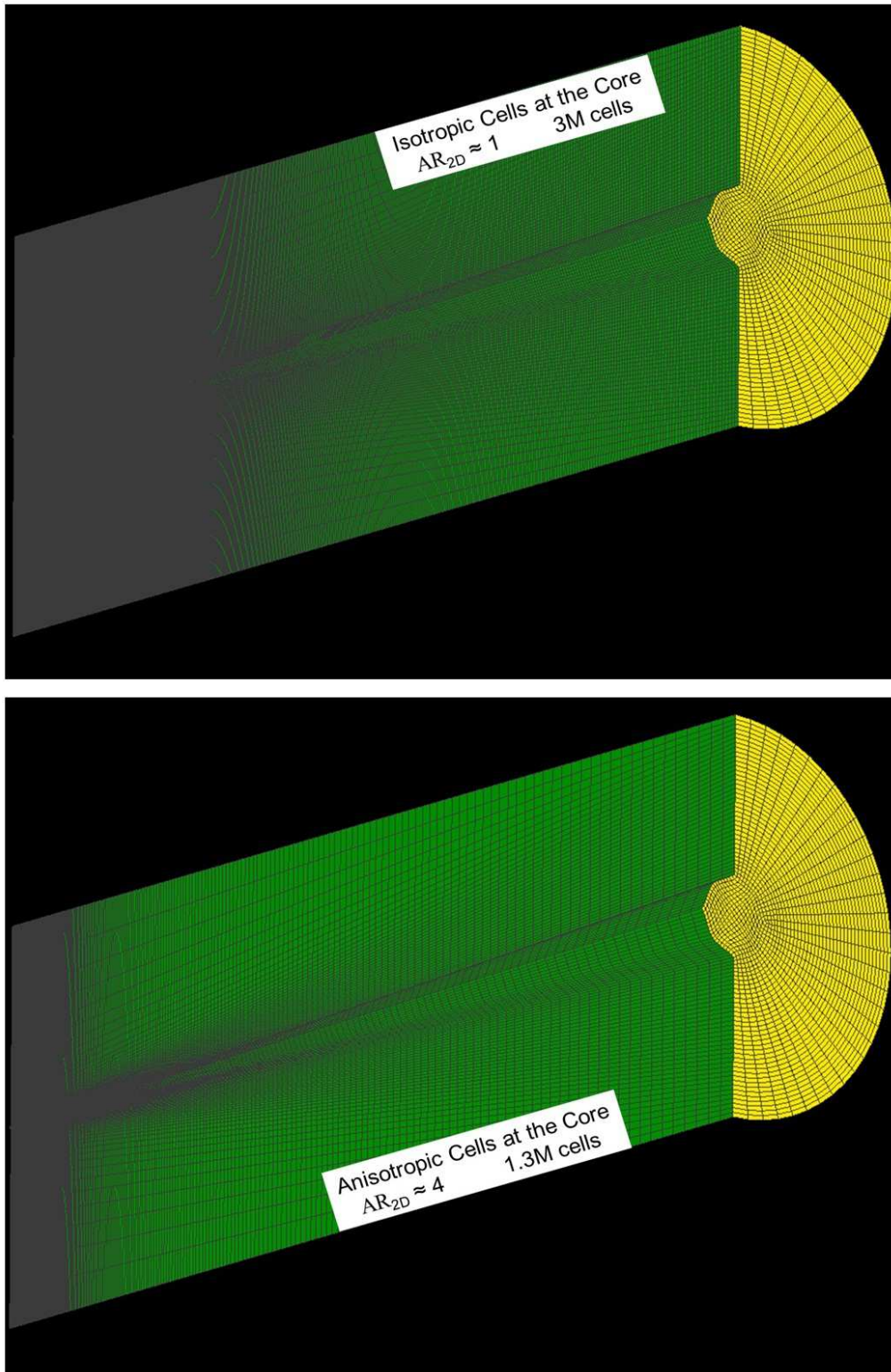


Figure 7.1: Volume cut of the two meshes participated in the *Far-field Mesh Isotropy* study. The isotropic mesh is presented on the left and the anisotropic mesh on the right.

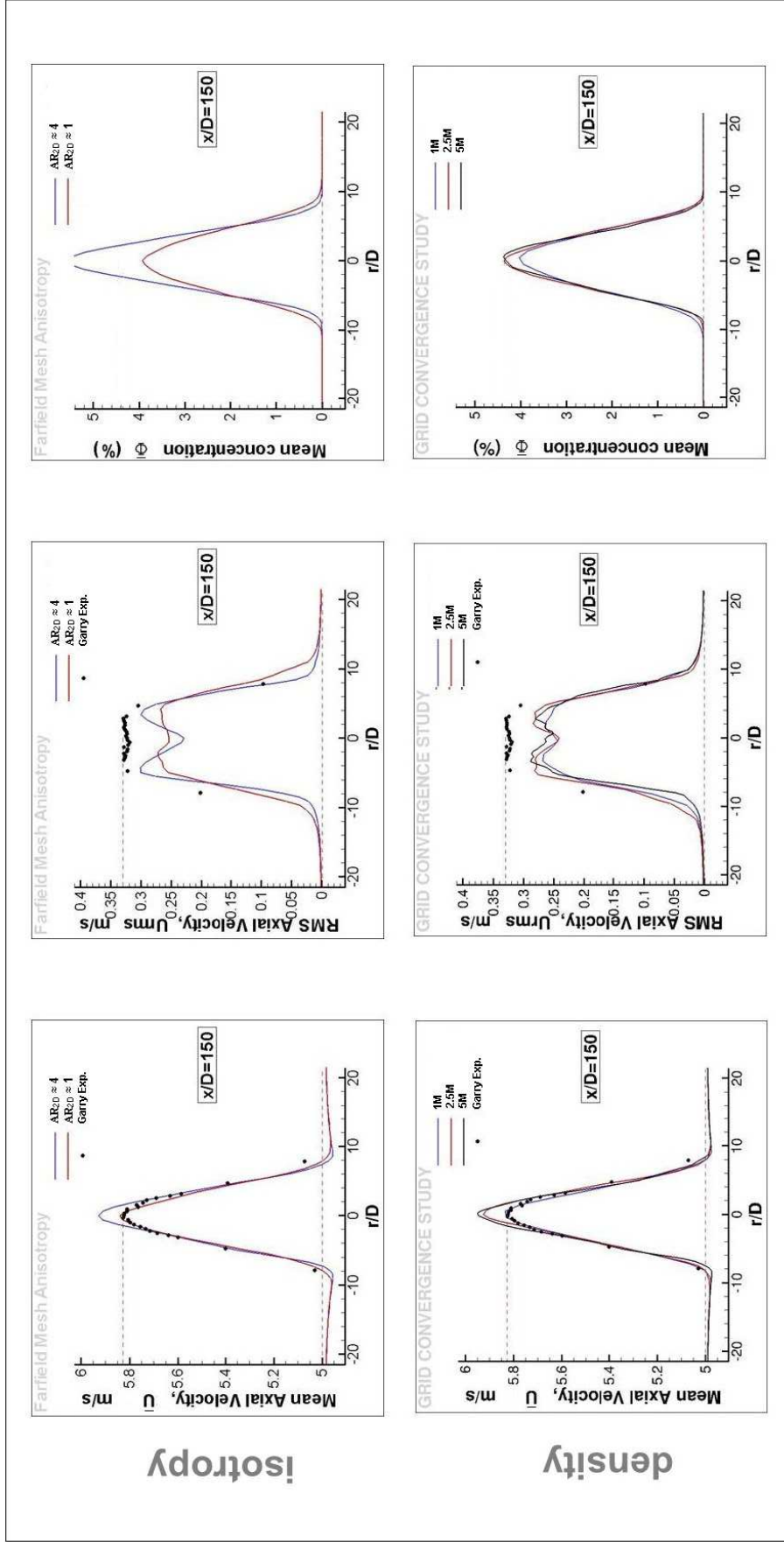


Figure 7.2: Results of the farfield mesh isotropy test (top), and grid sensitivity analysis (bottom). The plots illustrate profiles of $\bar{U}$ (left), U_{rms} (middle) and $\bar{\Phi}$ (right). Present data correspond to $X/D = 150$. Experimental data from (Garry & Holt, 2008) are compared with $\bar{U}$ and U_{rms} plots. The mesh densities of the top plots are 1.3M and 3M for the anisotropic and isotropic mesh respectively. The two dimensional aspect ratio of the grid sensitivity cases (bottom) is $AR_{2D} \sim 1$.

7.1.10 Comparison with previous LES

Previous *LES* of the experiment of (Garry & Holt, 2008) has been performed by (Dinesh *et al.*, 2009). Their simulations were performed on a well-established flow simulation code, *PUFFIN*, originally developed by Kirkpatrick and later extended by Ranga Dinesh. This code directly computes the temporal evolution of the largest energy-containing scales of motion in the flow by numerically solving equations for the spatially filtered mass, momentum, and passive scalar, and uses model approximations for the effect of unresolved smaller scale mixing. A 2.4×10^6 cells mesh was used, with a variable timestepping ensuring $CFL < 0.7$. The rest of the numerical setup was similar to the one selected here.

The results of the simulation of Dinesh are compared with present *LES* for the medium mesh density of the “*Grid Convergence Study*”. Plots are presented in (Fig.A.11), following a similar format with the previous tests. The two solutions agree well in terms of mean axial velocity $\bar{U}$ in the full length of the computational domain. However, the situation is different for the *RMS* axial velocity U_{rms} as well as for the mean passive scalar concentration $\bar{\Phi}$.

In particular, *PUFFIN*s predictions of U_{rms} are considerably overestimated in the full range of the computational domain, with this effect increasing with downstream distance, where at $X/D = 150$ the curve pick of *PUFFIN* becomes double the experimental one. On the contrary, present *LES* under-predicts U_{rms} . This is, however, justified by the modelled, *SGS*, part of the turbulent activity, as well as the numerical diffusion accompanying discrete solutions, also observed on the $\bar{U}$ profiles. The lateral extent of the U_{rms} profile of this work agrees well with the experimental measurements in the farfield². However it is interesting to note that turbulent activity of *PUFFIN* occurs at spanwise extends where pure free-stream (laminar) flow exists, which is physically unjustified.

Finally, a continuous increment of the relative overestimation of the passive scalar concentration $\bar{\Phi}$ is observed on the results of *PUFFIN*, while, in the same time, the lateral extent of the profiles is narrower than that of $\bar{U}$. This

²Near field inaccuracies have been discussed in the previous paragraphs.

phenomenon was previously observed in the results of study “*Far-field Mesh-Anisotropy*” discussed in section 7.1.9. Observing the three-dimensional flow visualisations within the report of (Dinesh *et al.*, 2009), it can be seen that the streamwise expansion of the mesh is high, forming high aspect ratio cells in the farfield. This is possibly contributing to the behaviour of $\bar{\Phi}$ and U_{rms} of the results of *PUFFIN* discussed above.

7.1.11 Comparison with Theoretical Trends

An attempt to modify the analytical trends for the mean flow characteristics of axisymmetric jets in order to account for the co-flow, was made in chapter 5. In this way, equation (5.1) was suggested for the theoretical estimation of the maximum (centreline) mean velocity U_m , and equation (5.4) for the mean velocity half-width $z_{1/2}$, with varying the streamwise distance from the nozzle exit. These equations have been used for the estimation of the different time intervals of the simulation (refer to equation 5.5).

This section aims to validate these theoretical estimations by comparison with the already validated *LES* predictions. The *LES* results of the medium mesh density of the “*Grid Convergence Study*” are used here. The results are plotted in (Fig.7.3). The left plot illustrates the streamwise variation of U_m on double logarithmic axes. The theoretical logarithmic decay agrees well with the *LES* solution as an average throughout the computational domain. In particular, the *LES* predicted decay is slightly higher in the near-field, and becomes lower in the farfield. The same decay rate is observed at $X/D \approx 25$. (Fig.7.3, right) illustrates the variation of $z_{1/2}$ with downstream distance. Similar to U_m , the jet expansion is in a good agreement between theory and *LES*, with a slight over-prediction of the *LES*-predicted jet growth in the near-field and a slight under-prediction in the farfield.

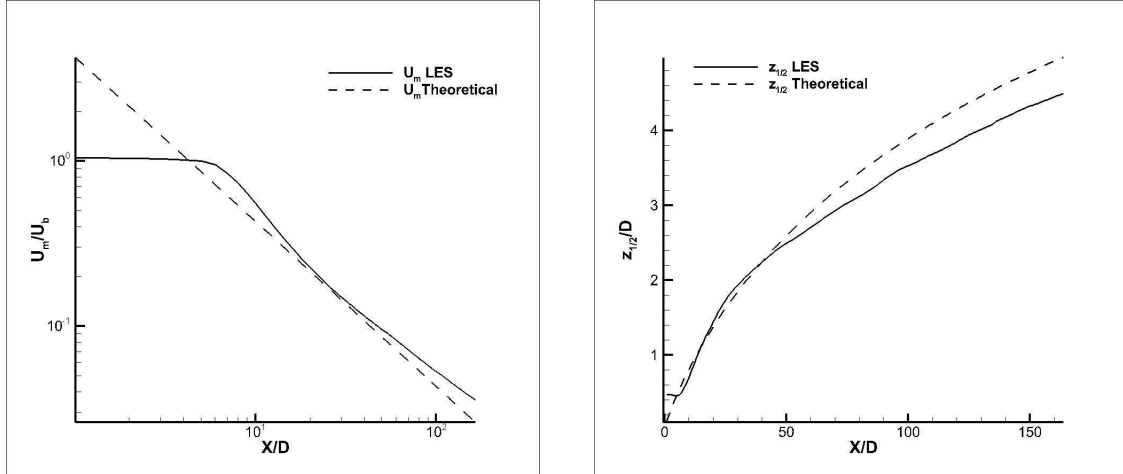


Figure 7.3: Comparison of theoretical mean flow characteristics with the corresponding *LES* predictions. Streamwise centreline velocity distribution (left), and streamwise variation of velocity half-width (right).

7.2 HIGH SPEED COAXIAL JET

The experiment of (Garry & Holt, 2008), which was used for the *LES* validations so far in this chapter, includes a relatively simple setup, with the Reynolds number being low ($Re = 10,000$) and the shape of the nozzle simple (cylindrical). However, the efflux of a turbofan engine is much more complex. Not only the Reynolds numbers formed are larger, but the shape of the nozzle from which the jet issues is highly complicated. In this way, the setup of (Eastwood *et al.*, 2008) presented in section 6.1.2 is being replicated in order to extend the free-shear validation tests in more realistic conditions ($Re = 300,000$).

7.2.1 An Advanced Technique to Initialize the Solution

For the low Reynolds number simulations performed in the previous studies, a simplistic method of initialising the solution was selected. Constant velocity and scalar values were applied throughout the computational domain, while the jet was left to develop naturally all the way to the outflow plane. Due to the high Reynolds number of the present setup, however, unrealistic conditions are forming at the beginning of the simulation. This is followed by serious stability issues and subsequent divergence of the solution.

In order to overcome this inconvenient situation, an alternative initialization method is employed. A steady-*RANS* simulation is initially performed, the results of which can be used as initial conditions for the large eddy simulation. However, direct initialization of the *LES* with a *RANS* solution is not appropriate due to the spatially smooth and symmetric nature of the latter. Therefore, the *RANS* fields are disturbed by the built-in function of FLUENT “*initialize-instantaneous-velocities*”, which eventually forms realistic fields for *LES* computations to initiate.

(Fig.7.4) illustrates streamwise velocity contours at the end of each one of the three stages of the simulation. The top image shows the *RANS* results, characterized by smoothness and axis symmetry. The image in the middle illustrates the same results after applying the “*initialize-instantaneous-velocities*” algorithm. Finally the bottom image shows the statistically steady instantaneous *LES* results.

Considering the absence of co-flow in the present setup ($U_f = 0$)³, which has a great impact to the flow establishment time $T_0 a_0$ (refer to section 5.4), the above described technique of solution initialisation has allowed to considerably reduce this time interval of the simulation.

³in order to improve consistency of the *RANS* simulation an elementary streamwise velocity component $U_f = 0.5m/s$ has been included to the numerical setup.

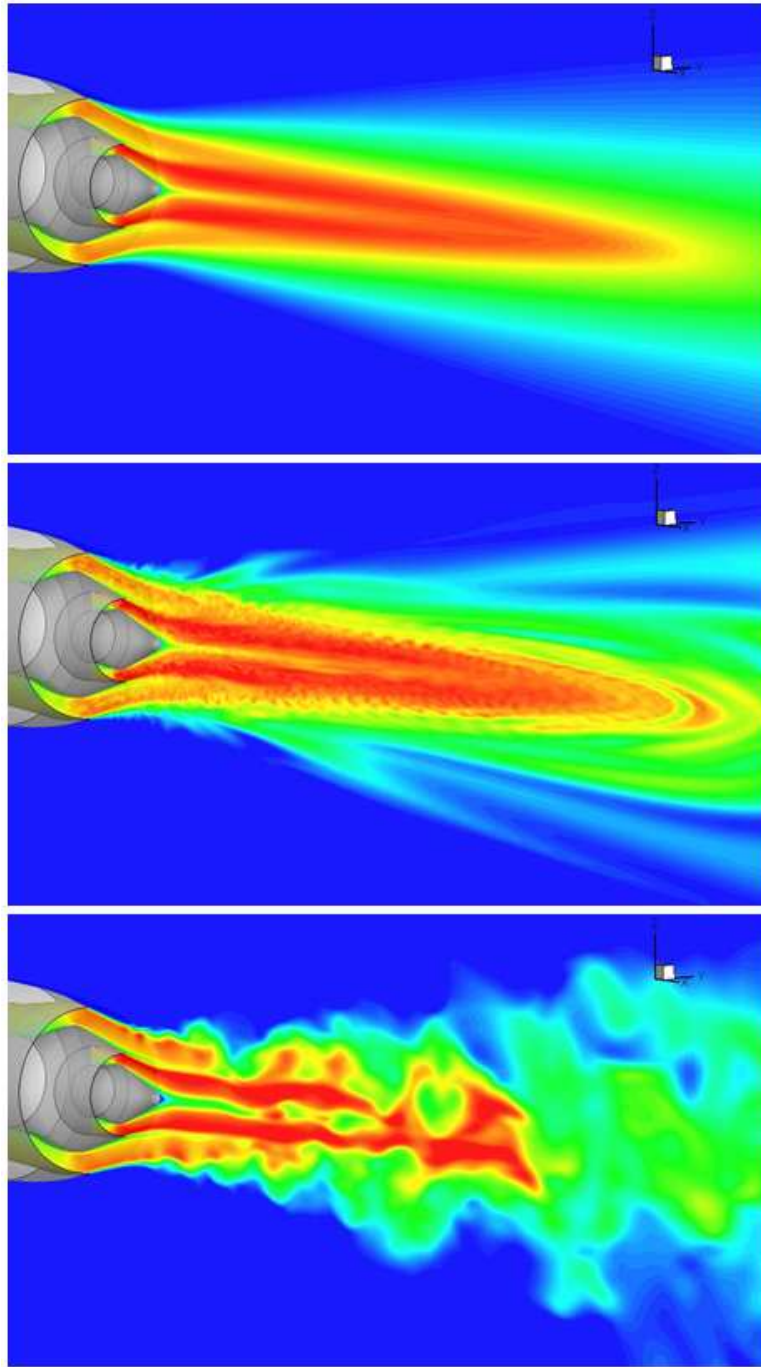


Figure 7.4: Three stages of the simulation. Converged *RANS* results (top), *RANS* results after initializing instantaneous velocity fields (middle), and *LES* instantaneous results (bottom). The contour shows the streamwise velocity component.

7.2.2 Modelling the Nozzle

In order to reduce the complexity of the problem, an attempt to simplify the inlet boundary is made. In this way, the complex shape of the nozzle is substituted by a top-hat-like profile for velocity, and a top hat profile for the passive scalar. During this modification it was ensured that the inflow momentum and velocity ratios are the same as in the “*Nozzle Included*” case. Two different realisations of the “*Modelled inlet*” setup are made; one without inlet perturbations, and one with the application of the “*spectral synthesizer*” algorithm, which was introduced at the previous section, setting the intensity of the fluctuations at $I = 5\%$. The computational domains extend $100D$ and $50D$ in the streamwise and lateral directions respectively. Meshing is implemented based on the O-Type approach. Following the techniques presented in section 4.2.1 to optimize the mesh quality, the two resulted meshes consist of approximately 2.5×10^6 cells. Other numerical parameters include: convergence criteria 10^4 , *SGS* modelling by *DSM*, and pressure-velocity coupling by the *SIMPLE* algorithm. A summary of this test’s numerical parameters and history of the simulations is presented on table 7.4.

Table 7.4: Variables and History of the Simulations of Study “*Modelling the Nozzle*”.

Case	I	Δt	Simulated Time
Nozzle Included	0% <i>inlets inside the nozzle</i>	3×10^{-6}	0.38s
Modelled Inlet 1	0%	3×10^{-6}	0.10s
Modelled Inlet 2	5%	3×10^{-6}	0.32s

Instantaneous Flow Considerations

The nozzle modelling procedure is visualized in (Fig.7.5). It is important to clarify that the modelled inlet is a flat plate, attempting to imitate the multiple jets exiting the three dimensional nozzle. This means that station $X = 0$ coincides with the inlet boundary for the “*Modelled inlet*” setups, while it lies on the interior of the computational domain for the “*Nozzle Included*” case.

(Fig.7.5, right) contains three groups of volumetric results at the near-field $-2 \leq X/D \leq 10$, including contours of pressure, stream-wise-velocity, and passive scalar concentration at the symmetry plane $Z = 0$, corresponding to the three cases tested respectively. The contours of pressure of the “*Nozzle Included*” case (group A) highlight the transitional mechanisms. The dipoles of pressure reveal the initial train of vortices of the external mixing layer and subsequent pairing. Although the velocity ratio between the by-pass and the core flow is very close to 1, the formation of multiple boundary layers inside the nozzle makes the effects of the multiple streams strong in the very-near-field. In particular, the velocity contours, presented at the figure underneath, reveals that the multi-stream jet develops into a uniform turbulent jet after transition is complete at $X/D \approx 5$. However, the appearance of a zero-concentration bubble at $X/D = 2$, is a clear indicator of turbulent activity at the inner mixing layer; something that is not clear from the contours of pressure.

Similarly, the volumetric results of case “*Modelled Nozzle 1*” are presented on group B of the same figure. The train of vortices of the external mixing layer is similar in terms of dimensions, except that in this case the jet growth begins directly, unlike the “*Nozzle Included*” case, where there is an initial region within which the round mixing layers converge inwards to occupy the gap left by the conical ending of the nozzle. The greatest drawback of the selected modelling method is seen at the contours of streamwise velocity. Here, the top-hat-like profile superimposed at the inlet causes the core flow to remain undisturbed up to the end of the potential core. There are only minor disturbances observed at the core flow. This is at regions where the helical pairing occurs, causing the flow to undergo massive expansions and contractions up to the end of the potential core. Finally, the contours of passive scalar are in close agreement with the “*Nozzle Included*” case. Encouraging is the appearance of a small engulfment at the inner mixing layer at $X/D \approx 2$, indicating that it begins to develop disturbances.

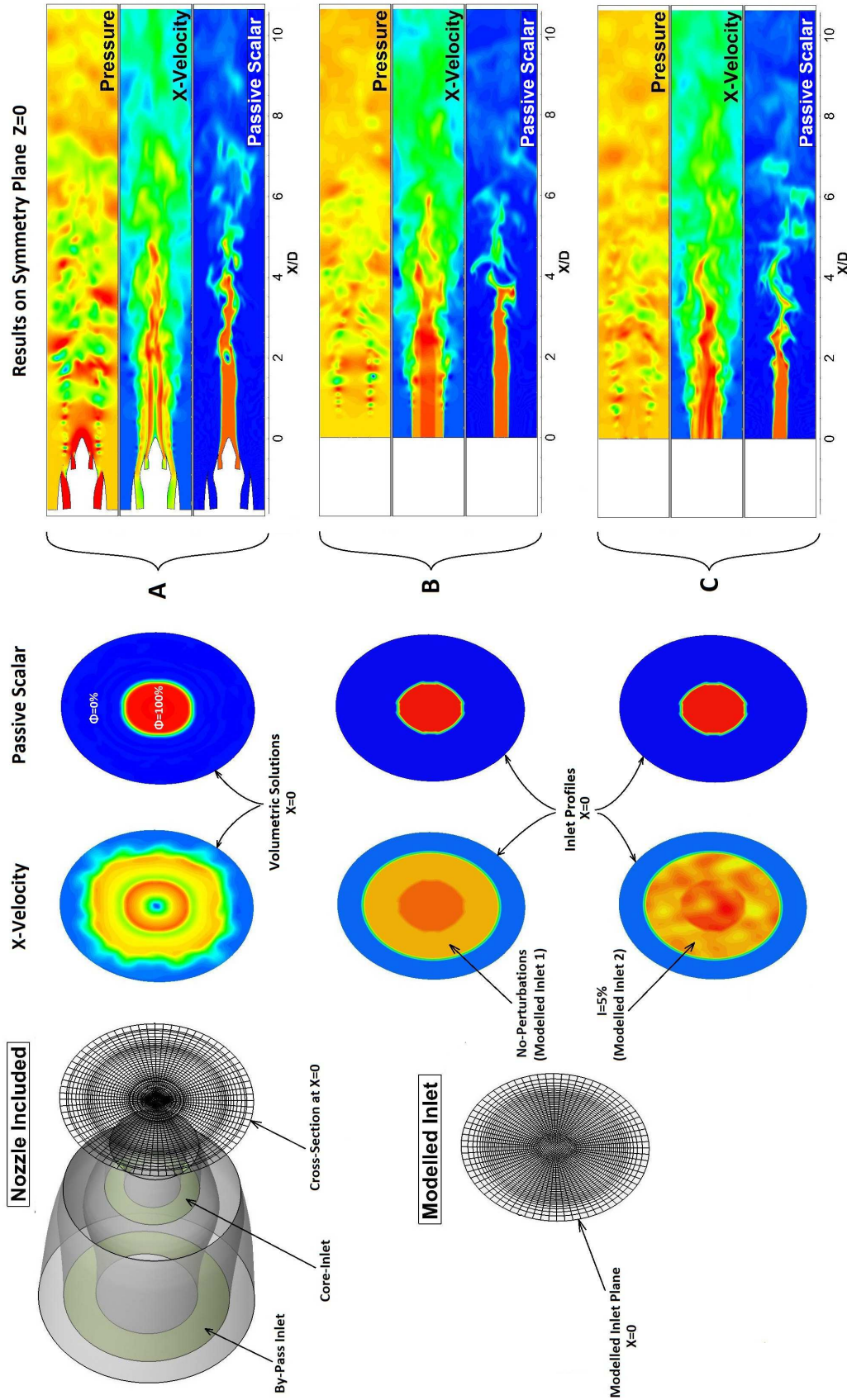


Figure 7.5: Visualisation and results of study “Modelling the Nozzle”. All round plates included represent cross-sections at $X = 0$ (end of the cone of the nozzle). The cross-sectional format of the two meshes at $X = 0$ is highlighted. Contours of streamwise velocity and passive scalar at $X = 0$ is also presented for all three cases tasted. The two-dimensional planes on the right of the figure, include contours of pressure, streamwise velocity, and passive scalar, respectively, on the symmetry plane $Z = 0$.

Finally, the results of case “*Modelled Nozzle 2*”, which is presented on group C, show the effects of the disturbance of the inlet top-hat-like velocity profile. The effects of such a flow modification are already present at the pressure field. In particular, the growth of the Kelvin Helmholtz instability is magnified, having caused the first pairing to occur more than $0.5D$ upstream comparing to the undisturbed case. Furthermore, velocity contours show that the core velocity is significantly disturbed by the inlet perturbations which are causing the whole jet to oscillate randomly, already from the “*laminar*” part of the jet. This phenomenon was not as extreme at the single round jet of the previous section, and it is attributed to the multiple streams of jet, which make the flow more susceptible in losing stability under the influence of external, asymmetric, disturbances. Finally, the passive scalar field reveals that the inlet perturbations contribute to faster decay of the scalar concentration.

Analysis of Statistical Quantities

So far, the instantaneous picture of the flow at the very near-field $-2 \leq X/D \leq 10$ has been discussed, and general assessment of the results of the three selected methods has been made. The analysis is now extended to statistical quantities, including mean and *RMS* velocities, as well as mean passive scalar concentrations. Experimental data of $\bar{U}$ and U_{rms} are available at the streamwise stations $X/D = 1, 2, 3$, and 5 from (Eastwood *et al.*, 2008). Their work also included high resolution *NLES*. In particular, their simulation includes a computational mesh of 12×10^6 cells, all concentrated at the very near-field, with their analysis being limited at $X/D = 5$, and a timestep size as small as $\Delta t = 3.7 \times 10^{-9}$.

Comparison plots of the results of this work with the experimental data, as well as with the *NLES* results of (Eastwood *et al.*, 2008), are presented in (Fig.A.12). In particular, plot (A.12.a), which refers to the immediate vicinity at the nozzle exit $X/D = 1$, confirms the correct definition of the nozzle geometry, as well as the appropriate specification of the two velocities corresponding to the two streams exiting the nozzle. The top-hat-like shape of the velocity profiles of the cases “*Modelled Nozzle 1*” and “*Modelled Nozzle 2*” are still observable at this distance

from the boundary; the span-wise extend of the former being in agreement with the “*Nozzle Included*” case, while the inlet perturbations appear to have affected the growth of the latter, something that was expected following the discussion of the instantaneous contours of $\bar{U}$ of (Fig.7.5) made above.

Plot (A.12.b) presents the U_{rms} distribution along R at the same station. The effects of the relatively low mesh density and large timestep size comparing to the *NLES* results of (Eastwood *et al.*, 2008), are visible here. This is, the turbulent activity of the inner mixing layer is highly diffused comparing to Eastwood’s *NLES*. An additional factor possibly affecting the disagreement of U_{rms} at this flow station is a minor modification of the geometry of the nozzle (smoothing the angle of the cone edge), which was necessary for the local quality of the O-Type mesh core. No turbulence activity is visible at U_{rms} for the modelled setups. A homogeneous amplification of the “*Modelled Nozzle 2*” curve is observed, revealing the additional turbulence intensity superimposed at the velocity inlet boundary.

Plot (A.12.c) presents the $\bar{\Phi}$ distribution along R at the same station. Although the study of (Eastwood *et al.*, 2008) did not involve scalar measurements, it is useful here in order to compare the effects of the nozzle modelling on the downstream mixing. In particular, for the “*Nozzle Included*” setup the pollution coming out from the inner stream (core jet) has not undergone any turbulence mixing yet at $X/D = 1$, because the jet is still laminar. Similar is the situation for case “*Modelled Nozzle 1*”. However, the additional perturbations of U at the inlet of case “*Modelled Nozzle 2*” has forced mixing of the inner stream with the outer, which is present to the profile of $\bar{\Phi}$.

The *LES* predictions of stations $X/D = 2$ and $X/D = 3$ are in close agreement with the experimental data and *NLES* of (Eastwood *et al.*, 2008). There is also a good agreement in terms of scalar concentrations between the “*Nozzle Included*” and the “*Modelled Nozzle 1*” cases, the latter being only slightly under-predicted. However, the results of case “*Modelled Nozzle 2*” are significantly under-predicted, a situation following the early initiation of the core stream mixing with the bypass stream.

Station $X/D = 5$ lies at the “*braid region*”, where small scale anisotropy dom-

inates. Mesh density, and extensively the contribution of *SGS* modelling play a dominant role at the formation of the turbulence structures therein. As expected, the moderate mesh density and relatively large timestep of the present *LES* have caused increased modelling errors at this station as revealed by plots (A.12.j) and (A.12.k). Although there are no experimental data available further downstream, the experience of the previous study tells us that these transitional errors are gradually recovered downstream. It is also observed that the large deviations of $\overline{\Phi}$ between the simulations of this study are completely recovered in the region $15 \leq X/D \leq 30$, as revealed by figures (A.12.o) and (A.12.r). This is an additional factor validating previous statements regarding the recovery of transitional errors.

Conclusions

Concluding, this study has extended the validation of the selected numerical approach to simulate free-shear jet mixing to a higher Reynolds number, and a more complex geometry. An additional feature of FLUENT for *LES*-initialization has helped overcoming simulation stability issues and achieving reduction of the computational cost. Finally, modelling (simplifying) the complex inlet was found to affect the near-field computations, but the solution is gradually becoming independent after transition is complete. Therefore, it can be safely decided to use simple circular jets for the representation of the turbofan engine efflux for the rest of this work, where the *FLI* is at the very farfield $X/D \gg 100$.

Chapter 8

Validation Tests

Part 2: *Wall Bounded Jets*

The presence of a solid surface in proximity to the path of development of an axisymmetric jet causes major changes to the jet dynamics as discussed in section 1.4.3. Furthermore, *LES* of wall jets is of higher complexity than their free-shear counterpart due to the near-wall anisotropic turbulence, which on the one hand increases the mesh density requirement, and on the other hand it increases the contribution of physical modelling to the mixing process (refer to section 3.5). Due to the relatively low values of Reynolds number at the investigated setups it has been decided to implement pure *LES*; therefore, not considering the application of near wall *RANS* (*DES* model).

The problem described in section 6.1.3 is used here to perform sensitivity studies based on a series of numerical and physical parameters. The aim is to ensure that the numerical solution is consistent and independent of the investigated numerical parameters, as well as to improve understanding of the impact of certain physical characteristics of the *3DOJ* in co-flow. Although the co-flow magnitude is large $\frac{U_f}{U_b} \approx 22\%$, not representative of early takeoff conditions, it is convenient for the implementation of this study as it reduces the computational cost significantly according to the analysis presented in section 5.4. (Fig.8.1) presents the history of the simulations performed and the parameters examined. It also shows the 13 studies that emerge from the simulations.

PARAMETER/ CASE NAME	SGS Model	Y Plus	Convergence Criterion	Mesh Type	Ground BL at Inlet	Time-step Size	Mesh Density																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																														
								Three-Dimensionality Effect	Wall Effect	Co-Flow Effect	SGS Model (O-Type)	Inlet Ground BL (O-Type)	O-Type Mesh Density	H-Type Vs O-Type	H-Type Mesh Density	Inlet Ground BL (H-Type)	Temporal Resolution	Yplus	Convergence Criterion	SGS Modelling (H-Type)																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																	
Case 1	A	A	A	A	A	B	B	●																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																																													</

Table 8.1 : Summary of the simulations of the wall bounded validation tests and studies carried out. The grid on the left marks the simulations compared for each study.

Further to the sensitivity studies implemented in this chapter, optimization of the mesh has also been attempted (discussion in section 4.2.2). Additionally, the post-processing method has been improved to increase comprehensiveness of the analysis plots, allowing for universal comparisons to be made between the results of different simulations.

8.1 POST PROCESSING METHODS

8.1.1 Conventional Post-Processing

Post-processing of the *LES* results is implemented using the commercial software TecPlot 360. The conventional approach follows a similar procedure to the previous studies. In this way, flow quantities of the *LES* solution are plotted at nine different streamwise stations being: $X/D = 5, 15, 30, 45, 60, 75, 90, 120$ and 150 . A substantial difference, however, between the axisymmetric jet and the *3DOJ* is that for the latter the direction of the data extraction is essential. Therefore, for each station we are extracting two series of data; one along the vertical direction $y = 0$, and one along the horizontal one $z = 0$ (Fig.8.1). The extracted variables here are the mean x-velocity, the *RMS* x-velocity, the mean passive scalar and the *RMS* passive scalar. In addition to this, profiles of mean wall-shear are also extracted from the wall surface at the stations corresponding to the wall jet region as shown in the same figure. An example of the application of this method is presented in figures (B.1), (B.2), and (B.3). Here, the results of the cases 5, 6, 9 and 10 are presented as part of the H-Type mesh density study. It can be observed that there are two different variables representing the passive scalar concentration in figures (B.1) and (B.2). Following a similar method as in the previous studies, Φ_1 is defined by a homogeneously distributed passive scalar tracer exiting the circular nozzle with a constant value of $\Phi_1 = 1$. On the other hand, Φ_2 is distributed at the top-half of the jet exit cross section, but having the value of $\Phi_2 = 2$. The axisymmetric nature of the jet velocity inlet ensures that the scalar inflow rate is the same in both cases. (Fig.8.2) illustrates the two methods

of specification of Φ at the inlet. Additionally, in order to avoid loss of information at the inlet (especially for coarse meshes) mesh adaptation has been applied at the first two rows of cells at the inlet boundary. The effect of this process is also highlighted in (Fig.8.2).

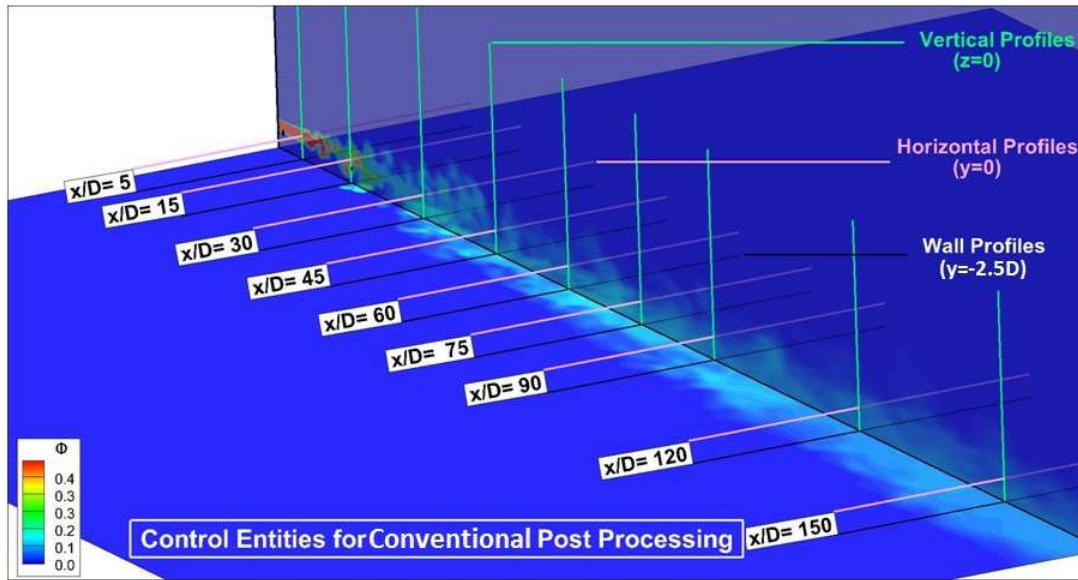


Figure 8.1: Control entities for the conventional post-processing of the *3DWJ*.

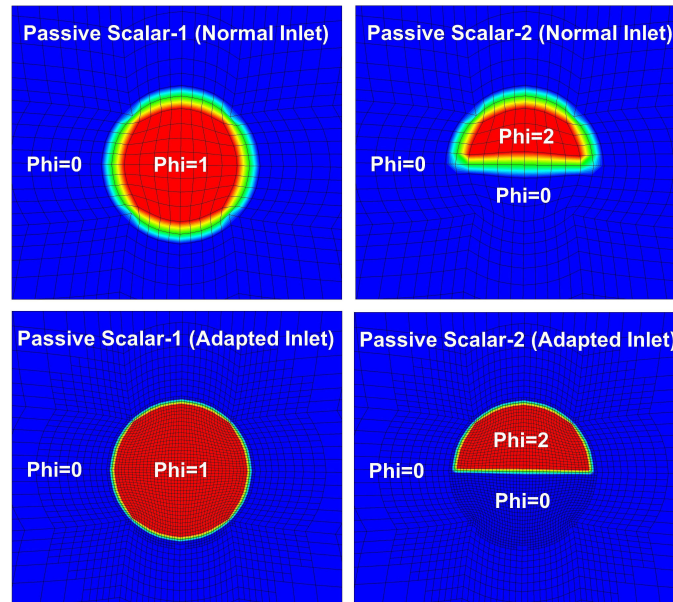


Figure 8.2: Two different methods of specification of Φ at the nozzle exit (Domain Inlet). Boundary adaptation is also illustrated.

8.1.2 Advanced Post-Processing

It can be seen that with the conventional post-processing approach the resulted group of plots for every study is large. Furthermore, due to the vertical asymmetry of *3DWJs*, the vertical position where the maximum velocity occurs varies with X . Therefore, the horizontal profiles, which with the conventional approach are taken at the offset of the nozzle $Z = 0$, do not always intersect the position of U_m as illustrated in (Fig.8.3). Furthermore, the analysis of the conventional approach is limited at nine streamwise locations and no information is analysed in between those stations. On the other hand, the mean flow characteristics of a *3DOJ* are well described by the value of velocity and length scales for a certain streamwise location. In particular, a *3DOJ* involves multiple transitions, which in order to be observed require a large number of streamwise stations to be analysed. Achieving such a post-processing detail in the streamwise direction is also expected to be essential for the results of the final simulations, where obstacles (baffles) lie at various farfield locations.

For the reasons mentioned above, it has been decided to apply a more sophisticated post-processing method utilizing the TecPlot scripting language (refer to Appendix G.2 for the code). In this way, the three-dimensional computational domain is subdivided into a user-defined number of two-dimensional, constant X , sections; i.e. in steps of $1D$ (Fig.8.3). Then, a vertical row of data is extracted at $Y = 0$ (symmetry line) on each section and is analysed separately.

In a first place the local maximum velocity U_m , the maximum pollution concentration Φ_m , and their vertical positions above the ground are calculated. Next, the vertical position where the half-mixing velocity $U_{0.5} = \frac{U_m - U_f}{2}$ occurs is calculated. This height is also referred to as the “*vertical velocity half-width*” $z_{1/2}$. Similarly, the vertical position where the half-mixing concentration $\Phi_{0.5} = \Phi_m/2$ occurs is calculated. This height is referred to as the “*vertical concentration half-width*” $z_{1/2\Phi}$. A horizontal row of data is then extracted at $z_{1/2}$ and the same process is repeated horizontally calculating the values of the horizontal half-widths $y_{1/2}$ and $y_{1/2\Phi}$ of the velocity and the concentration respectively.

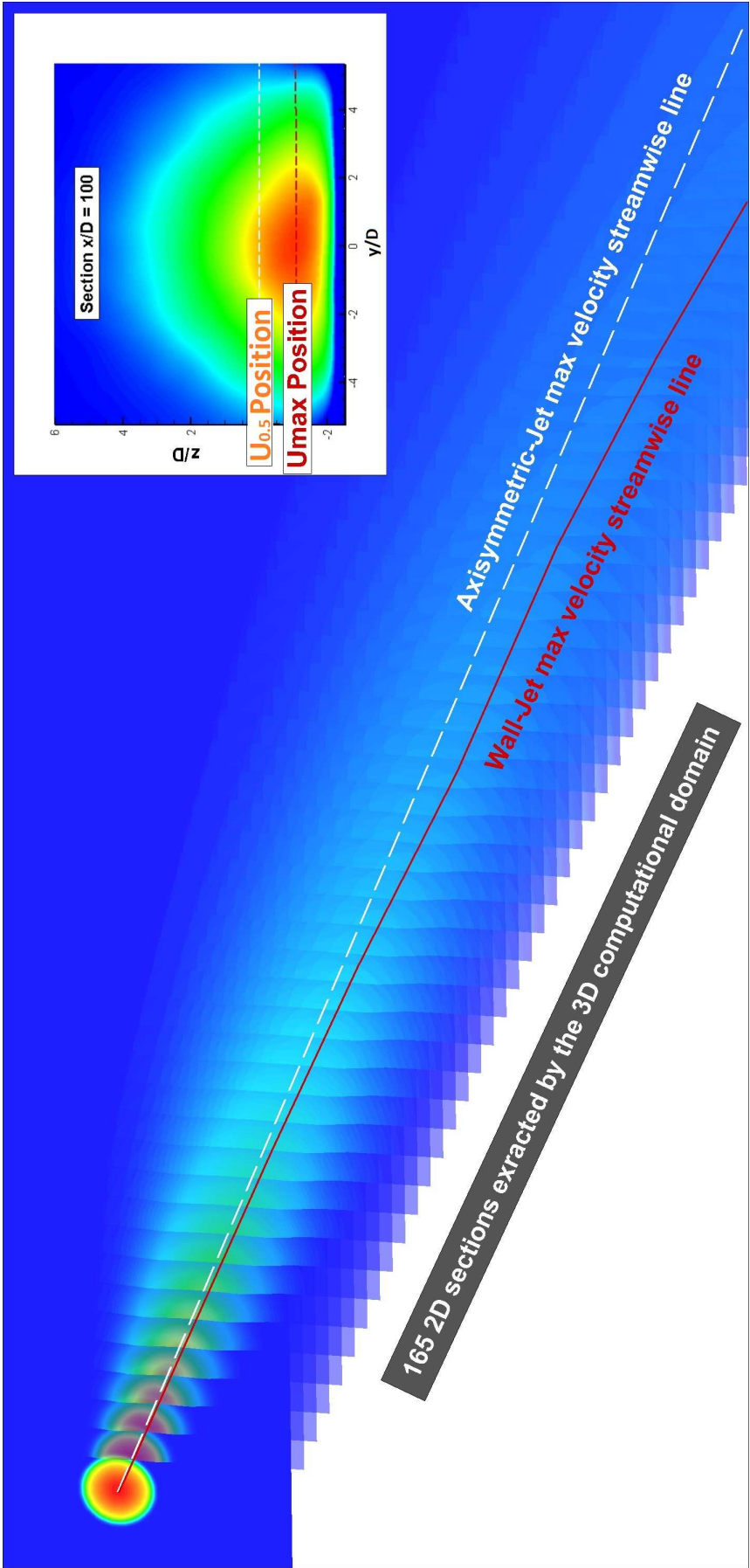


Figure 8.3: Visual representation of the advanced post-processing method.

The value and position of the maxima is calculated by a 2nd order polynomial interpolation applied at the three grid points of the one dimensional row of data surrounding the imminent peak, rather than keeping the information of the cell with the maximum value. Similarly, a linear two point interpolation is performed to locate the half-widths. The interpolations performed are essential in order to ensure mesh independence of the post-processing method.

Data output of the advanced post-processing method is presented in eight plots. The results of Case-15 are used in order to explain these plots. The first two plots illustrated in (Fig.8.4) show the streamwise evolution of U_m and Φ_m . The results are plotted in double logarithmic axes. It can be observed that there are two different transitions taking place. The first one is the typical transition to turbulence T_1 , which in this numerical setup occurs before the jet attaches to the ground. The second transition T_2 is that from the free round jet to a fully developed $3DWJ$. Due to the strong co-flow of the present case setup the jet appears to be still developing towards a $3DWJ$ in the investigated range.

Streamwise evolution of the vertical positions of U_m and Φ_m is presented in (Fig.8.5). At the top plot it can be observed that U_m initially descends in the range $X/D < 60$. After this position the jet is fully attached and begins to develop into its self-similar state, and while growing, the position of U_m also rises. Similar is the situation for the position of Φ_m . The only difference here is that, after the jet attaches completely on the ground, the maximum concentration Φ_m is formed at the lowest level. The flow dynamics of the $3DWJ$ contribute so that mixing intensity on the ground, near the symmetry plane, is low. This can be understood by observing the coherent structure model of (Namgyal & Hall, 2013) shown in (Fig.1.17). In addition to the curves, the plots of (Fig.8.5) include two-dimensional contours of the examined quantities in order to improve understanding of the data calculation method.

A similar format is used at the plots of (Fig.8.6) illustrating the streamwise evolution of the vertical and horizontal velocity half-width, $z_{1/2}$ and $y_{1/2}$. Although the jet is still developing towards its self-similar state, the high lateral growth rate, comparing to the vertical one, is already present. The passive scalars half-widths

are not presented in this explanatory format. It has to be noted that, particularly for the explanation plots, $Z = 0$ is set on the ground. In the actual plots of the sensitivity analysis, which are presented on Appendix C, $Z = 0$ is set at the nozzle offset h .

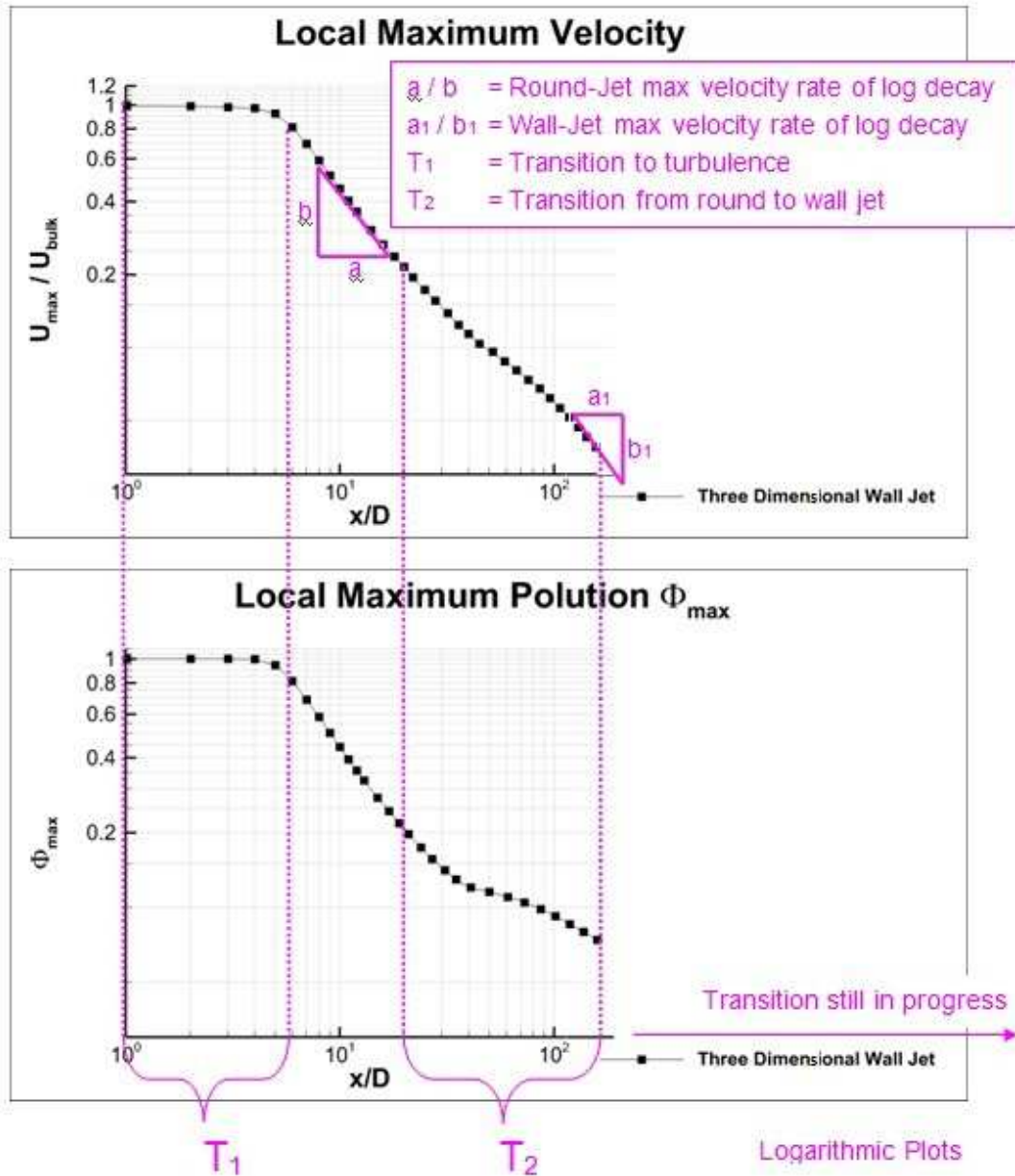


Figure 8.4: Explanation of the plots of maximum velocity and scalar streamwise evolution. The curves are plotted in double logarithmic axes.

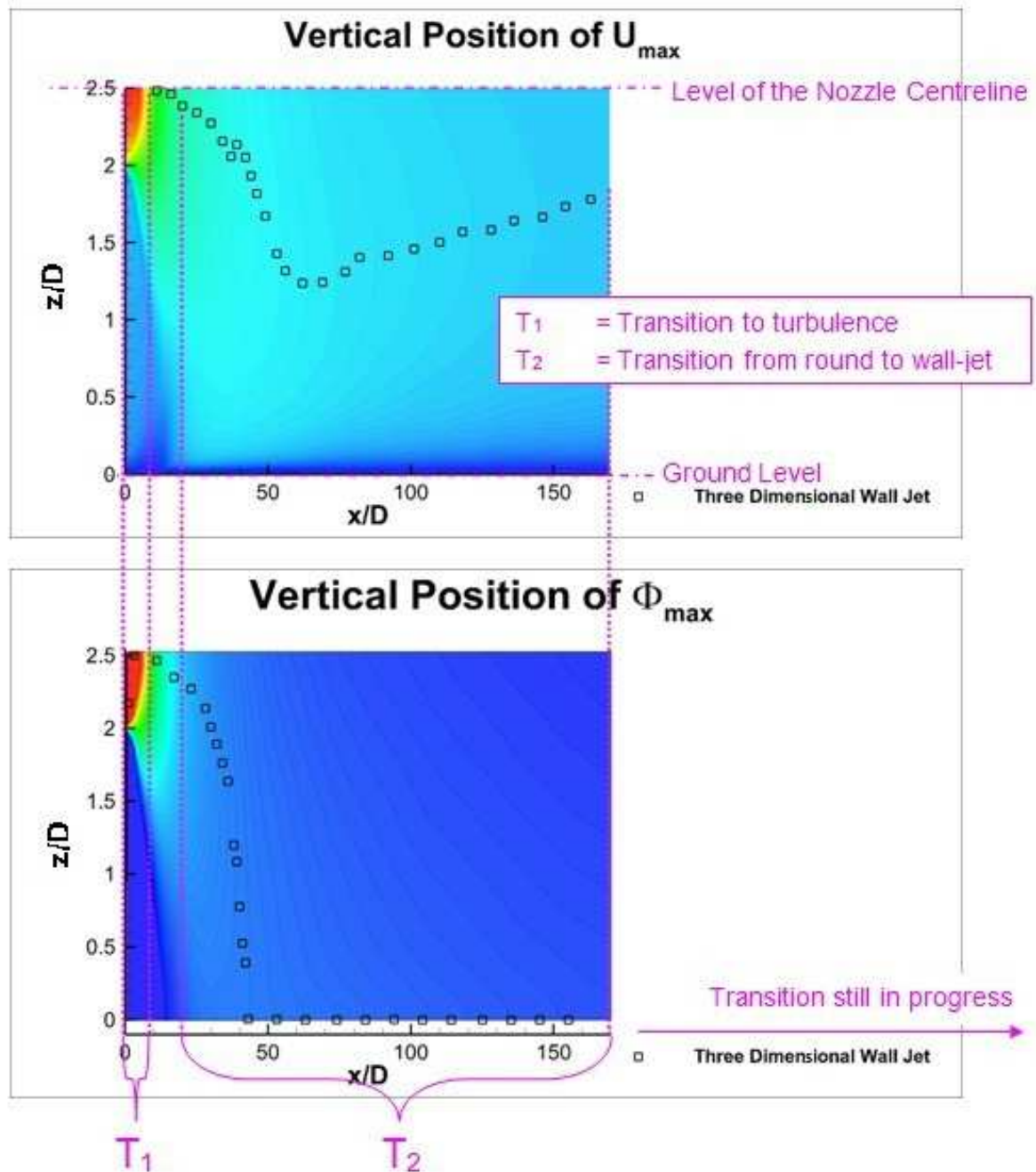


Figure 8.5: Explanation of the plots of streamwise evolution of the position of maximum velocity and passive scalar. Two-dimensional contours of the examined quantities are superimposed in the plot areas to improve understanding of the post-processing method.

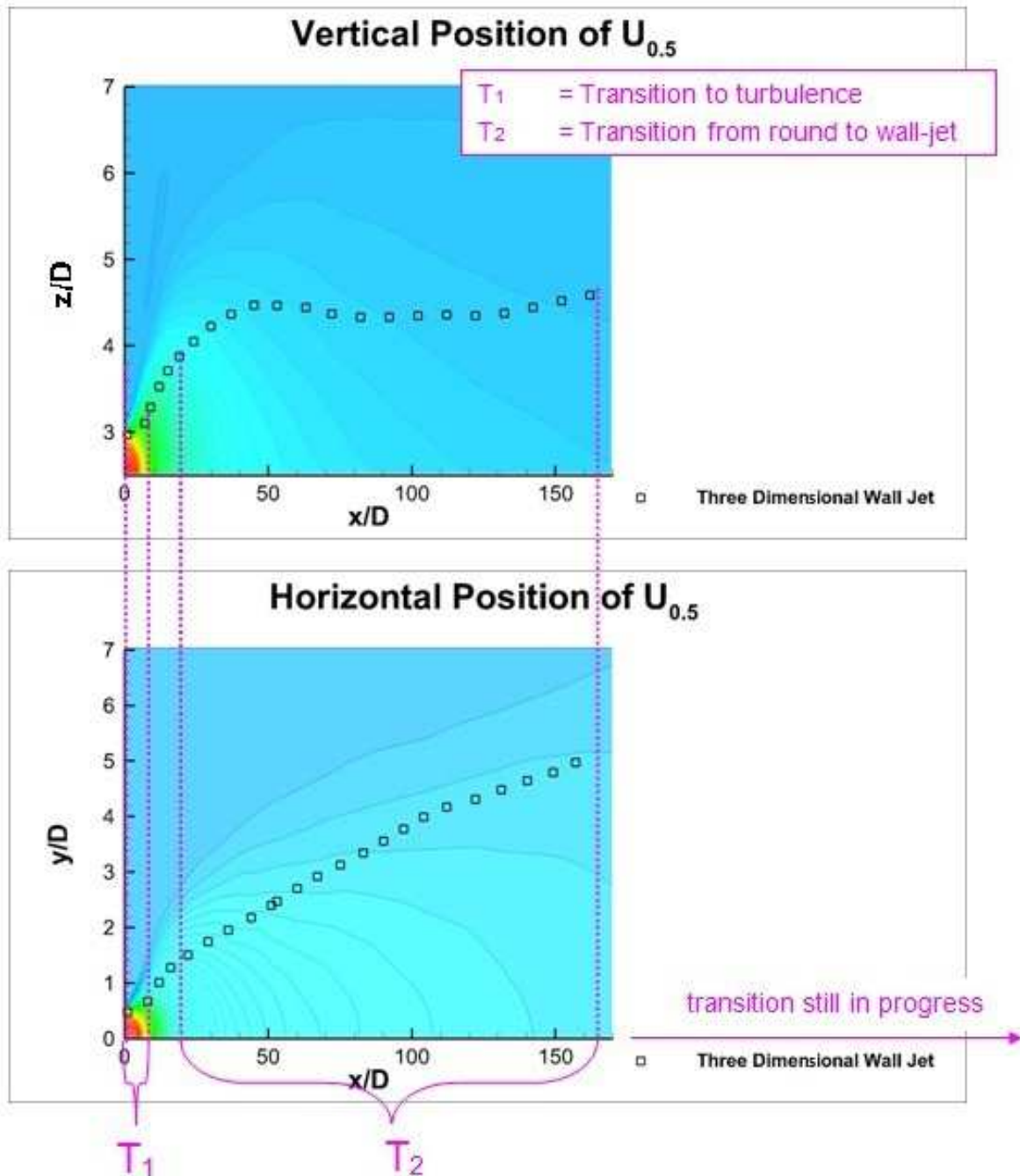


Figure 8.6: Explanation of the plots of streamwise evolution of the vertical and horizontal half-widths. Two-dimensional velocity contours are superimposed in the plot areas to improve understanding of the post-processing method.

8.2 ANALYSIS OF THE RESULTS

This section discusses the results of the sensitivity studies which appear in table 8.1; the number of cases participating in each study, as well as the setup parameters, are included therein. The complete set of comparison plots is located separately in Appendices B and C.

8.2.1 Study Nr.1: SGS Modelling (H-Type)

The FLUENT package supplies several models for the approximation of the *SGS* turbulent stresses of the *LES* algorithm. The majority of these models have been already tested against experimental data at the free-shear validation tests in previous chapter. The outcome was that the dynamic models are more accurate in the transitional jet. Differences on the fully turbulent regime were infinitesimal between different models. The dominant model, *DSM*, is now compared with *WALE*, a model based on the square of the velocity gradient tensor, which is oriented for wall-bounded flows. The aim is to ensure that near-wall mixing is not susceptible to major changes due to different *SGS* modelling. The numerical parameters of the simulations participating to this study are summarized on table 8.1.

In figures (C.1.a) and (C.1.b) it is observed that the streamwise distribution of U_m and Φ_m is identically predicted by the two models. Similar is the situation for the streamwise evolution of the vertical positions above the ground where these two values occur as it can be observed in figures (C.1.c,d). The vertical half-widths for velocity $z_{1/2}$ and concentration $z_{1/2\Phi}$ are presented in figures (C.1.e) and (C.1.f) respectively. It can be observed that $z_{1/2\Phi}$ still descends in the investigated streamwise range, which is a clear indicator that the *3DWJ* has not become self-similar in terms of Φ . Both $z_{1/2}$ and $z_{1/2\Phi}$ are observed to descent faster for *DSM* comparing to *WALE* in the examined range. The horizontal half-widths are similarly presented in figures (C.1.g,h). What appears here is a measure of the lateral extent of the jet, which both *SGS* models have similarly predicted. In particular, $y_{1/2}$ is slightly shorter for *WALE* and $y_{1/2\Phi}$ is identical for the two

models. It has to be reminded that the offset where the horizontal profiles are extracted is variable (refer to section 8.1.2).

The two models examined use different philosophy near the wall. On the other hand, the simulations of this test have been performed on a 1×10^6 cells H-Type mesh with $y^+ \approx 30$. It means that, during these simulations the part of the solution that depends on the near-wall model approximations is large. For this reason, the test is being repeated with a dense O-Type mesh and the results are discussed in section 8.2.10. Meanwhile, *WALE* is selected for the following sensitivity tests.

8.2.2 Study Nr.2: Convergence Criteria

As explained in section 3.1 the convergence of the solution is achieved through an iterative process for every timestep. Each iteration attempts to reduce predefined criteria, which represent the residual error of the solved equations. The speed of convergence depends on several parameters, such as the quality of the mesh, which can be improved pre-processing, and the complexity of the problem. The main factor determining the complexity here is the utilisation of the structured meshing approach. The large variation of the *CFL* condition with downstream distance is unavoidable with this meshing approach. The *CFL* streamwise distribution of a typical simulation of this work is shown in (Fig.8.7). It can be seen that the suggested, for *LES*, $CFL < 1$ is not possible to achieve universally. Such an attempt would require very small timestep size. On the other hand, the residual error reported by the solver represents an average of the entire computational domain. Therefore, the influence of the convergence criteria to the final solution is not expected to be homogeneous. The free-shear validation tests (refer to chapter 7) showed that the computational cost difference between the convergence criteria values 10^{-3} and 10^{-4} is not significant. It also showed that the value 10^{-5} brings about an essential increase at the cost, however, not significantly improving the solution; therefore the value 10^{-4} was selected.

In this test it is attempted to validate these findings of the free-shear validation test, in the presence of the wall. Of course, the value 10^{-3} is excluded as there is no significant gain in terms of computational cost following its application. The

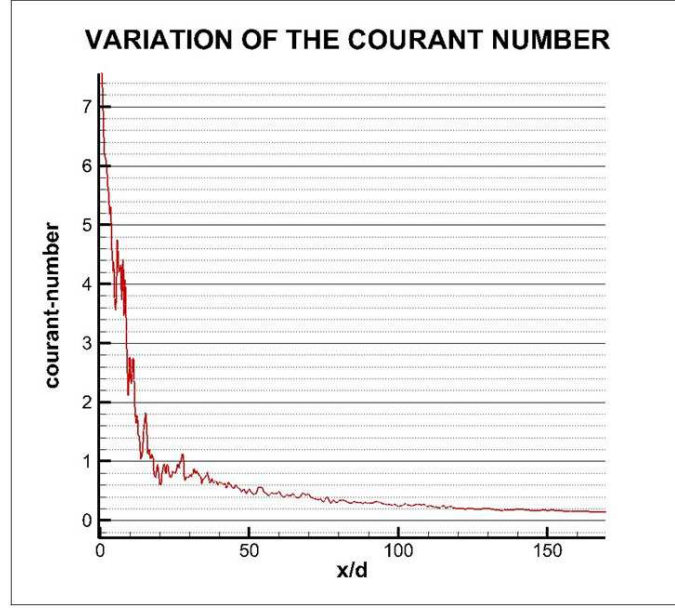


Figure 8.7: Typical streamwise variation of Courant-number in this study.

numerical setup of the previous study is repeated, this time varying the value of the convergence criterion for pressure, momentum, and passive scalar. In the first case, all three parameters are set to the value 10^{-4} and in the second case to the value 10^{-5} . The complete numerical parameters of the simulations participating to this study are summarized on table 8.1.

In figures (C.2.a) and (C.2.b), the streamwise variations of U_m and Φ_m reveal that the less accurate case 10^{-4} accelerates the transition process, something that was not straightforward to observe at the results of chapter 7, where the conventional post-processing approach was used. However, as discussed therein, the transitional errors are gradually recovered downstream for both the velocity and the passive scalar fields. The streamwise variation of the vertical position of U_m and Φ_m is plotted in figures (C.2.c) and (C.2.d) respectively. It is observed that the convergence criteria did not influence the position of the maximums either. The only difference between the two cases is again the transitional shift, corresponding to the accelerated transition of the lower accuracy case. Nevertheless, the farfield predictions are identical. Finally, comparison of the streamwise variation of the vertical and horizontal velocity and concentration half-widths is presented in figures (C.2.e-h). The differences are very small and suggest that

the growth of these scales is independent of the convergence criteria below a certain level (e.g 10^{-4}). Agreeing with the conclusions of the free shear validation tests it is suggested that reducing the jet-inlet turbulence-intensity will affect the solution in the opposite way than the increment of the convergence criteria. The acceleration of the transition can be avoided in this way.

8.2.3 Study Nr.3: Non-Dimensional Wall Distance (y^+)

The non-dimensional wall-distance y^+ is a measure of the near-wall resolution of the mesh and is given by:

$$y^+ = \frac{u_\tau \delta Z}{\nu} \quad (8.1)$$

where u_τ is the local friction velocity, δZ is the vertical distance of the first adjacent cell above the solid surface and ν is the local kinematic viscosity. Since viscosity is constant in the present problem, the value of y^+ is affected only by u_τ and δZ . It has to be clarified that the coordinate system used at this work has Y in the lateral and Z in the normal direction. However, the term y^+ is used instead of z^+ as it is a well established term in the literature for the non-dimensional wall distance.

As presented in section 3.5, FLUENT near-wall modelling uses different ways to model the wall shear stress at flow regions of different y^+ . However, the nature of *LES* suggests that U varies both spatially and transiently; therefore, the value of y^+ is not constant in time at certain locations where turbulent flow exists. Furthermore, the value of δZ is neither a constant throughout the computational domain, but it increases downstream following the vertical expansion of the jet. For this reason, parametrising the value of y^+ is not a straightforward task; it is instead defined as an approximate maximum value at the *3DWJ* farfield.

Three different simulations have been performed in order to assess the behaviour of the solution for different values of y^+ . (Fig.8.8) illustrates the y^+ distribution on the wall boundary of the computational domain, for the three different cases compared in this study. In Case 2 it can be seen that the value of y^+ is on

average less than 30. Similarly, Case 4 and Case 5 have y^+ less than 3 and y^+ less than 1 respectively. The complete numerical parameters of the simulations participating to this study are summarized on table 8.1. (Fig.8.9) is a repetition of (Fig.8.8), but now the explicit influence of the mesh is removed and the calculated wall-shear is illustrated for all three simulations.

The contradiction made here shows that for very large values of y^+ the shape of the turbulent structures reflected on the wall surface is not realistic. However, decreasing the value of y^+ at the level of $y^+ \leq 3$, the presence of the stream-wise “horseshoe-like” vortices is clearly visualised on the wall surface. Further decrease of y^+ improves further the reproduction of this flow characteristic.

Although the two-dimensional results of figures (8.8) and (8.9) reveal the superiority of $y^+ \leq 1$, the advanced post-processing method is necessary in order to make robust comparisons of the interior flow. In figures (C.3.a) and (C.3.b), the streamwise variations of U_m and Φ_m are presented respectively. In the near-field, when the jet is still away from the ground, all three cases behave the same. However, as soon as the jet has reattached, low values of y^+ lead to lower velocity and concentration decay. This can be attributed to the increased local mesh density following the reduction of δZ , which brings about reduced numerical diffusion and reduced contribution of the *SGS* model (*WALE*) to the solution. The streamwise variation of the vertical location of U_m and Φ_m is plotted in figures (C.3.c) and (C.3.d). The differences observed in the position of Φ_m are infinitesimal; however, the case with the large y^+ has failed to predict the position of U_m and the error is more than 50%. Comparison of the streamwise variation of the vertical velocity and concentration half-widths is presented in figures (C.3.e-f). Again, the largest deviations appear for $y^+ \leq 30$ while the results of $y^+ \leq 3$ and $y^+ \leq 1$ are comparable. Finally, the streamwise variation of the horizontal velocity and concentration half-width is presented in figures (C.3.g-h). There is a trend observed for the horizontal scales to increase with reducing the value of y^+ . This is in contradiction with the contours of wall-shear of (Fig.8.9), suggesting that simple observation of the contours may lead to incorrect conclusions, while the application of the advanced post-processing method provides clear trends.

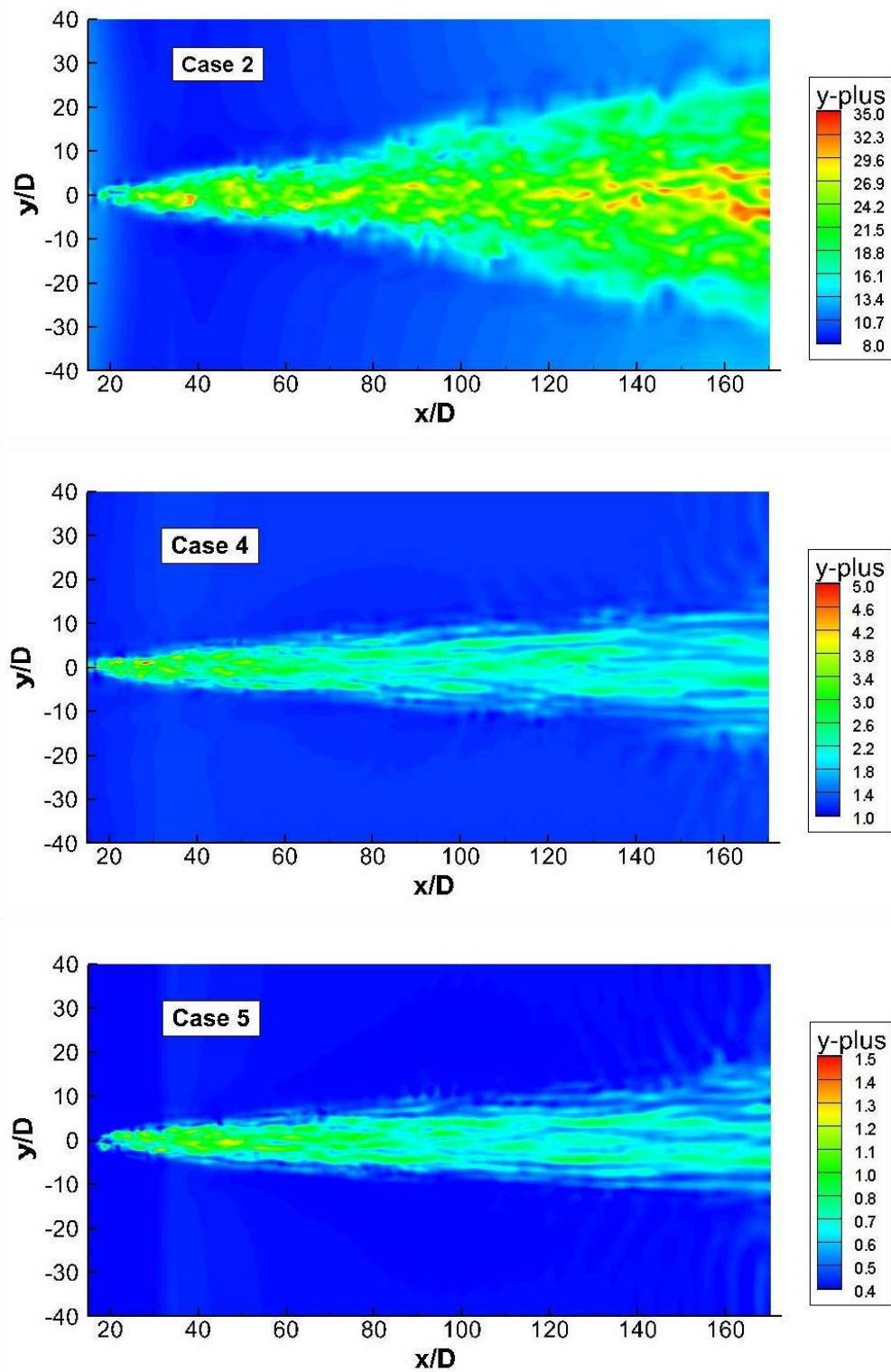


Figure 8.8: y^+ distribution along the solid boundary of the computational domain. Comparison between snapshots of three different cases. H-Type meshing is used for this study.

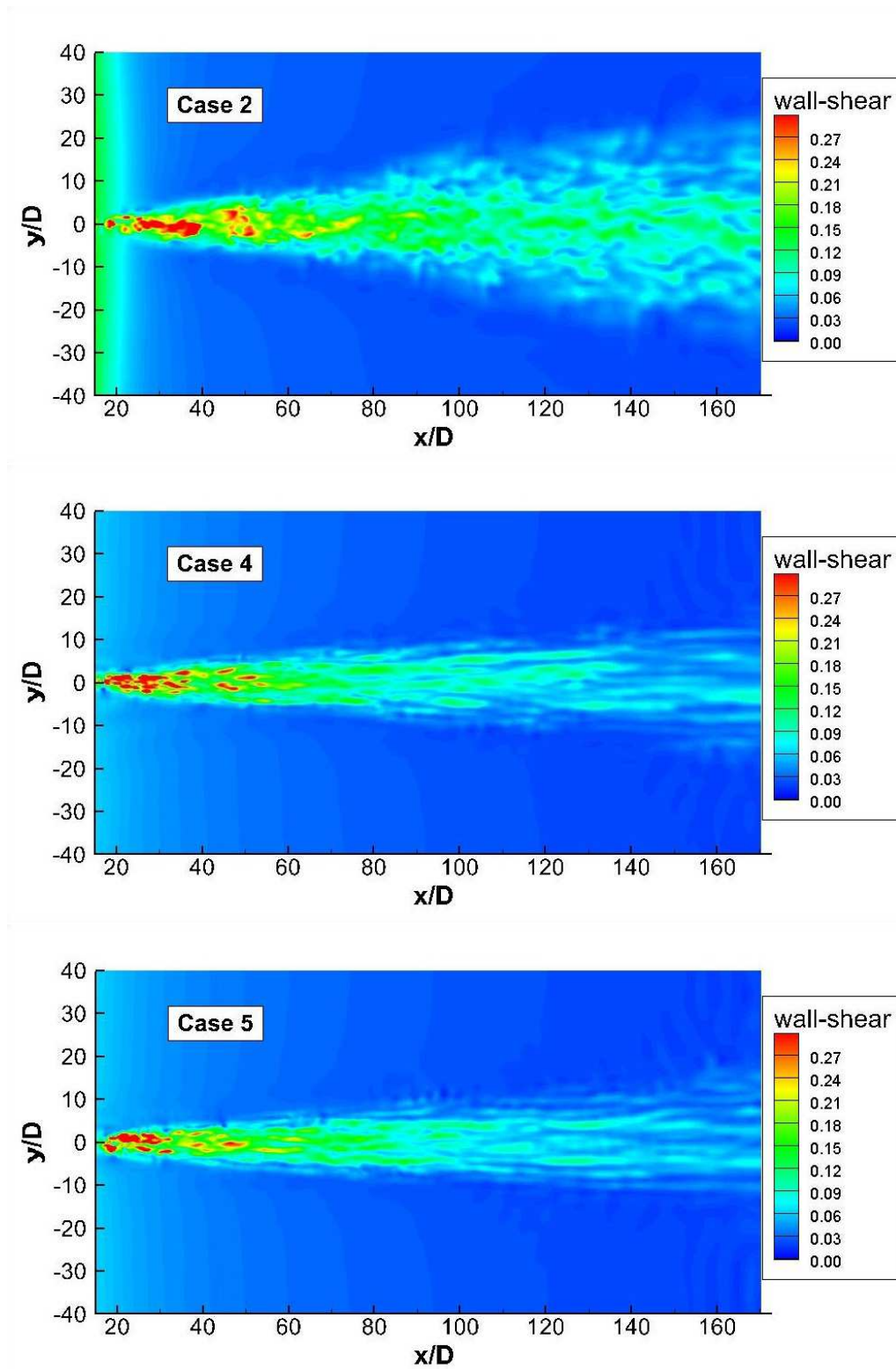


Figure 8.9: Wall-shear distribution along the solid boundary of the computational domain. Comparison between snapshots of three different cases. H-Type meshing used for this study.

It can be concluded that for large values of y^+ , when the influence of the near wall modelling increases, the turbulent mechanisms responsible for flattening the shape of the three-dimensional wall-jet, are underestimated. This causes the jet to spread faster in the vertical direction and drop the value of U_m .

8.2.4 Study Nr.4: Temporal Resolution

The present test is a repetition of the test “*Timestep Size*” of the free-shear validation tests presented in sub-section 7.1.3. Here, it is attempted to investigate the influence of the temporal resolution on the mean flow characteristics of a *3DOJ* by application of the advanced post-processing method described in section 8.1.2. Two different values of timestep size investigated are $\Delta t = 1.5 \times 10^{-4}$ and $\Delta t = 0.85 \times 10^{-4}$. The numerical parameters of the simulations participating to this study are summarized on table 8.1.

Streamwise evolution of the vertical positions of U_m and Φ_m is presented in figures (C.4.a,b). It can be observed that both variables are similarly predicted with the two timestep sizes. The descending of U_m and Φ_m at the reattaching region is sharper with the smaller timestep size as observed in figures (C.4.c,d). However, the farfield predictions are similar between the two cases. Finally, figures (C.4.e,h) reveal that there is a slight over-prediction of the vertical half-widths and an under-prediction of the horizontal ones when a large timestep size is selected.

It has to be considered that the coarse H-Type mesh has been used for the two simulations of this study, where large values of y^+ form at the wall boundary. Based on the findings of the previous test it can be claimed that the near wall modelling contribution was large in these simulations, therefore, it is possible that the effects observed here are magnified because of the mesh.

8.2.5 Study Nr.5: Inlet Ground Boundary Layer (H-Type)

So far the simulations include a homogeneous co-flow $U_f = 5m/s$ at the velocity inlet boundary. However, this is an unrealistic condition since it does not take into account the natural boundary layer formation, unavoidable when the no-slip

condition is selected on the adjacent boundaries, which is the case here. Of course, the improper specification of U_f at the inlet has been handled by the solver, with the solution having converged, and a BL having gradually developed downstream. In this test it is attempted to incorporate a realistic near-wall gradient of the inlet velocity profile and examine the effects on the mean flow characteristics by the application of the advanced post-processing method. The numerical parameters of the simulations participating to this study are summarized on table 8.1.

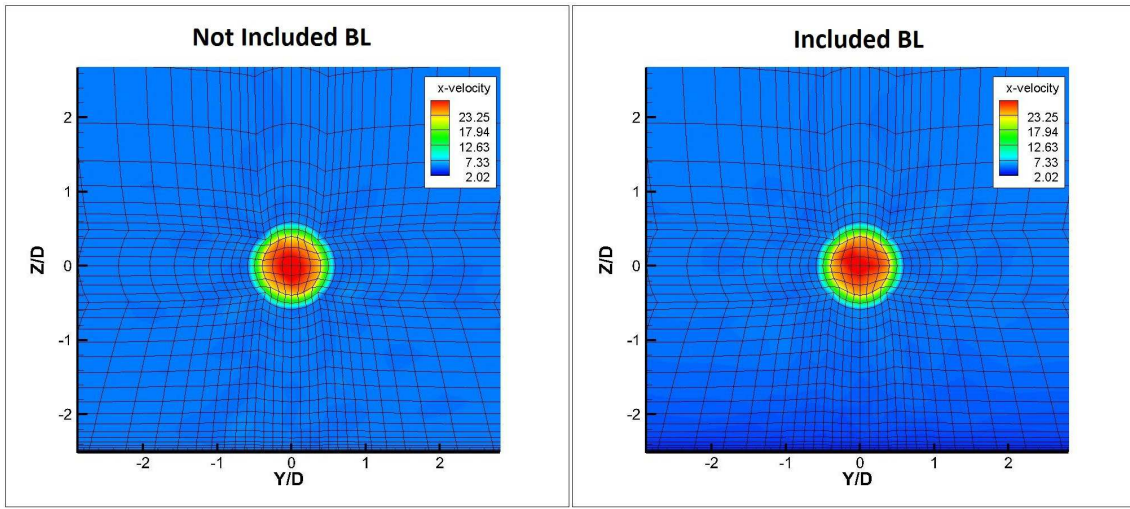


Figure 8.10: Contours of streamwise velocity at the inlet plane. Case 7 (Not-Included BL) on the left, and Case 8 (Included BL) on the right. H-Type mesh-lines are included.

(Fig.8.10) presents contours of instantaneous steam-wise velocity on the inlet plane. The left of the figure shows the homogeneous co-flow velocity inlet, and the right one shows the inlet including a velocity gradient near the wall. The co-flow velocity is described by the $1/7$ wind profile power law relationship:

$$\frac{U_f}{U_r} = \left(\frac{Z}{Z_r} \right)^{(1/7)} \quad (6.1)$$

where the control velocity U_r , and control height Z_r of equation (6.1) have been adjusted to the values of $5m/s$ and $0.012m$ respectively.

Streamwise evolution of U_m and Φ_m is presented in figures (C.5.a,b). It can

be observed that inclusion of the BL causes the velocity to decay faster in the farfield. Of course this is expected due to the low values of U_f close to the wall, where the U_m forms. However, Φ_m have not been significantly affected. The vertical position where U_m and Φ_m occur is plotted along the streamwise direction in figures (C.5.c,d) respectively. A considerable deviation of the position of U_m is observed. This implies that the absolute jet velocity U_m is more effective than the co-flow velocity ratio U_f/U_m to the vertical position of U_m . In other words, the inclusion of the BL at the inlet has mainly affected the position of U_m by increasing its decay rate; therefore, reducing the wall forces flattening the jet. Finally, the velocity and concentration half-widths presented in figures (C.5.e-h) appear to grow slightly faster when the inlet BL is included. This is attributed to the relatively lower value of U_f close to the ground, where the $3DWJ$ develops, which according to equation (5.4) is highly responsible for the growth of these scales.

8.2.6 Study Nr.6: H-Type Mesh Density

A necessary test to ensure validity of the simulations is the grid convergence study. It is desirable that the selected mesh density is high enough so that the final solution is closely independent from the computational mesh resolution. The H-Type meshing approach has been applied so far at this chapter. Here, four different mesh densities of the H-Type mesh are compared. The four meshes consist of 10^6 , 1.6×10^6 , 5×10^6 , and 7×10^6 cells respectively. The complete numerical parameters of the simulations participating to this study are summarized on table 8.1.

Results of the conventional post-processing approach are presented in figures (B.1), (B.2), and (B.3). An overall close agreement is observed at the velocity and concentration profiles in the full streamwise range examined. As far as the boundary definition of Φ is concerned, the results presented in figures (B.1) and (B.2) verify that the distribution of the pollution within the $3DWJ$ region is independent of the pollution inhomogeneity at the nozzle exit. Due to the low number of cells containing the scalar information at the near-field for Φ_2 it was possible to en-

counter loss of information due to numerical diffusion. However, both $\bar{U}$ and U_{rms} of figures (B.1) and (B.2) are converging to the same solution before $X/D = 60$ indicating that this effect is very small. Finally, in close agreement are the wall-shear profiles for all the mesh densities examined, as presented in (Fig.B.3). The only difference observed here is an increase of the streamwise mean wall-shear τ_x with increasing the mesh density. This, however, is attributed to the large y^+ values of the coarse meshes.

Results of the advanced post-processing method are presented in (Fig.C.6). The streamwise evolution of U_m and Φ_m , illustrated in figures (C.6.a,b), reveals that the lower the mesh density the higher the decay rate of both quantities. However, the differences between the 5×10^6 and 7×10^6 cells meshes are infinitesimal, indicating that the solution converges with mesh density. This statement is supported by the vertical positions of U_m and Φ_m in figures (C.6.c,d), where again the *LES* predictions converge with increasing the mesh density. A large scattering of the *LES* predictions is observed to the velocity half-widths in figures (C.6.e-h). The reason is that the H-Type meshing approach does not offer adequate control to the mesh densification process, therefore, the added cells are not uniformly distributed in the jet developing volume. Furthermore, a large part of the added cells is unavoidably consumed in regions of the mesh surrounding the jet development. Therefore, at this stage of the study it is decided to change the meshing approach and develop the modified O-Type meshing for the simulation of the *3DWJ*, as presented and discussed in section 4.2.2.

8.2.7 Study Nr.7: H-Type Vs O-Type Meshing

The present paragraph highlights the advantages following the application of the O-Type meshing approach for the simulation of the *3DOJ*. (Fig.8.11) illustrates contours of streamwise velocity at the cross-section $X/D = 45$ for two different mesh densities of each meshing approach. The red lines highlighting the contours of half-velocity $U_{0.5}$ and co-flow velocity U_f are taken from case 15 (densest O-Type mesh). Additionally, the H-Type mesh results (top images) do not include the inlet *BL* while the O-Type ones (bottom images) do, therefore the results dif-

fer close to the ground. It is observed that the O-Type meshing approach offers

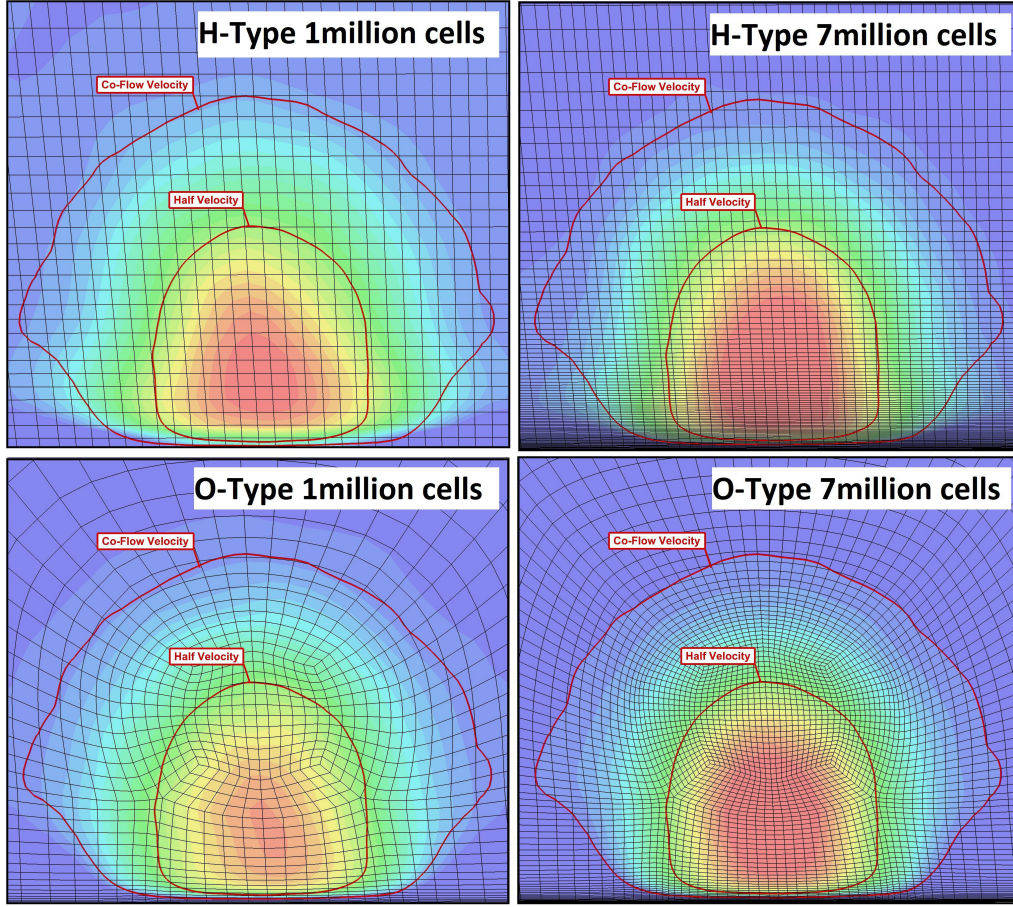


Figure 8.11: Computational mesh of cross-section $X/D = 45$ for four different meshes. Contours of $\bar{U}$ are colouring the planes. The red lines highlighting the half-velocity $U_{0.5}$ as well as the co-flow velocity U_f are taken from case 15.

higher homogeneity of the cell sizes within the jet-developing region, while it allows for a considerable reduction of the number of cells away from the jet. On the other hand, the H-Type mesh is not flexible to concentrate the cells at the jet-region, therefore, high mesh resolution lies at regions surrounding the jet. Consequently mesh densification with the H-Type meshing approach requires a high percentage of the added cells to be consumed to the surrounding regions, while the O-Type densification of the mesh is more efficient. A quantitative view of the above statement is given in table 8.2. The computational domain of all participating meshes is of similar volume. The number of cross-sectional cells describing the jet is considerably larger with the O-Type meshing approach comparing to H-Type. Additionally, this number increases more effectively with increasing the

Mesh type	Mesh Density	Nr of Cells Describing the Jet	Outer Layer Resolution	Half Core Resolution
H-Type	1 million	100	7	7.5
	1.6 million	100	6	15
	5 million	289	7	26
	7 million	432	9	31
O-Type	0.38 million	96	8	8
	1 million	240	11	11.5
	5 million	448	16	17
	7 million	1056	24	24

Table 8.2: Parametrization of the cross sectional cell distribution for the computational meshes used at the wall bounded validation tests.

mesh density (or mesh size), when O-Type meshing is used. Finally, due to the increased mesh isotropy within the jet region of the O-Type approach it can be observed that the individual mesh dimensions are increasing proportionally to the mesh density, which is not the case for the H-Type meshing approach¹. It has to be mentioned that mesh homogeneity is possible to be achieved with the H-Type approach by certain block subdivision of the volume, however, it is not possible to avoid the large number of cells at the jet-surrounding area. The numerical parameters of the simulations participating to this study are summarized on table 8.1.

(Fig.C.7) compares the results of two meshes of equal density designed with the two different approaches. In particular, the first mesh has been designed with H-Type meshing and consists of 1×10^6 cells (coarsest H-Type mesh). On the other hand, the second mesh has been designed with the O-Type meshing approach, also consisting of 1×10^6 cells (intermediate O-Type mesh). The results of the two meshes are similar for the full set of examined mean flow characteristics. Noteworthy is the smoother shape of the streamwise evolution of the vertical position of U_m with the O-Type approach. However, a better view of the direct advantages of the O-Type meshing approach, with respect to the *LES* solution,

¹the individual mesh dimensions are calculated within the contours of $U_{0.5}$ and U_f at $X/D = 45$ for case 15 (red-lines in figure (8.11)).

is going to be obtained by the repetition of three of the above validation tests in the following paragraphs.

8.2.8 Study Nr.8: O-Type Mesh Density

The H-Type mesh density study in section 8.2.6 presented a large scattering of the comparison plots, not making it possible to observe whether grid convergence occurs in all mean-flow characteristics. In this test the results of four different mesh densities are compared. The four meshes consist of 0.38×10^6 , 1×10^6 , 2.5×10^6 , and 7×10^6 cells respectively. Each mesh densification has been attempted to be as proportional as possible with the previous one, by a factor of approximately 2.5. The coarsest mesh here has been designed with 0.38×10^6 cells and consists of about 20 times less cells than the densest mesh which has been designed with 7×10^6 cells. The complete numerical parameters of the simulations participating to this study are summarized on table 8.1.

Results of the advanced post-processing are presented in (Fig.C.8). In particular, the streamwise evolution of U_m and Φ_m , illustrated in figures (C.8.a,b), reveals that the lower the mesh density the higher the decay rate of both quantities, in agreement with the H-Type mesh density test in section 8.2.6. The differences between the results of the 2.5 and the 7 million cells meshes are infinitesimal, indicating that the solution is nearly independent of the mesh resolution. This statement is supported by the vertical positions of U_m and Φ_m in figures (C.8.c,d), where again the *LES* predictions tend to converge with increasing the mesh density. *LES* predictions of the vertical half-widths for velocity and concentration in figures (C.8.e,f) are very close for all mesh densities.

The horizontal velocity half-width $y_{1/2}$, in (Fig.C.8.g) tends to converge regularly with increasing the mesh density. However, the horizontal concentration half-width $y_{1/2\Phi}$, in (Fig.C.8.h) follows a more irregular pattern. This, however, is attributed to the varying offset of the extraction of the horizontal profiles, which according to the definition of the advanced post-processing method is implemented at the variable $Z = z_{1/2}$. According to (Fig.C.8.c) there is a significant variation of $z_{1/2}$ with varying the mesh density, especially for the densest mesh of 7×10^6

cells.

8.2.9 Study Nr.9: Inlet Ground Boundary Layer (O-Type)

The present test is a repetition of study Nr.5. Here an O-Type mesh, with the same density as in the H-Type realization of the test (1×10^6 cells), is used. (Fig.8.12) is a repetition of (Fig.8.10) presenting contours of instantaneous stream-wise velocity on the inlet plane of the O-Type mesh. The left of the figure shows the homogeneous co-flow velocity inlet, and the right one shows the inlet including a velocity gradient near the wall. The co-flow velocity is described by the $1/7$ wind profile power law relationship (refer to equation 6.1).

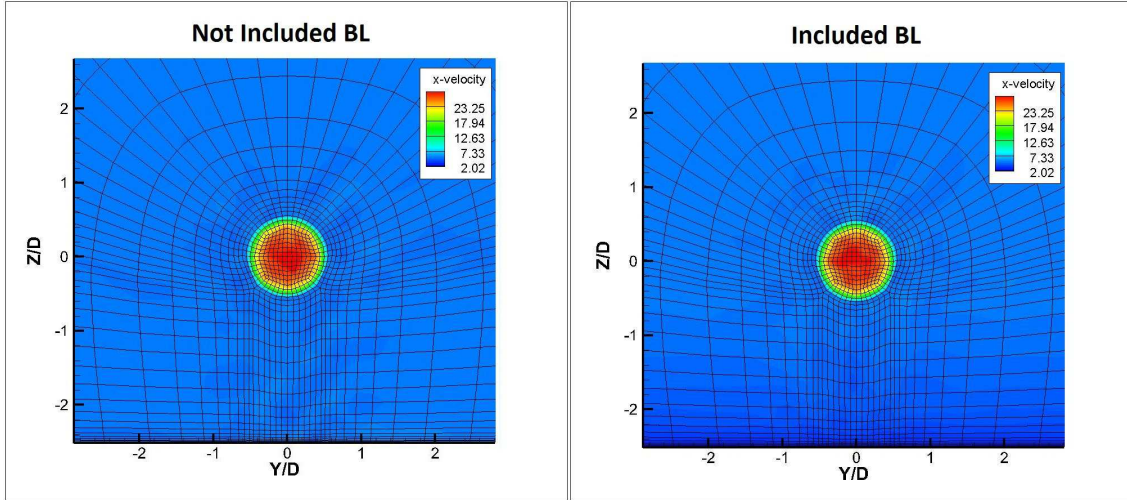


Figure 8.12: Contours of streamwise velocity at the inlet plane. Case 13 (Not-Included BL) on the left, and Case 12 (Included BL) on the right. O-Type mesh-lines are included.

There are minor differences between the results of the advanced post processing of this test and the ones of study Nr.5. A noteworthy difference is the streamwise evolution of the vertical position of U_m which is illustrated in (Fig.C.9.c). In particular, the O-Type approach has predicted smaller offset difference between the included and non-included BL cases than the H-Type approach did. Figures (C.9.e,g) reveal that although the vertical position of U_m is higher for the included BL case, both velocity half-widths $z_{1/2}$ and $y_{1/2}$ are smaller. This tells that although the free-stream co-flow velocity is the same in the two cases $U_f = 5m/s$,

the reduced velocity near the wall due to the formation of BL causes the jet to decelerate faster and to also develop faster into a self-similar state. Recall that the horizontal to vertical growth rate ratio of a self-similar $3DWJ$ is extremely large (refer to section 1.4.3). Finally, although the velocity scales appear to have been affected by the inclusion of the BL at the inlet, the effects observed to the concentration half-widths $z_{1/2\Phi}$ and $y_{1/2\Phi}$, are smaller as revealed by figures (C.9.f,h).

8.2.10 Study Nr.10: SGS Modelling (O-Type)

The present test is a repetition of study Nr.1. The value of the non-dimensional wall-distance of study Nr.1 was large $y^+ \leq 30$ adding uncertainty to the test according to the findings of section 8.2.3. This study includes $y^+ \leq 1$. Additionally, the inlet ground BL was not included in study Nr.1, while here it is included. Finally, the mesh density of the H-Type mesh was 1×10^6 cells. Here a dense O-Type mesh is used consisting of 7×10^6 cells. The complete numerical parameters of the simulations participating to this study are summarized on table 8.1.

Comparison plots of the advanced post-processing are presented in figures (C.10.a-h). It can be observed that the LES predictions of the mean flow characteristics do not vary at all with the two different SGS models tested. Additionally, neither the computational cost was observed varying.

The $WALE$ model is selected for the implementation of the wall-bounded simulations of this thesis.

8.2.11 Studies Nr.11-13: Physical Effects

Co-flow Effect

Co-flow velocity, representative of a head-wind configuration at central airports, normally corresponds to much lower velocity ratio $U_f/U_b \ll 0.22$. Before carrying on with the implementation of the actual numerical experiments of this thesis, study Nr.11 aims to identify numerical issues of such a case modification. In this way, the co-flow velocity of the present setup has been gradually reduced and two additional simulations have been performed with $U_f/U_b \approx 0.13$ and $U_f/U_b \approx 0.04$.

In order secure mesh independence the same computational mesh has been used for all three studies of this test.

The *LES* results of the three cases are compared by means of the advanced post-processing method, and the comparison plots are presented in figures(C.11.a-h) . (Fig.C.11.a) illustrates the streamwise evolution of U_m . It is observed that reducing the value of U_f leads to an earlier transition to turbulence. Additionally, the larger velocity difference across the initial mixing layer $U_b - U_f$, which is the case for low values of U_f , enhances the local turbulent activity, consequently, increases the local growth rate, and extensively the peripheral suction forces causing the Coandă effect. For this reason low values of U_f are characterized by earlier reattachment as indicated by the vertical position of U_m in (Fig.C.11.c). Similar observations are made for the streamwise evolution of Φ_m and its vertical position in figures (C.11.b) and (C.11.d) respectively. The reattachment length becomes significantly shorter with reducing the velocity ratio. This is well observed in figures (C.11.a,c). In particular, the original setup with $U_f/U_b \approx 0.22$ results at a reattachment length $\approx 35D$. This length reduces to $\approx 20D$ for $U_f/U_b \approx 0.13$, and to $\approx 5D$ for $U_f/U_b \approx 0.04$.

Jet reattachment is followed by the transition towards a self-similar $3DWJ^2$ as schematically illustrated in (Fig.1.18). The original setup, where $U_f/U_b \approx 0.22$, has already been analysed in previous sections and the jet does not achieve self-similarity in the investigated range (refer to figures 8.4, 8.5, and 8.6). The intermediate co-flow velocity ratio $U_f/U_b \approx 0.13$ causes that the mean velocity length scales obtain a closely linear increasing rate; however, self-similarity is not clear. The vertical concentration half-width $y_{1/2\Phi}$ still descends at the end of the computational domain $X/D = 170$. Finally, further reduction of the co-flow velocity ratio to $U_f/U_b \approx 0.04$ allows for the jet to develop in self-similarity, at $X/D \approx 70$. It is made clear that in the present discussion the self-similarity origin is defined at the position where the vertical half-widths begin to increase in a regular pattern³.

²Self-similarity of $3DWJs$ has not been tested in a co-flow configuration in the reviewed literature. It is possible that the co-flow is a restricting parameter for self-similarity to occur. It is believed by the author that for low values of U_f a high level of self-similarity is expected in the investigated streamwise range.

³in the absence of co-flow it would be linear growth according to theory, while in the presence of

It is noted that the jet growth rates of case $U_f/U_b \approx 0.04$ were very large, resulting to jet boundary interactions at $X/D \approx 70$. The O-Type mesh configuration is appropriate for such a flow as explained in 4.2.2. Therefore, the actual numerical experiments of the next chapter are performed at a hybrid H-Type/O-Type mesh, presented in (Fig.4.12).

Wall Effect

The *3DOJ* setup is a repetition of the low speed free-shear axisymmetric jet which has already been simulated in the free-shear validation tests (refer to section 7.1). It is interesting to contrast the mean flow characteristics of the two cases in order to, first of all, extend the advanced post-processing method to the axisymmetric jet results, also, observe the effects on the jet mean flow characteristics due to the presence of the solid boundary.

Comparison plots can be found in figures (C.12.a-h). The centreline velocity decay U_m is only slightly affected by the presence of the wall (Fig.C.12.a) and the differences are limited to the reattachment region. However, this is not the case for the maximum passive scalar concentration Φ_m , in which case (Fig.C.12.b), the decay rate of Φ_m drops considerably as soon as the jet reattaches. It is known, however, from the previous study that this “*delay*” of the concentration decay is related to the reattachment of the jet.

Theoretically, the vertical positions of U_m and Φ_m are encountered at $y, z = 0$ for the axisymmetric jet. Slight deviation of these curves, of order $< 0.2D$, is observed in figures (C.12.c,d). This deviation is not generally related to the error of the advanced post-processing method. On the contrary, it is attributed to the effect of mesh expansion to the numerical interpolation performed to calculate the imminent profile's peak and its position. Consider that the size of a cell lying at the axisymmetric jet centreline in the farfield is of order $1D$, while the vertical dimension of a cell lying at the offset where U_m occurs is more than one decade smaller.

co-flow it would be described by asymptotically reducing growth rates as drawn in the theoretical analysis of this work and illustrated in (Fig.5.2).

With regards to the vertical and horizontal velocity and concentration half-widths, figures (C.12.e-h) illustrate the flattening of the jet after it has attached to the wall. In particular, the near-field ($0 \leq X/D \leq 40$) expansion of the *3DOJ* is quite similar to the one of the axisymmetric jet in both directions. However, after the reattachment point, $X_{rp} \approx 40D$, the vertical half-widths are getting smaller, while the horizontal ones are getting magnified comparing to the ones of the axisymmetric jet. Flattening of the concentration half-widths is more intense than that of the velocity ones.

(Fig.8.13) illustrates contours of $\Phi = 6.5\%$ for three different cases, including the axisymmetric jet in strong co-flow (blue colour), the *3DOJ* in strong co flow (red colour), and the *3DOJ* in reduced co flow (green colour). It is observed that the presence of the wall causes the pollution to concentrate and develop close to it, considerably delaying the dispersion process. Additionally, reduction of the co-flow speed accelerates mixing, therefore, the jet spreads faster, and the pollution disperses earlier.

Three-Dimensionality Effect

In the previous test, direct comparisons between the axisymmetric jet and the *3DOJ* were performed. In a similar way, this study compares the *3DOJ* with its two dimensional counterpart. A plane jet with infinite span and vertical description identical to the diametrical description of the slow round jet used at this setup, is simulated. Symmetry condition is set to the lateral boundaries, and the jet offset $h = 2.5D$ has been reduced to $h = 0.5D$ in order to ease the near-field computations.

Results of the advanced post-processing are presented in (Fig.C.13). In particular, the streamwise evolution of U_m and Φ_m are illustrated in figures (C.13.a,b). Both plots reveal that the length of transition to turbulence of the plane jet is about double that of the *3DOJ*. The decay rates of U_m and Φ_m are both smaller for the two-dimensional jet, which is explained by the single mixing layer apparent. Finally, the vertical length scales of figures (C.13.c-e) reveal that besides the identical co-flow velocity of the two cases $U_f = 5m/s$, the plane jet has early obtained

a self-similar behaviour at $X/D \approx 35$, while, as previously discussed, the $3DOJ$ is still developing at $X/D = 170$.

Conclusions

A modification to the free-round jet setup of (Garry & Holt, 2008) has been made in order to carry out the wall bounded validation tests of this chapter. The main target of this study was to parametrically observe the response of certain solution variables to a set of selected numerical and physical parameters, therefore, lack of experimental data for direct cross-validation of the numerical predictions does not reduce credibility of this study. Repetition of the near-field accuracy tests of the free shear validation tests was observed to behave in a similar pattern as in free-shear validation tests, most importantly indicating that errors obtained in the transitional regime are quickly recovered suggesting independence of the solution in the farfield. An additional finding of this study was the importance of universally adjusted low values of the non-dimensional wall-distance ($y^+ \approx 1$). Apart from the immediate effects in near-wall region, high values of y^+ have been observed to affect both the mean velocity and concentration characteristics. Further to the numerical parameters, the effect of three physical parameters has been assessed, including the co-flow velocity, the wall effect and the three dimensionality of the plume. From these tests it has been observed that the dispersion of the pollution reduces significantly by increasing the co-flow velocity and reduces even further when the plume attaches to a solid surface, while the three dimensionality of the plume increases the rate of mixing, agreeing with the reviewed literature. The necessity to develop a post-processing code which calculates the stream-wise evolution of the mean velocity and mean concentration characteristics, by performing volumetric analysis of the three-dimensional solutions, was realised during this study. This is expected to reduce considerably the post-processing effort for the analysis of the complete setup results which follows.

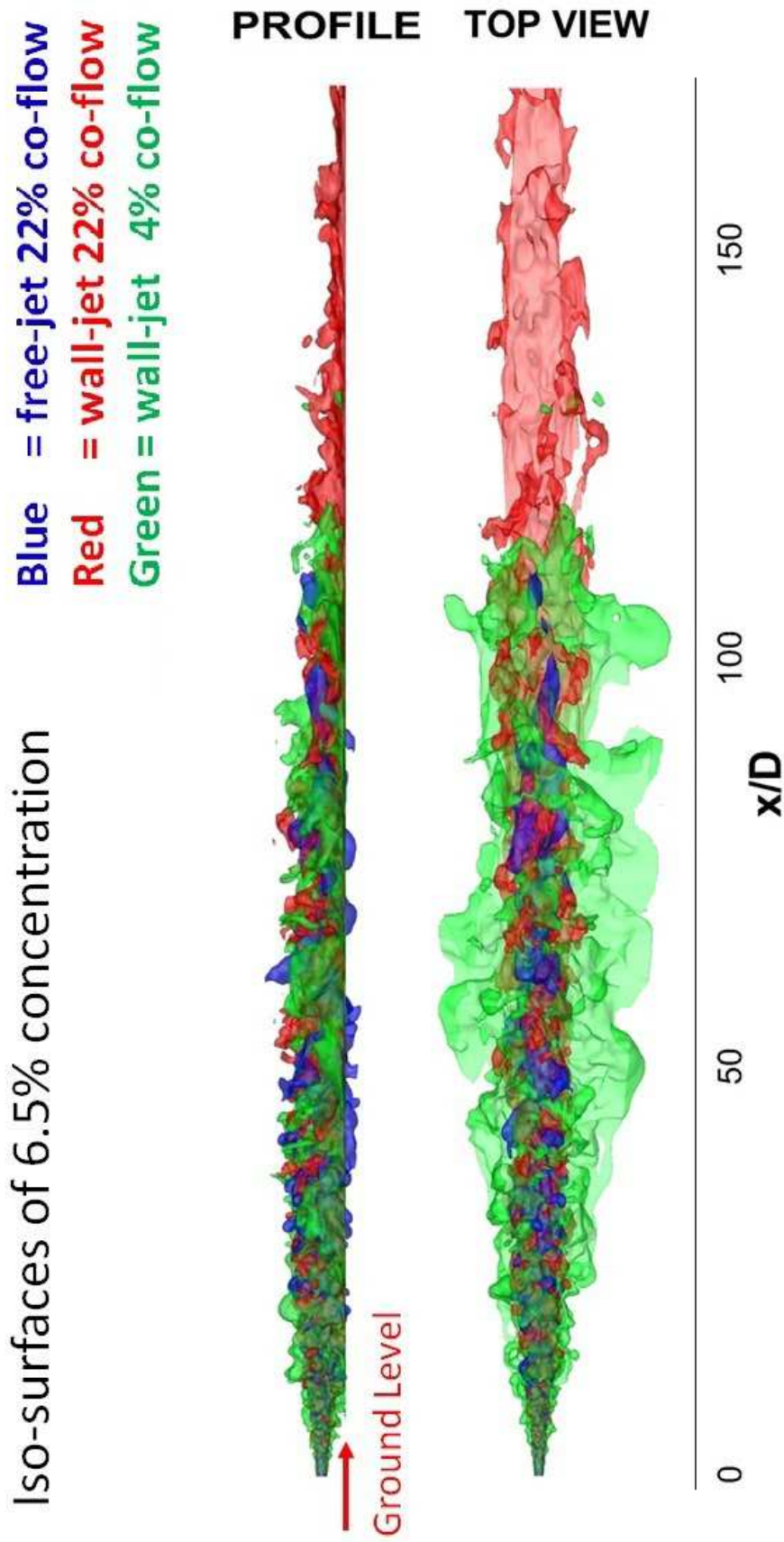


Figure 8.13: Iso surfaces of $\Phi = 6.5\%$ for three different cases of the physical parameter testing.

Chapter 9

Baffle Validation

This chapter is dedicated to the validation of the selected approach achieving momentum reduction of the plume in order to accelerate its lift-off and extensively the dispersion of the pollution. Initial investigations of the baffle inclusion in the path of the *3DWJ* development, and primary validations of the proposed model, *DRAMBI* (refer to section 4.3 for description of the model), are carried out based on low resolution drag-force and pollution concentration measurements in the atmospheric boundary layer wind tunnel (*ABLWT*) of Cranfield University. These experiments have been performed by (Velikov, 2013) on parallel studies aiming to optimise the baffle shape and geometrical characteristics of the original baffle arrangement. There are two different experimental setups employed for the initial baffle validations of this chapter, the details of which are presented in sections 6.2.1 and 6.2.2 respectively. Significant details of the setup and history of the simulations reported are summarised in table 9.1.

Before proceeding to the analysis of the *LES* results it is essential to first take a closer look at the temporal behaviour of *DRAMBI*. A three dimensional representation of all three control volumes, within which the model operates, is illustrated on the top of (Fig.9.2). The magnified view at the bottom of the same figure, highlights the shape of V_C and V_A , the latter being coloured with contours of instantaneous pressure. It is observed that in locations where a sudden change of pressure appears, the boundary of the active volume V_A gets deformed. The model identifies the deformation, from the local change of momentum, and re-

sponds by changing the value of the weighting factors a_i in this region. Therefore, in the following timestep the percentage of F_D distributed in those cells changes in order for V_A to return at its equilibrium shape, which is dictated by the volume ratio Ψ and enhanced by the geometrical function b_i .

Changes of the total momentum included in V_A are handled by changing the value of F_D . (Fig.9.1) presents time-series of data of the baffle representation model extracted during the simulation. In particular, instantaneous values of the volume ratio Ψ_n , the slope λ of the asymptotic function $a_{\Delta F}$, and the predicted drag force F_D are plotted for a short fraction of the simulated time. The zoom-in view reveals that the oscillation of Ψ is adequately small, varying as: $-1\% \leq \Psi \leq 1\%$. It has to be noted that the radii of the three baffle arrays have been set to infinity in Cases 3-5 of this chapter.

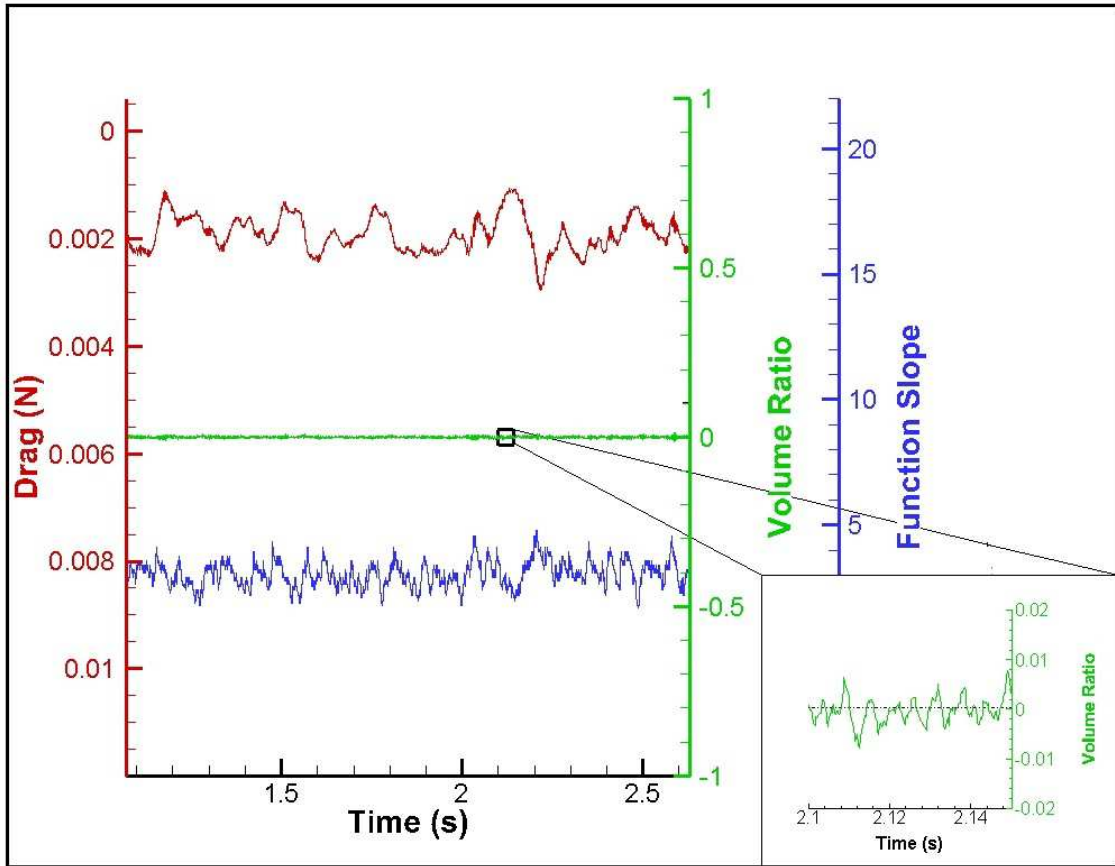


Figure 9.1: Time-series of data of *DRAMBI* extracted during the simulation. Instantaneous values of Ψ_n , λ , and F_D are plotted for a short fraction of the simulated time. The zoom-in window is used to highlight the magnitude of the oscillation of Ψ .

Table 9.1: Summary of the simulations' history and setup.

Baffle Arrangement	First Setup			Second Setup	Double Size
Simulation Name	<i>Case-1</i>	<i>Case-2</i>	<i>Case-3</i>	<i>Case-4</i>	<i>Case-5</i>
Baffle Representation	Designed		Modelled	Modelled	Modelled
Wind Direction	Co-flow	Co-flow	Co-flow	Cross-flow	Co-flow
Mesh Size (cells$\times 10^6$)	19	3.7	2.2	2.2	2.2
Δt (s$\times 10^{-4}$)	2.2	2.9	3.2	2.1	3.2
Simulated Time (s)	~ 25	~ 45	~ 45	~ 30	~ 45
Nr. Processors	128	64	64	64	64
Simulation Time (Hrs.)	~ 600	~ 300	~ 150	~ 150	~ 150

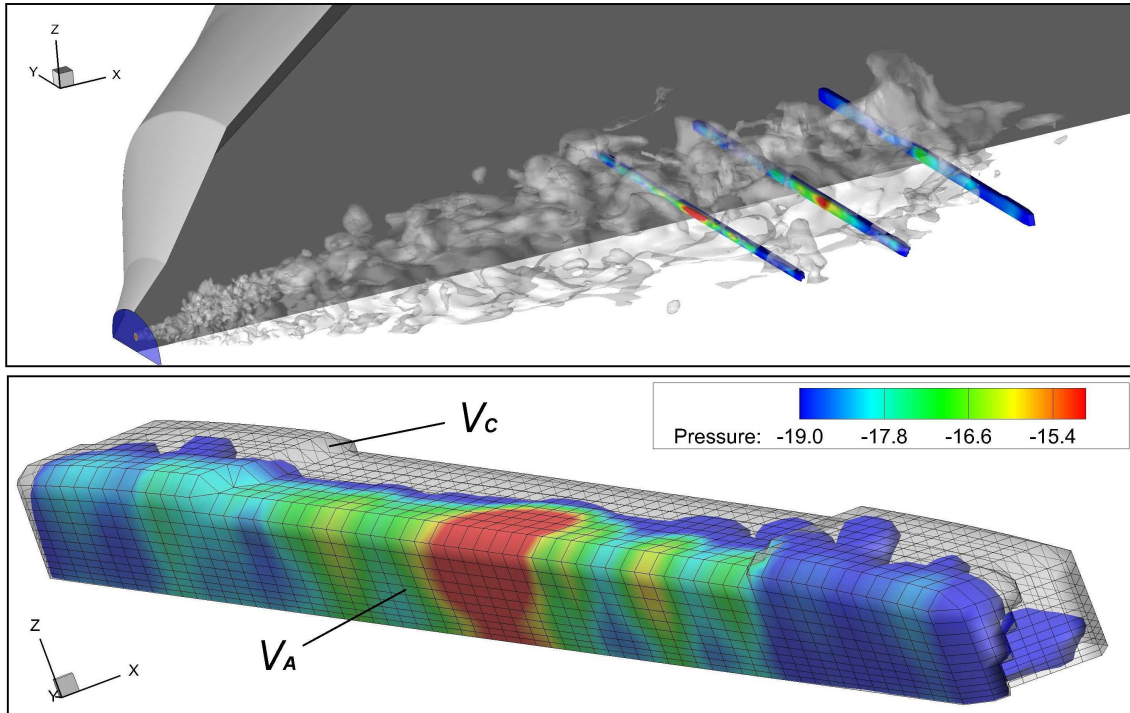


Figure 9.2: Three dimensional representation of the control volumes of *DRAMBI* (top), and magnified view of a single control volume (bottom). Contours of instantaneous pressure are colouring the boundary of V_A . Iso-surface of $\Phi = 3.5\%$ is included in the top figure. Present data are taken from an instance of the simulation *Case-3*.

9.1 FIRST SETUP: 1/100 Scale

The validation of the *LES* results, is based on the time averaged drag force $\overline{F}_D$ recorded by the balance platform. For the present setup, the value $\overline{F}_D = 0.052 \pm 0.002N$ has been measured. The uncertainty of the measurement $\pm 0.002N$ was evaluated based on the repeatability recorded during experiment. $\overline{F}_D$ represents the total drag force, including the individual drag of each baffle row, in addition to the viscous drag imposed by the horizontal surface of the balance platform.

A grid-sensitivity test is implemented using two different meshes. In this way, *Case-1* and *Case-2*, with mesh densities 19 and 3.7 million cells respectively, are performed. Then, the baffle geometries were removed from the coarse mesh, also coarsening the resolution in the near-baffle region, and with the resulted mesh of 2.2 million cells the simulation *Case-3* was performed. Table 9.2 presents the results of *Case-1*, *Case-2* and *Case-3* in comparison to the *ABLWT* measurement. The red coloured values represent the distribution of $\overline{F}_D$ (excluding the balance platform) on each baffle row separately.

Table 9.2: Time averaged drag forces for three baffle arrays and balance platform individually. Comparison between the *ABLWT* measurement and *LES* predictions. Forces are presented in N ($\times 10^2$).

	EXPERIMENT	<i>Case-1</i>		<i>Case-2</i>		<i>Case-3</i>	
Entity	$\overline{F}_D$	$\overline{F}_D$	(%)	$\overline{F}_D$	(%)	$\overline{F}_D$	(%)
1st B. Array	—	1.188	30.0	1.090	31.0	1.385	31.3
2nd B. Array	—	1.423	36.0	1.274	36.2	1.589	36.0
3rd B. Array	—	1.349	34.0	1.153	32.8	1.439	32.7
B. Platform	—	1.143	—	0.970	—	0.986	—
SUM (baffles)	—	3.960	100.0	3.517	100.0	4.413	100.0
SUM (all)	5.2±0.2	5.103		4.487		5.399	

The accuracy of *LES* is evaluated with respect to the experimental measurement from equation 9.1. Based on the uncertainty of the balance platform measurement $\pm 0.002N$, the reported accuracies also include an uncertainty of

$$\frac{0.002}{0.052} = \pm 3.85\%.$$

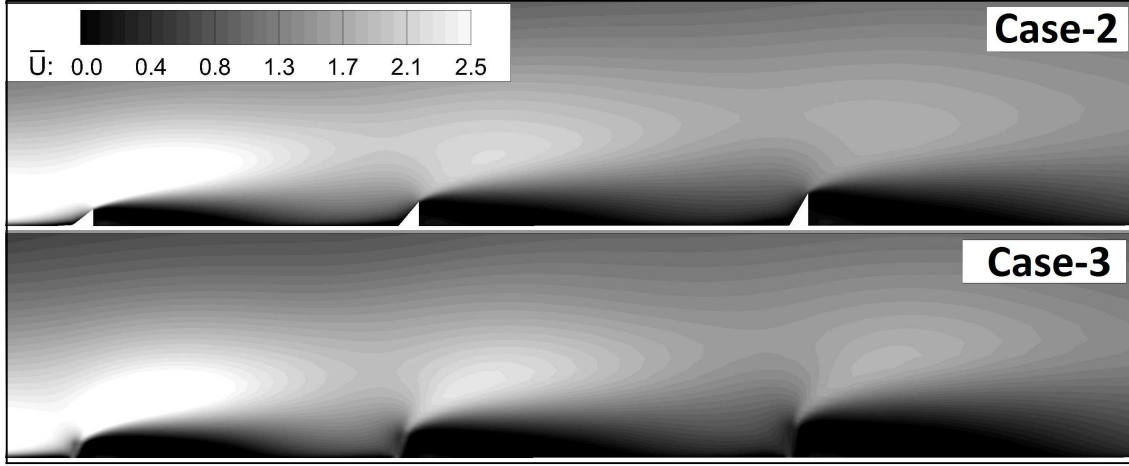


Figure 9.3: Contours of $\bar{U}$ on the plane $Y = 0$. Comparison between *Case-2* (top) and *Case-3* (bottom).

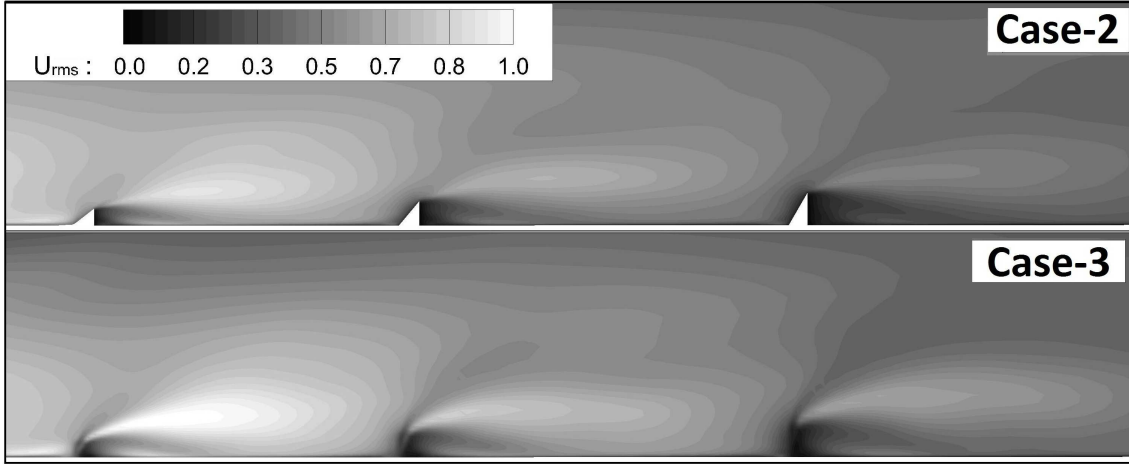


Figure 9.4: Contours of U_{rms} on the plane $Y = 0$. Comparison between *Case-2* (top) and *Case-3* (bottom).

$$Accuracy = \left(1 - \frac{|ABLWT - LES|}{ABLWT} \right) \times 100 \pm 3.85\% \quad (9.1)$$

The “designed baffle” approach has efficiently reproduced the experiment. Particularly, the accuracy of *Case-1* is $(98.1 \pm 3.85\%)$, while *Case-2* is $(86.3 \pm 3.85\%)$ accurate comparing to the experiment. It is observed that the higher the mesh density, the higher the value of $\bar{F}_D$, tending to converge to the experimental measurement. Additionally, the analytical force distribution is similar for the two mesh

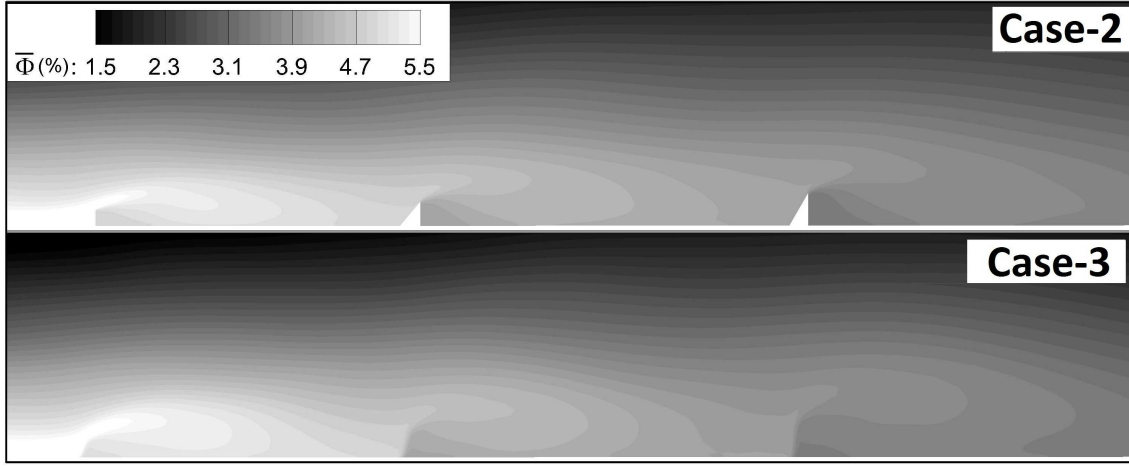


Figure 9.5: Contours $\bar{\Phi}$ on the plane $Y = 0$. Comparison between *Case-2* (top) and *Case-3* (bottom).

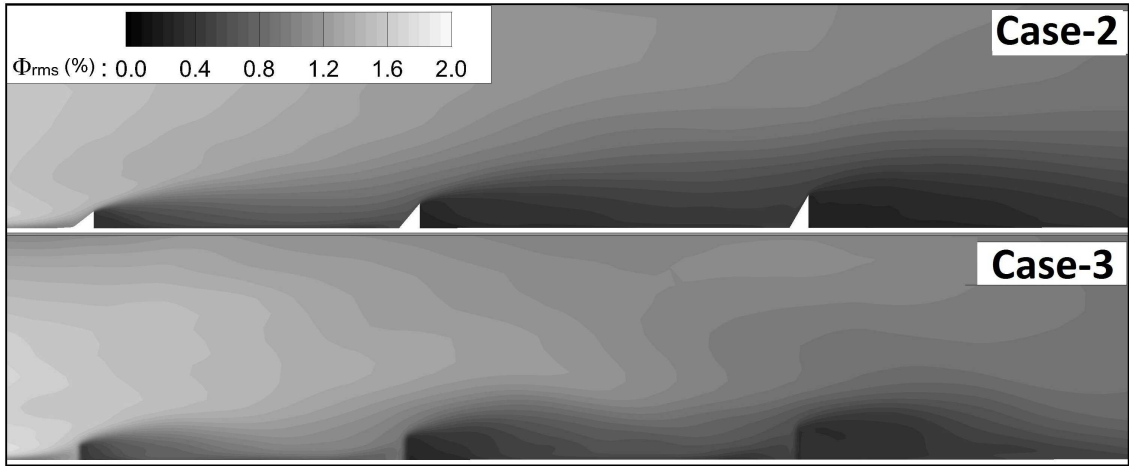


Figure 9.6: Contours of Φ_{rms} on the plane $Y = 0$. Comparison between *Case-2* (top) and *Case-3* (bottom).

densities. To continue with table 9.2, *Case-3* has slightly over-predicted the drag force. This over-prediction is justified by the limited resolution of the mesh in V_C (Fig.9.2), especially at the top side of the volume which defines the height of the baffle, extensively, of the frontal surface area of V_C . In other words, the higher the resolution of the mesh within V_C , the smaller the deviation of V_C from V_{C_0} . Based on the current results, the accuracy of *Case-3* on the prediction of $\bar{F}_D$ is $(96.2 \pm 3.85\%)$. The distribution of the force to the three individual rows of baffles is similar to *Case-1* and *Case-2*.

In order to give a better insight at the results of *DRAMBI*, the investigation is

extended on statistical quantities of the region affected by the model. (Fig.9.3) illustrates contours of mean streamwise velocity $\bar{U}$ on the vertical symmetry plane which lies on the center of the computational domain $Y = 0$. The results of *Case-2* are compared with the results of *Case-3*. Following the same format, figures (9.4), (9.5) and (9.6) illustrate the *RMS* streamwise velocity U_{rms} , the mean passive scalar concentration $\bar{\Phi}$, and the *RMS* passive scalar concentration Φ_{rms} respectively.

Observing the $\bar{U}$ contours in (Fig.9.3), it is observed that the modelled baffle representation method predicts all three separations corresponding to the three rows of baffles. The turbulent velocity field in (Fig.9.4), reveals increased turbulent activity on the wakes of the baffles, as predicted by *Case-3*. In fact, the increased level of U_{rms} at the wake of a baffle, reflects the unsteadiness of the separation edge, which in the case of *DRAMBI* is not robustly adjusted in space, but it is flexible to transiently oscillate according to the instantaneous flow conditions. However, it is essential to examine whether this feature of the model has major effects on passive scalar mixing. The contours of $\bar{\Phi}$ in (Fig.9.5) show that the mean scalar field has not been affected. There is only a slight homogeneous decrease of $\bar{\Phi}$ with downstream distance as predicted by *Case-3*. This is attributed to the fact not taking into account the radii of the baffle arrays (Fig.6.5), but instead assuming that the baffles are organized in straight rows (Fig.9.2). The effect of the baffle radius on $\bar{\Phi}$ can be observed in (Fig.9.7), where iso-surfaces of $\bar{\Phi} = 5.2\%$ are illustrated for both *Case-2*, and *Case-3*. More specifically, in the simulation of *Case-3* there is a region at the bottom of the front side of V_C where passive scalar spreads towards the lateral sides of the baffle. This is not the case for *Case-2*, where the curvature of the baffle array restricts the jet from spreading in the lateral directions. This is believed to be the reason for the slightly increased dispersion rate with downstream distance, previously observed in (Fig.9.3). Finally, the unsteadiness of the passive scalar mixing process is presented in (Fig.9.6). The contours of Φ_{RMS} show that the pollution reattaches faster according to the predictions of *Case-4*, in comparison to *Case-1*. To better examine that, the evo-

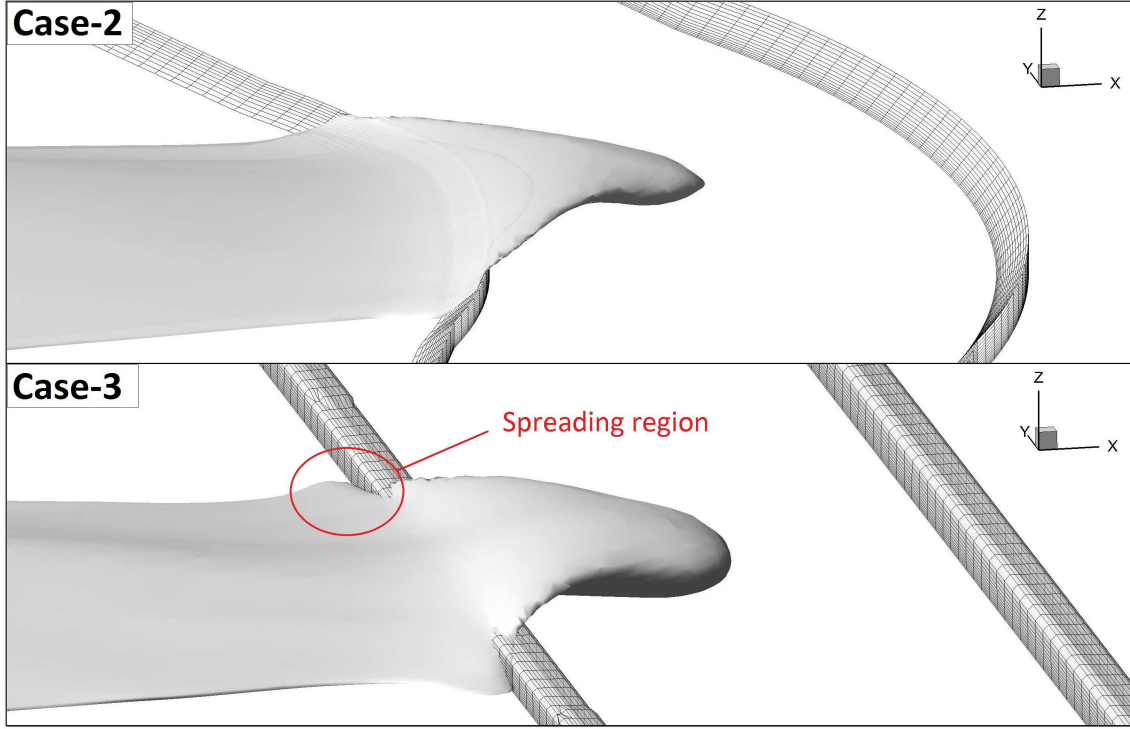


Figure 9.7: Iso-surfaces of mean passive scalar concentration $\bar{\Phi} = 5.2\%$. Comparison between *Case-2* (top), and *Case-3* (bottom).

lution of the velocity half-width¹ $z_{1/2}$ in the streamwise direction is presented in (Fig.9.8). It is seen that the curve of $z_{1/2}$ of *Case-1* (dense mesh) is permanently at higher level comparing to the curve of *Case-2* (coarse mesh). The $z_{1/2}$ curve of *Case-3*, the computational mesh of which is much coarser in the near-baffle region, develops below the one of *Case-2*. The above suggests that the coarser the mesh in the wake region, the higher the influence of the wall (Coandă effect); consequently, the faster the reattachment of the plume to the ground.

The results of baffle representation model have accurately reproduced the drag force measured in the *ABLWT*, also having significantly reduced the computational cost as presented on table 9.1. The robustness of the shape of V_A was found to have an effect on passive scalar mixing. Future improvement of the model in this respect, should include a sectionalized definition of V_{C_0} . This would enhance the effect of the geometrical function b_i on the shape of V_A , therefore, minimize its deformation due to flow unsteadiness. This modification is not

¹The velocity half-width $z_{1/2}$ is the position along a vertical profile of $\bar{U}$, at which the half peak value of $\bar{U}$ occurs.

expected to have any effect on the computational cost.

In order to minimize the lateral spreading of Φ , the curvature of the baffle array should be included at the model. This modification has been performed to the model and has been included to the description of *DRAMBI* in section 4.3, in order to to apply to the final simulations of the next chapter.

9.2 SECOND SETUP: 1/200 Scale

The validation is extended to the replication of the 1/200 scale wind tunnel experiment. The computational mesh of *Case-3* is scaled to the dimensions of the present setup, and modified in order to accommodate the cross wind simulation (Fig.6.8). The pick values of mean passive scalar concentration predicted by *Case-4* are presented on table 9.3 in comparison to the *ABLWT FID* measurements. The uncertainty of the *FID* data presented on table 9.3 is $\pm 0.2\%$ based on the repeatability of the measurements. The accuracy of the *LES* predictions, calculated by equation 9.1, is also presented on table 9.3.

Table 9.3: Maximum local concentrations. Comparison between the *ABLWT* measurements (Velikov, 2013) and *Case-4* predictions. Data at four different flow stations (X/D), as well as four different offsets from the ground (Z/D) are presented.

X/D	Z/D	Experiment $\bar{\Phi}$ (%)	Case-4 $\bar{\Phi}$ (%)	Accuracy (%)	
80.8	4.2	4.45	3.76	under-predicted	84.5
	8.3	2.96	2.96	—	100.0
114.2	4.2	2.76	3.25	over-predicted	82.3
	8.3	2.45	2.30	under-predicted	93.9
131.7	4.2	2.66	2.60	under-predicted	97.8
	8.3	2.28	2.12	under-predicted	93.0
148.3	4.2	2.19	2.41	over-predicted	90.0
	8.3	1.99	2.05	over-predicted	97.0
	12.5	1.70	1.70	—	100.0
	16.7	1.12	1.30	over-predicted	83.9

Considering the locations of the baffles, as defined by the experimental setup (Fig.6.5), we can see that the first flow station $X/D = 80.8$ lies at the wake of the first row of baffles, and the second station $X/D = 114.2$ lies at the wake of the third row of baffles. The third and fourth stations, $X/D = 131.7$ and $X/D = 148.3$ are located even more downstream, away from the balance platform.

According to the data presented on table 9.3, the predictions of *LES* are similar to the experimental measurements. The accuracy is usually above 90% except for the wake regions $X/D = 80.8$ and $X/D = 114.2$. At the wake regions, the influence of the baffles increases with decreasing the vertical distance above the ground (Z). This explains the reduced accuracy of the *LES* predictions at $Z/D = 4.2$. At the most remote location $X/D = 148.3$, the predictions close to the ground present high accuracy, while the concentration at the highest offset $Z/D = 16.7$ is slightly over-predicted. A more in depth investigation of the buoyant plume at the very farfield is presented in chapter 10.

9.3 LIFT-OFF OF THE PLUME

From the previous studies, it can be concluded that *DRAMBI* is a suitable model for the representation of the baffles in the flow field. Therefore, here it is used in order to examine the response of the selected method in greater resistance, and investigate the effect of this modification on the lift-off of the plume.

The new simulation, *Case-5*, incorporates the details of the first setup. The difference here is the size and location of the baffles. In particular, the new baffles are double in height and width, comparing to the first setup. The baffle locations had to be readjusted in order to maintain aerodynamic efficiency. We have, therefore, performed test simulations in order to ensure that the second baffle is adequately far from the first so that it is not affected by its wake; similarly for the third baffle. The new locations are $X/D = 61.6$, 107.3 and 141.7 respectively. In order to keep the baffle sizes within realistic scales, the baffles lie underneath the three degrees glide path landing slope. This is an extreme case very close to the aerodrome safety limits, and it is only tested for the sake of the investigation.

It is known that drag is directly proportional to the projected area of the obstacle opposing the flow; therefore, with this configuration we expect to measure double the drag force than in the previous setup. The averaged forces calculated by *Case-5* are given in N ($\times 10^2$), $F_D = 4.708, 2.145$ and 1.373 respectively. Summing the three forces we have a total force $F_D = 8.227$. This value is 1.86 times greater than the one measured at the first setup, adequately meeting our expectation of doubling the force. However, the distribution of the force has changed significantly, now being 57.2%, 26.1% and 16.6% for the three baffles respectively. This can be explained by the location readjustment which has brought the first baffle closer to the jet, where the plume momentum is larger, while the second and third baffles have been moved downstream, being exposed to a weaker flow.

The *ABL* is a crucial factor opposing the lift-off of the plume. Whatever the amount of momentum extracted by the plume is, the co-flowing wind is a permanent source of momentum enhancing the Coandă effect. Under these conditions it is difficult for the plume to detach from the ground. However, the evolution of $z_{1/2}$ has been widely used in the past four decades for the identification of the jet growth rate. For instance, Launder *et al.* (Launder & Rodi, 1983) reported the growth rate $dz_{1/2} / dx = 0.048$ for a *3DWJ* with $U_f = 0$. Here, $dz_{1/2} / dx$ will allow us to identify the lift-off of the plume.

In this way, a series of vertical profiles of $\bar{U}$ is extracted along the jet centre-line. From each profile, the value of $z_{1/2}$ is calculated as described in section 8.1.2. The same procedure is followed for *Case-1*, *Case-2*, *Case-3*, as well as the unresisted jet simulation. The post-processing results are illustrated on the plot of (Fig.9.8). After the end of the transition to a fully developed *3DWJ*, at approximately $X/D=40$, the unresisted jet grows linearly with a rate of $dz_{1/2} / dx = 0.055$. This value is only slightly higher than the one reported by Launder *et al.* for a non-buoyant jet, however, this is justified by the co-flowing wind of this investigated setup. The same pattern is observed at the near-field of *Case-1* and *Case-2*. However the appearance of the baffles causes the position of $z_{1/2}$ to elevate locally. The momentum reduction induced by the baffles allows the plume to spread faster; however, it is not enough in order to interrupt the wall-jet pattern. Even-

tually, after the end of the baffle region, $dz_{1/2}/dx$ rapidly decreases, tending to converge on the results of the unresisted jet. However, this is not the case for *Case-5*. In this case, the growth rate increases rapidly with the appearance of the baffles, finally converging to the value $dz_{1/2}/dx = 0.17$. This value remains after the end of the baffle region, all the way to the end of the computational domain, meaning that buoyancy has become dominant and the plume has stopped propagating as a *3DWJ*. In other words, the setup of the baffles selected for *Case-5*, which is an extreme case of our methodology, has managed to achieve lift-off of the plume on a realistic environment, encouraging in this way the continuation of this investigation.

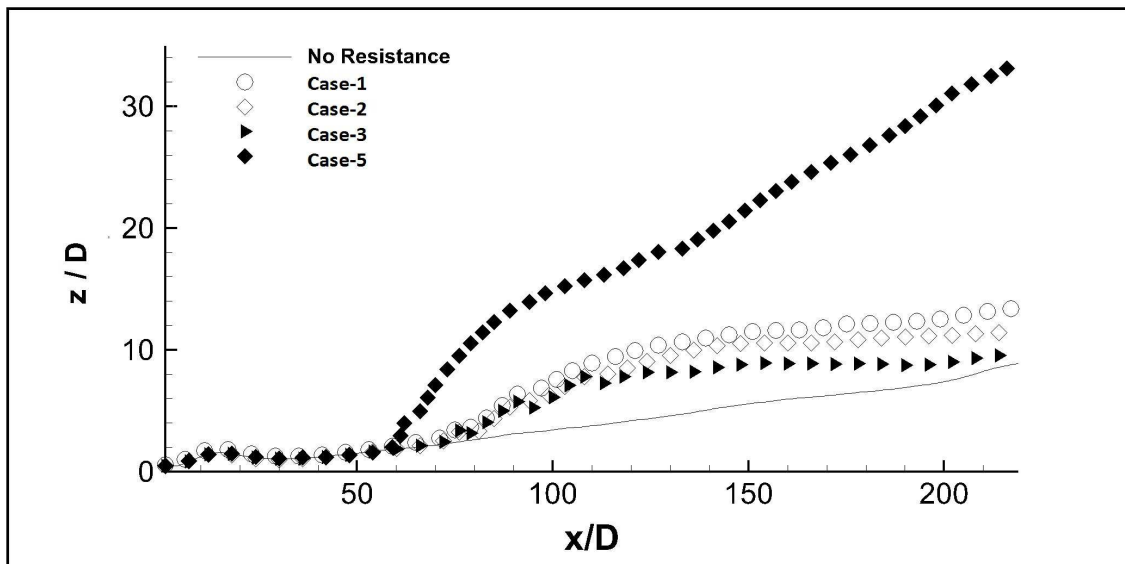


Figure 9.8: Streamwise evolution of the velocity half-width $z_{1/2}$. Results of *Case-1*, *Case-2*, *Case-3*, *Case-5*, as well as the unresisted jet, are included.

Chapter 10

Results and Analysis

The main objective of this thesis is the numerical investigation of the hot *3DOJ* development in the presence of a wind-brake system, an array of baffles suitably arranged at the path of the jet, which aims to accelerate the natural lift-off process. Due to the high complexity of the problem, progressive parametric validations have been initially performed by gradually increasing the problem complexity. In this way, the analyses presented in previous chapters 7,8, and 9, have provided the required experience, also raised the authors' confidence to setup and perform the final simulations with high fidelity. The material of this chapter is subdivided in four sections. Section 10.1 is dedicated to necessary simulation validity tests as well as two additional tests, one for the effect of buoyancy to the numerical solution, and one for the spectral analysis of velocity time series aiming to assess the *LES* filtering width. In the following, section 10.2 includes a parametric testing of the baffle arrangement supported by extensive cross-validations between the *LES* solutions and the experimental data obtained at the *ABLWT*. Then, section 10.3 extends the cross-validation studies of section 10.2, and two additional numerical tests are performed without the explicit support of experimental data. Finally, section 10.4 provides comparisons of the designed baffle approach and *DRAMBI*. A summary of the final simulations history and setup is highlighted on table 10.1.

Table 10.1: Summary of the setup and history of the final simulations.

Case Name	Scaling of the Field Trials Setup	Experimental Data	Baffle Setup		Buoyancy	Domain Length (X/D)	Co-Flow	Timestep Size (E-04) (s)	Mesh Density (E+06)	Simulated Time (s)		
			Baffle Setup	Baffle Representation						Establ.	Averaging	Total
Case 1	1/100	NO		NO	NO	220	original	3	2.2	4.8	35.1	39.9
Case 2	1/100	NO		NO	NO	220	original	2.5	5.0	3.8	16.5	20.3
Case 3	1/100	NO		NO	NO	220	original	2.2	10.0	3.3	7.0	10.3
Case 4	1/100	NO		NO	YES	220	original	3.2	2.2	5.0	15.0	20.0
Case 5	1/200	$\bar{u}, \bar{\phi}$		NO	YES	380	original	2.5	~ 6	9.0	12.9	21.9
Case 6	1/200	$\bar{u}, \bar{\phi}$	ORIGINAL	DESIGNED	YES	380	original	2.5	~ 7	8.0	22.8	30.8
Case 7	1/200		ORIGINAL	DRAMBI	YES	380	original	2.5	~ 7	3.8	12.4	16.2
Case 8	1/200	$\bar{u}, \bar{\phi}$	LARGE	DESIGNED	YES	380	original	2.5	~ 7	5.5	13.0	18.5
Case 9	1/200		LARGE	DRAMBI	YES	380	original	2.5	~ 7	6.5	18.5	25.0
Case 10	1/200	NO	THRESHOLD	DESIGNED	YES	380	original	2.5	~ 7	3.8	14.9	18.7
Case 11	1/200	NO	THRESHOLD	DESIGNED	YES	380	half	2.5	~ 7	7.8	31.2	39.0
Case 12	1/200	$\bar{u}, \bar{\phi}$	THRESHOLD LARGE	DESIGNED	YES	380	original	2.5	~ 7	3.8	12.4	16.2
Case 13	1/200		THRESHOLD LARGE	DRAMBI	YES	380	original	2.5	~ 7	5.0	12.5	17.5
Case 14	1/200	NO	ADDITIONAL ARRAYS	DESIGNED	YES	380	original	2.5	~ 8	5.0	20.0	25.0

10.1 SIMULATION VALIDITY TESTS

This section discusses the results of four validity tests preceding the final simulations of the hot jet plume under the baffle influence. Cases 1-4 have been performed on a scale 1/100 of the field-trials in the absence of the baffle-induced resistance, following the setup details of section 6.2.1. The final setup was not available during the implementation of the present validity tests, therefore, no experimental data are available for cases 1-4. However, the objectives of the four validity tests presented are fulfilled in the absence of experimental data.

10.1.1 Temporal Convergence Study

A theoretical approach to estimate the simulated time requirement STR is presented in section 5.3.2. The high importance of the co-flow (wind) velocity magnitude is highlighted therein. The co-flow velocity of this work is represented by the 1/7 wind profile power law relationship of equation (6.1), with control height $Z_r/D = 6.67$. Only for the sake of the present theoretical estimations, and since the selected value of the control height is an approximate average of the vertical velocity half-width $z_{1/2}$ in the investigated range $FLI = 200D$ (Fig.5.4, left), it can be assumed that U_r represents the co-flow velocity U_f throughout the computational domain. Additionally, the theoretical estimations of STR in section 5.3.2 are made upon an average growth rate of the vertical and horizontal scales of the $3DWJ$, in order to have a direct comparison with the axisymmetric jet. However, here we calculate two different values of STR , one based on the vertical velocity half-width $dz_{1/2}/dx = 0.048$ and one based to the horizontal one $dy_{1/2}/dx = 0.26$, values recommended by (Launder & Rodi, 1981).

The procedure described in section 5.3.2 is repeated using the details of Case 1 (refer to table 10.1 and section 6.2.1), also considering the discussion above. The estimated establishment time for this case, considering $FLI = 200D$, is $T_0a_0 = 5.2s$, and the two different estimations of the averaging time (collection of sample for 1st order statistics) are: $T_{11}a_1 = 3.14s$ (based on $dz_{1/2}/dx = 0.048$) and $T_{11}a_1 = 17.0s$ (based on $dy_{1/2}/dx = 0.26$).

The actual flow establishment time of simulation Case 1 was selected $4.8s$, (refer to table 10.1), according to the theoretical prediction. After this point of the simulation the averaging process started and four intermediate averaged solutions were captured. The results have been analysed using the advanced post-processing method (refer to section 8.1.2). The results compared in figures (D.1.a-h) represent the values of $T_{11a_1} = 8.1s, 17.2s, 25.4s$, and $35.1s$. Apart from the results of $T_{11a_1} = 8.1s$ which appear undergoing considerable changes with extending the averaging time, the results of the other three stops present an overall agreement (temporal convergence), with only the vertical concentration half-width in (Fig.D.1.g), varying close to the exit plane. However, this is attributed to the outflow boundary affecting mixing at the proximate flow (the domain size in this case was $220D$).

In conclusion, it can be claimed that the *STR* estimations based on the theoretical model of section 5.3.2, are reliable, therefore, appropriate to manage the time intervals of the simulations of this study. It has to be noted that due to high traffic of the University supercomputer-Astral, in which the simulations were run, it was difficult to control the time intervals of the simulation with accuracy. Analytical details of the simulated time are included on table 10.1 for all the simulations of this chapter.

10.1.2 Grid Convergence Study

In the second validity test the consistency of the *LES* predictions with varying the mesh density is investigated. In particular, the computational mesh of Case 1 has undergone two successive isotropic refinements. The results of Cases 1, 2, and 3, corresponding to the mesh densities of 2.2×10^6 , 5×10^6 , and 10×10^6 cells, are compared in terms of the advanced post-processing method output. In particular, plots of the streamwise evolution of the mean flow characteristics are illustrated in figures (D.2.a-h). An overall agreement is observed in terms of all eight quantities compared. It is, therefore, suggested that the mesh density is adequately large to avoid substantial numerical dissipation, or effects on the turbulent mixing mechanisms.

The streamwise extent of the computational domain of the simulations of this test is limited at $X/D = 220$. However, the final mesh, upon which Cases 5-14 are performed, extends further downstream, at $X/D = 380$. The density of the latter was selected within the investigated range of this study, and its value around 7×10^6 cells. Non-significant variations around this value (refer to table 10.1) depend on the baffle arrangement installed.

10.1.3 Incorporation of Buoyancy to the Numerical Simulation

Buoyancy has been incorporated to the simulations reported in chapter 9. It has been achieved through different temperatures of the jet and ambient flows in the boundary conditions. However, the studies therein did not explicitly test the effect of buoyancy. In this section the effect of buoyancy is parametrically observed by comparing the results of a cold (Case 1) and a hot (Case 4) *3DOJ*. Selection of the values of the jet and ambient flow temperatures are referred in section 6.2.

The results of this study are being analysed using the advanced post-processing method and are plotted in figures (D.3.a-h). In particular, figures (D.3.a) and (D.3.e) present the streamwise evolution of U_m and Φ_m respectively. Both quantities decay faster for the hot jet comparing to the cold one, with more intense the difference of the decay of Φ_m . The faster decays observed at the hot jet are attributed to buoyant fluxes preferring moving upwards; therefore, resisting to contribute to the formation of U_m and Φ_m . The effects of this “*preferred*” motion are reflected to the vertical velocity half-width $z_{1/2}$ shown in (Fig.D.3.c), being larger for the hot jet, as well as to the horizontal one $y_{1/2}$ shown in (Fig.D.3.d) which is smaller for the hot jet. Similarly, the streamwise evolution of the concentration half-widths $z_{1/2\Phi}$ and $y_{1/2\Phi}$ are illustrated in figures (D.3.g) and (D.3.h) respectively. Although $y_{1/2\Phi}$ is significantly affected by buoyancy, similar to $y_{1/2}$, the situation is different for $z_{1/2\Phi}$, which is identical in both the cold and hot jets.

To understand the difference between $y_{1/2}$ and $y_{1/2\Phi}$ we need to realize the physics involved at the external mixing layer. Fluid exchange is a common process, responsible for growing of mixing layers. When a bucket of fluid, part of the Kelvin-Helmholtz-like eddy, is travelling outwards and is hot comparing to the sur-

rounding flow, it is more likely to get away from the circulation field and continue its buoyant trajectory upwards. In order for continuity to be satisfied, a bucket of ambient fluid of the same volume to the one escaped, has to replace it. Under this mechanism the momentum content of the jet is not affected compared to the cold jet, unlike the passive scalar which is gradually reduced, with immediate effect to the upper side of the external mixing layer. This statement is validated by the vertical profiles of the hot unresisted jet (Case 5) presented in (Fig.E.1), where the vertical profiles of $\bar{U}$ (left) and $\bar{\Phi}$ (right) of Case 5 are plotted. The velocity profiles indicate a vertical jet expand $Z/D < 20$, while the concentration profiles reveal the presence of Φ up to $Z/D \approx 23$ in the most distant regions.

In conclusion, it has been observed that density variation between the jet and the ambient flow, which here is formed due to different temperatures specified at the domain inlets, has caused distinguishable changes to the unresisted $3DWJ$ mean flow characteristics. It is, therefore, expected that inclusion of the baffle arrays to the path of development of the jet will magnify these effects due to both the reduced streamwise momentum and the subsequently reduced wall forces (less intense Coandă effect) achieved by the baffles. However, due to limited computational resources, the buoyancy effect study has not been extended to the deflected jet simulations, but they have only been implemented with the hot jet.

10.1.4 Spectral Analysis of Velocity Time Series

The fourth, and final test of this section, is to perform spectral analysis to the turbulence velocity signals at various positions of the free mixing layer of the *3DOJ*. The aim is to identify the filtering position within the turbulent spectrum, at different farfield locations.

Numerical probes have been set at $X/D = 3.5, 6, 15, 30, 100$, and 300 , at approximately the centre of the upper (free shear) mixing layer, collecting the instantaneous velocities for every timestep. The six signals have been analysed using the built-in fast-Fourier-transform *FFT* algorithm of Tecplot 360, and the calculated energy spectra $E_{(k)}$ are plotted against wavenumber k in figures (10.1.a-f) respectively.

(Fig.10.1.a) presents the energy spectra of the signal collected at $X/D = 3.5$. At this location, the Kelvin-Helmholtz rings have developed and are on the stage undergoing helical pairing (refer to section 1.4 for theory). As stressed by (Pope, 2000), this is a challenging region to define the filtering length due to the fact that the “*long-wave*” and the “*dissipative*” intervals of the turbulent spectrum intersect. The inertial range, where isotropic turbulence exists, and is an appropriate width to perform *LES* filtering, is identified by a segment of the spectrum parallel to $k^{-5/3}$. It is not difficult to observe in (Fig.10.1.a) that this slope has not formed yet. Similar is the energy spectrum at $X/D = 6$ presented in (Fig.10.1.b). Both $X/D = 3.5$ and $X/D = 6$ lie at the developing jet region. Moving downstream, (Fig.10.1.c) refers to $X/D = 15$. At this station, jet transition to turbulence is complete and the inertial range has formed a clear $k^{-5/3}$ slope. The position along k where the large oscillations begin represents the filtering window width. The filter width (cell size) lies inside the inertial range at $X/D = 15$. Moving further downstream, at $X/D = 30, 100$, and 300 , illustrated in figures (10.1.d-f), the filtering width appears getting gradually deeper in the inertial and the dissipative range, ensuring that *SGS* modelling assumptions are increasingly acceptable moving towards the domain exit.

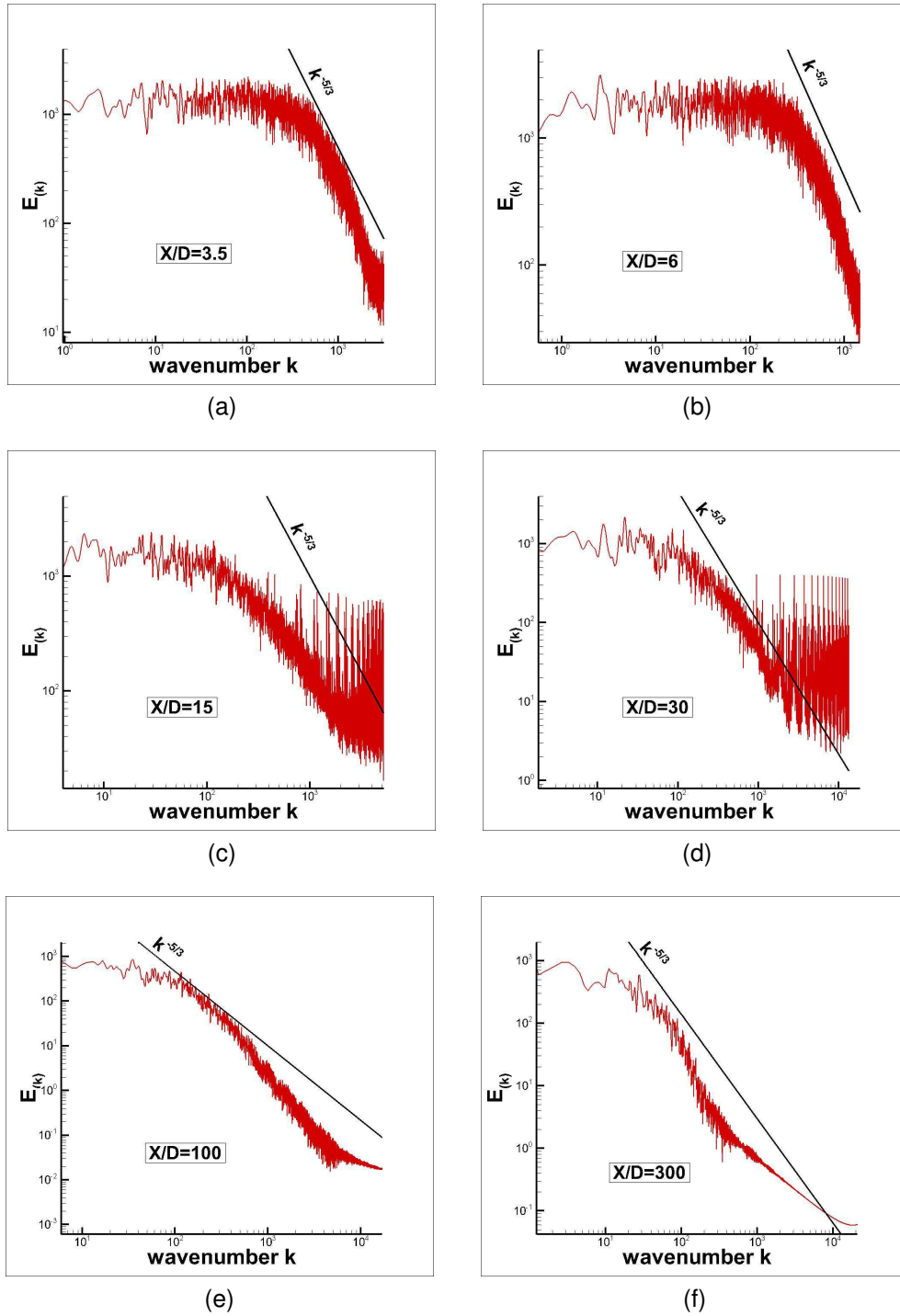


Figure 10.1: Velocity spectra of time-series extracted within the top side of the jet mixing layer, at six streamwise locations in the range $3.5 \leq X/D \leq 300$. Data are collected from simulation (Case 5).

10.2 CROSS-VALIDATIONS

Extensive cross-validations of the *LES* results and experimental measurements are essential to enhance understanding of the geometrical characteristics of the wind-brake system. A close cooperation of the authors with a separate experimental group, applying relevant research in the atmospheric boundary layer wind tunnel (*ABLWT*) of Cranfield University (Velikov, 2013), has lead to the formation of the final experimental setup, which is analytically described in section 6.2.3. The results of the final simulations replicating the wind tunnel experiments are presented and analysed in this section.

The cross validations of this section analysis involve mean streamwise velocity $\overline{U}$ and mean passive scalar concentration $\overline{\Phi}$, plotted in vertical profiles at various farfield locations, in two dimensional contours at specific farfield cross-sections, and in plots of streamwise evolution of mean flow characteristics obtained by the advanced post-processing method (plots are included in Appendix D, refer to section 8.1.2 for details about this method). The computational domain of the final simulations has been extended further downstream ($X/D = 380$), in order to observe the plume at regions away from the direct influence of the baffles. The streamwise extent of the experimental measurements, however, is restricted at $X/D = 200$ due to experimental limitations.

10.2.1 Unresisted Hot 3DOJ

Prior to installation of the baffles in the *ABLWT*, experimental measurements of $\overline{U}$ at $X/D = 90, 90, 90, 150$, and 170 and of $\overline{\Phi}$ at $X/D = 90, 110, 130, 150, 170$ and 200 have been collected for the unresisted *3DOJ*. The experimental setup has been numerically replicated by Case 5. Comparison of the experimental measurements (*ABLWT*) and the numerical calculations (*LES*), in terms of vertical profiles along the symmetry plane, are plotted in (Fig.E.1). $\overline{U}$ profiles are plotted on the left plot and $\overline{\Phi}$ on the right plot. *LES* is observed to have significantly under-predicted $\overline{U}$ in all five flow stations examined. It is, however, important to note that there is a considerable disagreement of the free stream velocity magni-

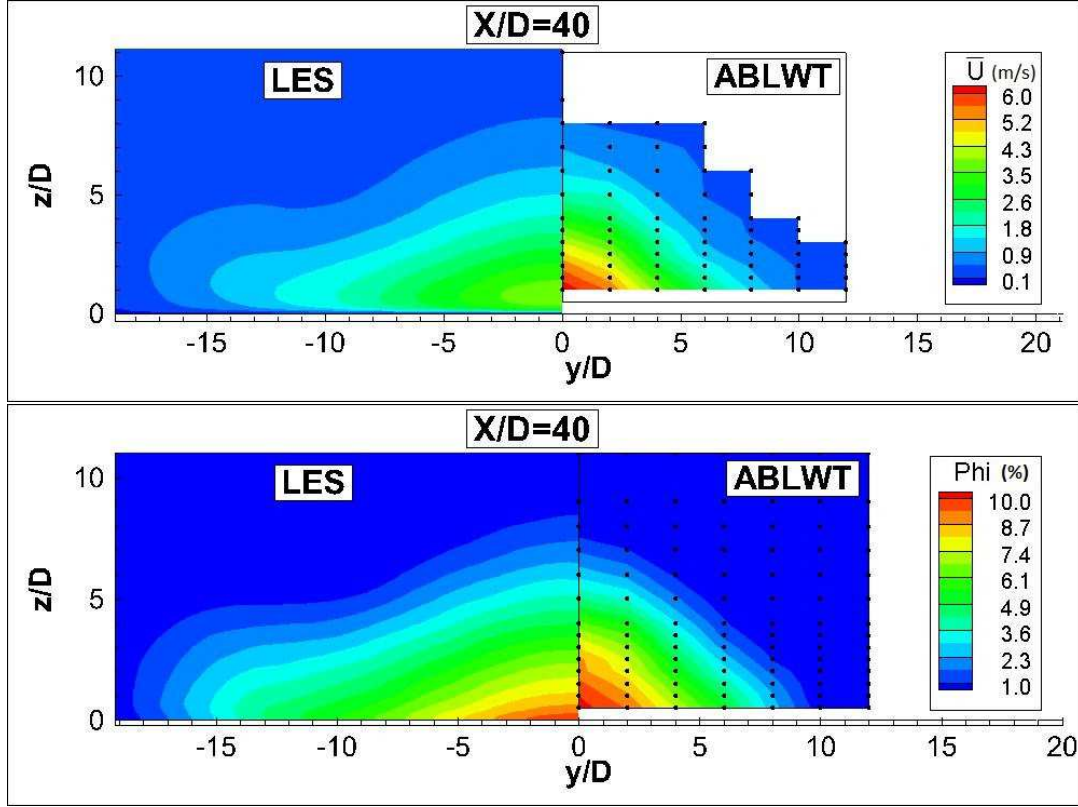


Figure 10.2: Contours of mean streamwise velocity $\bar{U}$ (top), and mean concentration $\bar{\Phi}$ (bottom) at the cross-section $X/D = 40$ of the “UNRESISTED” 3DOJ. Case 5 results are illustrated on the left and the measurements of the ABLWT experiment are on the right of the figure. The scattered points on the right indicate the grid of measurements in the wind tunnel.

tude at the highest offset $Z/D = 22$, approximately 25% between experiment and *LES*. The experimental co-flow speed is higher than the one of the simulation.

Cross-sectional contours of $\bar{U}$ are presented at the top of figures (10.2) and (10.3), corresponding to stations $X/D = 40$ and $X/D = 170$ respectively. In both stations the lateral spread of the jet is over-predicted by *LES*. At $X/D = 170$ it is observed that the velocity field surrounding the jet was stronger in the experiment than the one in the simulation. As shown in the theoretical analysis of section 5, as well as the relevant validation test in section 8.2.11, co-flow velocity is a crucial parameter for the 3DOJ development. In particular, lower value of co-flow results in faster transition to turbulence, earlier jet reattachment and increased growth rates of the jet.

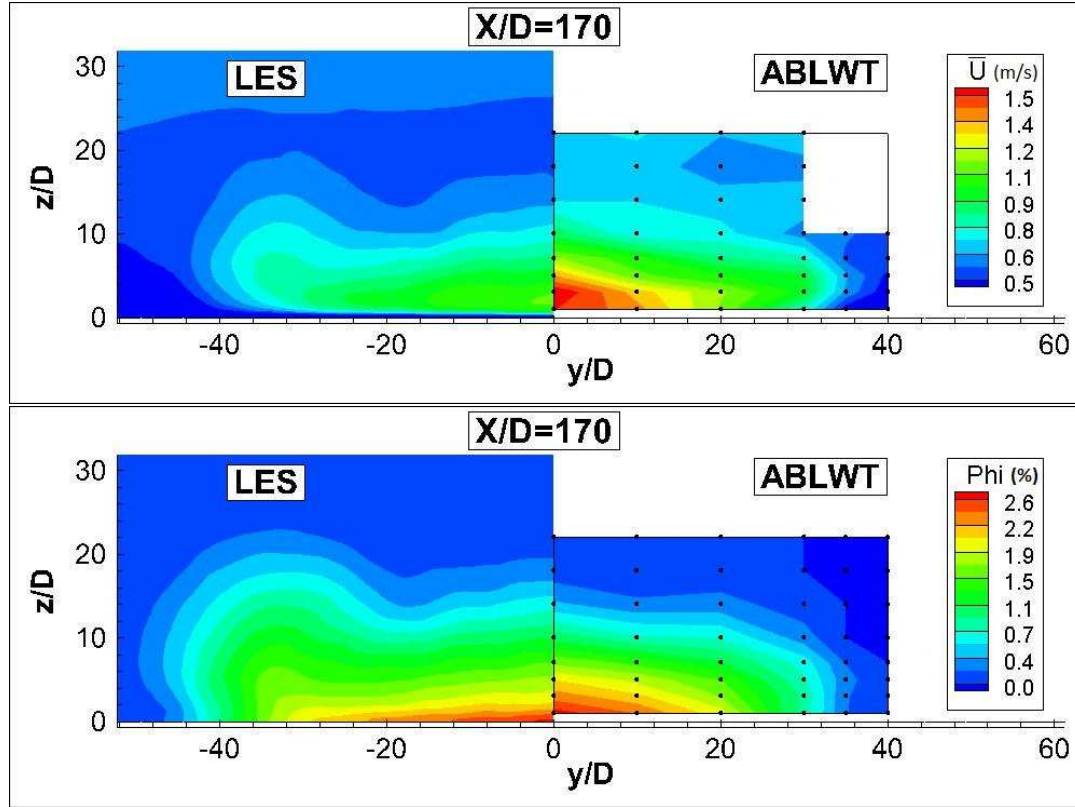


Figure 10.3: Repetition of figure 10.2 at $X/D = 170$.

The problem in question is the cause of the disagreement of the co-flow velocity magnitude between *LES* and experiment, although the experimental setup is accurately reproduced in the inlet boundary of the simulation. The velocity contours on the horizontal plane illustrated in (Fig.10.7) of section 10.3.1, are borrowed in order to help answering this question. Although, wind brake system is included to the numerical simulations analysed in this figure, the free stream flow should be identical to the setup which is discussed here. The key observation is that although the horizontal plane is located at constant offset from the ground $Z/D = 2$, the co-flow velocity varies along the streamwise direction X . This is a clear indicator that the velocity profile, set at the inlet boundary, continues to develop downstream causing the velocity to continuously drop at the examined offset. To reduce this effect, a more detailed description of the nature of the boundary layer in the wind tunnel is required. Then the co-flow velocity unsteadiness recorded in the wind tunnel should be reproduced by an unsteady inlet boundary in *LES*.

On the other hand, the vertical concentration profiles presented in (Fig.E.1, right) are in close agreement between *LES* and experiment. There is an approximately constant 10% under-prediction of the ground concentration by *LES*, but the fact that the position of $\Phi_{0.5}$ is almost identical between *LES* and experiment (Fig.D.4.g) tells that this under-prediction is not related to numerical dissipation, but to earlier transitions of the *3DOJ*. This can be validated by the cross-sectional contours of $\bar{\Phi}$ at the bottom of figures (10.2) and (10.3), corresponding to stations $X/D = 40$ and $X/D = 170$ respectively. Although the lateral distribution of $\bar{\Phi}$ at $X/D = 40$ (Fig.10.2) differs significantly between *LES* and experiment, it meets greater agreement at $X/D = 170$ (Fig.10.3), complying with overall observations of the validation tests, stressing that transitional errors are recovered downstream.

With respect to the streamwise evolution of U_m illustrated in (Fig.D.4.a), there is a sudden drop observed at $X/D \approx 200$. This is attributed to early initiation of the sample collection process. Specifically, table 10.1 reports that the establishment time of the Case 5 is $9s$. Although this is the longest flow establishment time from the cases reported, the initial conditions of this case were formed by a streamwise sweep of the co-flow boundary condition. This means that the jet had to naturally develop in a low speed environment, all the way to the exit, which has possibly increased $T_0 a_0$ of equation (5.5). On later simulations, Cases 6-14, the volume solution variables of Case 5 have been interpolated as initial conditions.

10.2.2 Original Baffle Arrangement

The analysis continues with the installation of the “*ORIGINAL*” baffle arrangement (refer to table 6.2 for details). Experimental measurements are collected in a similar way as for the unresisted jet. The experimental setup has been numerically replicated by Case 6. Comparison of the vertical profiles along the symmetry plane are plotted in (Fig.E.2). The numerical under-predictions of $\bar{U}$ and $\bar{\Phi}$ follow similar trends to the unresisted jet. However, the local deflection of the plume in the baffle region $50 \leq X/D \leq 81.7$ has affected the shape of the vertical profiles of both quantities, which appear to undergo a new transition. The experimen-

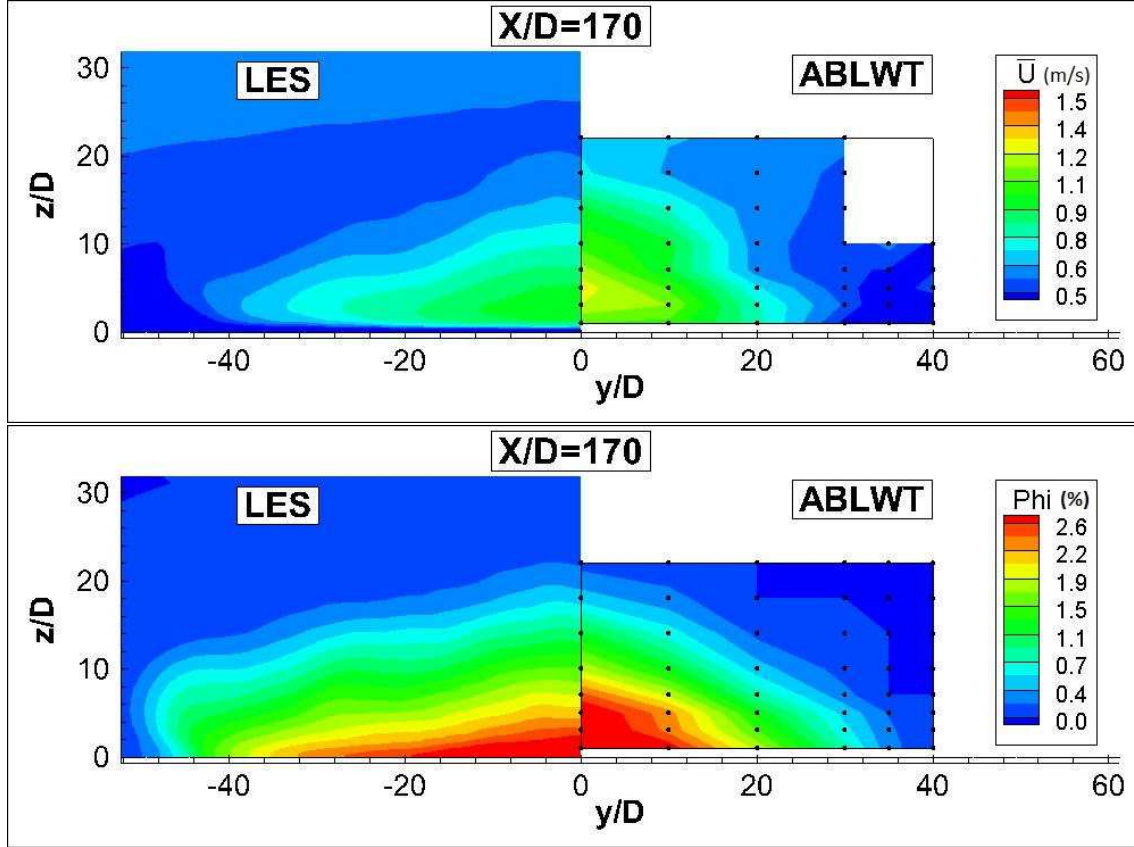


Figure 10.4: Contours of mean streamwise velocity $\bar{U}$ (top), and mean concentration $\bar{\Phi}$ (bottom) at the cross-section $X/D = 170$ of the “ORIGINAL” baffle setup. Case 6 results are illustrated on the left and the measurements of the *ABLWT* experiment are on the right of the figure.

tal profiles of $\bar{\Phi}$ appear less developed than the numerical ones, which again is attributed to the co-flow velocity disagreement.

Quantitative plots of the streamwise evolution of the mean flow characteristics, including comparisons with the experimental data where available, are illustrated in figures (D.5.a-h). The streamwise evolution of Φ_m (Fig.D.5.e) is again in remarkable agreement between *LES* and experiment. Additionally, the co-flow effect on the jet scales is reflected on the vertical concentration half-width $z_{1/2\Phi}$, shown in (Fig.D.5.g).

Comparison of the *LES* results and the experimental measurements in the form of cross-sectional contours of $\bar{U}$ and $\bar{\Phi}$ are available in (Fig.10.4), corresponding to station $X/D = 170$. Taking into account the lower co-flow speed present in the simulation and its subsequent effects to the jet development dis-

cussed above, the comparison of (Fig.10.4) reveals an overall agreement between numerical predictions and experimental measurements, therefore, enhancing the validity of present *LES*. The effect of the installation of the baffles is separately discussed, in terms of the streamwise evolution of the jet mean flow characteristics, in section 10.3.1.

10.2.3 Large Baffle Arrangement

The “*ORIGINAL*” baffle arrangement is substituted by its “*LARGE*” counterpart (refer to table 6.3 for details) and the analysis of the previous section is repeated. The experimental measurements are cross-validated with the *LES* results of Case 8. In this baffle arrangement, not only the vertical deflection of the plume is larger due to the larger baffles, but also the streamwise extent of the detached jet region is larger due to the larger streamwise extent of the arrays ($50 \leq X/D \leq 134.5$). For this reason, it can be claimed that turbulence in this case is dominated by the free-shear jet dynamics, which means that the wall contribution is considerably reduced.

Comparison of the vertical profiles of $\bar{U}$ at $X/D = 170$ and of $\bar{\Phi}$ at $X/D = 170$, and 200 are plotted in (Fig.E.5). It is observed that the reduction of the wall-contribution to the jet development has resulted to an increased agreement of the presented curves. Particularly, it is observed that the two velocity profiles of (Fig.E.4), “experiment versus *LES*” have closely similar shapes and are separated by an approximate constant deviation. This deviation is the response of the previously discussed co-flow velocity difference between *LES* and experiment.

Highly informative are the cross-sectional comparisons at $X/D = 170$ (Fig.10.5). Particularly, the velocity contours on the top image indicate that the recirculation bubble of the third baffle array is still present at $X/D = 170$. The finite width of the baffle arrays allow for small portions of the plume to by-pass the arrays from the lateral sides. This mechanism can be observed at the horizontal planes of figures (F.3) and (F.4). The results of this flow feature are reflected at the velocity contours of (Fig.10.5) through the attached portion lying on the lateral sides of the cross-section. This attached portion appears to be larger for the *LES*

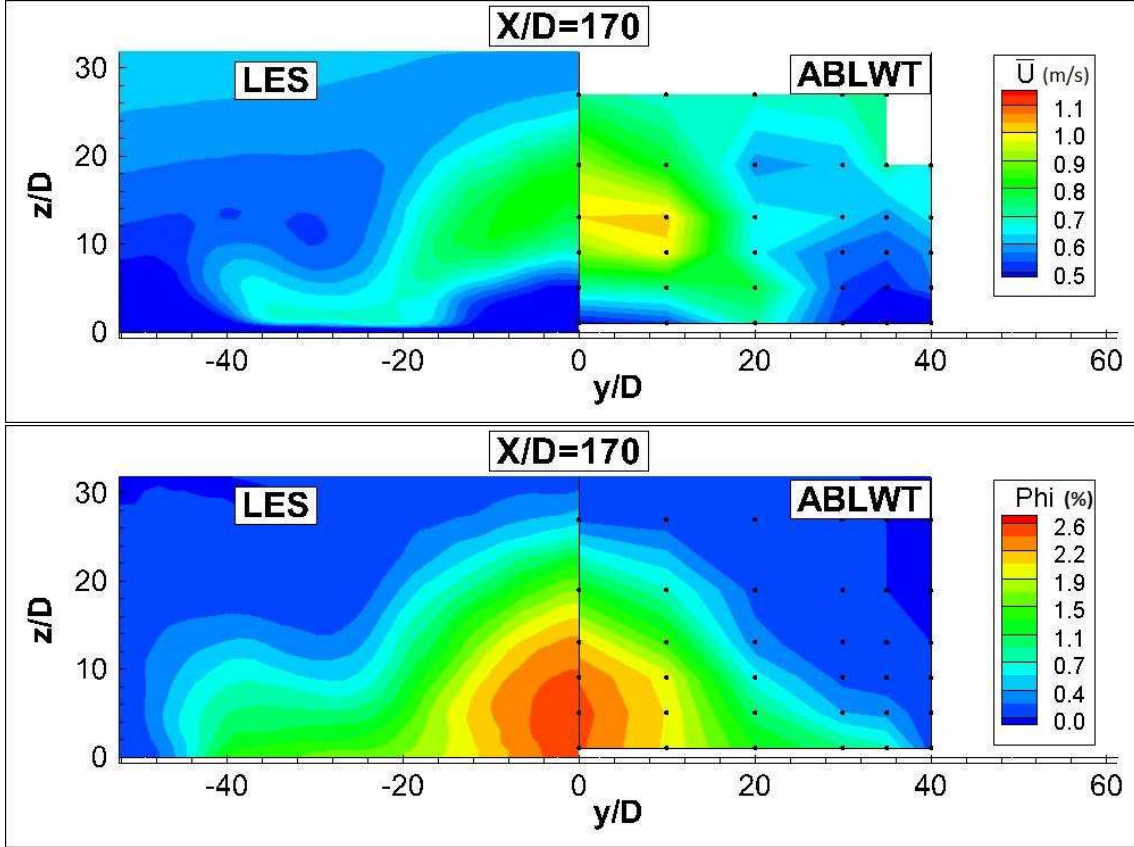


Figure 10.5: Repetition of figure 10.4 using Case 8 results “*LARGE*” baffle setup.

solution than the one measured in the experiment, something that is attributed to the slightly faster initial growth of the simulated jet which means that larger by-passed fluxes occurred in *LES* than in the experiment. The bottom image of (Fig.10.5) illustrates contours of mean concentration $\bar{\Phi}$ at the same cross-section. The agreement between *LES* and experiment is high for this variable, not only in the vertical direction, but in the lateral direction as well. Minor disagreements are attributed to the same two reasons referred above, being the deviation of U_f and the finite width of the baffle arrays.

Finally, improved cross-validations are observed on the streamwise evolution of the mean flow characteristics presented in figures (D.6.a-h). Specifically, focussing in comparisons between *LES* (designed baffles) and experiment, we observe a complete recovery of U_m at $X/D = 170$ (Fig.D.6.a), an overall agreement of Φ_m (Fig.D.6.e), and a significantly improved agreement of $z_{1/2\Phi}$ (Fig.D.6.g) comparing to previous setups.

10.2.4 Threshold Large Baffle Arrangement

An alternative approach to remove momentum from the plume has been put under investigation. According to this approach the multiple baffle arrays, so far located at the farfield, are replaced by a single baffle array located at the runway threshold¹, therefore, closer to the jet source. In this way, the “*LARGE-THRESHOLD*” baffle arrangement (refer to table 6.4 for details) has been installed to the wind tunnel surface, and a new set of measurements has been collected. The measurements are cross-validated with the *LES* results of Case 12 (designed-baffles).

A universal view of the *LES* flowfield can be obtained via two-dimensional contours of U , $\bar{U}$, Φ , and $\bar{\Phi}$, plotted at the vertical and horizontal planes in figures (F.5) and (F.6). In the experiment, the lateral extent of the baffles was sufficiently large $30D$ (refer to table 6.4). However, the conical shape of the *LES* computational domain imposes a restriction to the representation of this parameter. More specifically, in order to reduce interactions with the inlet plane, the lateral extent of the baffle array in the simulation was set $21.5D$, however, it was found insufficient, consequently, large fluxes are observed to by-pass the baffle array from the lateral sides.

Comparisons of the vertical profiles of $\bar{U}$ at $X/D = 90, 150$, and 170 and of $\bar{\Phi}$ at $X/D = 90, 150, 170$ and 200 are plotted in (Fig.E.4). The agreement of the numerical predictions with the experimental data in terms of $\bar{U}$ (left plot) is low at the early stations, and appears to improve with downstream distance. Regarding the $\bar{\Phi}$ profiles (right plot), the ground concentrations are in agreement, however, the vertical distribution suggests greater vertical expansion of the *LES* jet. To better understand these disagreements, (Fig.10.6) illustrates cross-sectional comparisons of $\bar{U}$ and $\bar{\Phi}$ at $X/D = 170$. The velocity contours indicate that the by-passed fluxes have not yet converged to a unique jet in this station, while the concentration contours disagree on the horizontal to vertical expand ratio of the local jet cross-section.

¹the array of baffles has to be retractable because the airport regulations do not allow any obstacle in such a close proximity to the runway (CAA, 2011).

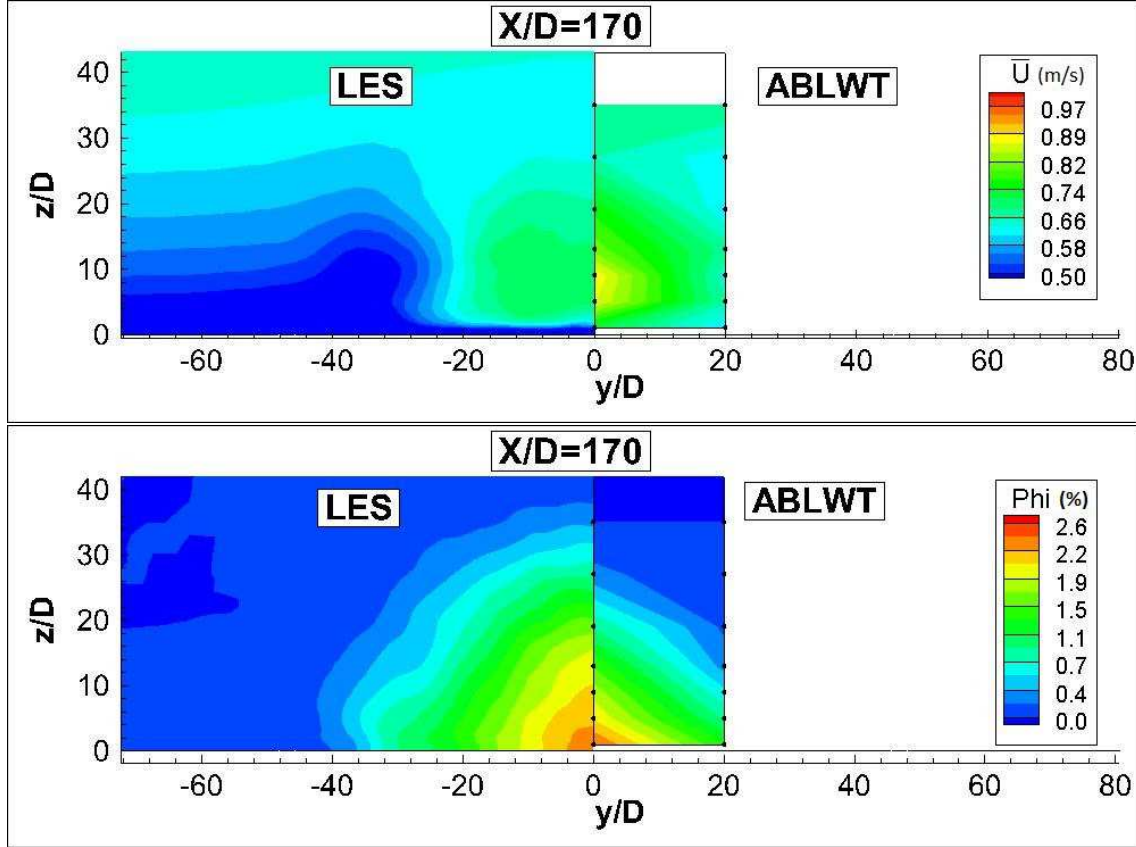


Figure 10.6: Repetition of figure 10.4 using Case 12 results “*THRESHOLD-LARGE*” baffle setup.

Similar to the previous setup, the deviation of U_f and the finite width of the baffle array are responsible for the disagreements between *LES* and experiment, but in this setup the finite width of the baffle array was found deterministic, having affected both the developments of $\bar{U}$ and $\bar{\Phi}$. For future cross-validations of the threshold baffle arrangement it is recommended to modify the *LES* computational domain to accommodate a longer array of baffles. Furthermore, a parametric study investigating the effect of the radius of the threshold baffle array (infinite radius was in this setup) is suggested.

10.3 EXTENDED NUMERICAL ANALYSIS

The *LES* predictions have been validated against experimental data and met good agreement as discussed in section 10.2. In this section the study is extended to the individual analysis of certain *LES* solutions. In particular, the subjects discussed in the following sub-sections include the evaluation of different baffle arrangements, with a deeper focus put to the “*ADDITIONAL*” baffle arrangement setup, as well as the investigation of the wind speed (co-flow) effect under the influence of a strong baffle resistance.

The comparisons of this section analysis involve mean streamwise velocity $\overline{U}$ and mean passive scalar concentration $\overline{\Phi}$, in plots of streamwise evolution of mean flow characteristics, which are obtained by the advanced post-processing method. Further to these plots, the comparisons also include two dimensional contours of U , $\overline{U}$, Φ , and $\overline{\Phi}$, plotted in vertical and horizontal juxtapositions between the compared cases.

10.3.1 Evaluation of the Different Baffle Arrangements

The results of Cases 5, 6, 8, 12 and 14, corresponding to the *UNRESISTED*, *ORIGINAL*, *LARGE*, *THRESHOLD-LARGE*, and *ADDITIONAL* baffle arrangement are compared in terms of the streamwise evolution of the *3DWJ* mean flow characteristics. The results illustrated in figures (D.9.a-h) have been obtained with the advanced post-processing method presented in section 8.1.2. It has to be noted that calculation of the velocity length scales have been implemented based on the difference $U_m - U_f$ instead of simply U_m , something that was not mentioned in section 8.1.2 due of the homogeneous co-flow used in the validation tests. On the other hand, the analysis of section 10.2 has revealed that U_f varies along X due to the boundary layer development in the simulation. The streamwise evolution of the velocity length scales plotted in figures (D.9.b-d) is calculated based on a fully developed boundary layer velocity distribution described by the $1/7$ wind profile power law relationship of equation (6.1). For this reason the plots illustrated in figures (D.9.b-d) include uncertainty; therefore, it is decided to focus

this discussion on the concentration mean flow characteristics.

The vertical concentration half-widths $z_{1/2\Phi}$ are compared in (Fig.D.9.g) and the horizontal concentration half-widths $y_{1/2\Phi}$ are compared in (Fig.D.9.h). The *UNRESISTED* jet results are used as reference for the evaluations. It is observed that installation of the *ORIGINAL* baffle arrangement has a small effect to both concentration half-widths. It is, however, encouraging that the lateral expansion rate drops significantly towards the end of the computational domain when the original baffles are installed. Doubling the size of the baffles (*LARGE* baffle arrangement), causes significant changes to the development of the concentration half-widths. In particular, the increased momentum reduction achieved by the large baffle resistance has allowed for the buoyancy forces to become dominant, something that is reflected by the considerable increase of the vertical growth rate, resulting to a negative horizontal growth rate. Even more effective is the *THRESHOLD-LARGE* baffle arrangement, having achieved even higher vertical jet expansion, and further reduction of the horizontal jet dimension.

Additional Baffle Arrangement

The discussion of the previous paragraph highlighted that the *ORIGINAL* baffle arrangement, originally developed through experimental tests by (Velikov, 2013), and field-trials (Bennett *et al.*, 2013), also complying with the regulations of aerodrome licensing (CAA, 2011), does not appear to be very effective with respect to accelerating lift off of the pollution. On the contrary, the *LARGE* baffle arrangement was found considerably more effective, however, the height of the baffles is beyond the aerodrome safety limitations. The *THRESHOLD-LARGE* baffle arrangement was also found very effective; but such a configuration has to be retractable, which increases the complexity and cost of the application operation.

In order to retain compliance with the regulations, without having to increase complexity of the application, the additional baffle arrangement is suggested. In this way, the *ORIGINAL* baffle arrangement is expanded adding three extra baffle arrays downstream. The height and spacing between the new baffle arrays has been calculated following trends of the already optimized by (Velikov, 2013) first

three arrays, as illustrated in (Fig.6.10). It is clear that all six baffle arrays lie well below the three degree glide-slope. Detailed information of the additional baffles are available in table 6.2.

The addition of extra baffles to the original setup has considerably increased the efficiency of the arrangement, with the vertical concentration half-width $z_{1/2\Phi}$ of (Fig.D.9.g) approaching the ones of the *LARGE* and *THRESHOLD-LARGE* baffle arrangements, and more importantly, this occurs while a negative growth rate is recorded at the horizontal half-width $y_{1/2\Phi}$ of (Fig.D.9.h) indicating lift off of the pollution.

The analysis based on the advanced post-processing method relies on the level of averaging achieved in the simulation. According to the theoretical estimations presented in section 5.3.3, the uncertainty of the solution depends on the time-scale of the integral turbulent motions. Addition of the baffles reduces the jet momentum, meaning that the frequency of the large scale motion drops, while it affects the *3DWJ* length scales as well. Therefore, in order to enhance analysis of this study, the *LES* solutions of Case 6 and Case 14 are also compared in terms of two dimensional contours of U , $\bar{U}$, Φ , and $\bar{\Phi}$ on a horizontal plane at $Z = 0$, and on a vertical plane at $Y = 0$ (axisymmetry plane).

The velocity comparisons are illustrated in (Fig.10.7). The comparison between the two cases is made on each image. In particular, Case 6 (*ORIGINAL* baffle arrangement) occupies the bottom side of the images, and Case 14 (*ADDITIONAL* baffle arrangement) occupies the top side. In exactly the same way the concentration comparisons are made in (Fig.10.8).

The presence of the three additional baffle arrays of the *ADDITIONAL* baffle arrangement, is identified by the wakes following each baffle array in (Fig.10.7). The six wakes are getting gradually bigger in both the streamwise and vertical direction following the increasing baffle size. The horizontal distribution of U and $\bar{U}$ highlight the considerable reduction of the streamwise velocity when the additional baffles are installed. It is noted that minor fluxes by-pass the baffle arrays from the lateral edges. These fluxes tend to merge again towards the centreline after the end of baffle region. This leads to the suggestion increasing the lateral

extent of each baffle array in order to help increasing the drag force, extensively the efficiency of the system. Finally, the excessive deflection of the plume due to the additional baffles seems to have caused non-recoverable changes to both the vertical and horizontal velocity distribution in the farfield, $X/D > 250$ which is attributed to the dominance of buoyancy.

The vertical profiles of appendix E show that when the *3DWJ* develops into a self-similar state the pollution tends to concentrate on the wall surface. Therefore, inclusion of the baffles not only removes momentum from the plume, but also brings this high concentration fluid, into higher level above the ground. The same high concentration fluid is also the hot-buoyant one. This means that the closer it is brought to the external mixing layer, where low-temperature flow exists, the higher the effect of buoyancy. In conclusion, if at the end of the baffle region momentum reduction is enough so that buoyancy dominates, the plume will not manage to undergo a transition towards a self-similar *3DWJ*. In (Fig.10.8) we can observe both occasions described above. Specifically, the resistance caused by the *ORIGINAL* baffle arrangement is not enough to allow buoyancy to dominate; therefore, after the end of the baffle region the plume begins a transition towards a self-similar *3DWJ*, and the maximum concentration returns into a steep formation on the ground. Of course there are certain improvements comparing to the unresisted jet according to previous discussion, but not considerable. On the contrary, inclusion of the additional baffles is considerably effective, causing buoyancy to dominate, extensively no longer allowing the pollution to form high concentrations on the ground. Instead, the pollution remains longer at high level where mixing rate with the ambient flow is higher.

In conclusion, although the original baffle arrangement has been found inadequate to achieve lift-off of the plume, certain modifications of the baffle arrangement have improved the wind-brake system and lift off of the plume was observed from within the presently investigated streamwise range.

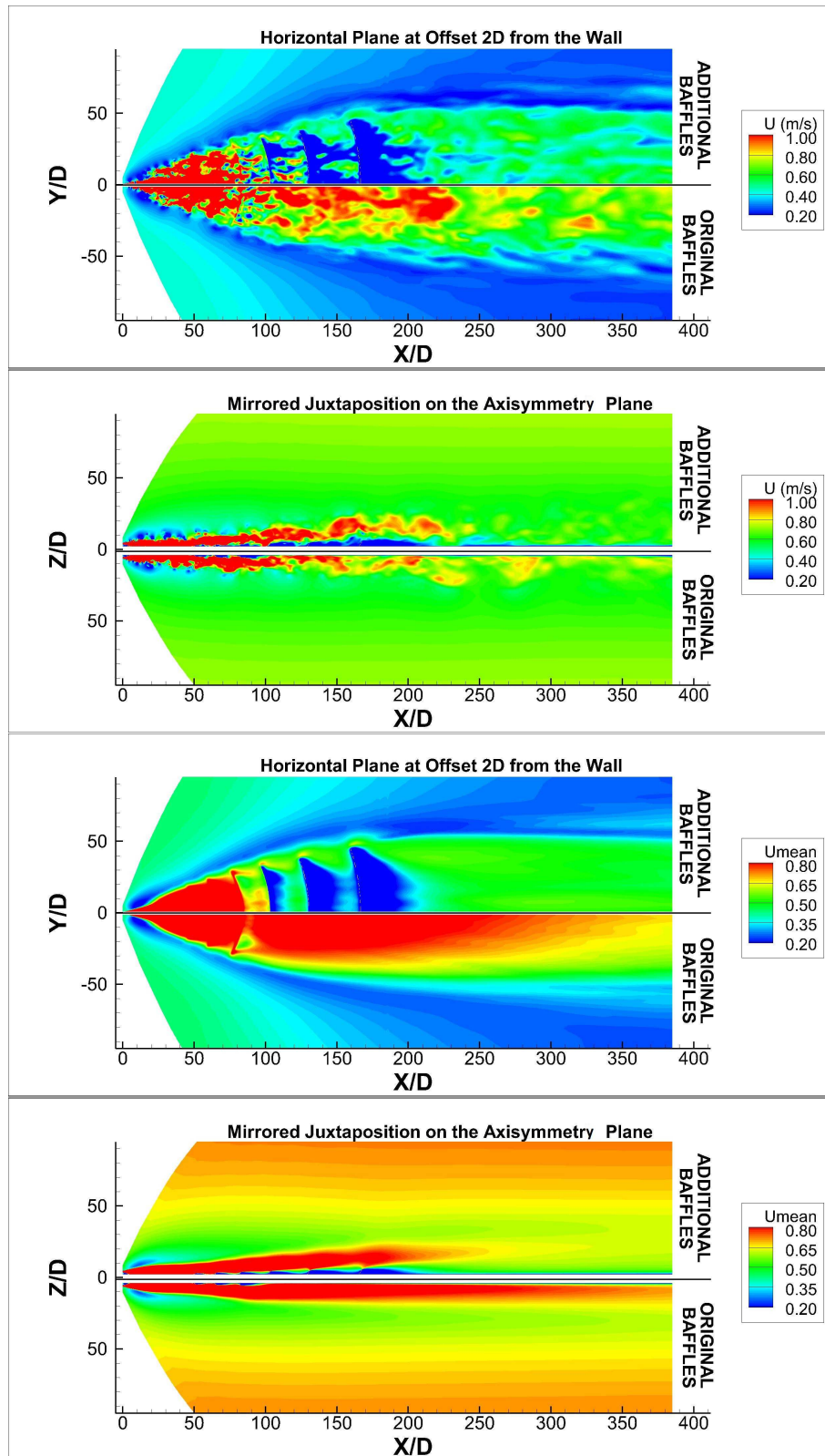


Figure 10.7: Axisymmetric-mirrored comparison of the “ORIGINAL” (Case 6) and “ADDITIONAL” (Case 14) baffle arrangements. Contours of the instantaneous and mean streamwise velocity, U and $U_{mean} = \overline{U}$, are plotted at the horizontal $Z = 0$ and axisymmetry $Y = 0$ planes.

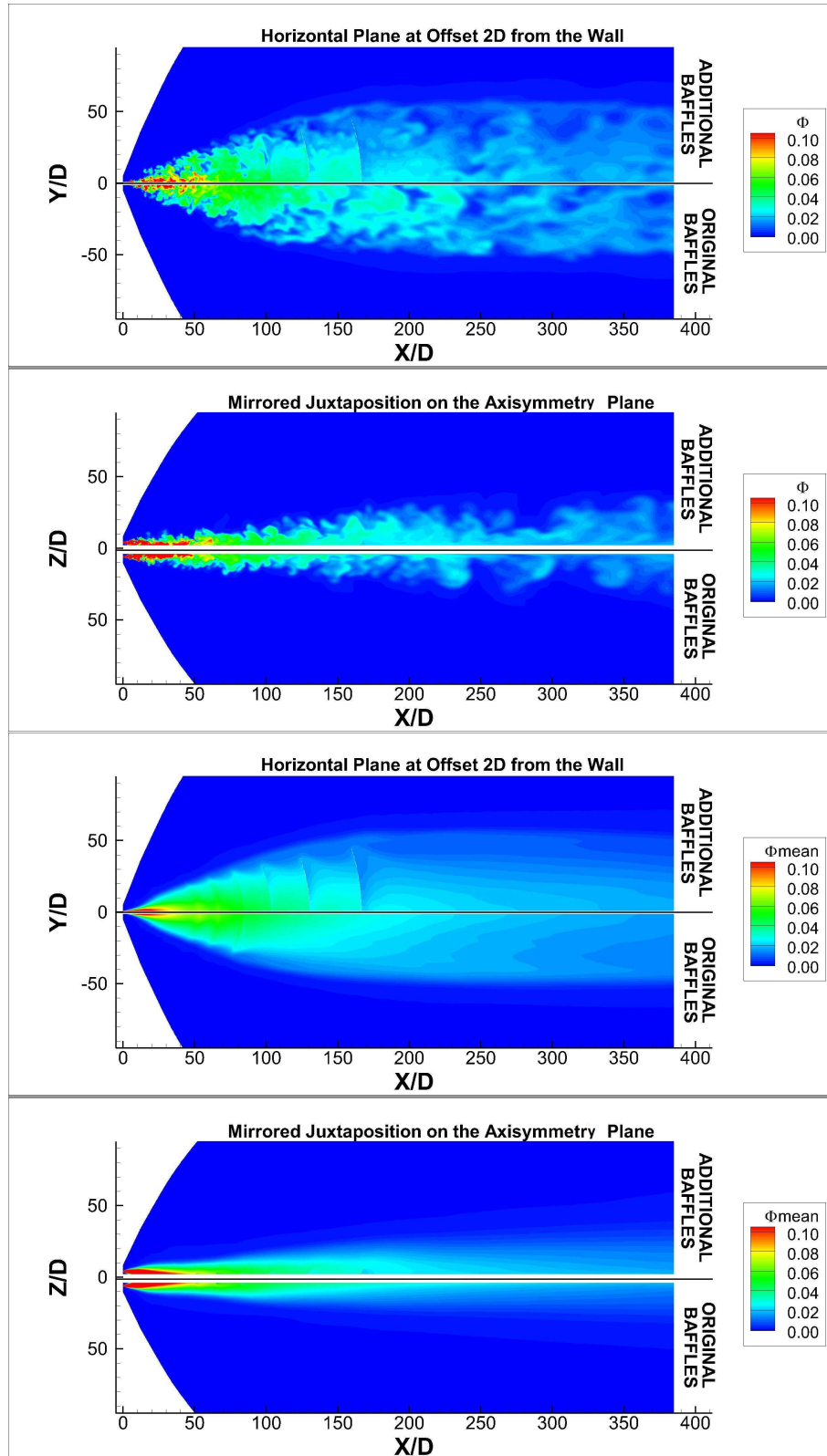


Figure 10.8: Axisymmetric-mirrored comparison of the “*ORIGINAL*” and “*ADDITIONAL*” baffle arrangements. Contours of the instantaneous and mean concentration, Φ and $\Phi_{mean} = \bar{\Phi}$, are plotted at the horizontal $Z = 0$ and axisymmetry $Y = 0$ planes.

10.3.2 Varying Wind (co-flow) Velocity on the THRESHOLD Baffle Arrangement

The final test of this section is the co-flow velocity effect to the plume development, when baffle resistance is present. In order to maximize the after-baffle region, the *THRESHOLD* baffle arrangement was chosen to perform this test. The difference with the *THRESHOLD-LARGE* baffle arrangement is the height of the baffles, which is now reduced to the half in order to reduce the by-passed fluxes which cause several problems as discussed in section 10.2.4. Since the co-flow velocity (U_f) is described by a 1/7 wind profile power law relationship, its modification is achieved through the control velocity (U_r). In particular, two different control velocities are set to the inlet boundary layer velocity profile, being $U_r = 0.5m/s$ and $U_r = 0.25m/s$ and Cases 10 and 11 are simulated respectively. Details are available on table 10.1. The analysis of this study follows the same procedure with section 10.3.1.

Streamwise evolution of the vertical concentration half-widths $z_{1/2\Phi}$ are compared in (Fig.D.8.g) and the horizontal concentration half-widths $y_{1/2\Phi}$ are compared in (Fig.D.8.h). It can be observed that $z_{1/2\Phi}$ becomes larger for the small wind speed immediately after the baffle array, however, the growth rate is the same for both cases. On the contrary, although $y_{1/2\Phi}$ also grows faster for the small wind speed, it encounters a negative growth rate in the farfield, which, according to the discussion of the previous section, indicates increased contribution of buoyancy.

The *LES* solutions of Case 10 and Case 11 are also compared in terms of two dimensional contours of U , $\bar{U}$, Φ , and $\bar{\Phi}$ on a horizontal plane at $Z = 0$, and on a vertical plane at $Y = 0$ (axisymmetry plane). The velocity comparisons are illustrated in (Fig.10.9) and the concentration comparisons are made in (Fig.10.10).

The application of different co-flows can be clearly observed in the interior flow at both the vertical and horizontal planes of (Fig.10.9). Furthermore, an overall reduction of the jet speed is observed. However, comparison of the jet shape is difficult to make as the same contour scale is used for both cases. However, such

observation is clearly made through the concentration contours of (Fig.10.10). A significant observation here is that decreasing the co-flow velocity promotes the jet for by-passing the baffle array from the lateral edges. This appears to have also increased the interaction of the jet flow with the lateral-inlet boundary and possibly contributes to the unaffected growth rate of $z_{1/2\Phi}$, by reducing the amount the jet travelling at the symmetry plane, which is where $z_{1/2\Phi}$ is calculated.

In conclusion, it was found out that, when a limited span baffle array is installed on the runway threshold, the volume of the plume by-passing the baffles from the lateral sides increases with reducing the wind speed. It is difficult to define a baffle extend which secures zero lateral fluxes, however, it is suggested that adding a curvature to the threshold baffle arrangement would contribute on a considerable increase of the efficiency of the system, especially for low values of the wind speed.

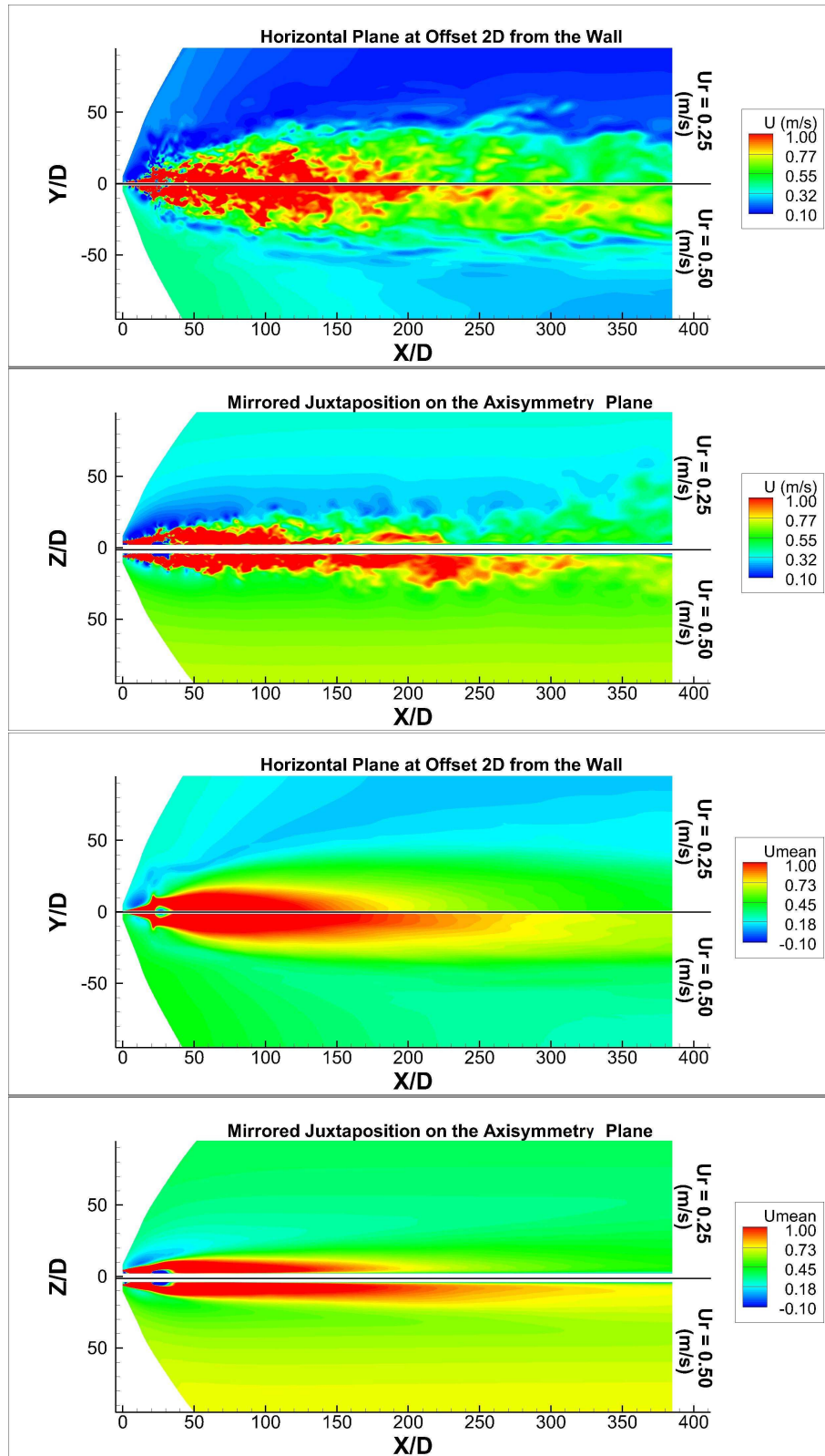


Figure 10.9: Comparison of two different co-flow velocities $U_r = 0.5$ (Case 10) and $U_r = 0.25$ (Case 11). Image setup repeated from figure 10.7.

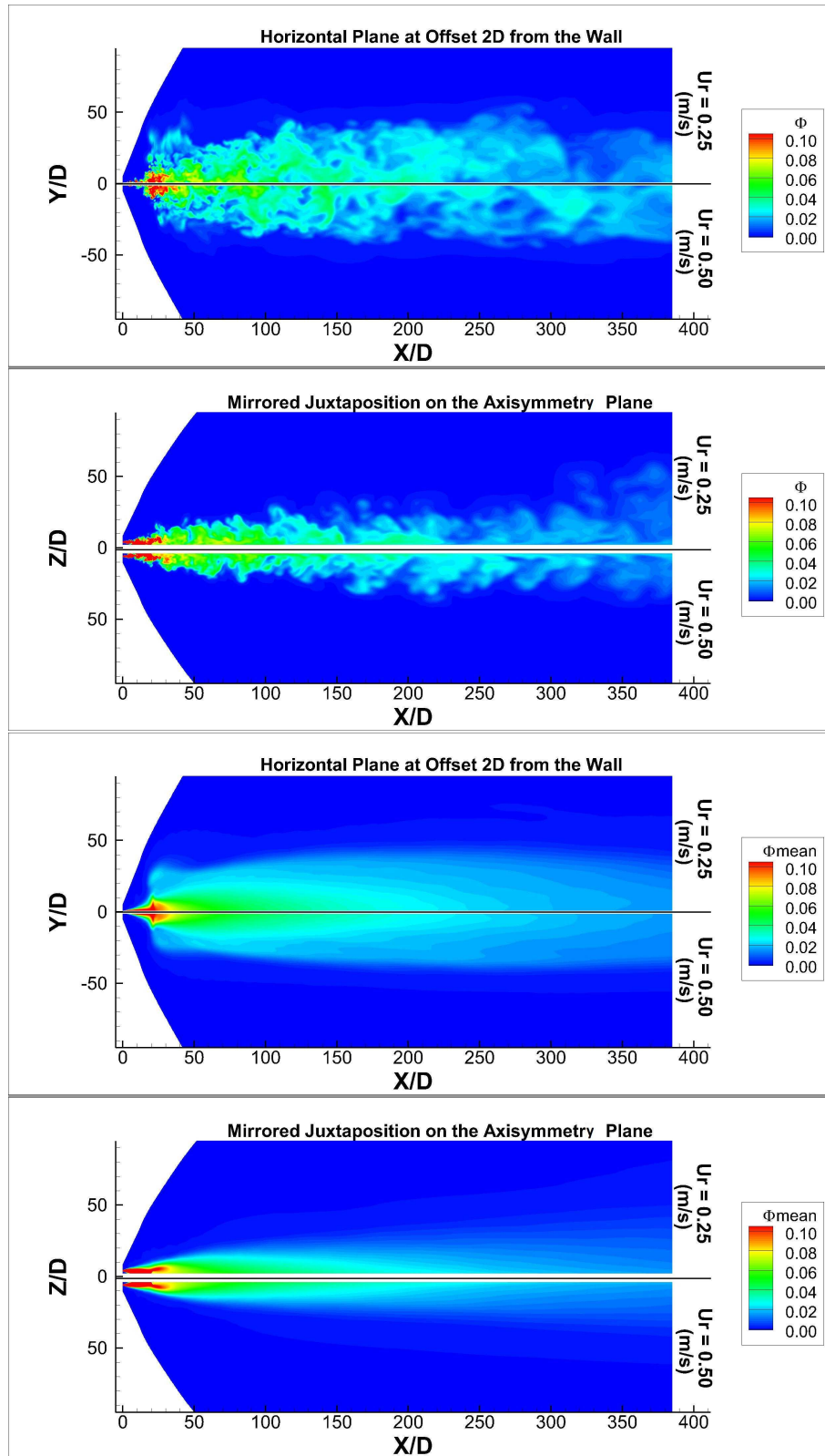


Figure 10.10: Comparison of two different co-flow velocities $U_r = 0.5$ (Case 10) and $U_r = 0.25$ (Case 11). Image setup repeated from figure 10.8.

10.4 BAFFLE REPRESENTATION METHOD

The baffle validation of chapter 9 completed with a positive outcome. In particular, it showed that two selected methods of representing the baffle-induced resistance to the numerical simulation produce comparable results in terms of drag force (F_D) and local maximum concentration (Φ_m). The initial definition of the modelled approach was diagnosed with certain deficits as discussed in (Spanelis *et al.*, 2013). The proposed, therein, modifications have been incorporated at the description of the model in section 4.3, and extensively applied for the simulation of Cases 7, 9, and 13 of this chapter. In this section, the predictions of these cases are directly compared with Cases 6, 8, and 12 respectively, corresponding their “*designed baffles*” equivalence.

The comparisons of this section involve mean streamwise velocity $\bar{U}$ and mean passive scalar concentration $\bar{\Phi}$, in plots of streamwise evolution of mean flow characteristics obtained by the advanced post-processing method (plots are included in Appendix D), as well as vertical profiles in various farfield locations (plots are included in Appendix E). Further to these plots, the comparisons also include two dimensional contours of U , $\bar{U}$, Φ , and $\bar{\Phi}$, plotted in vertical and horizontal juxtapositions between the compared solutions, available in Appendix F. Discussion of the comparisons is subdivided in three paragraphs, each one corresponding on one of these the three comparison approaches.

Mean Flow Characteristics

figures (D.5), (D.6), and (D.7) compare the results of the *designed baffles* with the *modelled baffles* for the *ORIGINAL*, the *LARGE*, and the *THRESHOLD-LARGE* baffle arrangement respectively. The differences between the predictions of the two methods are indistinguishable on the *ORIGINAL* baffle arrangement results. However, the *LARGE* baffle arrangement results suggest that the *modelled baffles* over-predict the vertical scales and under-predict the horizontal ones. In other words, *DRAMBI* accelerates lift-off of the plume when used for the representation of large baffles. This statement is also confirmed by the *THRESHOLD-*

LARGE results, where higher deviations are observed between the predictions of the two baffle representation methods. It has to be taken into account that the *THRESHOLD-LARGE* baffle arrangement involves further uncertainty due to the plume interactions with the inlet boundary, as discussed in previous sections; therefore, it is not advisable to justify all differences observed in (Fig.D.7) on weaknesses of the model, but on this effect as well.

Vertical Profiles

Figures (E.2) and (E.3) compare the two baffle representation methods in terms of vertical profiles of $\bar{U}$ and $\bar{\Phi}$ for the *ORIGINAL* baffle arrangement. The experimental data can be used as reference point for the comparisons between the two figures. Again, the differences are indistinguishable, agreeing with the previous paragraph. Repeating the comparison with figures (E.4) and (E.5) it is observed that the *modelled baffles* have lifted the profiles at slightly higher level, which, again, suggests that the vertical deflection is over-predicted by *DRAMBI* when the baffle size increases. figures (E.6) and (E.7) illustrate the *THRESHOLD-LARGE* baffle arrangement vertical profiles for the two methods. Major differences are observed in both $\bar{U}$ and $\bar{\Phi}$. However, as stated in the previous paragraph it is difficult to make direct model comparison in this baffle arrangement due to the effects of the inlet boundary, something that will be better observed on the interior planes of the following paragraph.

Interior Planes

The two baffle representation methods are also compared in terms of two dimensional contours of U , $\bar{U}$, Φ , and $\bar{\Phi}$ on a horizontal plane at $Z = 0$, and on a vertical plane at $Y = 0$ (axisymmetry plane). The velocity comparisons for the *ORIGINAL* baffle arrangement are illustrated in (Fig.F.1). The comparison between the two cases is made on each image. In particular, the *designed baffles* occupy the bottom side of the images, and the *modelled baffles* occupy the top side. In exactly the same way the concentration comparisons are made in (Fig.F.2). A first observation is the shape of the baffle arrays, which is reproduced with high accuracy

by *DRAMBI*. This is one of the improvements of the initially proposed model in (Spanelis *et al.*, 2013), where the baffle arrays did not include the curvature. Both the velocity and the concentration fields obey high similarity for the two compared approaches. The only observable differences can be identified on U at the wake of the baffles, where the modelled approach slightly over-predicts the reattachment length. Similarly, figures (F.3) and (F.4) repeat the comparisons for the *LARGE* baffle arrangement. Again the agreement is high between the two baffle representation methods. Here the most observable difference is identified on the laterally by-passed fluxes, which in the case of the *modelled baffles* it is larger, especially on the first baffle array. Large deviations between the *designed baffles* and the *modelled baffles* are observed on the *THRESHOLD-LARGE* baffle arrangement in figures (F.5) and (F.6). The main source of disagreement between the two baffle representation methods on this baffle arrangement is now clarified by the horizontal U . In particular, it is observed that the laterally by-passed fluxes are considerably larger for the *modelled baffles*, therefore, also magnifying the interactions with the inlet boundary. This early disagreement between the two cases is not recovered in the investigated range.

In conclusion, it can be claimed that the model is efficient when the lateral extent of the jet is smaller than the one of the baffles, and as long as the full jet passes above the baffles and not from the lateral edges. Additionally, the model appears to over-predict the vertical deflection of the plume,

Chapter 11

Conclusions and Recommendations for Further Work

This work investigated the horizontally emitted exhaust jets of civil airliners during takeoff, that make a large contribution to local air quality around airports, within the framework of Clean Sky project. Numerical simulations have been carried out in the commercial software FLUENT, and the large-scale turbulent events have been directly resolved by its *LES* algorithm. The main goal of the thesis was to setup a valid approach for the numerical simulation of this high complexity problem, while at the same time studying the recently proposed (Bennett *et al.*, 2013) installation of a wind-brake system, which is an array of baffles suitably arranged behind the aircraft, aiming to accelerate the natural lift-off of the plume. A large part of this work has been undertaken in a complementary manner alongside wind tunnel experiments carried out by a separate group (Velikov, 2013) in the atmospheric boundary layer wind tunnel of Cranfield University.

Apart from the inclusion of the newly proposed wind-brake system, present work has major differences compared to previous FLUENT-*LES* studies of exhaust jets in takeoff conditions (Aloysius *et al.*, 2007; Aloysius & Wrobel, 2009a). Previous studies have investigated the plume development at very long downstream locations, up to 2000 nozzle diameters, while the streamwise extent of present simulations only reached a maximum of 380 diameters. However, the results of previous work have only been validated by real world measurements in

terms of instantaneous concentration data, while present work results are thoroughly cross-validated by time-averaged velocity and concentration data. Despite the fact that the streamwise extent of this work was short comparing to previous ones, the wind-brake system accelerated the lift-off process enough so that it becomes visible from within the investigated range. The main achievements of this work, ordered by their importance to this thesis contribution to knowledge are:

1. Established credibility of the *LES* predictions by means of a staged validation following an increasing problem complexity: starting from the simplest “cold, free, round jet in strong co-flow”, and extending to the most complicated “buoyant, three dimensional wall-jet, developing inside a low speed boundary layer, under the influence of a wind-brake system”.
2. Enhanced understanding of the response of the geometrical characteristics of the wind-brake system to the behaviour of the exhaust jet in the farfield.
3. Shed light to the response of certain numerical parameters to the *LES* solution of axisymmetric and three-dimensional wall-jets, in a way to limit the computational cost without affecting the quality of the numerical predictions.
4. Devised and validated an algorithm for the representation of the baffles in the computational domain of *LES*, based on the dynamic evaluation of local source terms in the momentum equations.
5. Developed a post-processing code which performs volumetric analysis of the three-dimensional solutions, calculating the streamwise evolution of the jet mean velocity and mean concentration characteristics.
6. Developed an efficient meshing approach adaptable to both the simulations of axisymmetric jet and the three dimensional wall jet.
7. Established a simulation setup, automated by scripts, easily modifiable to apply at different cases, which was necessary to maintain robustness of the multiple parametric studies carried out.
8. Developed a method for *a-priori* definition of the simulated time, based on theoretical jet properties, when co-flow is present.

11.1 TOWARDS THE COMPLETE SIMULATIONS

The problem described by the central objective of this thesis is highly complicated due to the multiple physical parameters composing it, such as the presence of a solid surface parallel to the jet development, the boundary layer formed by the wind, the three dimensional obstacles opposing the plume (baffles), and the buoyant forces formed by the temperature difference between the exhaust jet and the ambient flow. An attempt to directly validate the *LES* predictions based on this complexity would increase uncertainty of the observations, raising questions about which parameters, numerical or physical, are responsible for the deviations between *LES* and experiment. For this reason, it was decided to carry out a staged validation of the numerical approach by progressively increasing the complexity of the investigated setup.

Validating Free-Shear Jet Simulations

The first validation problem was a cold, free, axisymmetric jet in strong co-flow. A high value of the co-flow velocity is desirable in the simulation due to the inverse proportionality of this variable with the computational cost of *LES* (refer to section 5.4). Experimental data taken by a previous relevant experiment (Garry & Holt, 2008), as well as results of previous *LES* (Dinesh *et al.*, 2009) were used for cross-validations during this study, the results of which are analytically discussed in section 7.1. Numerical parameters, such as the convergence criteria, timestep size, jet inlet turbulence intensity, shear layer resolution, under relaxation factors, temporal convergence, mesh density, pressure coupling, *SGS* modelling, and mesh isotropy in the farfield, were parametrically tested.

The main outcome of this set of tests was that the near-field computations $0 \leq X/D \leq 15$ strongly depend on most of the tested parameters. Specifically, accurate near-field predictions required the following: a high mesh density in order to capture the small anisotropic motions of the transitional jet, a small timestep to temporally resolve these motions, low values of the convergence criteria in order to ensure sufficiently converged equations in every timestep, and

similar adjustment of other numerical parameters. Such a high resolution numerical setup implies high computational cost. Alternatively, it was observed that although under-resolution of turbulence in the near-field brings about erroneous results at the end of transition, there is a strong tendency of such errors to gradually recover with increasing downstream distance. In other words, it was observed that the farfield of an expensive simulation is identical to the farfield of a moderate one.

Another significant observation made in this study was the high importance of isotropic mesh cells in free-shear regions with respect to accurate passive scalar mixing calculations. The streamwise dimension of the computational mesh contains the largest number of cells comparing to the cross-sectional ones. For this reason, a straightforward approach to limit mesh density is to trade-off streamwise resolution at the very farfield, where the dominant turbulent events have become very large. Such a simplification, however, was found to be incorrect because it results in the formation of elongated cells in the streamwise direction, which according to present observations, leads to erroneous scalar mixing calculations.

The second free-shear validation problem was a high speed co-axial jet in stagnant environment. This was a more difficult problem to simulate due to the high Reynolds number and the complex inlet which consists of a realistic turbofan engine model. Experimental data and previous high resolution *LES* results, available from (Eastwood *et al.*, 2008), were used to validate the *LES* predictions. The most important finding of this study was that modelling (simplifying) the complex geometry of the turbofan engine at the inlet, by substituting it with simple velocity and scalar profiles, only affects the near-field results, but the solution becomes independent after transition is complete.

Validating Wall Bounded Jet Simulations

At the time of the validation tests, experimental data of a three-dimensional wall jet in co-flow were not available. Therefore, a modification was made to the free-round jet setup of (Garry & Holt, 2008) in order to carry out the wall bounded validation tests of chapter 8. Lack of experimental data does not reduce credibility

of this study, because the main target was not to cross-validate the *LES* predictions, but to parametrically observe the response of certain solution variables to a set of selected numerical and physical parameters. Repetition of the near-field accuracy tests was followed by similar conclusions to the free-shear validation tests, most importantly having observed a similar recovery of transitional errors in the farfield. An additional finding of this study was the importance of universally adjusted low values of the non-dimensional wall-distance ($y^+ \approx 1$). Apart from the immediate effects in near-wall region, high values of y^+ were observed to affect both the mean velocity and concentration characteristics.

The analysis of the wall bounded validation tests identified the necessity to develop a post-processing code, which calculates the streamwise evolution of the mean velocity and mean concentration characteristics, by performing volumetric analysis of the three-dimensional solutions. This was a crucial input to the research, facilitating high level assessment of the non-axisymmetric fields which would otherwise have required enormous post-processing effort, or extremely long discussions for robust comparisons in terms of multiple horizontal and vertical profiles.

Selecting the Meshing Approach

Three different structured meshing approaches were tested during the validation tests, including revolution meshing, H-Type meshing, and block structured O-Type meshing. Although the revolution mesh is the easiest to design and control during isotropic densification, and its predictions were observed similar to the other two approaches, it is extremely difficult to apply in wall jet simulations. Furthermore, it was found out that although O-Type meshing involves higher complexity of design comparing to H-Type, it significantly reduces the required number of cells, extensively reducing the computational cost which has been a major constraint for this work. The most difficult task was to benefit from the advantages of the O-Type meshing approach in the three dimensional wall jet simulations, something that was achieved with the development of a hybrid H-Type/O-Type mesh.

Defining the Simulated Time Requirement

The duration of the simulation has been a major model parameter in this work, because of the long distances needed to be travelled by the investigated jets during the simulations, especially considering the sampling process required for the investigated statistical quantities to be formed. In order to be able and *a-priori* define the simulated time intervals, theoretical properties of jets in stagnant environment have been modified to account for a co-flowing wind in chapter 5. Predictions of this method were tested and met close agreement with the *LES* results in terms of both the streamwise evolution of U_m and $z_{1/2}$ for the axisymmetric jet in co-flow. It is possible to extend this test in the case of a three dimensional wall jet. It is, however, important to first improve the definition of U_f which in the latter case is not homogeneous around the mixing layer, but varies from zero (on the ground) to a non-constant maximum value depending on the vertical position above the ground where the free mixing layer lies.

The presence of the baffles reduces the passage frequency of the largest eddies, thereby increasing the simulated time requirement (*STR*). Further investigation is needed before being able to predict the streamwise convection frequency of these new turbulent scales based on the geometrical characteristics of the installed baffle arrangement.

It is interesting to stress the possibility of substituting the highly expensive near-field computations by an artificial (unsteady) inlet, representing the cross-section of a three-dimensional wall-jet, at the most distant possible streamwise location. Such an application is expected to reduce the simulation cost by a large factor, both due to the reduction of the number of computational cells, and due to the increased timestep size. Therefore, it is recommended that separate research is conducted with the aim to establish robust methods for defining an unsteady inlet in a turbulent region of the wall-developing plume, closer to the area of interest.

Representing the Baffles in the Flowfield

The numerical representation of the wind-brake system was initially investigated in chapter 9. Problem setup as well as experimental measurements, including aerodynamic drag and pollutant concentration, were provided by the experimental work of Velikov (Velikov, 2013). The conventional approach, which is to design the baffle geometries similar to the experimental setup and adjust their surfaces to the no-slip condition in the simulation, have efficiently reproduced the experiment.

An alternative method to represent the baffles in the flowfield was developed and validated. This algorithm is based on the immersed boundary method first introduced by (Peskin, 1972), and is based on the dynamic evaluation of local source terms in the momentum equations, causing the flow to respond aerodynamically as if the baffles were present in the flow (refer to section 4.3). Validation of the modelled baffle approach, which has also been published (Spanelis *et al.*, 2013), has met a high level of success. Suggestions for improvements made in this publication were applied to the model before extending its application to the final simulations of this thesis. Extensive comparisons between the modelled and the designed baffle results, for three different baffle arrangements, met reasonable agreement, including streamwise evolution of mean velocity and concentration characteristics.

The modelled baffle representation method is expected to reduce the computational cost by reducing the number of computational cells. However, the most significant advantage of this model is that it does not require new meshing for every geometrical change applied on the baffles. In this way it is expected to accelerate the numerical investigation of the baffle-resisted plumes in the future.

The main weakness of the model is on reproducing separations from the lateral edges of the baffle arrays, especially when the lateral extent of an array is short. This weakness would be insignificant in a sufficiently long baffle array, when the full jet passes above the baffles. Another disagreement between the modelled baffles and the designed ones was the over-prediction of the vertical deflection of the plume when modelled baffles are selected. These over-predictions

increased proportionally to the amount of the laterally by-passing plume. It is, therefore, believed that lateral extension of the baffle arrays would resolve both issues.

Including Buoyancy in the Simulation

It was observed that density variations due to the different temperatures specified at the jet and co-flow inlets, cause distinguishable changes to the unresisted three dimensional wall jet mean flow characteristics. It is expected that in the final simulations, where streamwise momentum is reduced by the baffles, the buoyancy effects would get magnified. However, due to limited computational resources the deflected jet simulations have only been performed with active buoyancy. A suggestion for future work is to repeat the baffle including simulations with buoyancy deactivated. Such a study is expected to facilitate the evaluation of the “*efficiencies*” of the tested baffle arrangements.

11.2 LIMITATION OF CROSS-VALIDATIONS

Further to the setup parameters of the free jet validation tests, numerical replication of the final experiments required two additional parameters to be reproduced in the simulation. These include a fully developed boundary layer, and a wind-brake system of certain geometrical dimensions.

Replicating the Boundary Layer

Boundary layer reproduction was not a trivial task to accomplish. The mean vertical velocity distribution recorded in the wind tunnel experiments, which was later recreated as a profile at the inlet boundary of *LES*, includes high uncertainty. The reason being a co-flow speed variation of about 10 – 15%, between different experimental measurements, due to the uncontrollable ambient (to the wind tunnel) conditions as stated by Velikov (Velikov, 2013). Furthermore, a highly significant parameter for disagreements between *LES* and experiments is the lack of a detailed turbulent description to be reproduced at the inlet boundary. This

has allowed a considerable streamwise development of the boundary layer in the simulation, resulting in a gradual reduction of the co-flow speed surrounding the simulated jet. This disagreement in the setup replication has caused the jet to expand slightly faster in the simulation, especially in the attached jet regime where the lateral growth rate is, in theory, very large.

It is suggested, for future investigations, that high fidelity measurements of U_f and $U_{f_{RMS}}$ are made in the wind tunnel, before being accurately replicated by *LES*. Such a detailed description of U_f would also give space for higher level analysis of the streamwise evolution of mean velocity characteristics. Specifically, the improved analysis would be implemented in terms of the “*net mean x-velocity*” ($\bar{U} - U_f$) instead of the mean x-velocity ($\bar{U}$). The most significant gain of such a post-processing method would be the ability to directly compare non-dimensional distributions between $(\bar{U} - U_f)$ and $(\bar{\Phi})$, and improve conclusions regarding the lift-off of the plume.

The Lateral Extent of the Baffle Arrays

A source of uncertainty, additional to the first one, is the limited lateral extent of the baffle arrays composing the wind-brake system.

Limited lateral extent of the baffle arrays was observed to allow the jet to bypass the array from the lateral sides. In some of the investigated setups this escape has imposed significant effects to the flow at the plane of symmetry. This uncertainty parameter add to the co-flow disagreement mentioned above. This can be understood by taking into account that the amount of the laterally by passed jet fluxes increases with increasing horizontal dimension of the jet, which also increases with decreasing co-flow velocity. It is therefore suggested for future work aiming to improve robustness of the *LES – Experiment* cross-validations, to further extend the lateral span of the baffle arrays, and to remove this uncertainty parameter from the analysis.

Although the above reasons have lead to significant disagreement in terms of mean velocity, the effects on passive scalar fields were, in the majority, insignificant. In particular, the overall level of disagreement remained below 10%, while

this percentage is well justified by the cross-sectional over-expansion of the plume in the simulation. A more efficient reproduction of the experimental conditions of the simulation is expected to decrease this percentage even further.

11.3 CURRENT ASSESSMENT AND FUTURE RECOMMENDATIONS

Current Assessment of the Wind Brake System

Further to the replication of the *ORIGINAL* baffle arrangement, which is a modelled version of the wind-brake system tested in the field-trial investigation of (Bennett *et al.*, 2013), this thesis work also considered additional setups. The *LARGE* and *THRESHOLD* setups have only been tested for the purpose of enhancing present understanding of the geometrical characteristics of the baffle arrangement. Considering that the standard glide-path has an approach angle of 3° , and that regulatory clear approach surfaces are typically 1.5° or less, such wind-brake configurations conflict with the civil aviation authority regulations (CAA, 2011). A threshold arrangement would only become a possible application in a retractable format, in cases where the original arrangement is not possible for other reasons.

The original baffle arrangement has been found inadequate to achieve lift-off of the plume in the investigated range; further extension of the computational domain is required. However, the increased drag and lift forcing of the larger baffles has accelerated the natural lift-off process, and the plume's vertical to horizontal scale ratio increases rapidly after the baffle region, indicating a gradual domination of buoyancy.

Following the suggestion made by Bennett *et al.* (Bennett *et al.*, 2013), three extra baffle arrays were added to the original setup to further accelerate the lift-off process. This setup, herein called *ADDITIONAL* baffle arrangement, follows the regulatory restrictions of CAA. With this addition, efficiency of the wind-brake system improved considerably, and gradual buoyancy domination is also observed.

Suggestions for the Continuation of the Study

This thesis has set a strong basis for the numerical investigation of the aerodynamic baffles' installation, as suggested by Bennett *et al.* (Bennett *et al.*, 2013), which aims to reduce near airport pollution. Further to the high level validations of the numerical approach carried out, preliminary investigation of the geometrical characteristics of the wind-brake system has been implemented. However, this has only been a first step towards a better understanding of this method. Results of future numerical investigation of larger and even more complex simulations, together with field-trial measurements, should be evaluated together in order to reach a consensus about the application of this method in the market.

In particular, the total number of the simulations of this work have been carried out on a scaled version of the real situation in order to maximize controllability of cross-validations. For the continuation of the study, the established numerical approach is suggested to be extended on a real scale environment, aiming to exclude scaling uncertainty. It is worth including the complete aircraft geometry in future simulations, which as suggested by the work of Aloysius and Wrobel (Aloysius & Wrobel, 2009a), plays a major role in the vertical dispersion of the plume.

In terms of the analysis method, it is recommended for future simulations to not consider overall statistical analysis. Instantaneous flow assessment at certain times after the engines begin to produce thrust, is suggested instead, with the statistical analysis limited to the early stages of the jet development, depending on the averaging time available for sample collection. It is also recommended that the computational domain be long enough so that the plume does not manage to reach the exit of the computational domain in the simulated time. The detailed plots of streamwise evolution of mean velocity characteristics, produced in this work, are valuable in such *a-priori* definitions.

The recommended parameters to be tested in full scale studies include the number of baffle arrays added to the *ORIGINAL* setup, the time of the operational jet, the direction and magnitude of the wind speed, and the ambient temperature.

A further study should also consider the effects of ground surface humidity on the lift-off length.

Finally, should the retractable *THRESHOLD* baffle arrangement be put under investigation, it is suggested that adding a curvature to the baffle array would contribute to a considerable increase of the efficiency of the system. Therefore, a parametric study, investigating the radius of the threshold baffle array, is recommended.

References

- ABRAHAMSSON, H., JOHANSSON, B., & LÖFDAHL, L. 1994. A turbulent plane two-dimensional wall jet in a quiescent surrounding. *European Journal of Mechanics B. Fluids*, **13**, 533–556.
- ABRAHAMSSON, H., JOHANSSON, B., & LÖFDAHL, L. 1996. An Investigation of the Turbulence Field in a Three-Dimensional Wall Jet. *Advances in Turbulence VI, Springer Netherlands*, 417–420.
- ADEL-RAHMAN, A. 2010. A review of effects of initial and boundary conditions on turbulent jets. *Wseas Transactions on Fluid Mechanics*.
- AGELIN-CHAAB, M., & TACHIE, M. F. 2011. Characteristics of Turbulent Three-Dimensional Offset Jets. *Journal of Fluids Engineering*, **133**(5), 051203.
- AHN, S., & FESSLER, J. A. 2003. Standard errors of mean, variance, and standard deviation estimators. *University of Michigan: EECS Department*.
- AIRBUS. 2014 (Feb.). *2Eyes-of-the-Vortex.jpg*. Image accessible at: <http://www.airbus.com/galleries/photo-gallery>.
- ALOYSIUS, S. S., PEARCE, D., & WROBEL, L. C. 2007. *ALAQs-Comparison of CFD and Lagrangian Dispersion Methods-Simple Scenario during Take-off*. Report EEC/SEE/2007/001. EUROCONTROL Experimental Centre.
- ALOYSIUS, S.S., & WROBEL, L. C. 2009a. Large eddy simulation of plume dispersion behind an aircraft in the take-off phase. *Environmental Fluid Mechanics*, **9**(4), 457–470.

- ALOYSIUS, S.S., & WROBEL, L.C. 2009b. Comparison of flow and dispersion properties of free and wall turbulent jets for source dynamics characterisation. *Environmental Modelling & Software*, **24**(8), 926–937.
- APTE, S. V., MAHESH, K., MOIN, P., & OEFELEIN, J. C. 2003. Large-eddy simulation of swirling particle-laden flows in a coaxial-jet combustor. *International Journal of Multiphase Flow*, **29**(8), 1311–1331.
- AUFFHAMMER, M., & KELLOGG, R. 2011. Clearing the air? The effects of gasoline content regulation on air quality. *The American Economic Review*, **101**(6), 2687–2722.
- BABU, P. C., & MAHESH, K. 2004. Upstream entrainment in numerical simulations of spatially evolving round jets. *Physics of Fluids*, **16**(10), 3699.
- BENNETT, M, CHRISTIE, S M, GRAHAM, A, GARRY, K P, VELIKOV, S, POLL, D I A, SMITH, M G, IQBAL, MEAD, POPOOLA, O A M, STEWART, G, & JONES, R L. 2013. Abatement of an aircraft exhaust plume using aerodynamic baffles. *Environ. Sci. Technol.*
- BEYER, R. P., & LEVEQUE, R. J. 1992. Analysis of a one-dimensional model for the immersed boundary method. *SIAM Journal on Numerical Analysis.*, **29**(2), 332–364.
- BOERSMA, J., BRETHOUWER, G., & NIEUWSTADT, F. T. M. 1998. A numerical investigation on the effect of the inflow conditions on the self-similar region of a round jet. *Physics of Fluids*, **16**(10), 899.
- BORGHI, R. 1975. Chemical reactions calculations in turbulent flows: application to a co-containing turbojet plume. *Advances in Geophysics*, **18**, 349–365.
- BRADSHAW, P. 1975. *An introduction to turbulence and its measurement*. Oxford: Pergamon Press.
- BRANCHER, P., CHOMAZ, J. M., & HUERRE, P. 1994. Direct numerical simulations of round jets: Vortex induction and side jets. *Physics of Fluids*, **6**, 1768–1774.

- BROGAARD, L., JACOBSEN, J. A., & LEMCHE, L. 2012. *AIR POLLUTION IN AIRPORTS*. The Danish Ecocouncil 77. Copenhagen Airports, Blegdamsvej 4B, 2200 Copenhagen N, Denmark.
- BROWN, G.L., & ROSHKO, A. 1971. On density effects and large structure in turbulent mixing layers. *Journal of Fluid Mechanics*, **64**, 775.
- BURATTINI, P., ANTONIA, R. A., & DANAILA, L. 2005. Similarity in the far field of a turbulent round jet. *Physics of Fluids*, **17**.
- BURTON, G. 2006. Large-eddy simulation of round turbulent jets using the Inertial LES method with multifractal subgrid-scale modeling. *10th IWPCTM PARIS (FRANCE)*, jul.
- CAA. 2011. Licensing of Aerodromes. *CAP 168*, Table 4.2. Aerodrome Standards Department, Safety Regulation Group, Civil Aviation Authority.
- CHAMPAGNE, F., & WYGNANSKI, I. 1971. An experimental investigation of coaxial turbulent jets. *International Journal of Heat and Mass Transfer*, **14**(9), 1445–1464.
- CHEN, J., KATZ, J., & MENEVEAU, C. 2005. Implication of mismatch between stress and strain-rate in turbulence subjected to rapid straining and destrain-ing on dynamic LES models. *Journal of Fluids Engineering-Transactions of the ASME*, **127**(5), 840–850.
- CHIANG, C.M. 2007. *Inviscid Instability Mechanism of Parallel Flows*. Mas-sachusetts Institute of Technology: Lecture Notes on Fluid Dynamics.
- CHIGIER, N.A., & BEER, J.M. 1964. The flow region near the nozzle in double concentric jets. *Trans. ASME, vol. 86D, J. Basic Eng*, 779–804.
- CITRINITI, J. H., & GEORGE, W. K. 2000. Reconstruction of the global velocity field in the axisymmetric mixing layer utilizing the proper orthogonal decom-position. *Journal of Fluid Mechanics*, **418**(2000), 137–166.

- COHEN, J., & WYGNANSKI, I. 1987. The evolution of instabilities in the axisymmetric jet. Part 1. The linear growth of disturbances near the nozzle. *Journal of Fluid Mechanics*, **176**(Mar.), 191–219.
- CRAFT, T. J., & LAUNDER, B. E. 2001. On the spreading mechanism of the three-dimensional turbulent wall jet. *Journal of Fluid Mechanics*, **435**(1), 305–326.
- CROW, S.C., & CHAMPAGNE, F.H. 1971. Orderly structure in jet turbulence. *Journal of Fluid Mechanics*, **48**(03), 547.
- DAHM, W.J.A., FRIELER, C.E., & TRYGGVASON, G. 1992. Vortex structure and dynamics in the near field of a coaxial jet. *Journal of Fluid Mechanics*, **241**, 371–402.
- DAVIDSON, P.A. 2004. *Turbulence : an introduction for scientists and engineers*. Oxford: Oxford University Press.
- DAVIS, MR, & WINARTO, H. 1980. Jet diffusion from a circular nozzle above a solid plane. *Journal of Fluid Mechanics*, **101**(1), 201–221.
- DEJOAN, A, & LESCHZINER, M A. 2005. Large eddy simulation of a plane turbulent wall jet. *Physics of Fluids*, **17**(2).
- DFT, UK. 2006. *Project for the Sustainable Development of Heathrow*. Tech. rept. Department for Transport, London SW1P 4DR, UK.
- DINESH, K. K. J. R., GARRY, K. P., HOLT, J. C., KIRKPATRICK, M. P., POLL, D. I. A., & SAVILL, A. M. 2009. Appendix 2: Technical Report. Study of turbulent coaxial jet efflux mixing using LES [in, Final Report: Dispersion of Aircraft Efflux in Proximity to Airports]. *OMEGA*.
- DINESH, K. K. J. R., SAVILL, A. M., JENKINS, K. W., & KIRKPATRICK, M. P. 2010. A study of mixing and intermittency in a coaxial turbulent jet. *Fluid Dynamics Research*, **42**(2).

- DINESH, K. K. R., SAVILL, A. M., GARRY, K. P., HOLT, J. C., POLL, D. I. A., & KIRKPATRICK, M. P. 2012. Modelling of Coaxial Jet Efflux Mixing using LES. *International Journal of Fluid Mechanics Research*, **39**(1), 20–39.
- DUCROS, F., NICOUD, F., & POINSOT, T. 1998. Wall-adapting local eddy-viscosity models for simulations in complex geometries. *In Proceedings of 6th ICFD Conference on Numerical Methods for Fluid Dynamics*, 293–299.
- DURAO, D., & WHITELAW, J.H. 1973. Turbulent Mixing in the Developing Region of Coaxial Jets. *Trans ASME, J. Fluids Engng*, **95**, 467–473.
- EASTWOOD, S, TUCKER, P, XIA, H, & CARPENTER, P. 2008. Comparison of LES to LDA and PIV Measurements of a Small Scale High Speed Coflowing Jet. *AIAA Paper*, 1–12.
- ERIKSSON, J. G., KARLSSON, R. I., & PERSSON, J. 1998. An experimental study of a two-dimensional plane turbulent wall jet. *Experiments in Fluids*, **25**(1), 50–60.
- EWING, D., & POLLARD, A. 1997. Evolution of the large scale motions in a three-dimensional wall jet. *In Proceedings of the 28th AIAA Fluid Dynamics Conference/Fourth AIAA Shear Flow Control Conference*.
- FISCHER, H. B., LIST, E. J., KOH, R. C. Y., IMBERGER, J., & BROOKS, N. H. 1979. Mixing in Inland and Coastal Waters. *Academic Press, San Diego (USA)*.
- FLUENT, ANSYS. 2012. 12.0 Theory Guide. *Ansys Inc 5*.
- FOIAS, C, MANLEY, O, ROSA, R, & TEMAM, R. 2001. *Navier-Stokes Equations and Turbulence*. Vol. 3. Cambridge: Cambridge University Press.
- FORSTALL, W., & SHAPIRO, A.H. 1950. Momentum and mass transfer in coaxial gas jets. *Transactions of ASME Journal of Applied Mechanics*, **17**, 399–408.
- FÖRTHMANN, E. 1936. *Turbulent Jet Expansion*. Tech. rept. 789. National Advisory Committee of Aeronautics.

- FREYMUTH, P. 1966. On transition in a separated laminar boundary layer. *Journal of Fluid Mechanics*, **25**(04), 683.
- FRÖHLICH, J., GARCÍA, M. V., & RODI, W. 2007. Scalar Mixing and Large-Scale Coherent Structures in a Turbulent Swirling Jet. *Flow, Turbulence and Combustion*, **80**(1), 47–59.
- GAMARD, S., JUNG, D., & GEORGE, W. K. 2004. Downstream evolution of the most energetic modes in a turbulent axisymmetric jet at high Reynolds number. Part 2. The far-field region. *Journal of Fluid Mechanics*, **514**, 205–230.
- GANIER, F., BAUDOIN, C., & LOUISNARD, N. 1997a. Engine emission alteration in the near field of an aircraft. *Atmospheric environment*, **31**(12), 1767–1781.
- GANIER, F., BRUNET, S., & JACQUIN, L. 1997b. Modelling exhaust plume mixing in the near field of an aircraft. *Springer-Verlag; In Annales Geophysicae*, **15**(11), 1468–1477.
- GAO, N., & EWING, D. 2007. Experimental Investigation of Planar Offset Attaching Jets With Small Offset Distances. *Exp. Fluids*, **42**, 941–954.
- GAO, N., & EWING, D. 2008. On the Phase Velocities of the Motions in an Offset Attaching Planar Jet. *J. Turbulence*, **9**(27), 1–21.
- GARRY, K P, & HOLT, J. 2008. Modelling the dispersion of aircraft engine efflux in proximity to airports in an atmospheric boundary layer tunnel. *Cranfield OMEGA Project Report*.
- GASTER, M., KIT, E., & WYGNANSKI, I. 1985. Large-scale structures in a forced turbulent mixing layer. *Journal of Fluid Mechanics*, **150**, 23.
- GERMANO, MASSIMO, PIOMELLI, UGO, MOIN, PARVIZ, & CABOT, WILLIAM H. 1991. A dynamic subgrid-scale eddy viscosity model. *Physics of Fluids A: Fluid Dynamics*, **3**(7), 1760.
- GILLILAND, T., DINESH, K. K. J. R., FAIRWEATHER, M., FALLE, S. A. E. G., JENKINS, K. W., & SAVILL, A. M. 2012. External Intermittency Simulation in

- Turbulent Round Jets. *Springer: Flow Turbulence Combust*, **89**(sep), 385–406.
- GIOVANNI, BISIGNANI. 2011 (feb). *Vision 2050*. Tech. rept. IATA, Singapore.
- GLAUERT, M. B. 1956. The wall-jet. *Journal of Fluid Mechanics*, **1**(6), 625–643.
- GRAHAM, A., BENNETT, M., & CHRISTIE, S. 2008. *Representing the dispersion of emissions from aircraft on runways*. Report 43(43/2), 563–568. Hrvatski meteoroloski casopis.
- GRINSTEIN, F.F. 2009. On integrating large eddy simulation and laboratory turbulent flow experiments. *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences*, **367**(1889), 2931–2945.
- GUTMARK, E., & HO, C. M. 1983. Preferred modes and the spreading rates of jets. *Physics of fluids*, **26**, 2932.
- HALL, J. W., & EWING, D. 2007. The Asymmetry of the Large-Scale Structures in Turbulent Three-Dimensional Wall Jets Exiting Long Rectangular Channels. *ASME J. Fluids Eng.*, **129**(7), 929–941.
- HALL, J. W., & EWING, D. 2010. Spectral Linear Stochastic Estimation of the Turbulent Velocity in a Square Three-Dimensional Wall Jet. *ASME J. Fluids Eng.*, **132**(5), 0512031–0512039.
- HAMMERSLEY, R.J. 1969. An experimental investigation of the turbulent characteristics of coaxial jet flows and their role in aerodynamic noise generation. *Ph. D. Thesis; University of Illinois*.
- HANNA, S. R., BRIGGS, G. A., & CHANG, J. C. 1998. Lift-off of ground-based buoyant plumes. *Journal of hazardous materials.*, **59**(2), 123–130.
- HAQ. 2011. *Heathrow Air Quality Strategy 2011-2020*. Tech. rept. Heathrow.
- HUSSEIN, H. J., CAPP, S. P., & GEORGE, W.K. 1994. Velocity measurements in a high-Reynolds-number, momentum-conserving, axisymmetric, turbulent jet. *J. Fluid Mech.*, **258**, 31–75.

- ILYUSHIN, B.B., & KRASINSKY, D.V. 2006. Large eddy simulation of the turbulent round jet dynamics. *Thermophysics and Aeromechanics*, **13**(1), 43–54.
- KADER, B A. 1981. Temperature and concentration profiles in fully turbulent boundary layers. *International Journal of Heat and Mass Transfer*, **24**, 1541–1544.
- KIM, S E. 2004. Large Eddy Simulation Using an Unstructured Mesh Based Finite-Volume Solver. *Fluid Dynamics*.
- KLEIN, M., SADIKI, A., & JANICKA, J. 2003. A digital filter based generation of inflow data for spatially developing direct numerical or large eddy simulations. *Journal of Computational Physics*, **186**(2), 652–665.
- KOSO, T., & OHASHI, H. 1982. Turbulent diffusion of a three-dimensional wall jet: 1st report mean and turbulent characteristics of velocity and temperature field. *Bulletin of JSME*, **25.200**, 173–181.
- KOUTSOURAKIS, N, BARTZIS, JG, & VENETSANOS, A. 2006. Computation of pollutant dispersion during an airplane take-off. *Environmental Modelling & Software*, **21**, 486–493.
- KRAICHNAN, R. 1970. Diffusion by a Random Velocity Field. *Physics of Fluids*, **11**, 21–31.
- KUDELA, H., & PAWEL, R. 2004. Study of vortex rings interaction by 3D vorticity particle-in-cell method. *Mechanics of 21st Century ICTAM04*, 15-21 August.
- LAI, M. C., & PESKIN, C. S. 2000. An immersed boundary method with formal second-order accuracy and reduced numerical viscosity. *Journal of Computational Physics*, **160**(2), 705–719.
- LAUNDER, B. E., & RODI, W. 1981. The turbulent wall jet. *Progress in Aerospace Sciences*, **19**, 81–128.
- LAUNDER, B. E., & RODI, W. 1983. The turbulent wall jet - measurements and modelling. *Physics of Fluids*, **15**, 429–459.

- LAW, A. W. K. 2002. Measurements of turbulent mass transport of a circular wall jet. *International Journal of Heat and Mass Transfer*, **45**(24), 4899–4905.
- LEE, J. H. W., & CHU, V. H. 2003. *Turbulent jets and plumes: a Lagrangian approach*. Springer.
- LESIEUR, M. 2007. *Turbulence in fluids, 4th edition*. London: Springer.
- LIEPMANN, D., & GHARIB, M. 1992. The role of streamwise vorticity in the near-field entrainment of round jets. *Journal of Fluid Mechanics*, **245**, 643–668.
- LILLY, D. K. 1992. A proposed modification of the Germano closure method. *Physics of Fluids A: Fluid Dynamics*, **4**, 633–635.
- LIPARI, G., & STANSBY, P. K. 2011. Review of experimental data on incompressible turbulent round jets. *Flow, turbulence and combustion*, **87**(1), 79–114.
- LIST, E. J. 1982. Turbulent jets and plumes. *Annu. Rev. Fluid Mech.*, **14**, 189–212.
- LIU, Y., TUCKER, P. G., & KERR, R. M. 2008. Linear and nonlinear model large-eddy simulations of a plane jet. *Computers & Fluids*, **37**(4), 439–449.
- LUBBERS, L., BRETHOUWER, G., & BOERSMA, B. J. 2001. Simulation of the mixing of a passive scalar in a round turbulent jet,. *Fluid Dyn. Res.*, **28**, 189.
- MAHESH, K., HOU, Y., & BABU, P. 2007. Three Problems in the Large-Eddy Simulation of Complex Turbulent Flows. *In Complex Effects in Large Eddy Simulations: Springer Berlin Heidelberg*, 99–115.
- MATSUDA, H., IIDA, S. I., & HAYAKAWA, M. 1990. Coherent structures in a three-dimensional wall jet. *Journal of fluids engineering*, **112**(4), 462–467.
- MCLWAIN, S., & POLLARD, A. 2002. Large eddy simulation of the effects of mild swirl on the near field of a round free jet. *Physics of Fluids*, **14**(2), 653.

- MCLEAN, J. D., & HERRING, H. J. 1974 (May). *Discrete wall jets in quiescent air*. Contractor Report 2388. National Aeronautics And Space Administration (NASA), Washington, D. C.
- MENEVEAU, C. 2012. Germano identity-based subgrid-scale modeling: a brief survey of variations on a fertile theme. *Phys. Fluids*, **24**.
- MENEVEAU, C., O'NEIL, J., PORTÉ-AGEL, F., CERUTTI, S., & PARLANGE, M.B. 1999. *Fundamental Problematic Issues in Turbulence; Physics and modeling of small scale turbulence for large eddy simulation*. Springer.
- MI, J., NOBES, D. S., & NATHAN, G. J. 2001. Influence of jet exit conditions on the passive scalar field of an axisymmetric free jet. *J. Fluid Mech.*, **432**, 91–125.
- MICHALKE, A. 1965. On spatially growing disturbances in an inviscid shear layer. *Journal of Fluid Mechanics*, **23**(03), 521.
- MITTAL, R., & IACCARINO, G. 2005. Immersed boundary methods. *Annu. Rev. Fluid Mech.*, **37**, 239–261.
- MMU. 2013 (Oct.). *OMEGA accessible at: <http://www.cate.mmu.ac.uk/omega-overview/>*.
- NAMGYAL, L., & HALL, J. W. 2013. Coherent Streamwise Vortex Structures in the Near-Field of the Three-Dimensional Wall Jet. *J. Fluids Eng.*, **135**(6), 7pages.
- NASR, A., & LAI, J. C. S. 1997. Comparison of Flow Characteristics in the Near Field of Two Parallel Plane Jets and an Offset Plane Jet. *Phys. Fluids*, **9**(10), 2919–2931.
- NASR, A., & LAI, J. C. S. 1998. A turbulent plane offset jet with small offset ratio. *Experiments in fluids*, **24**(1), 47–57.

- NEWMAN, B G, PATELL, R P, SAVAGE, S B, & TJIO, H K. 1972. Three-Dimensional Wall Jet Originating from a Circular Orifice. *Aero. Quart*, 188–200.
- NICOUD, F. 1999. Subgrid-scale stress modelling based on the square of the velocity gradient tensor. *Flow, Turbulence and Combustion*, 1–36.
- ONLINE. 2011 (Jan.). *2Eyes-of-the-Vortex.jpg*. Image accessible at: <http://www.artfulplayground.com/wp-content/uploads/2009/11>.
- ONLINE. 2013 (Oct.). *Taxiing of Transporters*: <http://sciencera.com/technology>.
- OR, C. M., LAM, K. M., & LIU, P. 2011. Potential core lengths of round jets in stagnant and moving environments. *Journal of Hydro-environment Research*, **5**(2), 81–89.
- PADMANABHAM, G., & GOWDA, B. H. LAKSHMANA. 1991. Mean and turbulence characteristics of a class of three-dimensional wall jets. II: Turbulence characteristics. *Journal of fluids engineering*, **113**(4), 629–634.
- PANCHAPAKESAN, N.R., & LUMLEY, J.L. 1993. Turbulence measurements in axisymmetric jet of air and helium. Part 1. Air jet. *J. Fluid Mech.*, **246**, 197–223.
- PELFREY, J. R. R., & LIBURDY, J. A. 1986a. Effect of Curvature on the Turbulence of a Two-Dimensional Jet. *Exp. Fluids*, **4**, 143–149.
- PELFREY, J. R. R., & LIBURDY, J. A. 1986b. Mean flow characteristics of a turbulent offset jet. *ASME Transactions Journal of Fluids Engineering*, **108**, 82–88.
- PESKIN, CHARLES S. 1972. *Flow patterns around heart valves: a digital computer method for solving the equations of motion*. Ph.D. thesis, Sue Golding Graduate Division of Medical Sciences, Albert Einstein College of Medicine, Yeshiva University.

- PETZOLD, A., STRÖM, J., SCHRDER, F. P., & KÄRCHER, B. 1999. Carbonaceous aerosol in jet engine exhaust: emission characteristics and implications for heterogeneous chemical reactions. *Atmospheric Environment*, **33**(17), 2689–2698.
- POLL, D I A, GARRY, K P, & SAVILL, M. 2009. Final Report: Dispersion of aircraft efflux in proximity to airports. *OMEGA*.
- POPE, S. B. 2000. *Turbulent Flows*. Cambridge: Cambridge University Press.
- RAMSDALE, S. A., & TICKLE, G. A. 2001. *Review of Lift-off Models for Ground Based Buoyant Clouds*. Report AEAT-4262 (2). AEA Technology Report for EC URAHFREP Project.
- RANDALL, D. A. 2006. Reynolds averaging. *Colorado State University, Fort Collins*.
- REBA, IMANTS. 1966. Applications of the Coanda Effect. *Scientific American*, **214**, 84–92.
- RIBAUT, C., & SARKAR, S. 2001. Large eddy simulation of evolution of a passive scalar in plane jet. *AIAA journal*, **39**(8), 1509–1516.
- RODI, W. 1975. A review of experimental data of uniform density free turbulent boundary layers. *Studies in Convection*.
- SADR, R., & KLEWICKI, J.C. 2003. An experimental investigation of the near-field flow development in coaxial jets. *Physics of Fluids*, **15**(5), 1233.
- SAIKI, E. M., & BIRINGEN, S. 1996. Numerical simulation of a cylinder in uniform flow: application of a virtual boundary method. *Journal of Computational Physics*, **123**(2), 450–465.
- SALIM, S M, ONG, K C, & CHEAH, S C. 2011. Comparison of RANS, URANS and LES in the prediction of airflow and pollutant dispersion. *Proceedings of the World Congress on Engineering and Computer Science*, **II**.

- SALINAS-VAZQUEZ, M, VICENTE, W, ESPINOSA, A., & BARRIOS, E. 2005. Large eddy simulation of an ammonia jet. *American Journal of Applied Sciences*, **2**(8), 1270–1273.
- SCHLENKER, W., & WALKER, W. R. 2011. Airports, air pollution, and contemporaneous health. *National Bureau of Economic Research*.
- SCHMID, P.J., & HENNINGSON, D.S. 2001. *Stability and transition in shear flows*. Vol. 142. New York: Springer.
- SCHNEIDER, M. E., & GOLDSTEIN, R. J. 1994. Laser Doppler measurement of turbulence parameters in a two-dimensional plane wall jet. *Phys Fluids* **6**, 3116–3129.
- SCHWARZ, W. H., & COSART, W. P. 1961. The two-dimensional turbulent wall-jet. *Journal of Fluid Mechanics*, **10**(04), 481–495.
- SFORZA, P.M., & HERBST, G. 1967. *PIBAL report: A Study of Three-dimensional, Incompressible, Turbulent Wall Jets*. Polytechnic Institute of Brooklyn, Department of Aerospace Engineering and Applied Mechanics.
- SHIRI, A., GEORGE, W. K., & NAUGHTON, J. W. 2008. Experimental study of the far field of incompressible swirling jets. *AIAA J.*, **46**(8), 2002–2009.
- SIVAKUMAR, S., SANGRAS, R., & RAGHAVAN, V. 2012. Characteristics of turbulent round jets in its potential-core region. *International Journal of Mechanical and Aerospace Engineering*, **6**, 119–125.
- SMAGORINSKY, J. 1963. General Circulation Experiments With The Primitive Equations. *Monthly Weather Review*, **91**(3), 99–164.
- SMIRNOV, A., & SHI, S. 2001. Random flow generation technique for large eddy simulations and particle-dynamics modeling. *Journal of Fluids Engineering*, **123**, 359.
- SPANELIS, A., GARRY, K. P., & SAVILL, A. M. 2013. Large Eddy Simulations of Turbofan Engine Exhaust Plume Dispersion on Take-Off Runway. *ASME*

- Turbo Expo 2013, San Antonio, USA, Conference Proceedings*, Paper No. GT2013–95006.
- STOCKIE, J. M., & WETTON, B. R. 1999. Analysis of stiffness in the immersed boundary method and implications for time-stepping schemes. *Journal of Computational Physics*, **154**(1), 41–64.
- SUN, H., & EWING, D. 2002. Development of the Large-Scale Structures in the Intermediate Region of the Three-Dimensional Wall Jet. *Proceedings of the Fluids Engineering Division Summer Meeting, Montreal, QC, Canada, ASME*, Paper No. FEDSM2002–31414.
- TESTA, E., GIAMMUSSO, C., BRUNO, M., & MAGGIORE, P. 2012. Fluid dynamic analysis of pollutants dispersion behind an aircraft engine during idling. *Air Quality, Atmosphere & Health*, 1–17.
- URBIN, G., & MÉTAIS, O. 1997. *Direct and Large-Eddy Simulation II; Large-eddy simulations of three-dimensional spatially-developing round jets*. Springer Netherlands.
- VELIKOV, S. 2013. Private communication. *PhD Thesis, School of Engineering, Cranfield University*.
- VERSTEEG, H. K., & MALALASEKERA, W. 1995. *An Introduction to computational fluid dynamics: The finite volume method*. Harlow: Pearson Education Limited.
- VOLKOV, K. N. 2008. Simulation of a subsonic isothermal turbulent submerged jet flowing from an annular nozzle. *Journal of Engineering Physics and Thermophysics*, **81**(2), 312–320.
- WAITZ, IAN, TOWNSEND, JESSICA, CUTCHER-GERSHENFELD, JOEL, GREITZER, EDWARD, & KERREBROCK, JACK. 2004 (December). *A National Vision Statement, Framework for Goals and Recommended Actions*. Report to the United States Congress 77. Massachusetts Institute of Technology, Massachusetts Avenue.

- WEISGRABER, T. H., & LIEPMANN, D. 1998. Turbulent structure during transition to self-similarity in a round jet. *Exp. Fluids*, **24**, 210–224.
- WILLIAMS, T.J., ALI, M. R. M. H., & ANDERSON, J.S. 1969. Noise and flow characteristics of coaxial jets. *J. Mech. Eng. Sci*, **11**(2), 133–142.
- WINANT, CD. 1974. Vortex pairing- The mechanism of turbulent mixing-layer growth at moderate Reynolds number. *Journal of Fluid Mechanics*, **63**, 237–255.
- WINTER, K. G., & GAUDET, L. 1973. *Turbulent boundary-layer studies at high Reynolds numbers at Mach numbers between 0.2 and 2.8*. Technical Report 70251. Royal Aeronautical Establishment, Bedford, UK.
- WOLFE, C. 2009. The Immersed Boundary Approach to Fluid Flow Simulation: a rapid, reliable and automated approach simulates complex flow fields. *ANSYS Advantage 2009 ANSYS ANALYSIS TOOLS*, accessible at: <http://www.ansys.com/staticassets/ANSYS/staticassets/resourcelibrary/article/AA-V3-I2-The-Immersed-Boundary-Approach.pdf>.
- WYGNANSKI, I., & FIEDLER, H. 1969. Some measurements in the self-preserving jet. *J. Fluid Mech.*, **38**(3), 79–114.
- WYGNANSKI, I., KATZ, Y., & HOREV, E. 1992. On the applicability of various scaling laws to the turbulent wall jet. *J Fluid Mech*, **234**, 669–690.
- XU, G., & ANTONIA, R. 2002. Effect of different initial conditions on a turbulent round free jet. *Exp. Fluids*, **33**(5), 31–75.
- XU, H., KHALID, M., & POLLARD, A. 2003. Large eddy simulation of turbulent flow in a confined square coaxial jet. *International Journal of Computational Fluid Dynamics*, **17**(5), 339–356.
- ZEMTSOP, C. P., STOELLINGER, M. K., HEINZ, S., & STANESCU, D. 2009. Large eddy simulation of swirling turbulent jet flows. *AIAA Journal*, **47**.

- ZHI-WEI, LI, HUAI, WEN-XIN, & QIAN, ZHONG-DONG. 2012. Large eddy simulation of flow structure in the near region of a circular wall jet. *Procedia Engineering*, **31**(Jan.), 713–722.
- ZHOU, XU, & LUO, KH. 2001. Large-eddy simulation of a turbulent forced plume. *European Journal of Mechanics*, **20**, 233–254.

Appendices

Appendix A

Free Shear Validation Tests

Cross-Sectional Profiles

The present Appendix includes all the axisymmetric jet cross-sectional profile comparisons of the 12 studies of the free-shear validation tests discussed in chapter 7. Description of the investigated setups corresponding to these results is available in sections 6.1.1 and 6.1.2, corresponding to a single circular and a coaxial jet respectively.

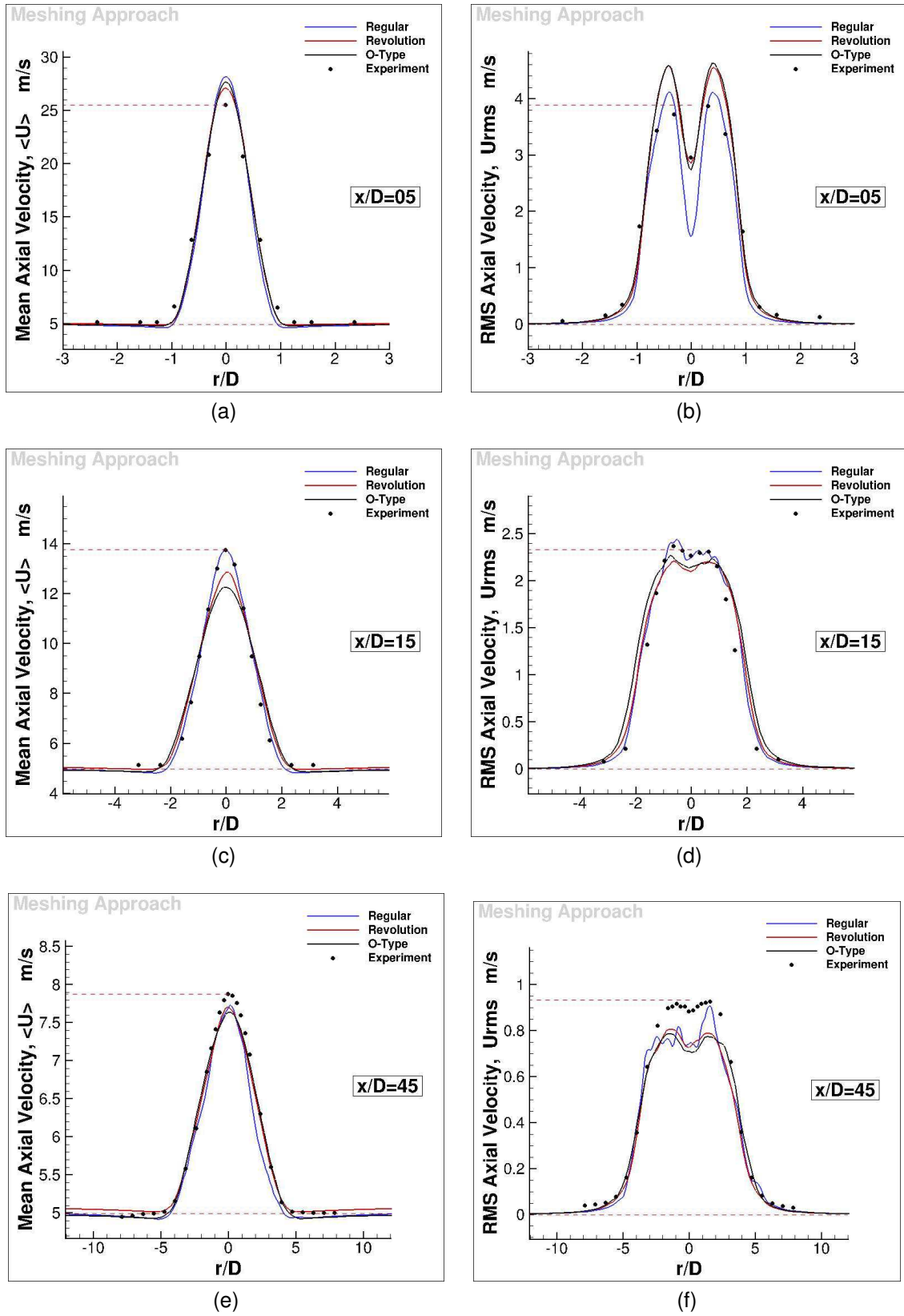


Figure A.1: Profiles of the study "Meshing Approach".

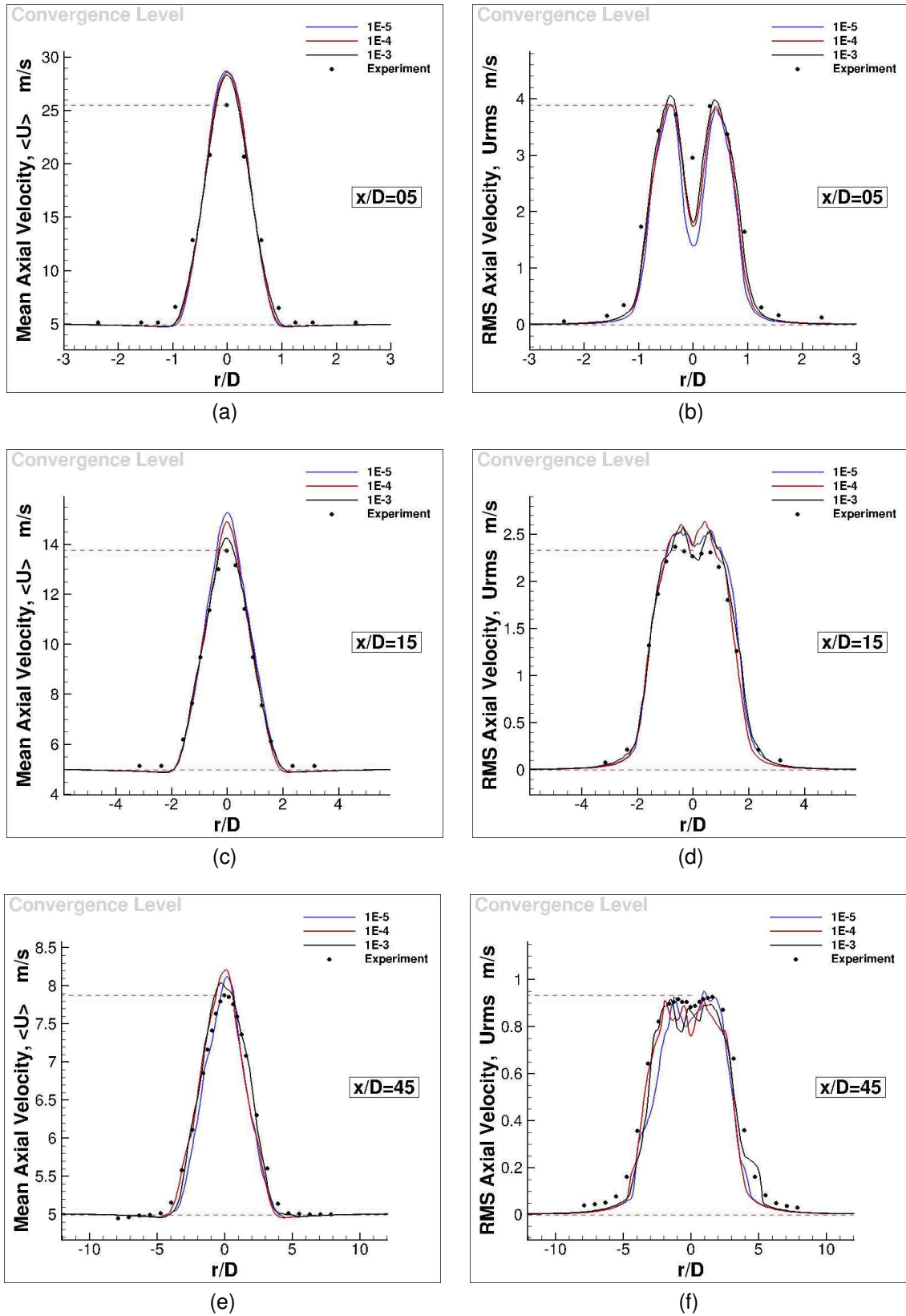


Figure A.2: Profiles of the study “Convergence Criteria”.

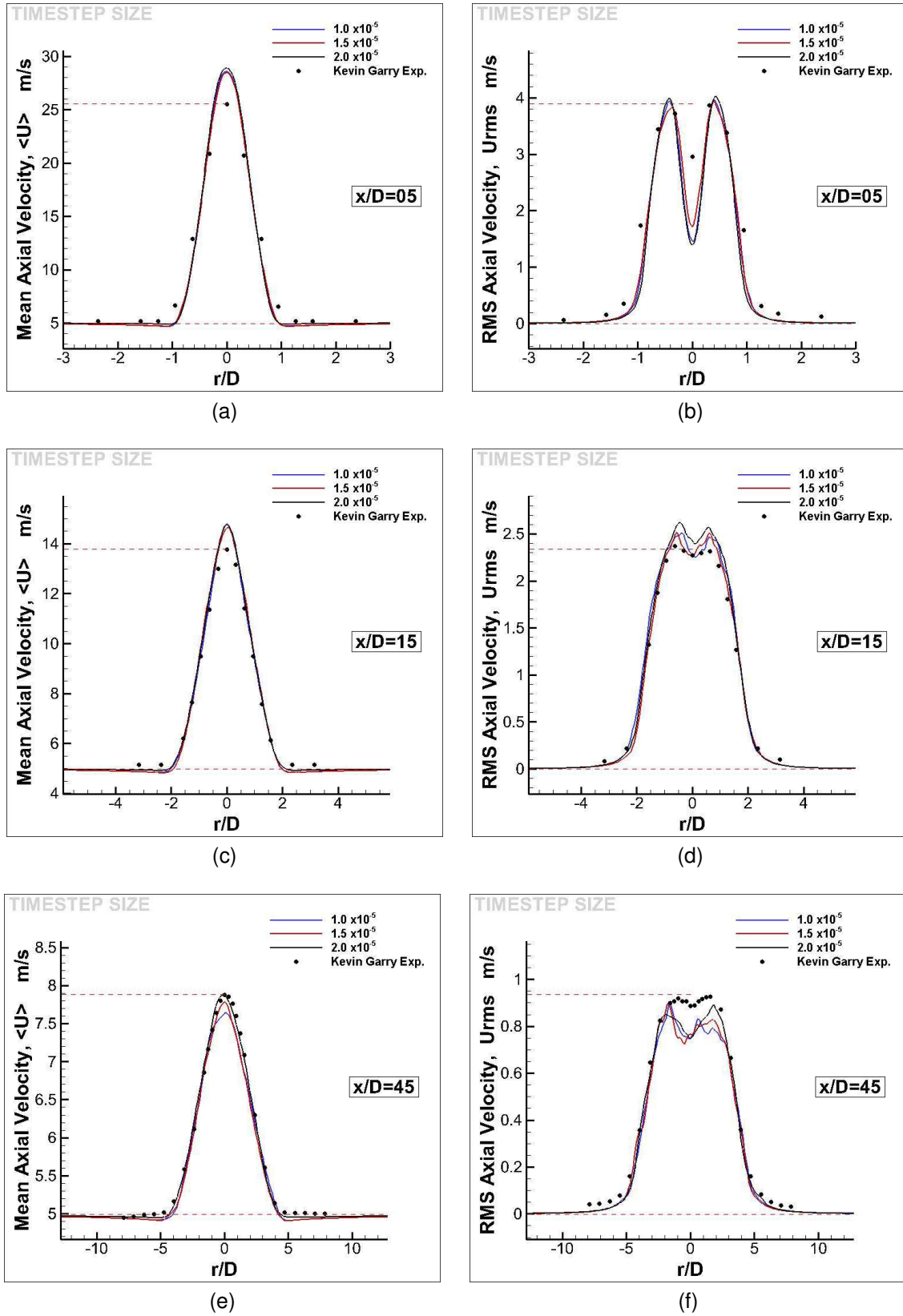


Figure A.3: Profiles of the study "Timestep Size".

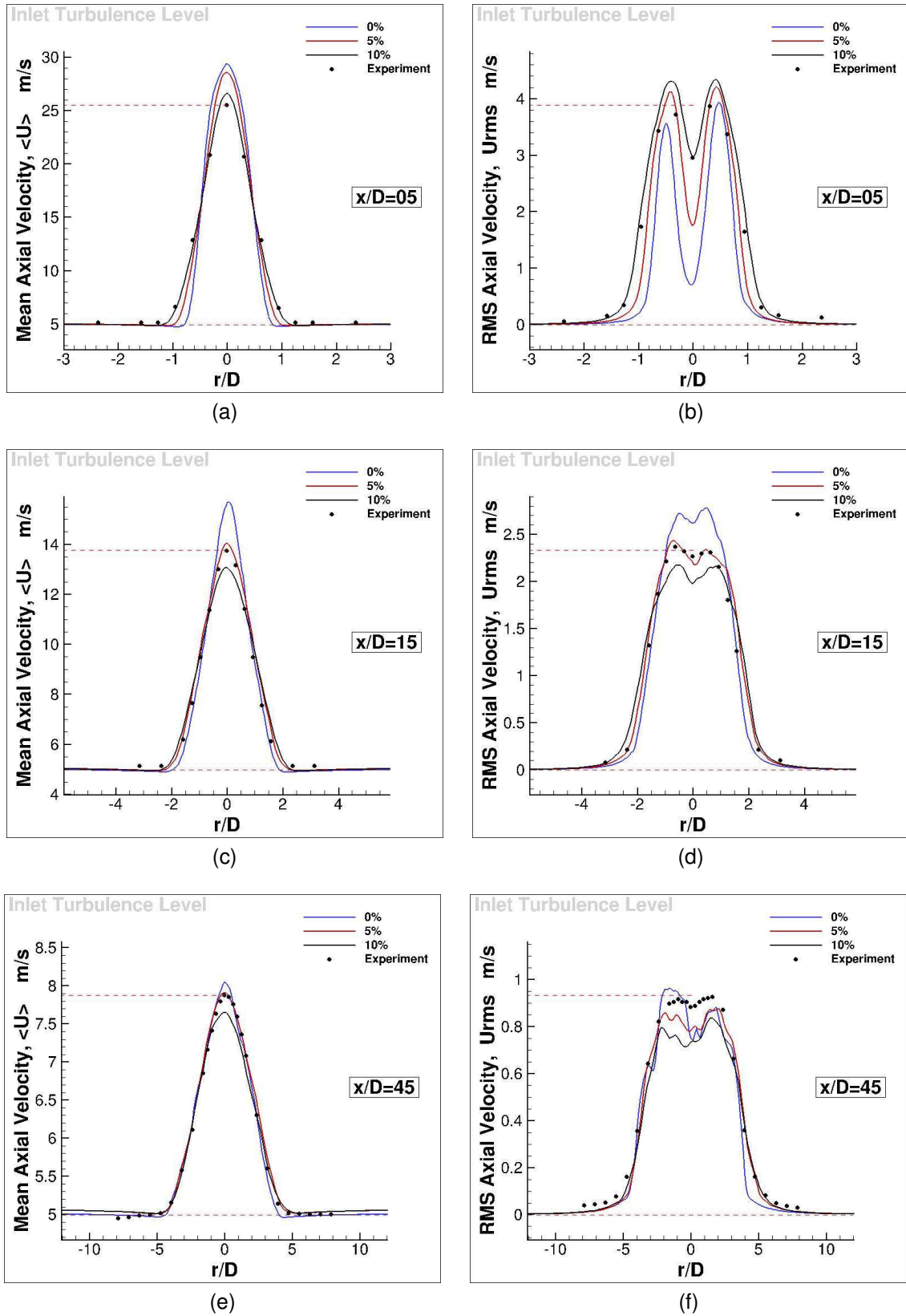
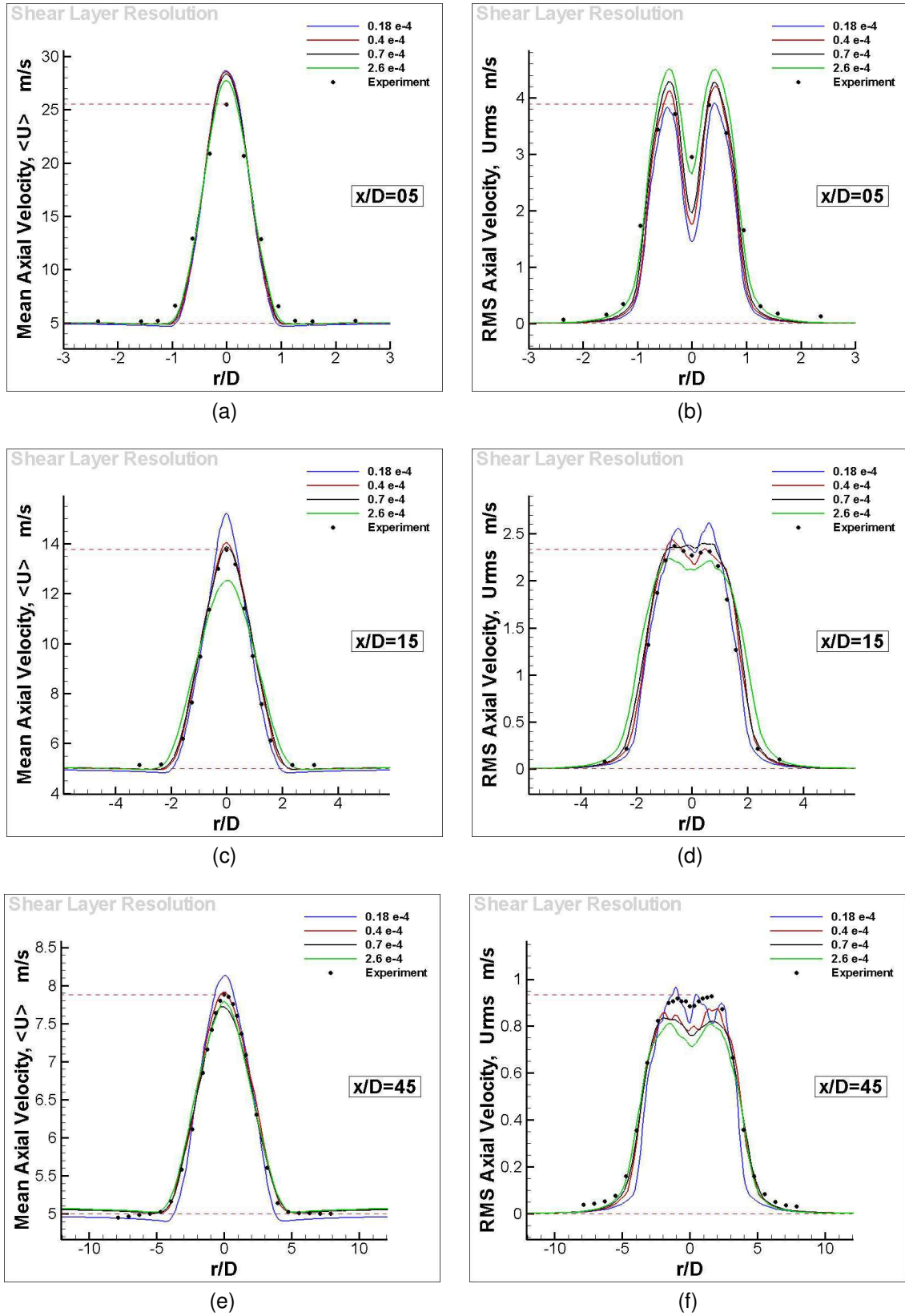
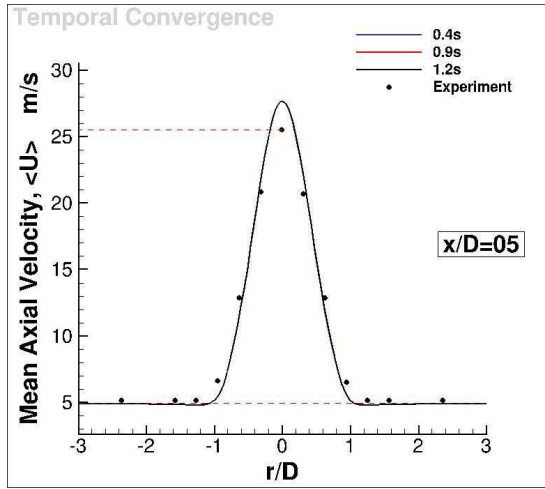
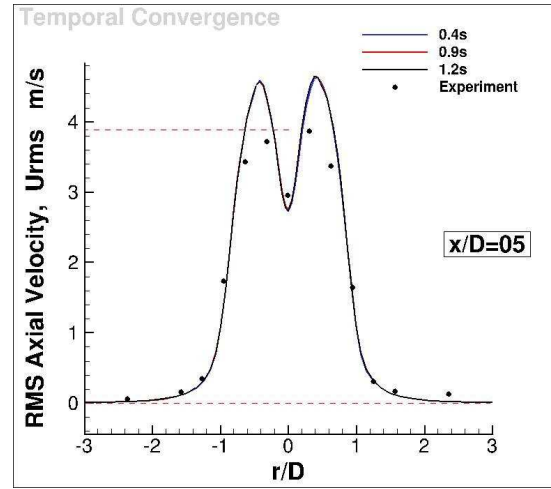


Figure A.4: Profiles of the study “Inlet Turbulence Intensity”.

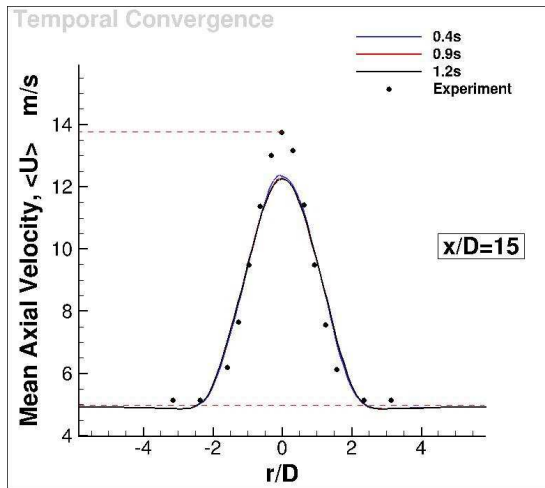
Figure A.5: Profiles of the study “*Shear Layer Resolution*”.



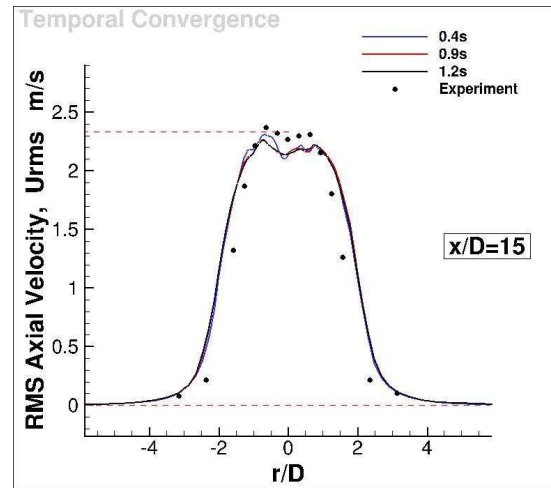
(a)



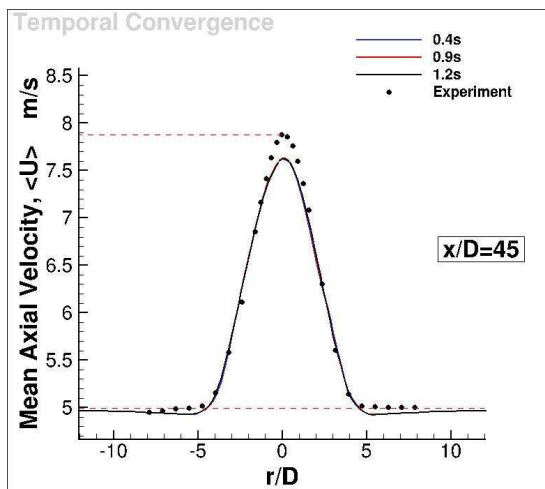
(b)



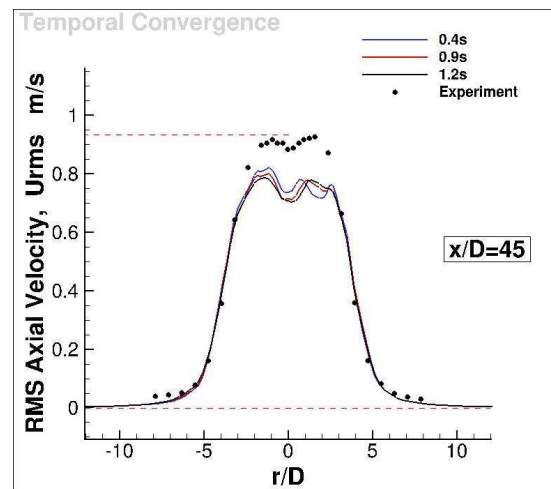
(c)



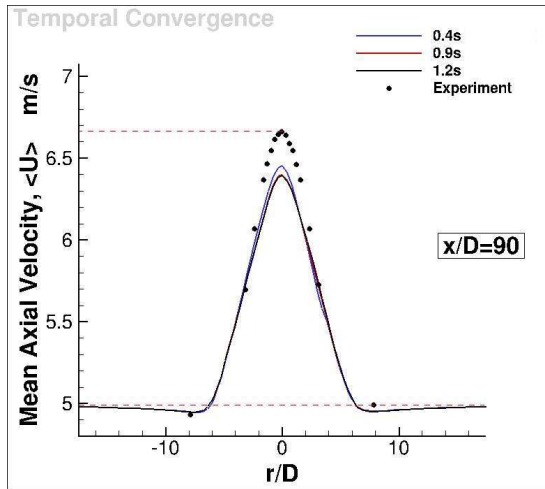
(d)



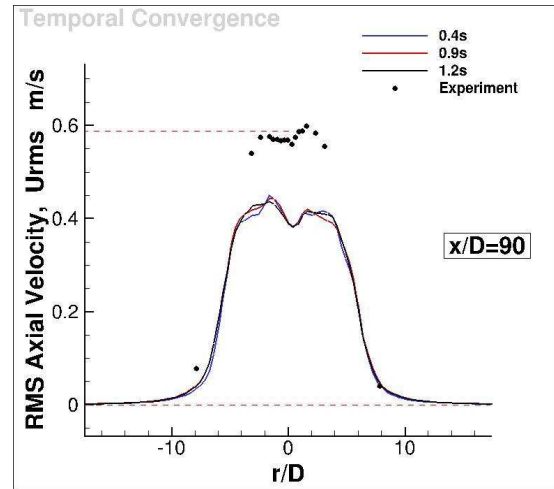
(e)



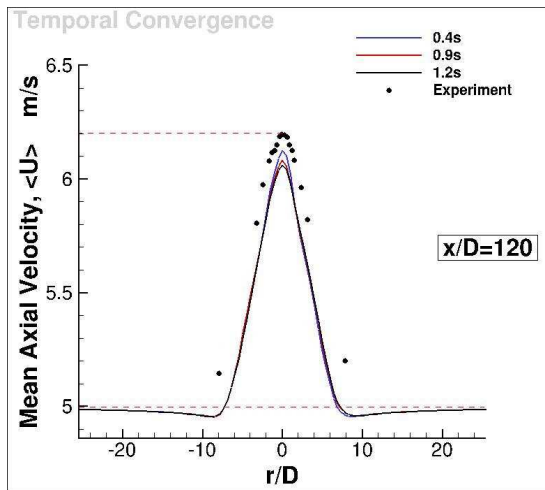
(f)



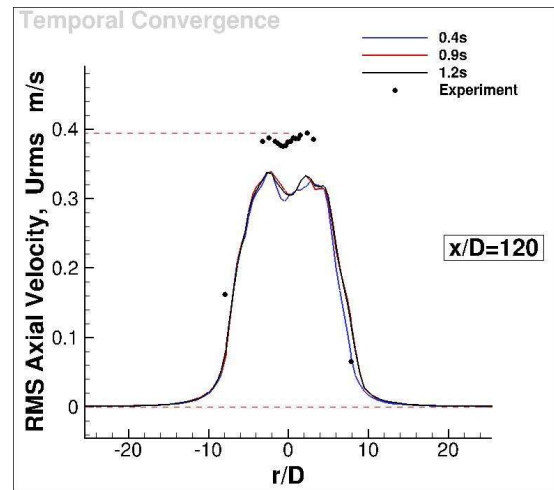
(g)



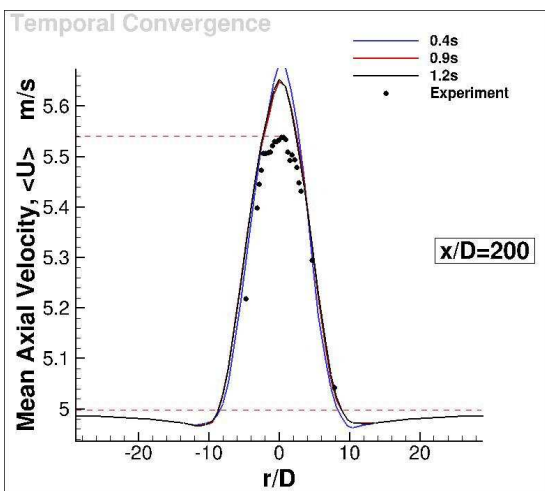
(h)



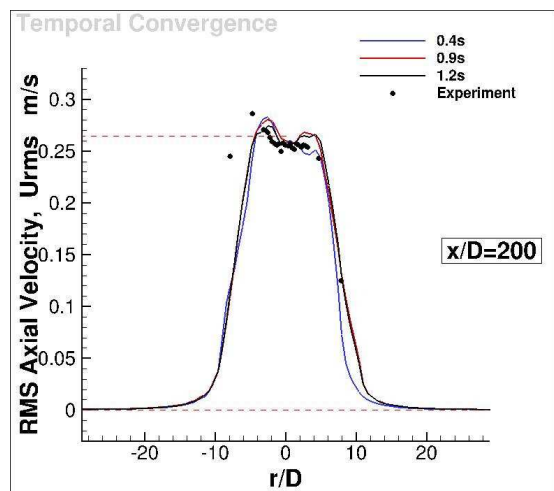
(i)



(j)

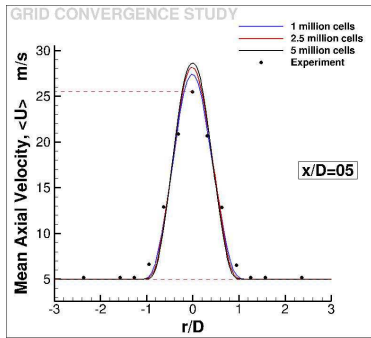


(k)

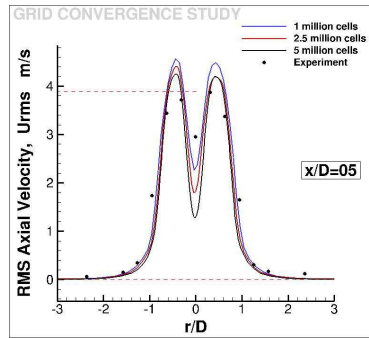


(l)

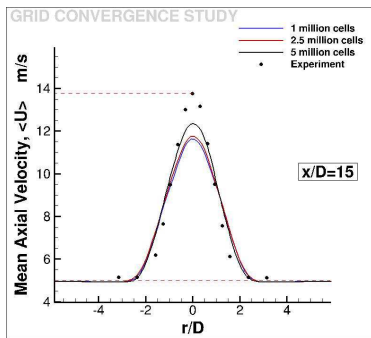
Figure A.6: Profiles of the study "Temporal Convergence".



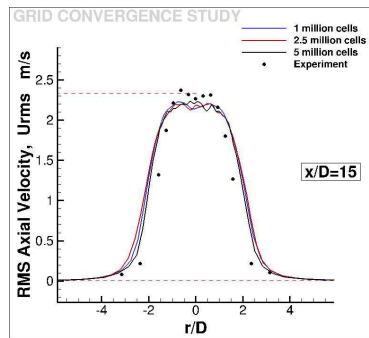
(a)



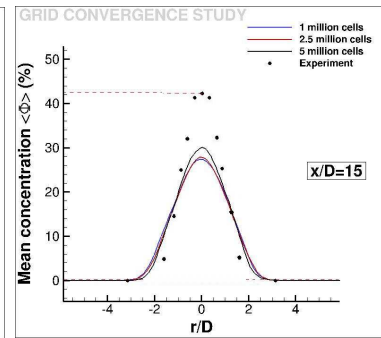
(b)



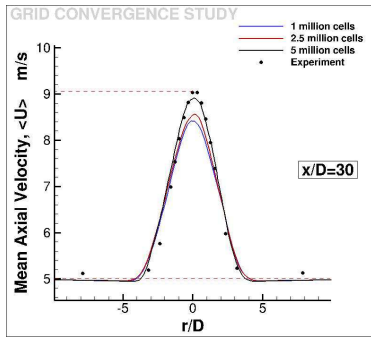
(c)



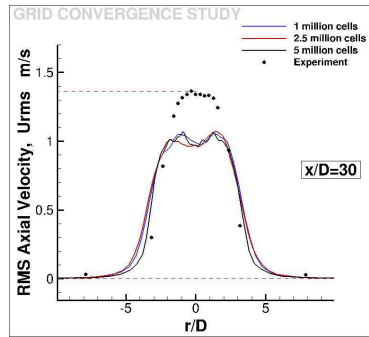
(d)



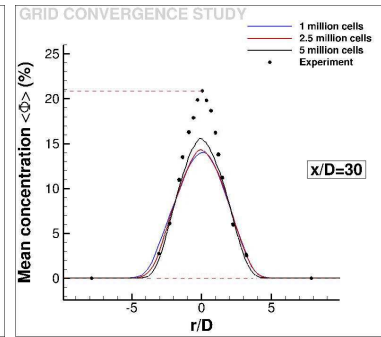
(e)



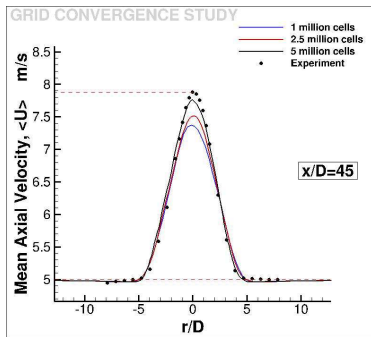
(f)



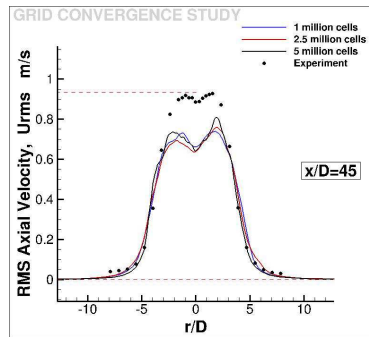
(g)



(h)



(i)



(j)

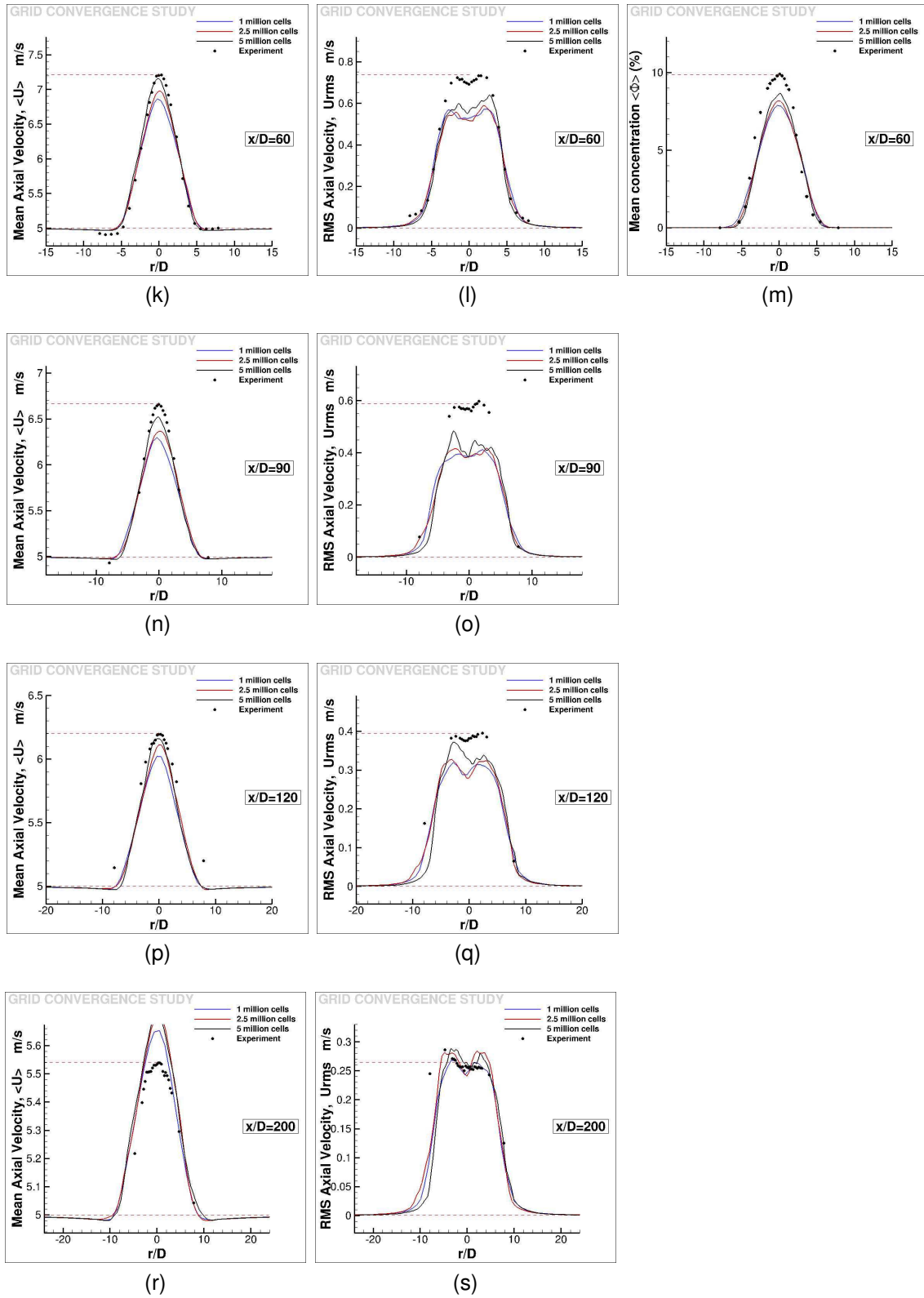
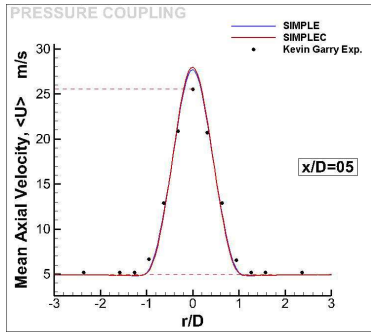
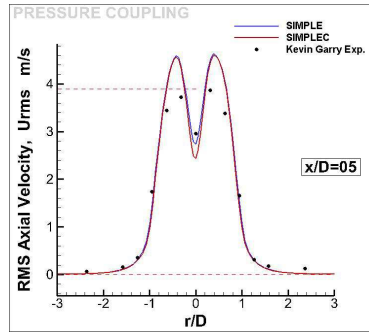


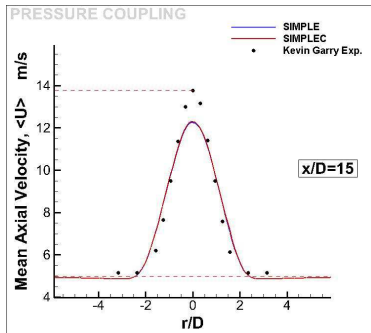
Figure A.7: Profiles of the study “Grid Convergence”.



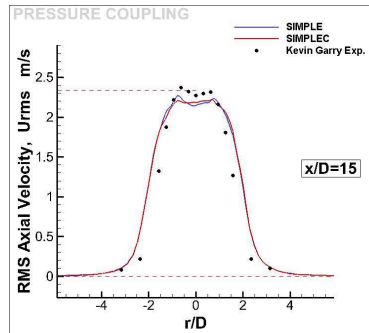
(a)



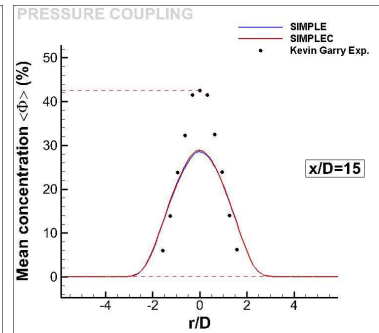
(b)



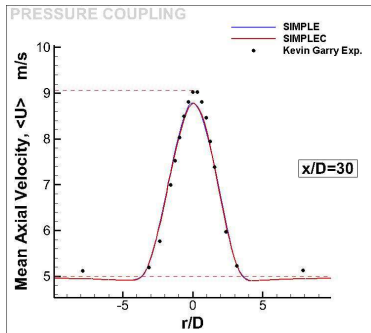
(c)



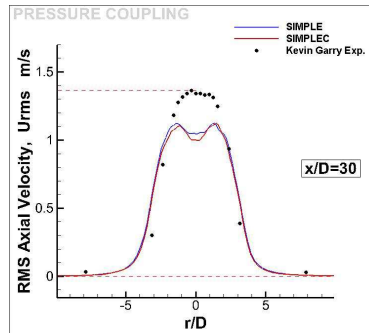
(d)



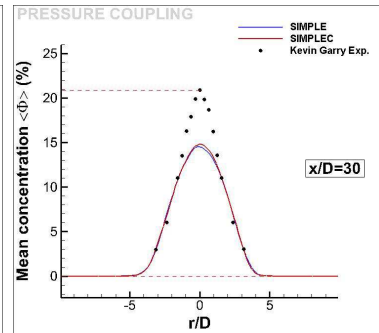
(e)



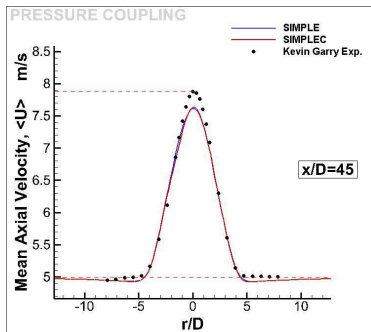
(f)



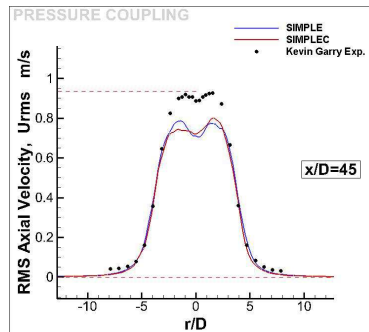
(g)



(h)



(i)



(j)

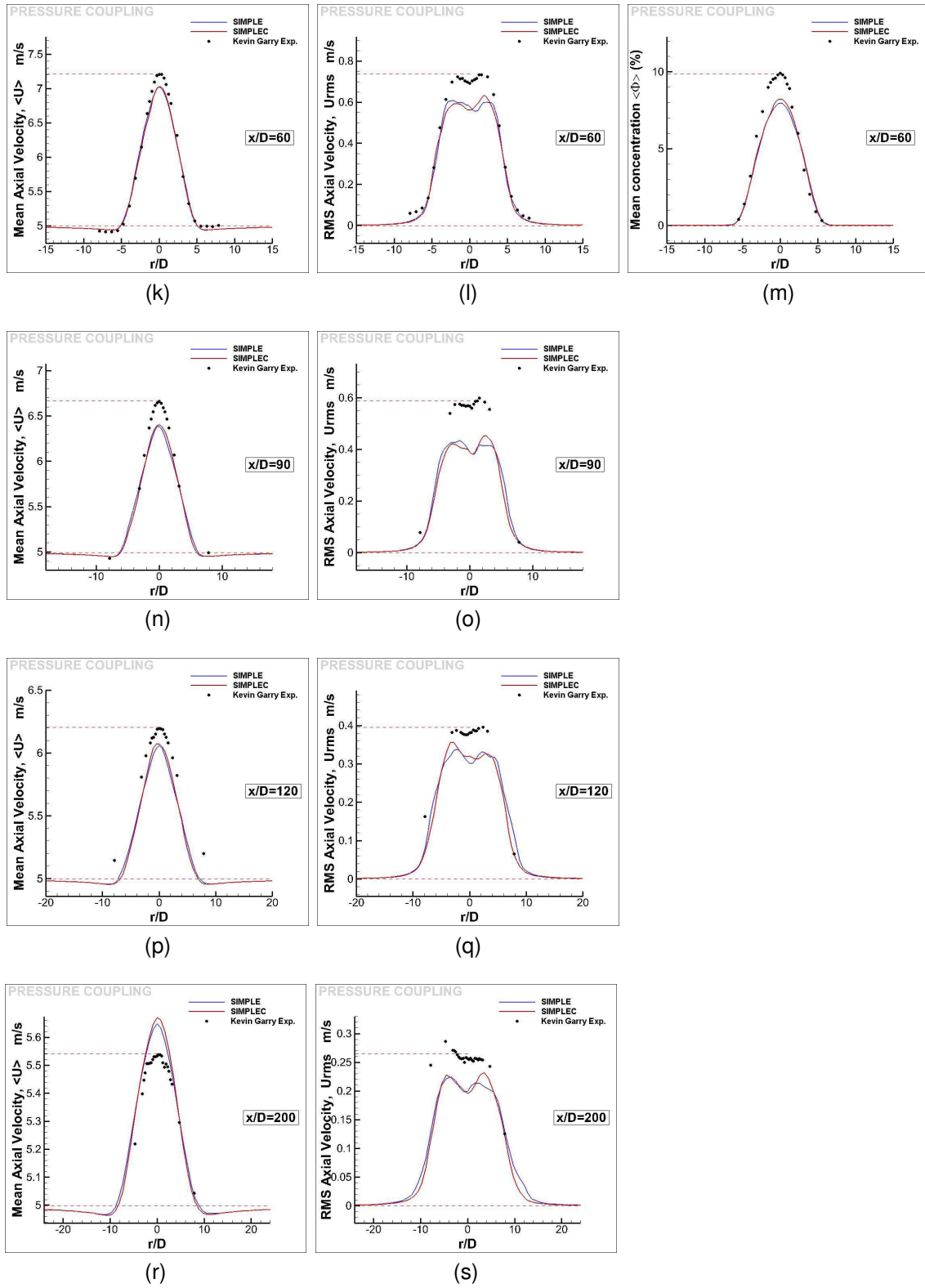
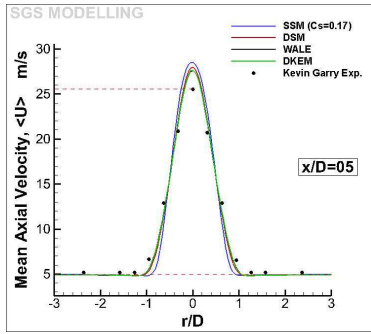
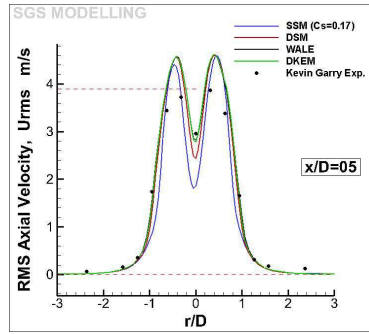


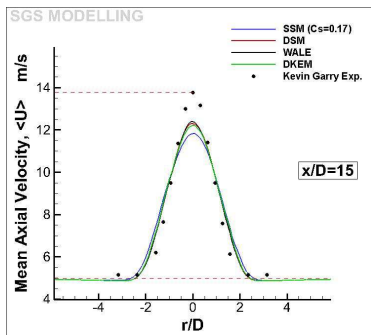
Figure A.8: Profiles of the study "Pressure Coupling Algorithm".



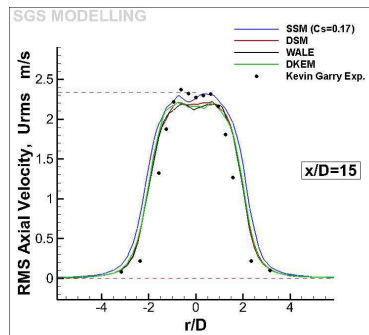
(a)



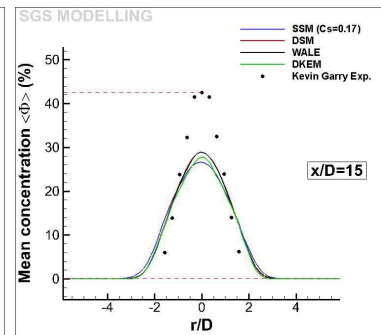
(b)



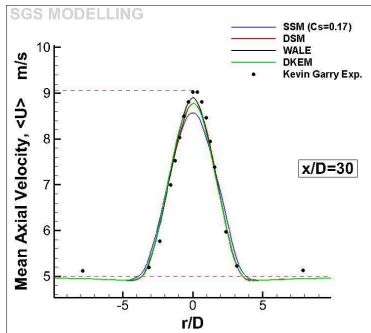
(c)



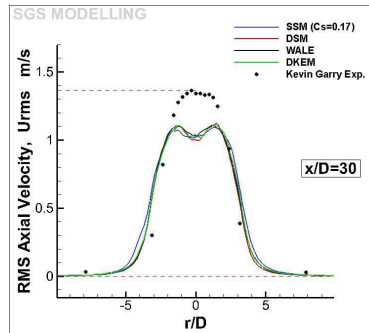
(d)



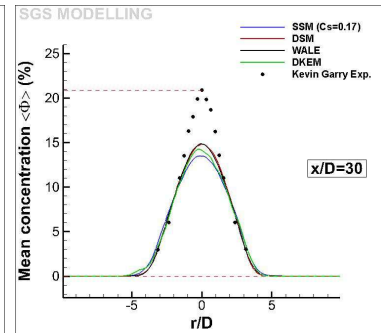
(e)



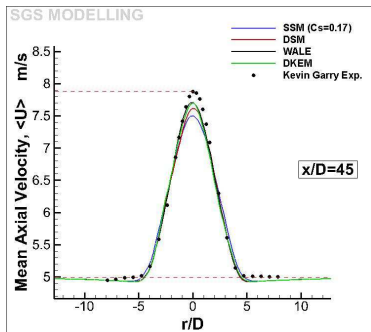
(f)



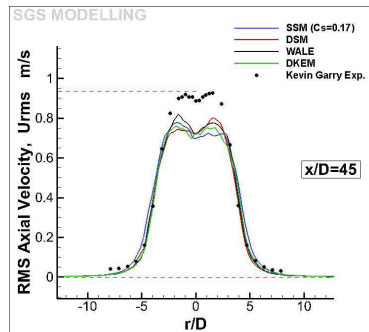
(g)



(h)



(i)



(j)

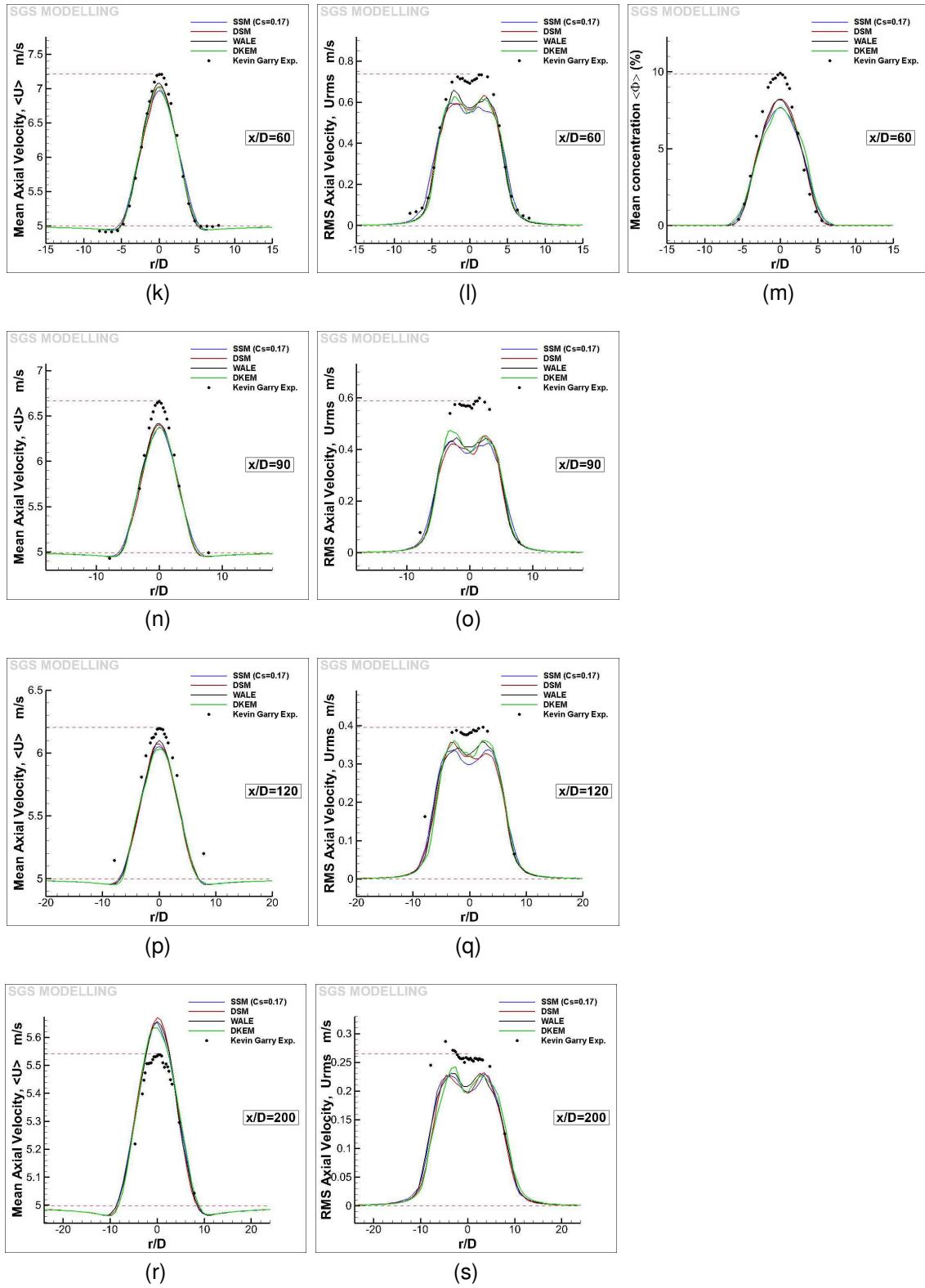
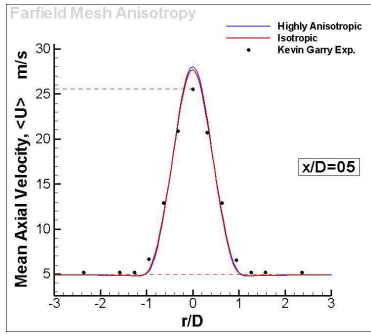
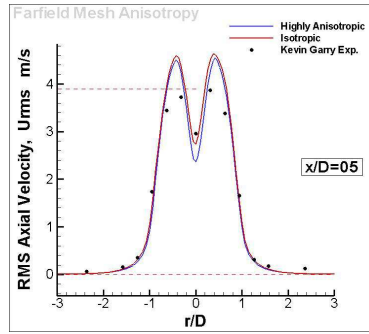


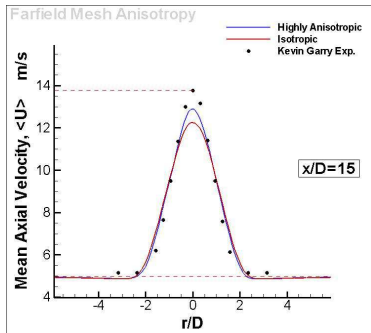
Figure A.9: Profiles of the study "SGS Modelling".



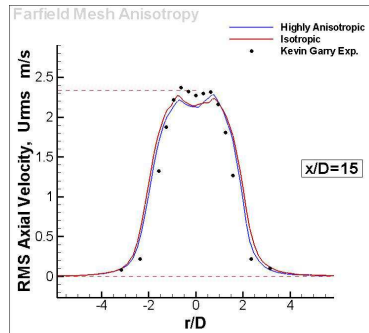
(a)



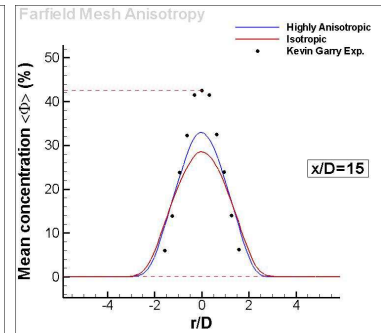
(b)



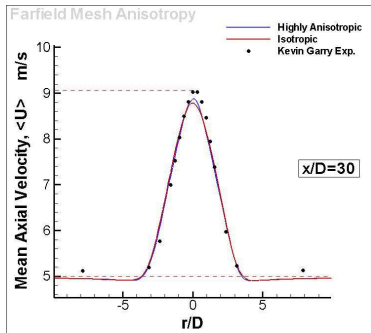
(c)



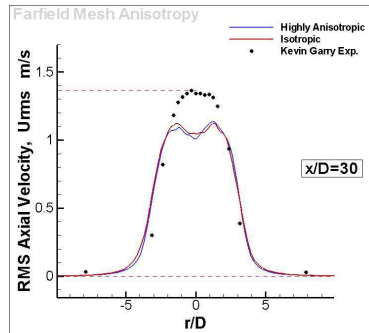
(d)



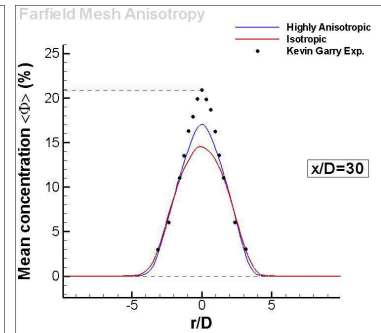
(e)



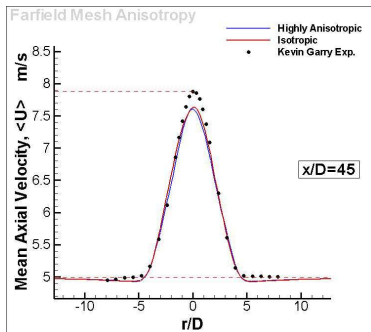
(f)



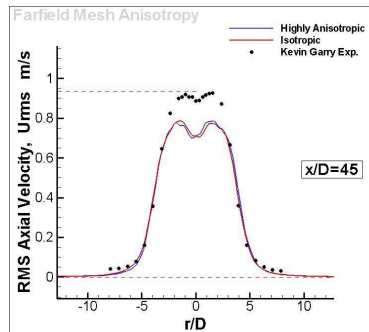
(g)



(h)



(i)



(j)

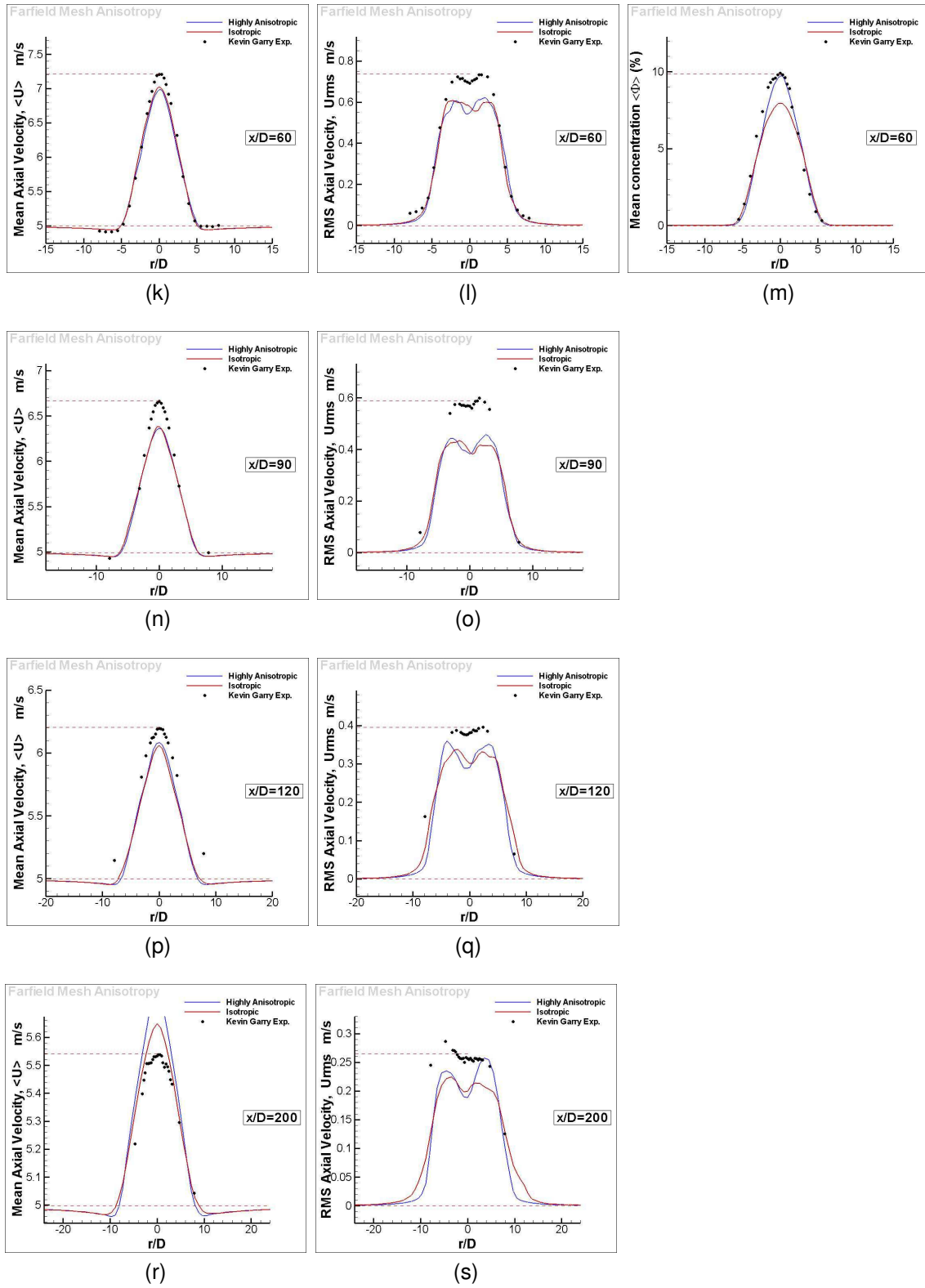
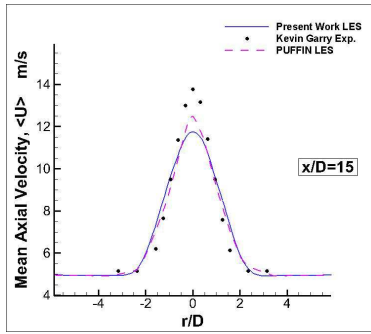
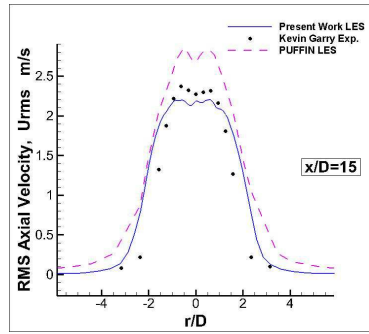


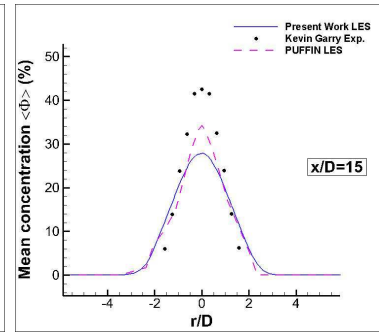
Figure A.10: Profiles of the study “Far-field Mesh-Anisotropy”.



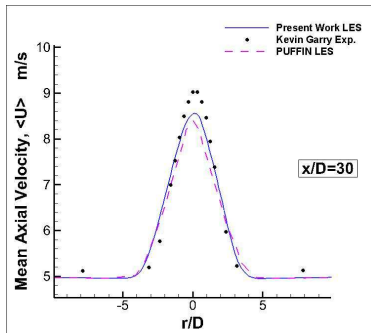
(a)



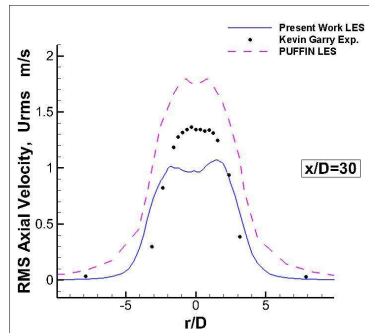
(b)



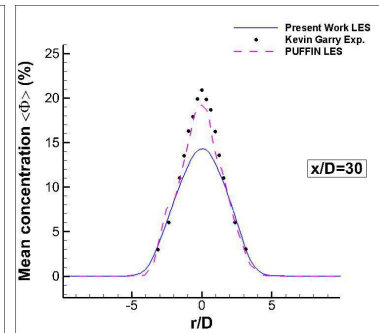
(c)



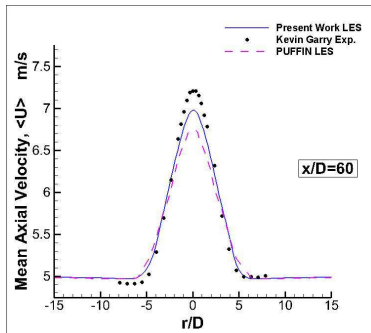
(d)



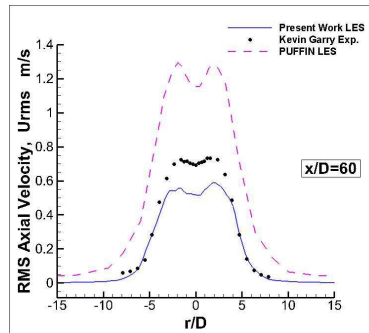
(e)



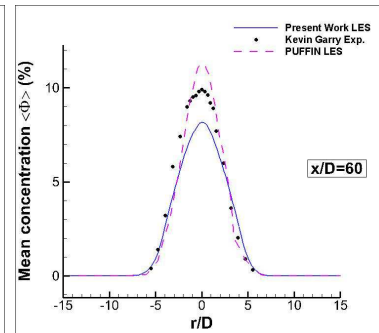
(f)



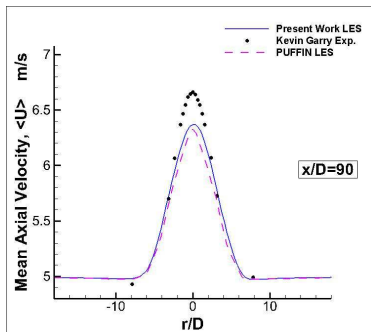
(g)



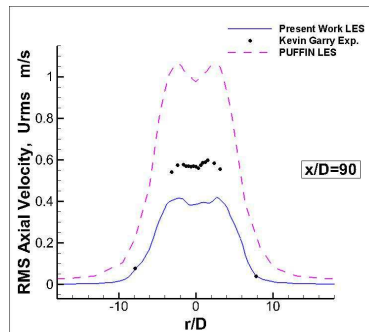
(h)



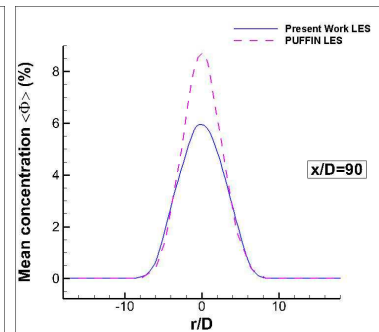
(i)



(j)



(k)



(l)

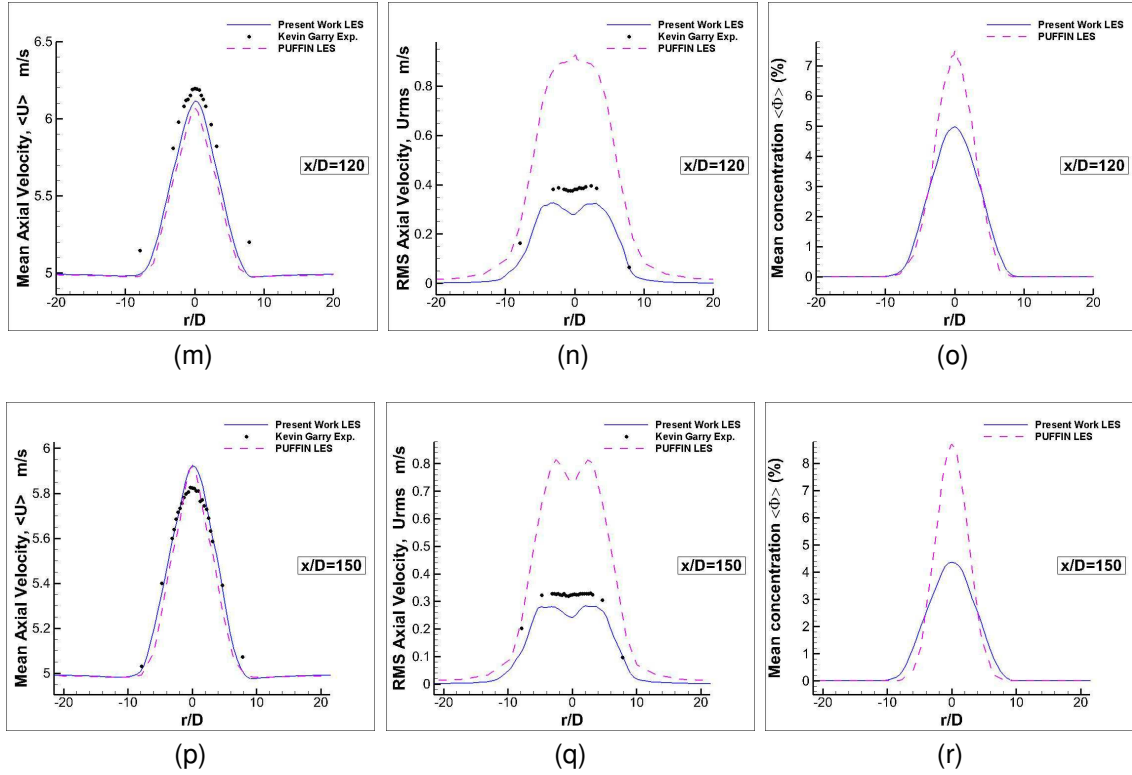
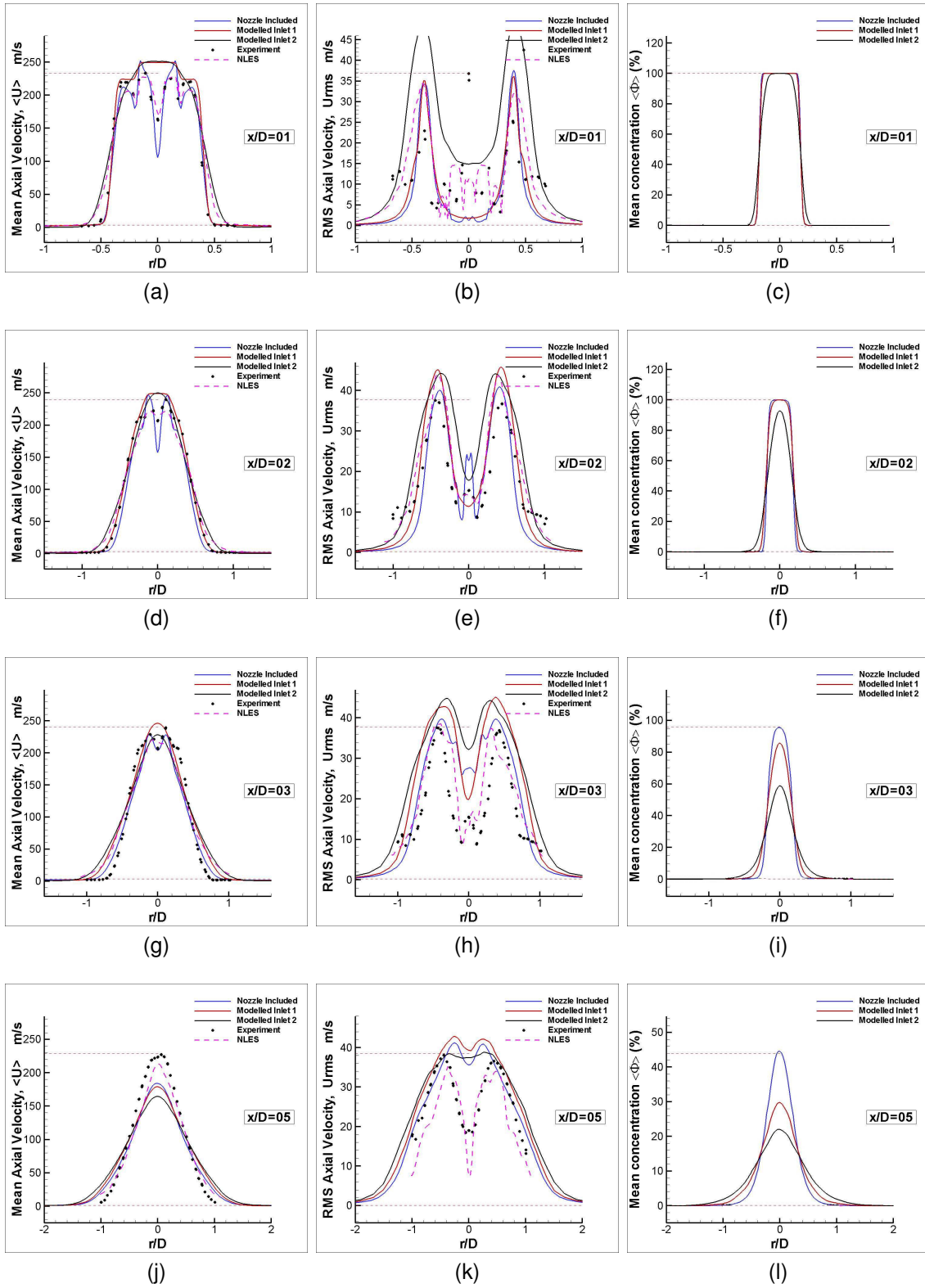


Figure A.11: Profiles of the study “Comparison with Previous LES”.



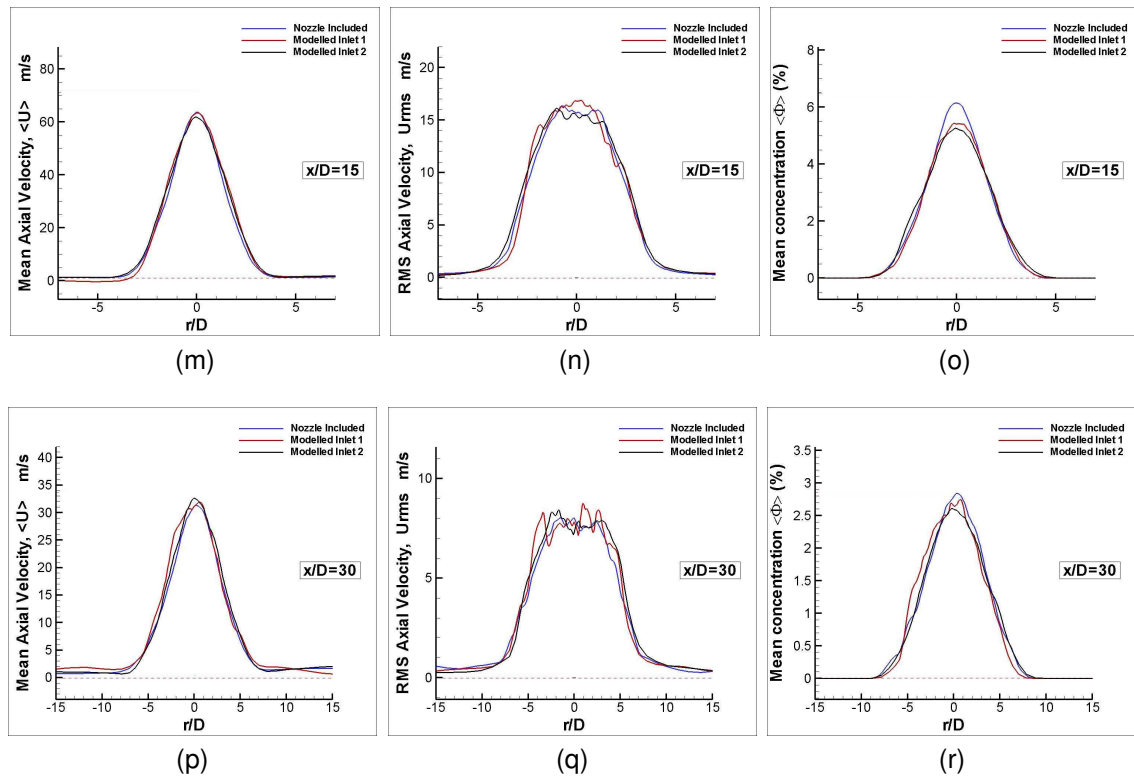


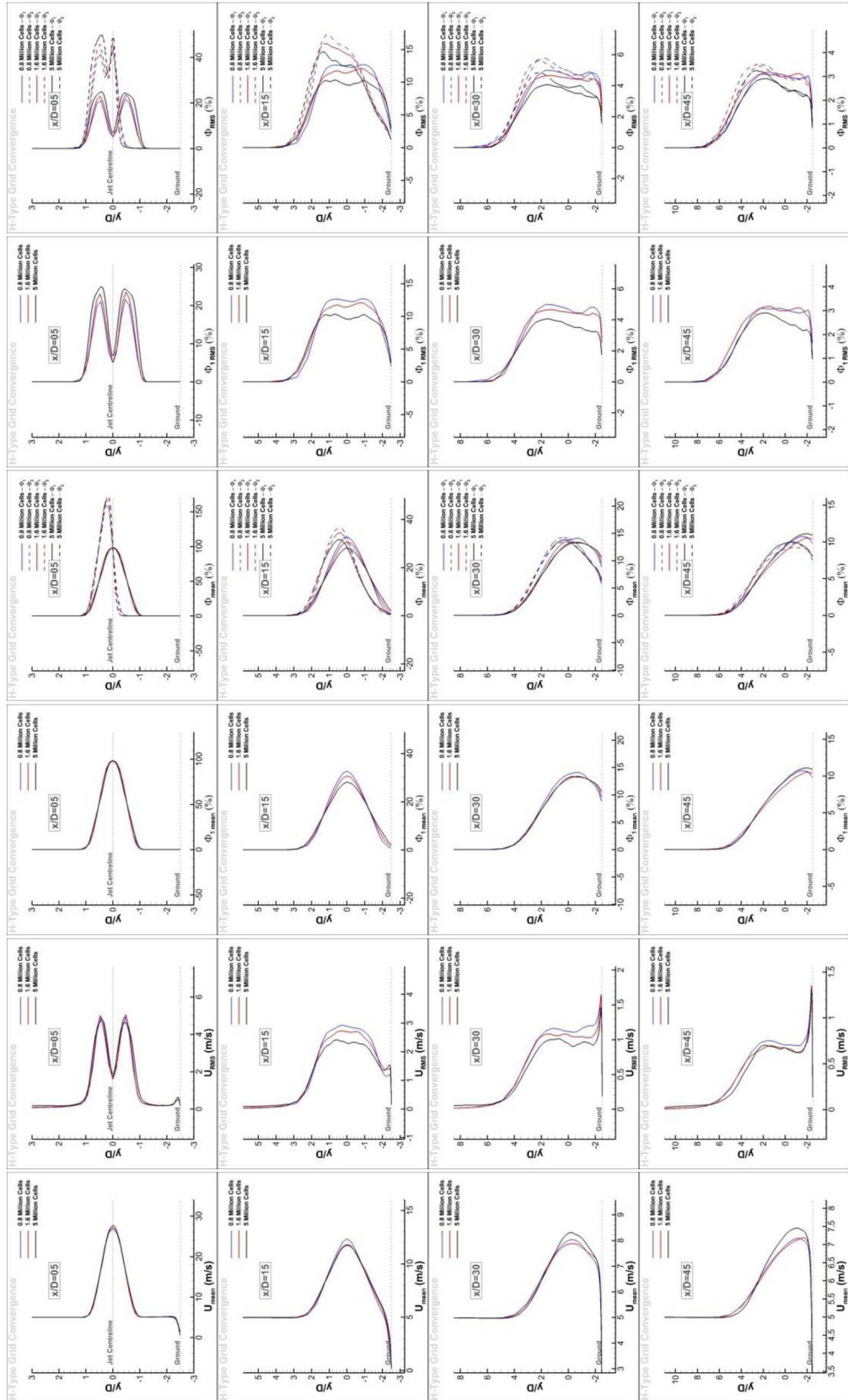
Figure A.12: Profiles of the study “Modelling the Nozzle”.

Appendix B

Wall Bounded Validation Tests: Example of the Conventional Post-Processing

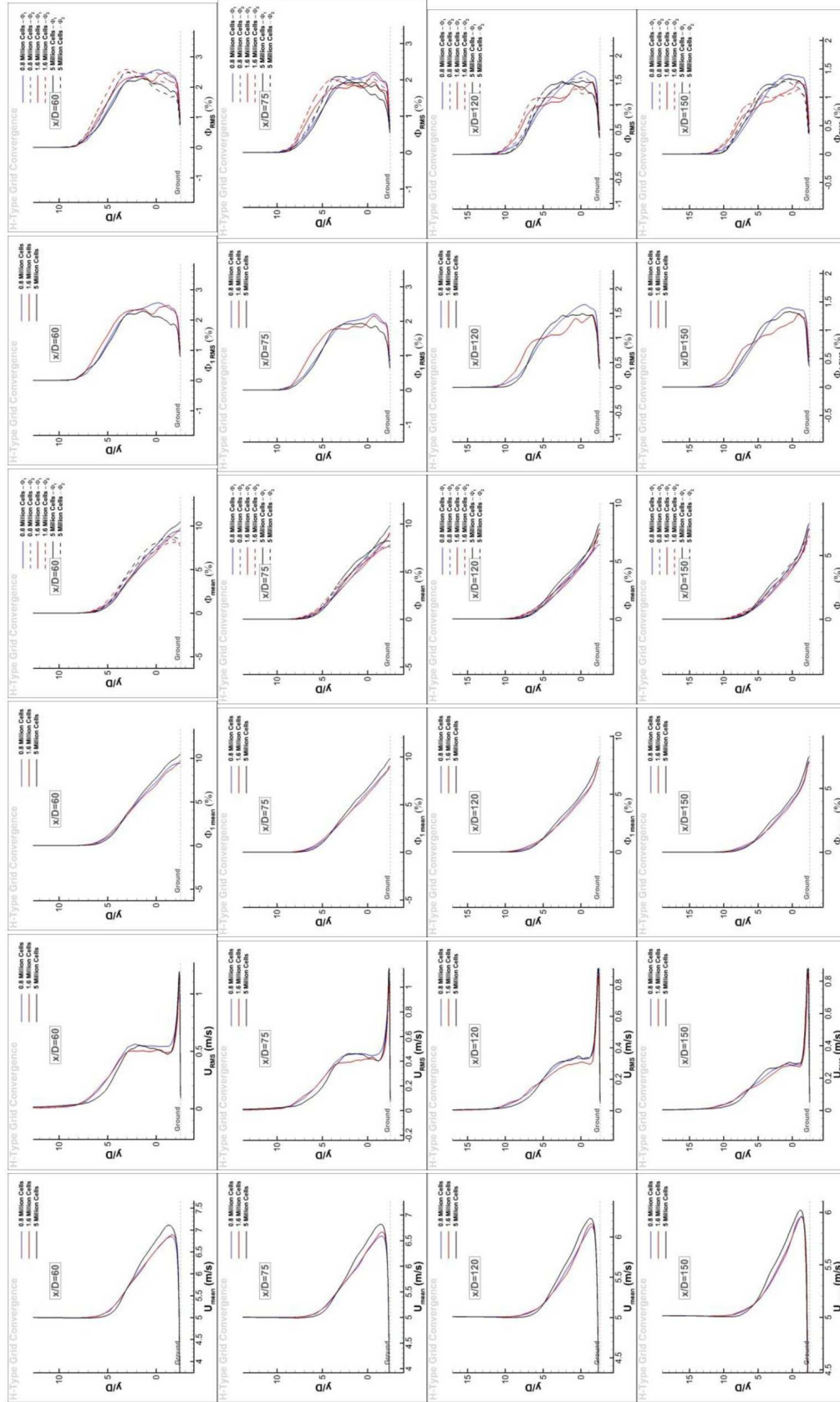
The present Appendix includes a sample of the conventional post processing method. Refer to section 8.1.1 for description of this approach. Description of the investigated setup corresponding to these results is available in section 6.1.3.

VERTICAL PROFILES



(a)

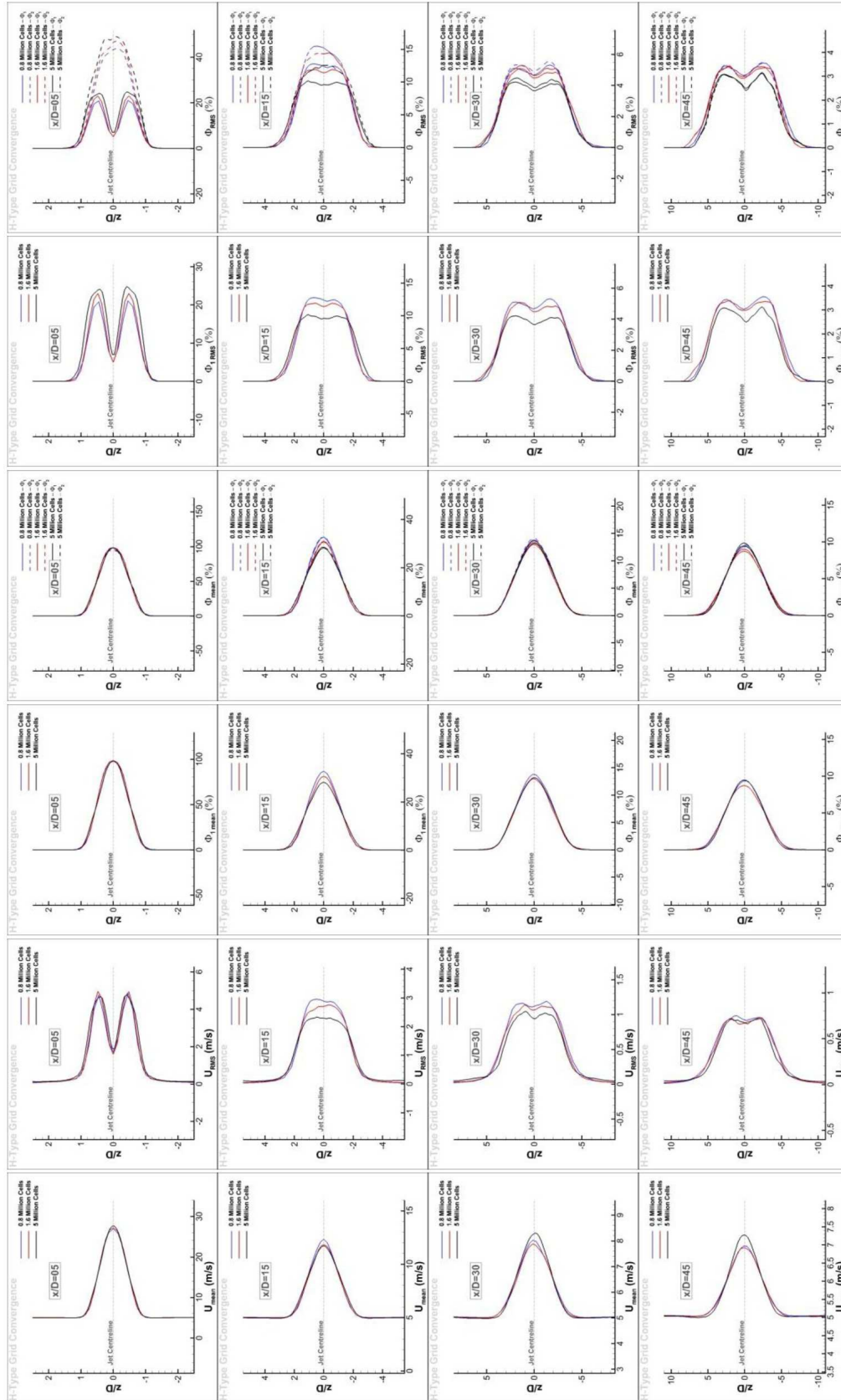
VERTICAL PROFILES



(b)

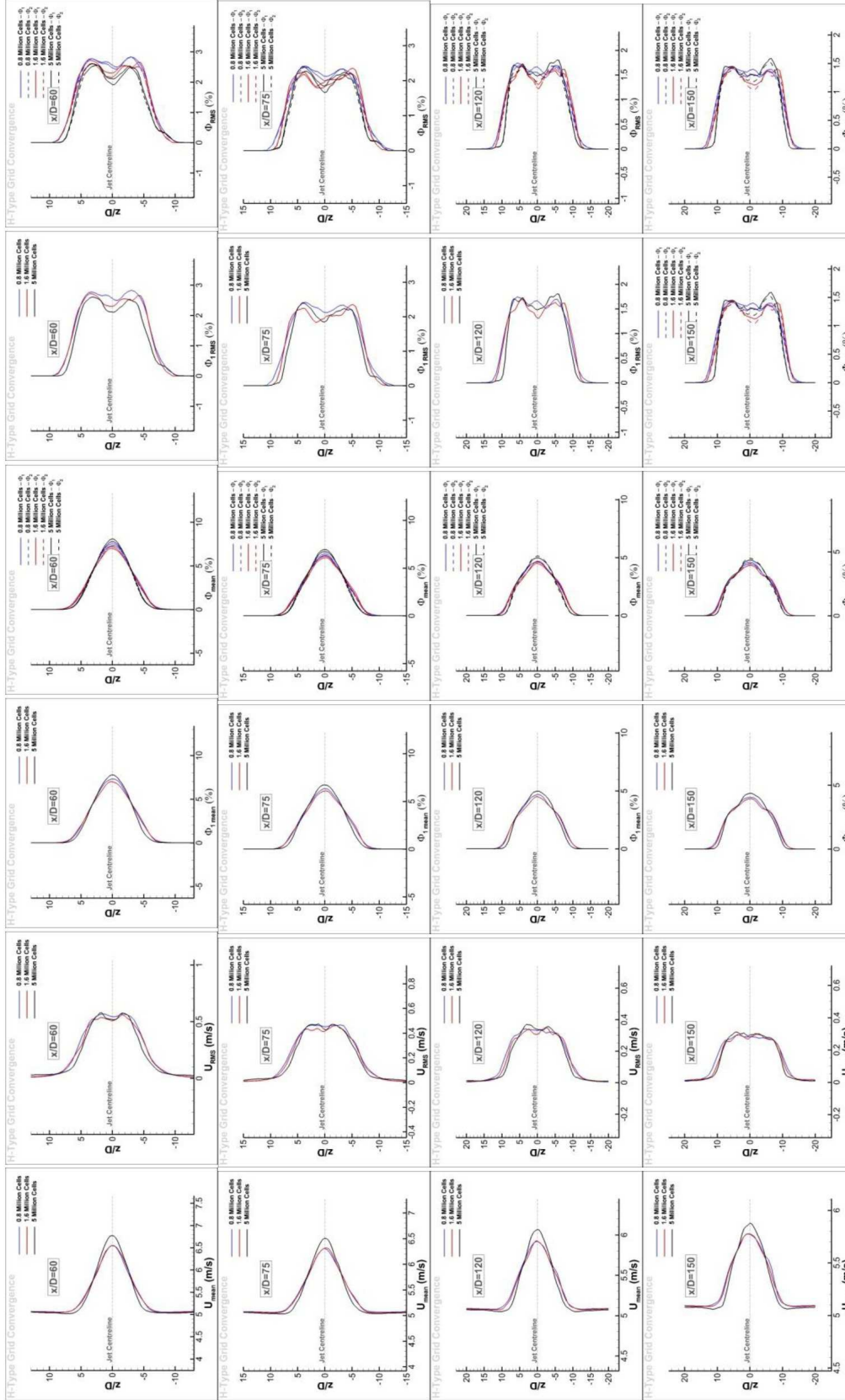
Figure B.1: Conventional Post-Processing of Study Nr.6: H-Type Mesh Density. Vertical Profiles of $\bar{U}$, U_{rms} , $\bar{\Phi}$, and Φ_{rms} are Plotted at the Selected Stations $5 \leq x/D \leq 150$.

HORIZONTAL PROFILES



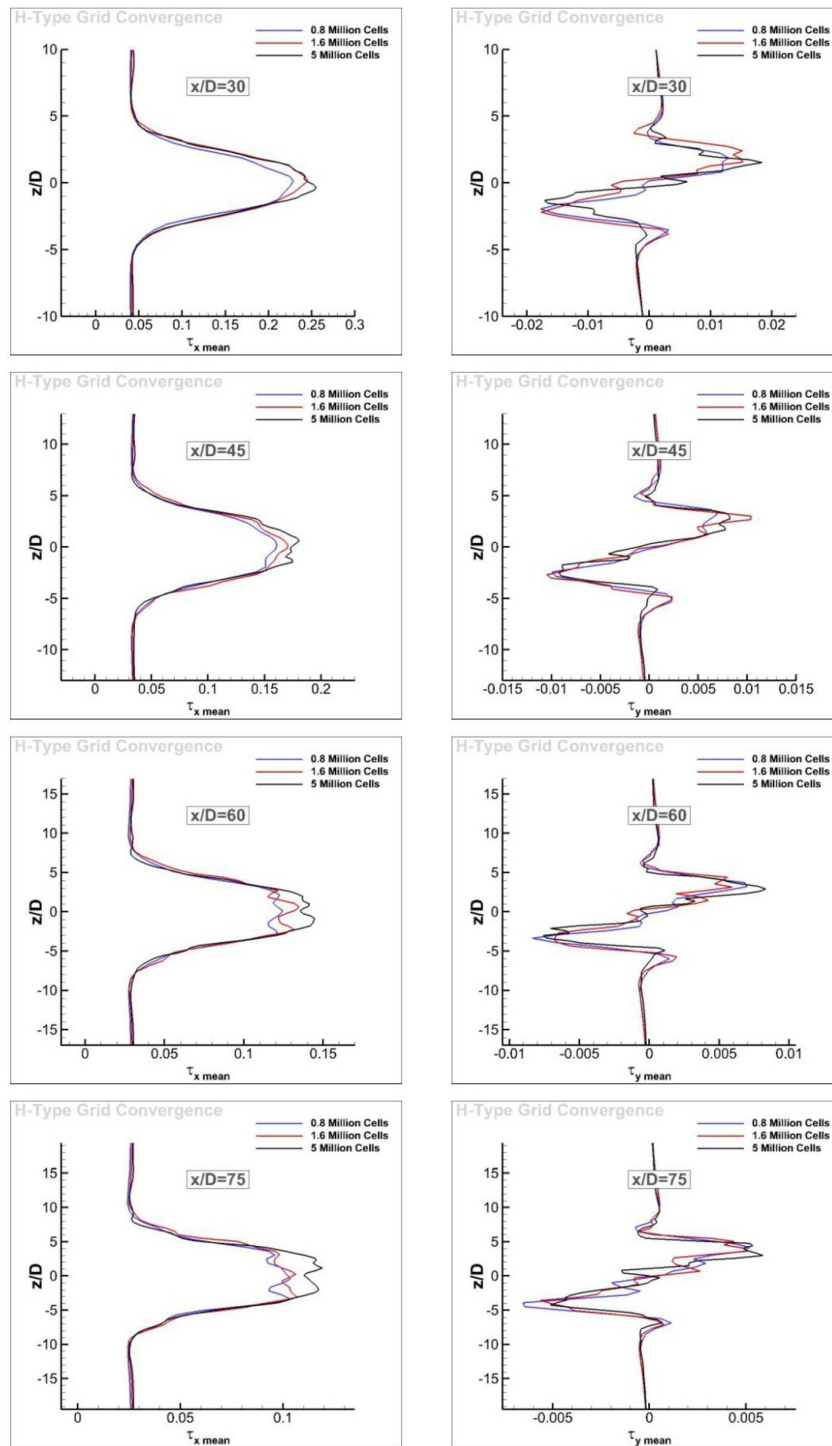
(a)

HORIZONTAL PROFILES



(b)

Figure B.2: Conventional Post-Processing of Study Nr.6: H-Type Mesh Density. Horizontal Profiles of $\bar{U}$, U_{rms} , $\bar{\Phi}$, and Φ_{rms} are Plotted at the Selected Stations $5 \leq X/D \leq 150$.



WALL PROFILES

(a)

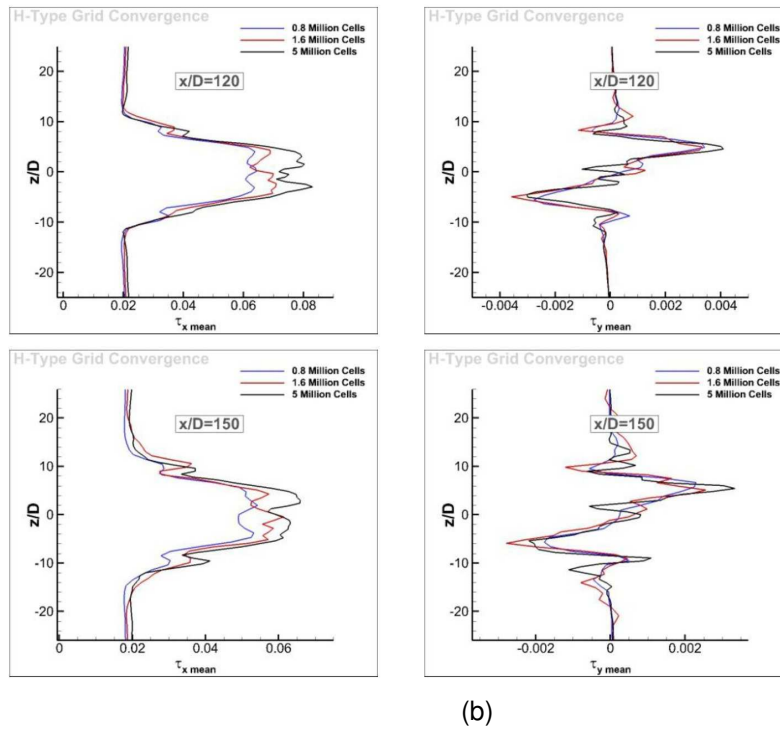
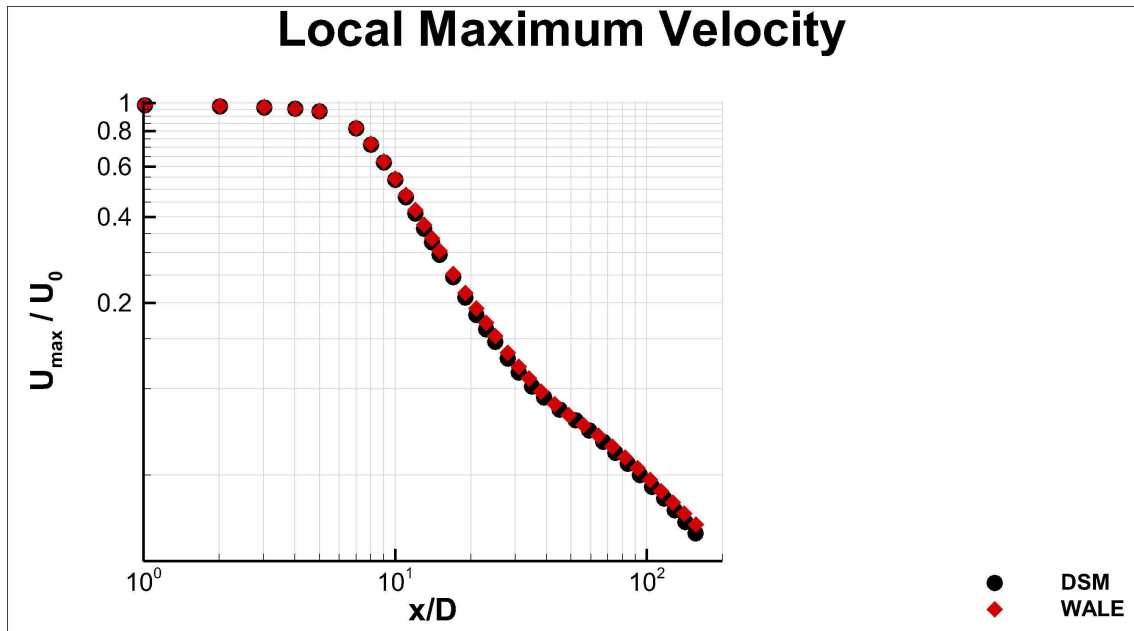

WALL PROFILES

Figure B.3: Conventional Post-Processing of Study Nr.6: H-Type Mesh Density. Wall Profiles of mean X- and Y-Wall-Shear are Plotted at the Wall-Jet Region $30 \leq X/D \leq 150$.

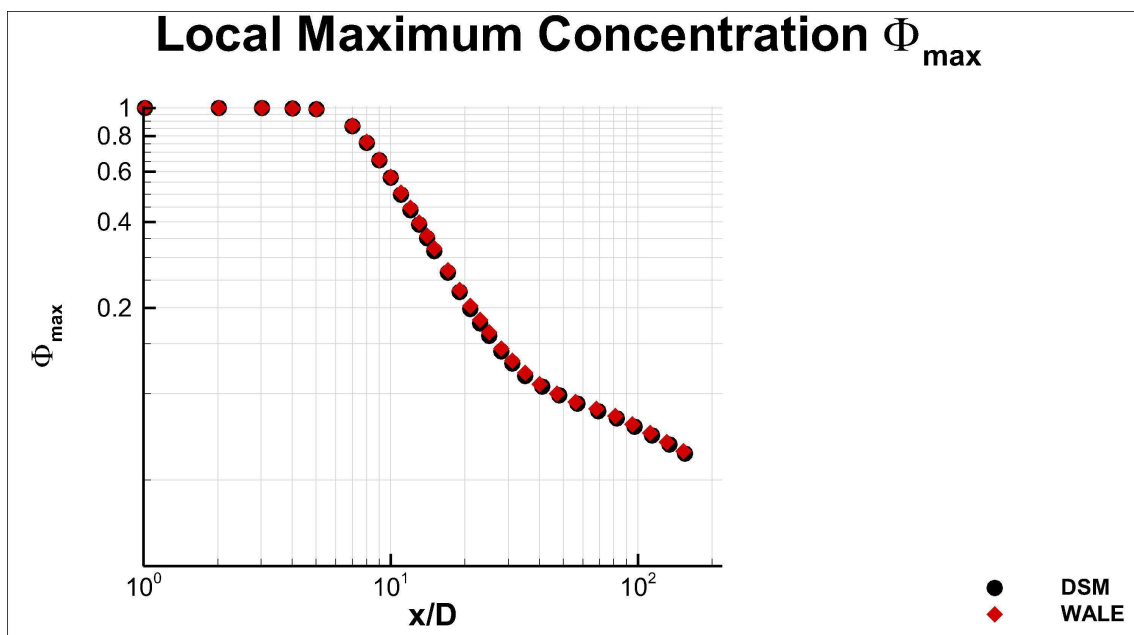
Appendix C

Wall Bounded Validation Tests: Advanced Post Processing

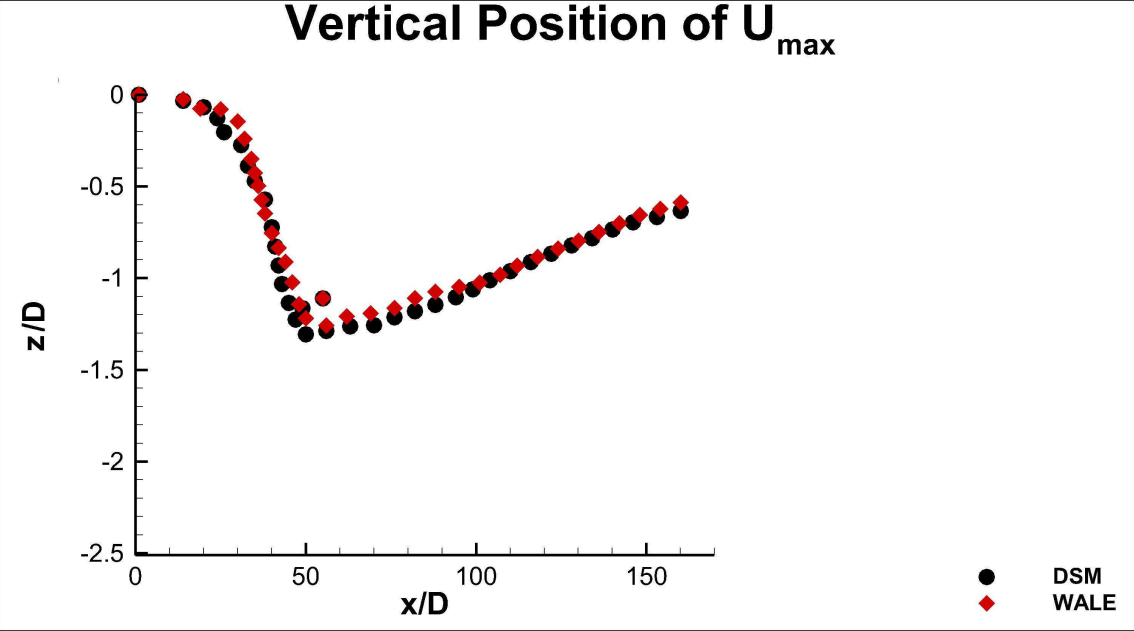
The present Appendix includes the streamwise evolution mean jet characteristics plots of all 13 studies of the wall-bounded validation tests discussed in chapter 8. This post processing approach is called advanced post-processing method; refer to section 8.1.2 for description of this approach. Description of the investigated setup corresponding to these results is available in section 6.1.3.

Study Nr.1: SGS Modelling (H-Type)

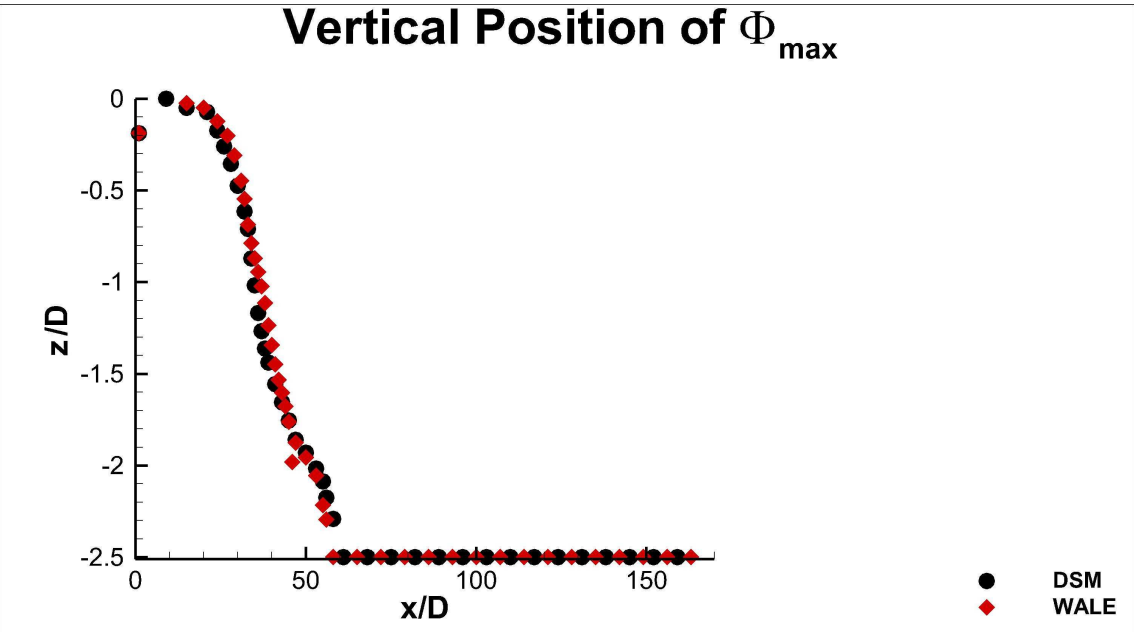
(a)



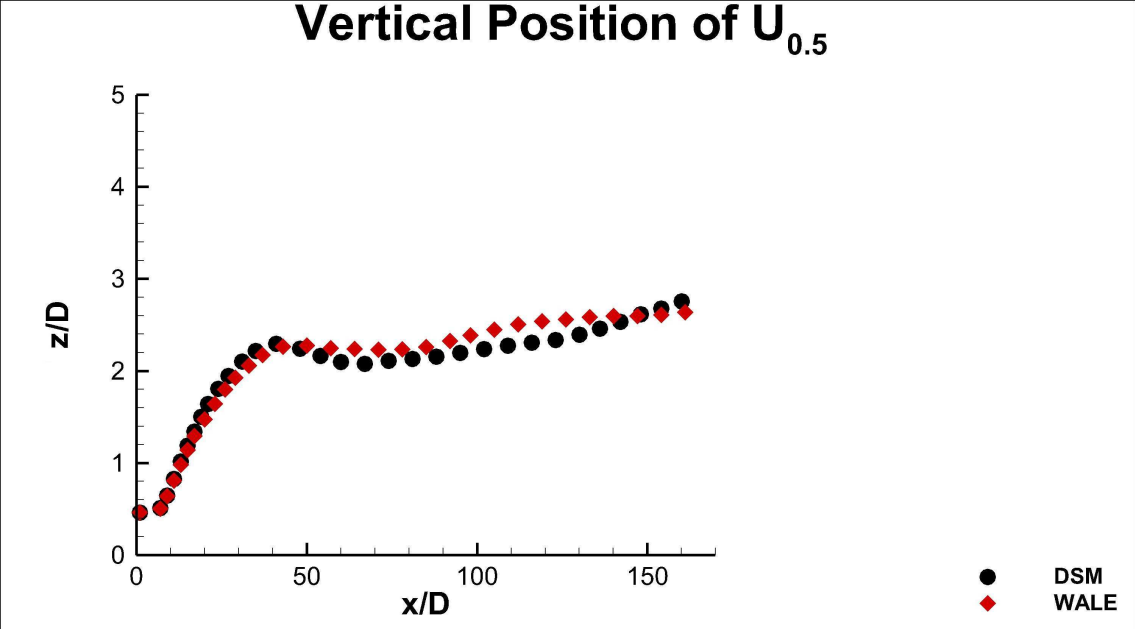
(b)



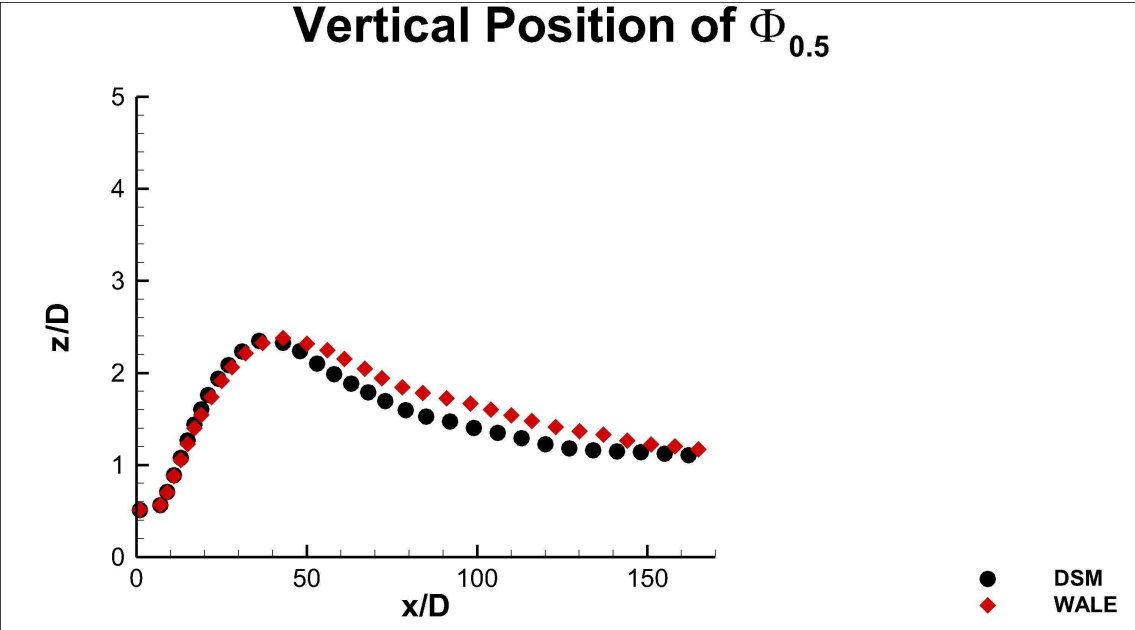
(c)



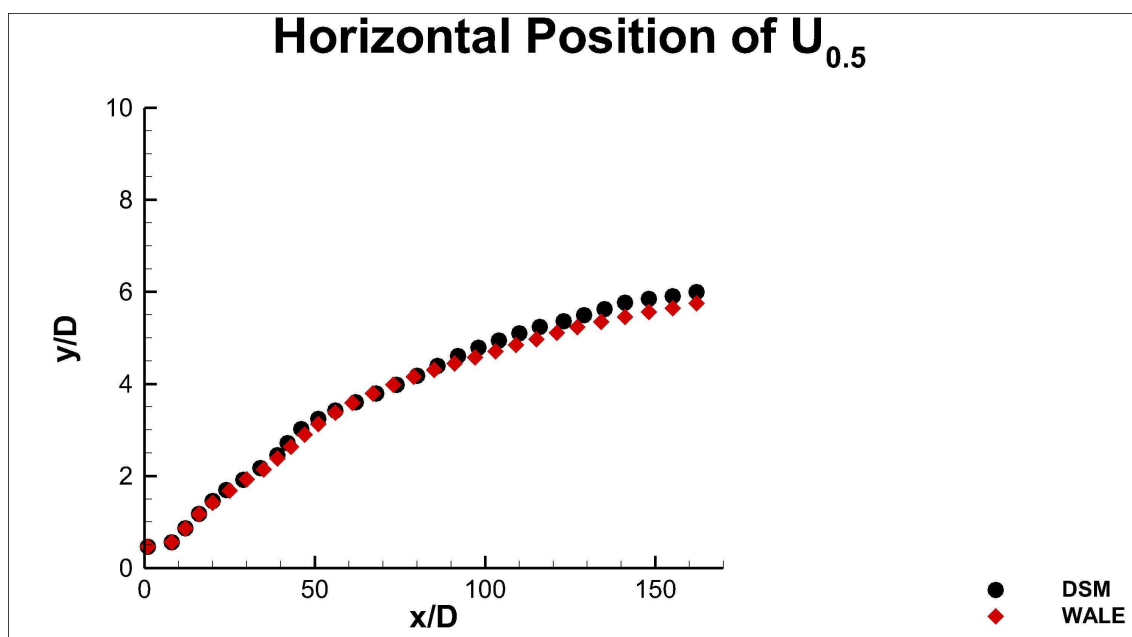
(d)



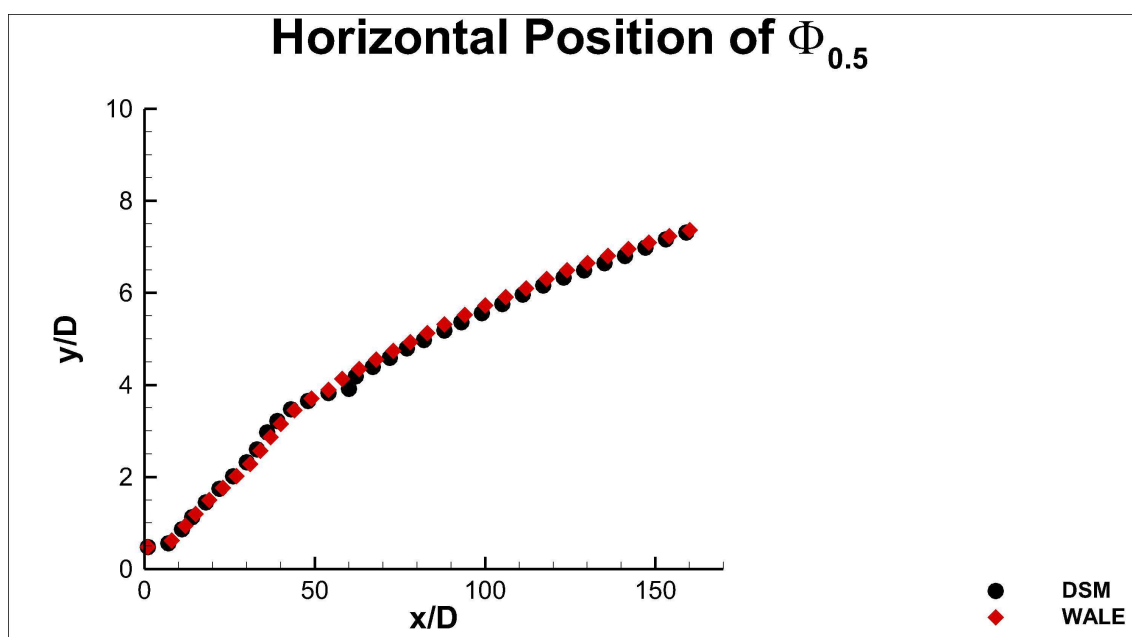
(e)



(f)



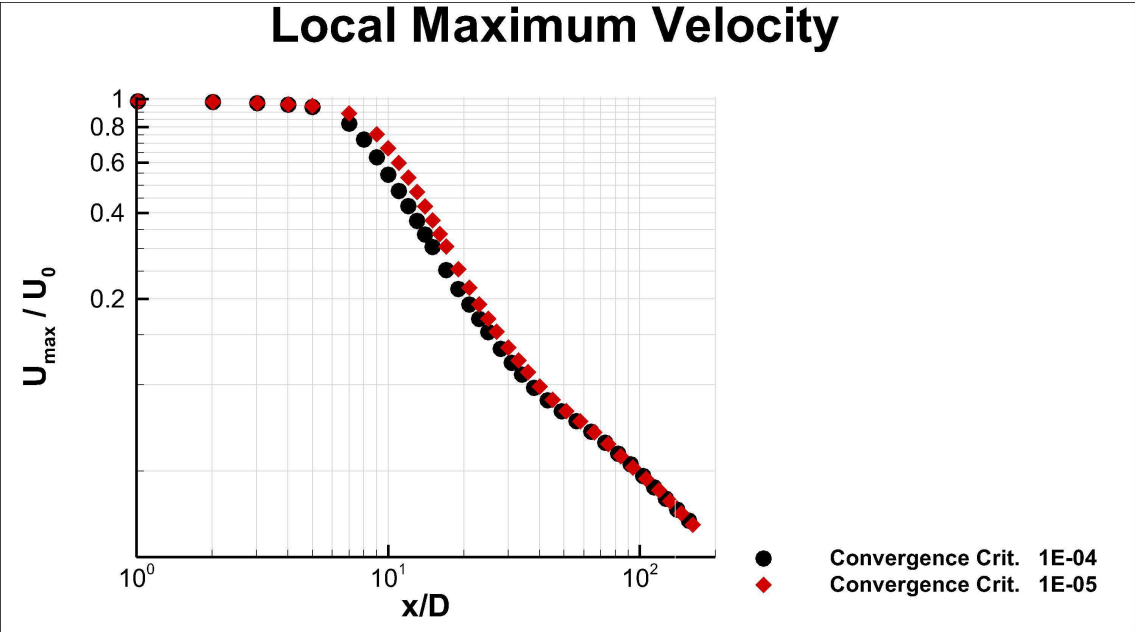
(g)



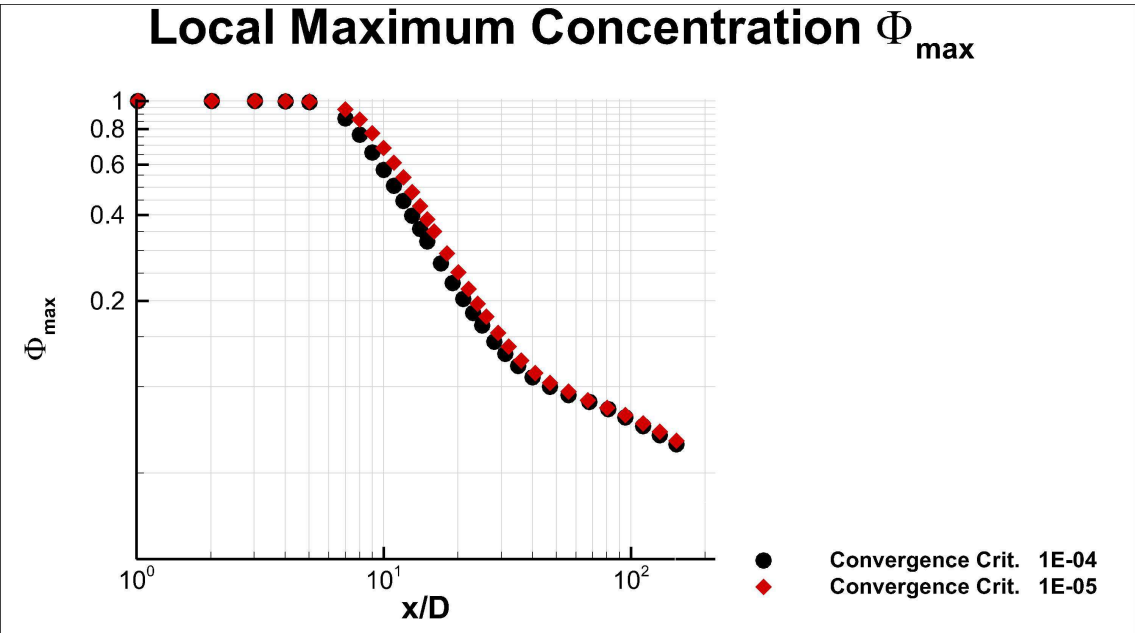
(h)

Figure C.1: Streamwise plots of study “SGS Modelling (H-Type)”.

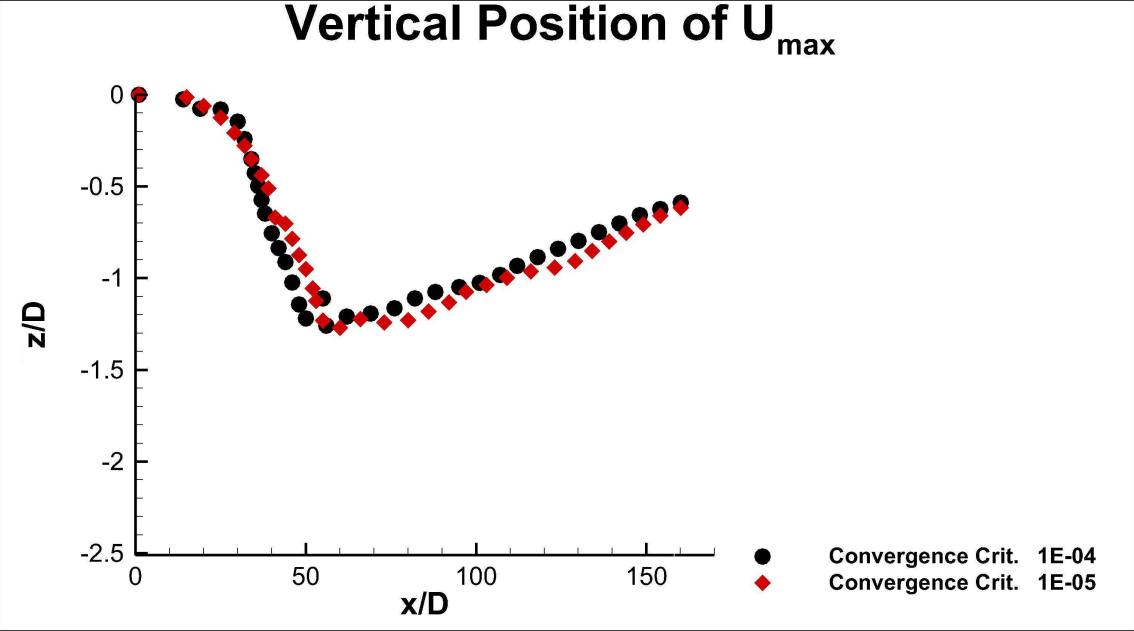
Study Nr.2: Convergence Criteria



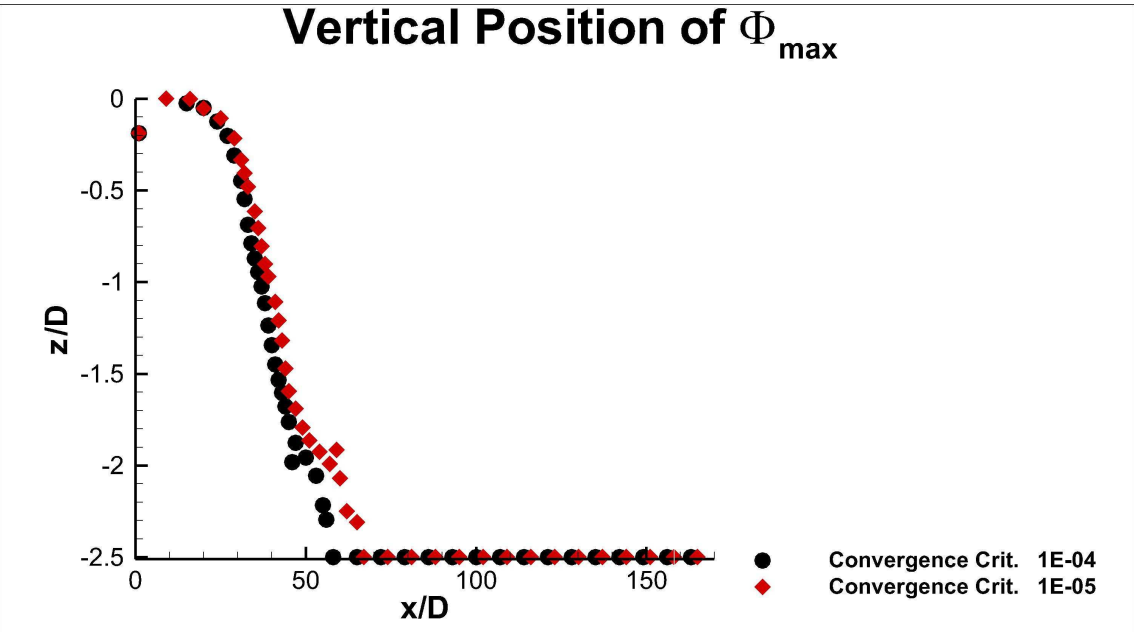
(a)



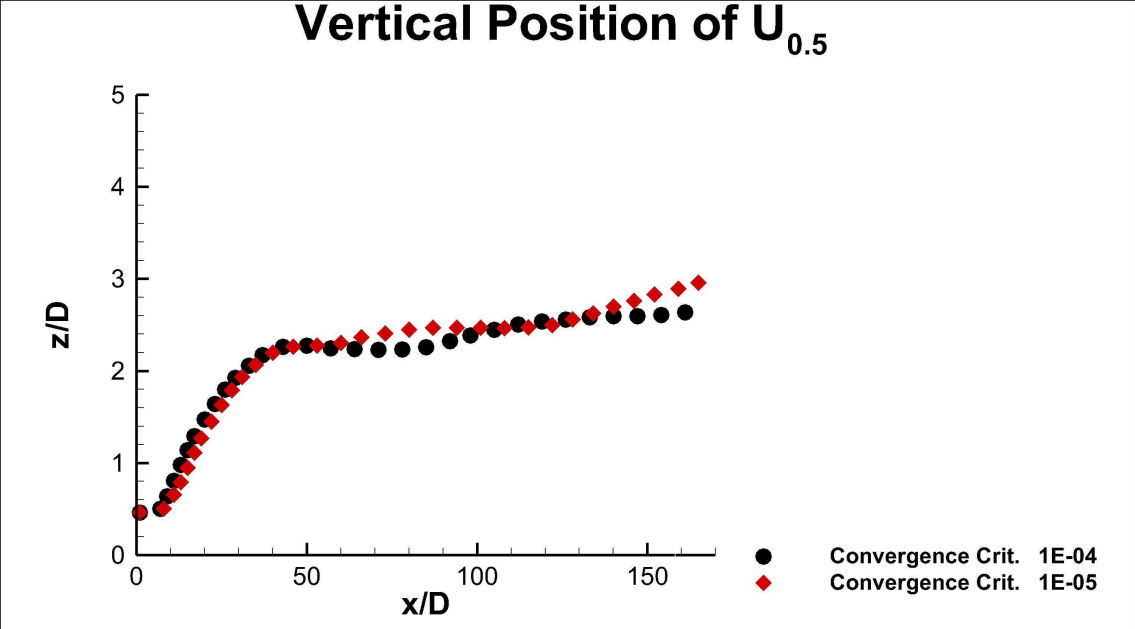
(b)



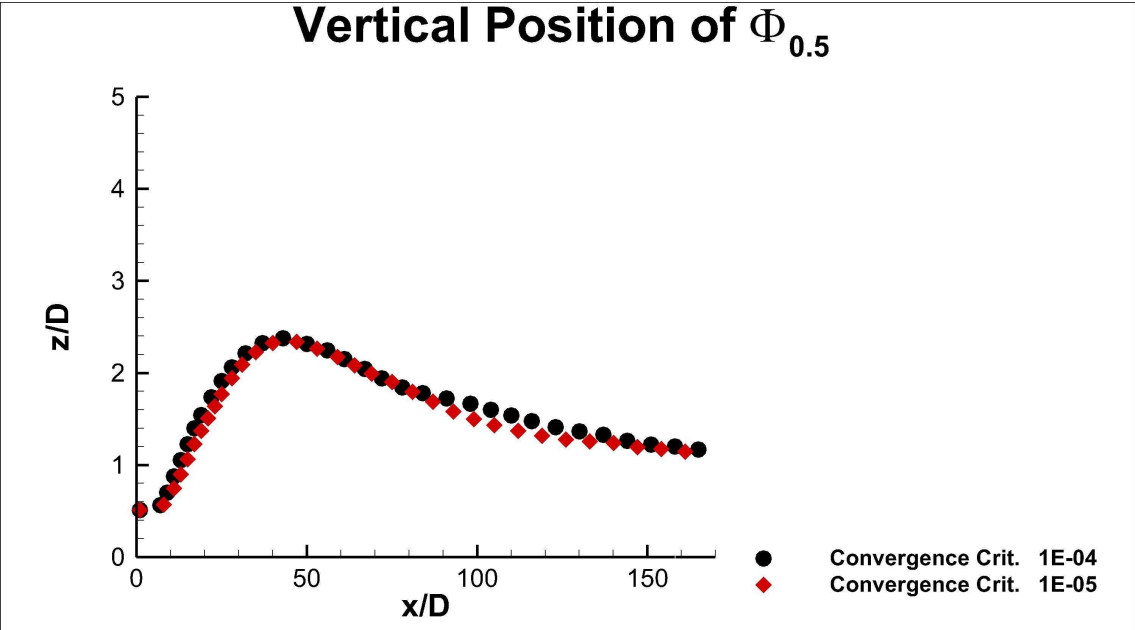
(c)



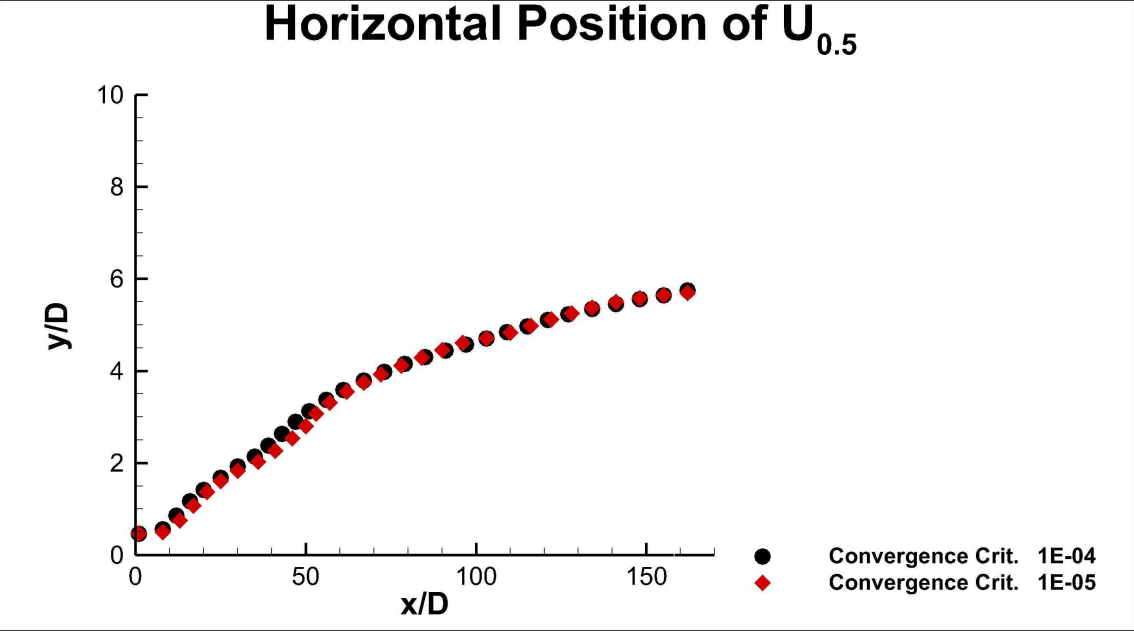
(d)



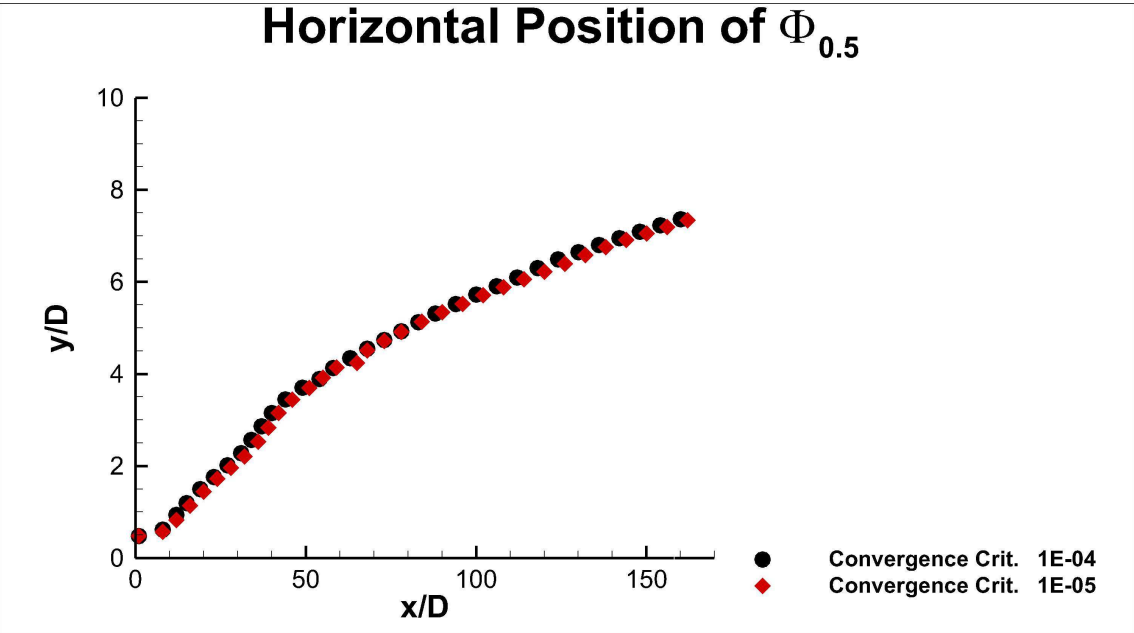
(e)



(f)



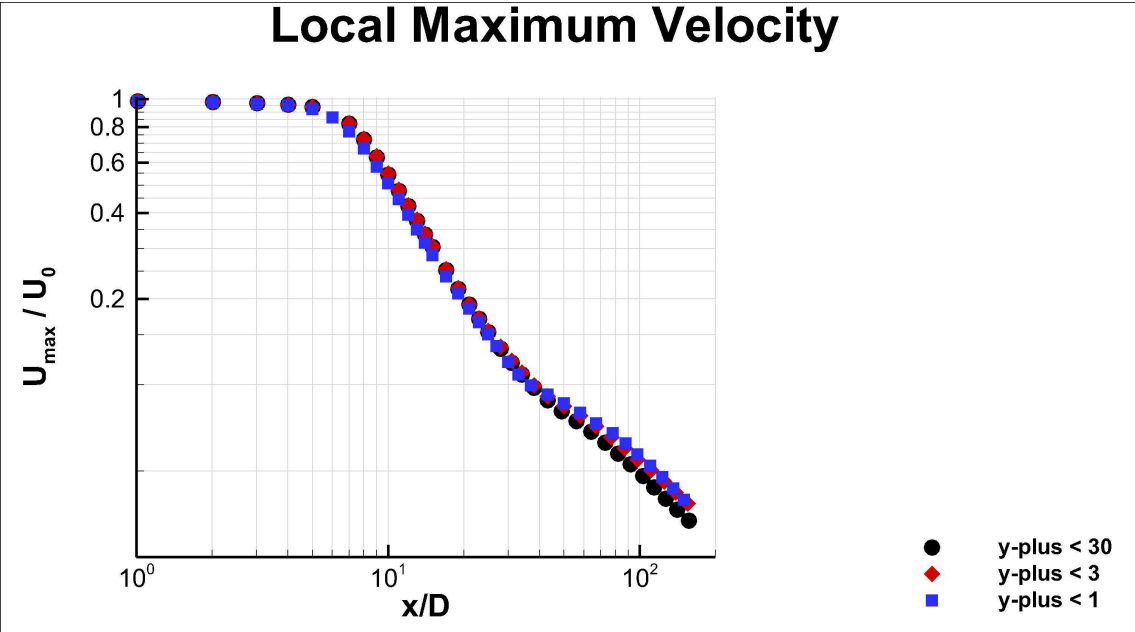
(g)



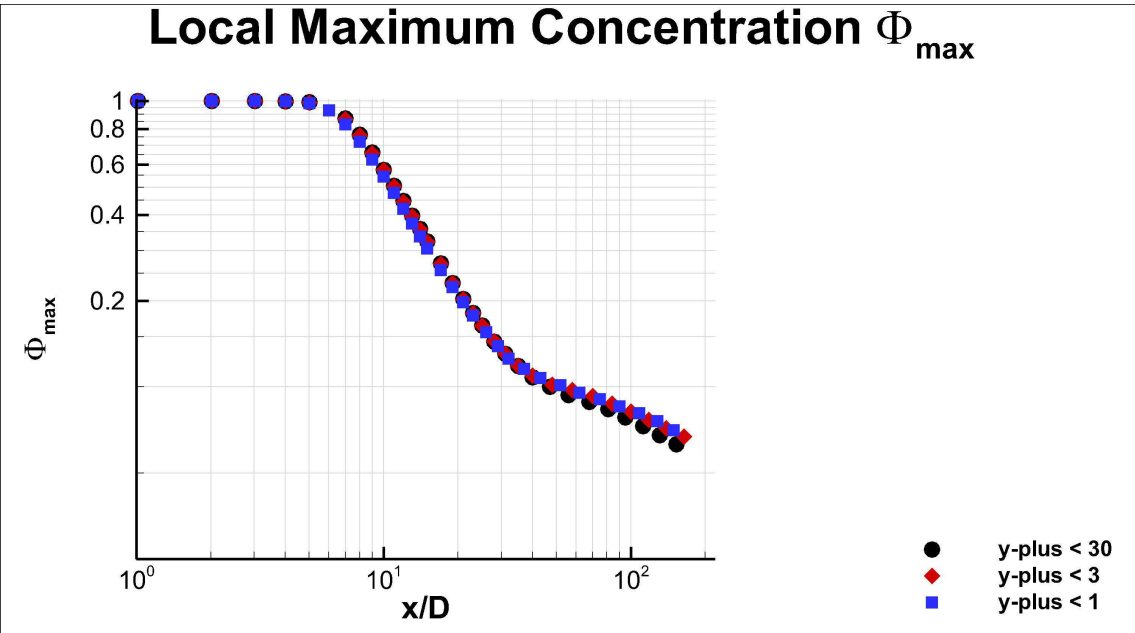
(h)

Figure C.2: Streamwise plots of study “*Convergence Criteria*”.

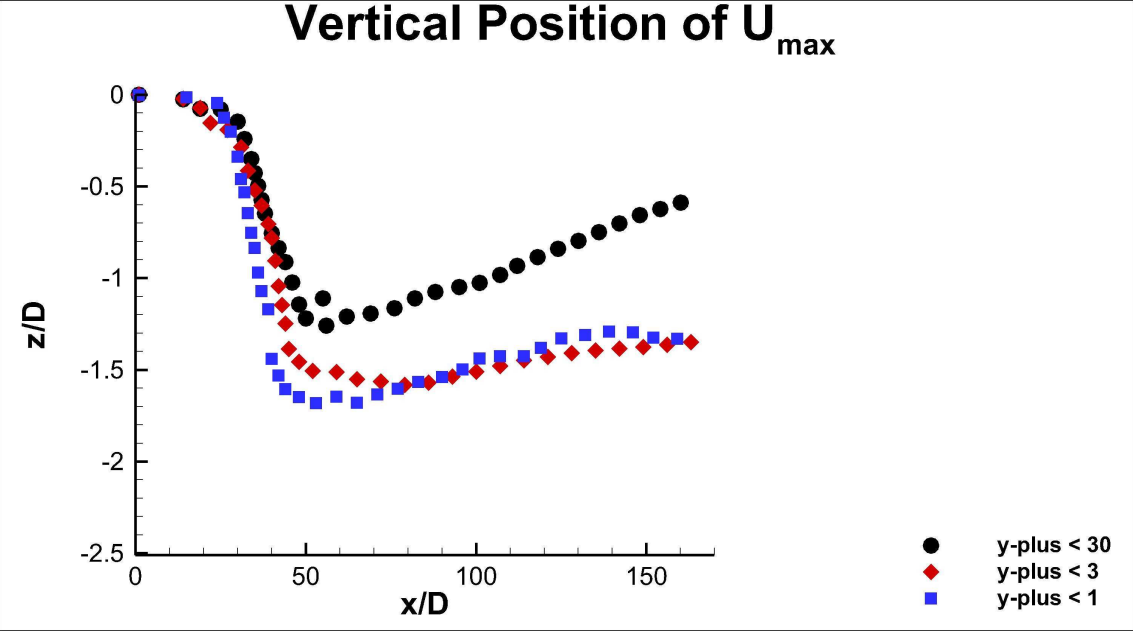
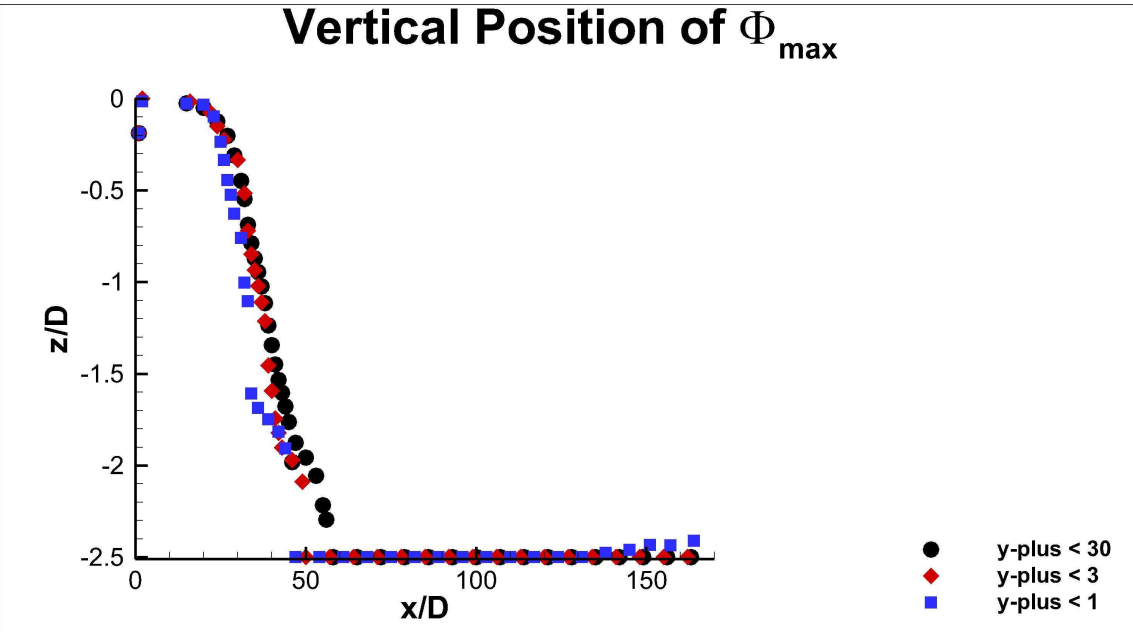
Study Nr.3: Non-Dimensional Wall Distance (y^+)

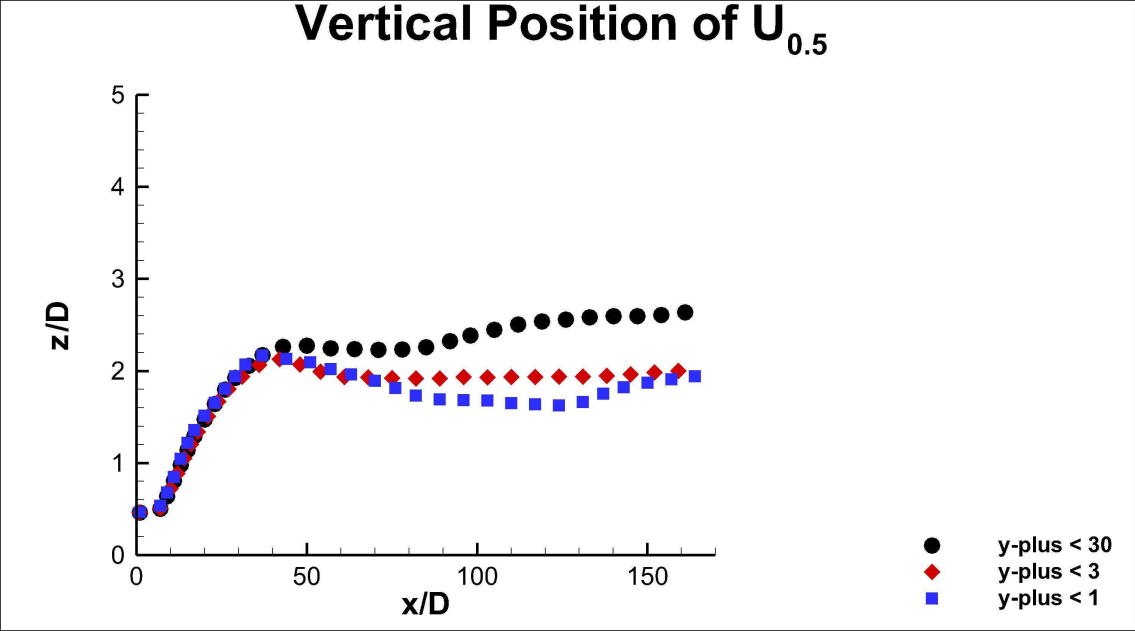


(a)

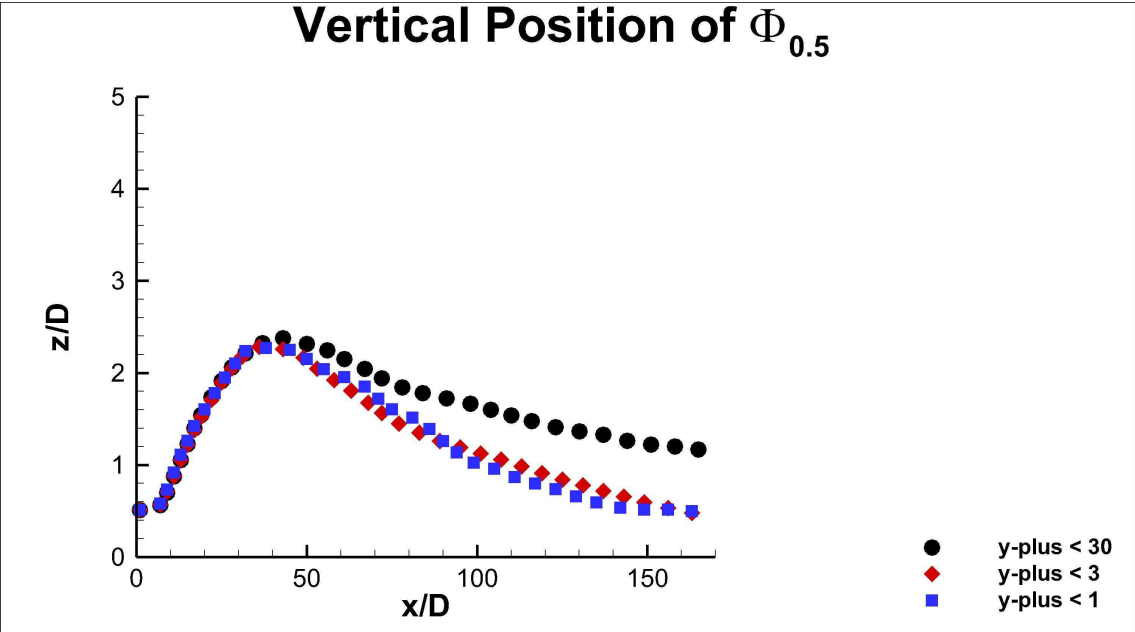


(b)





(e)



(f)

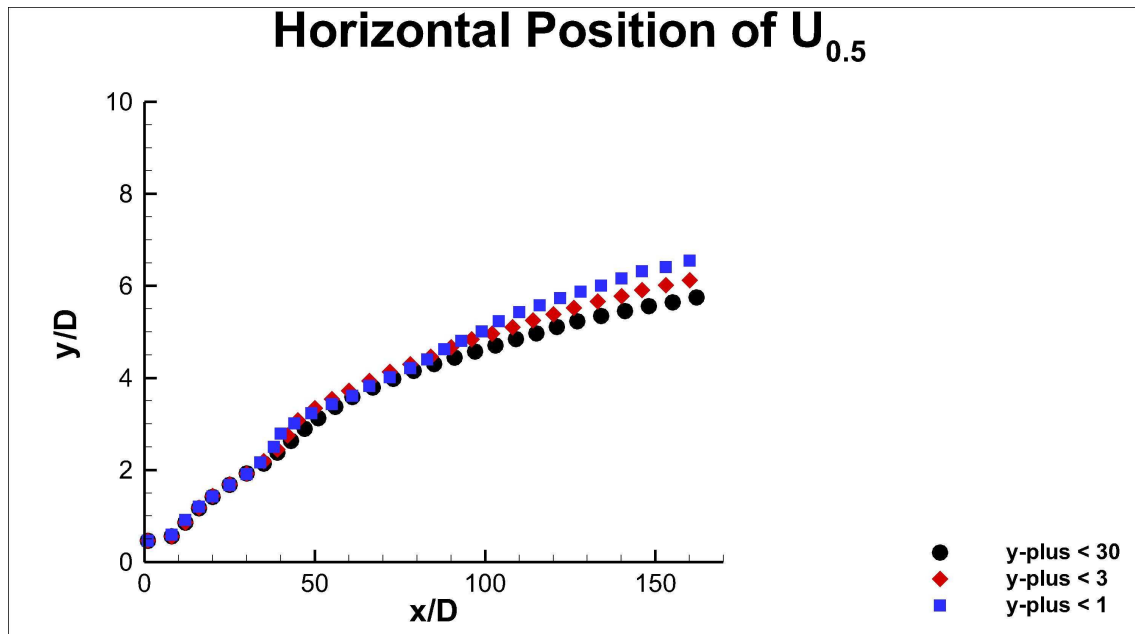
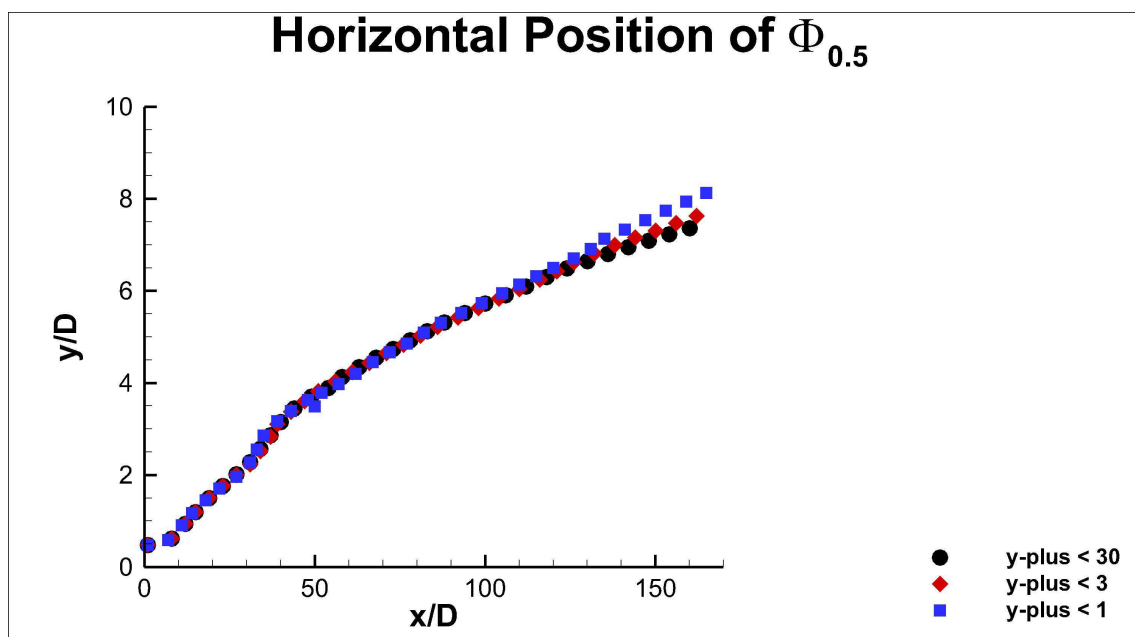
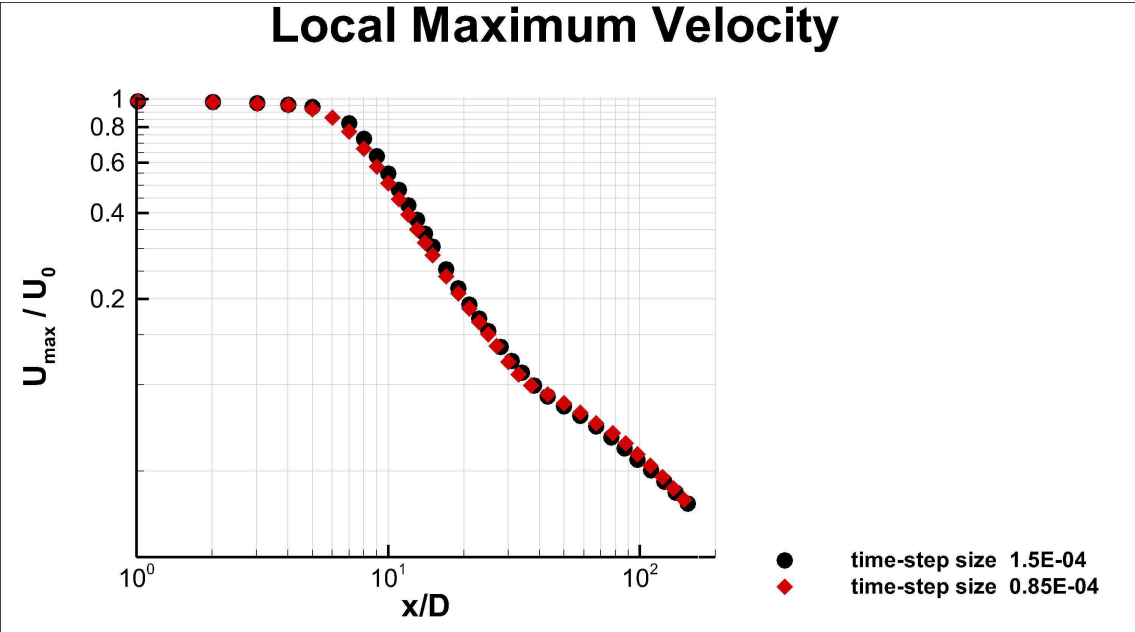
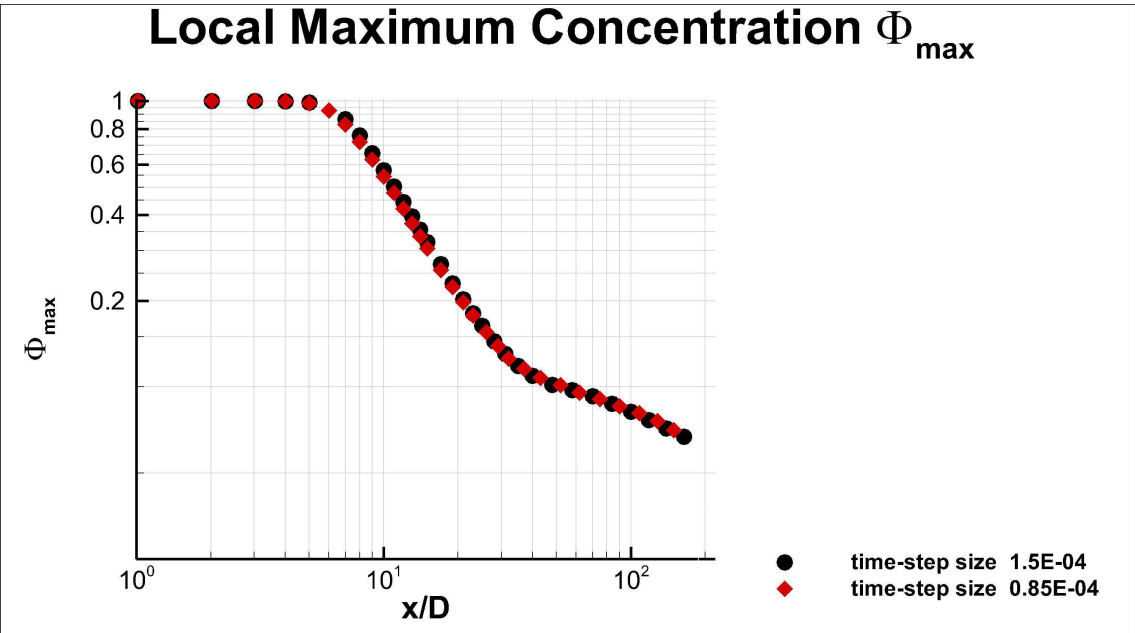
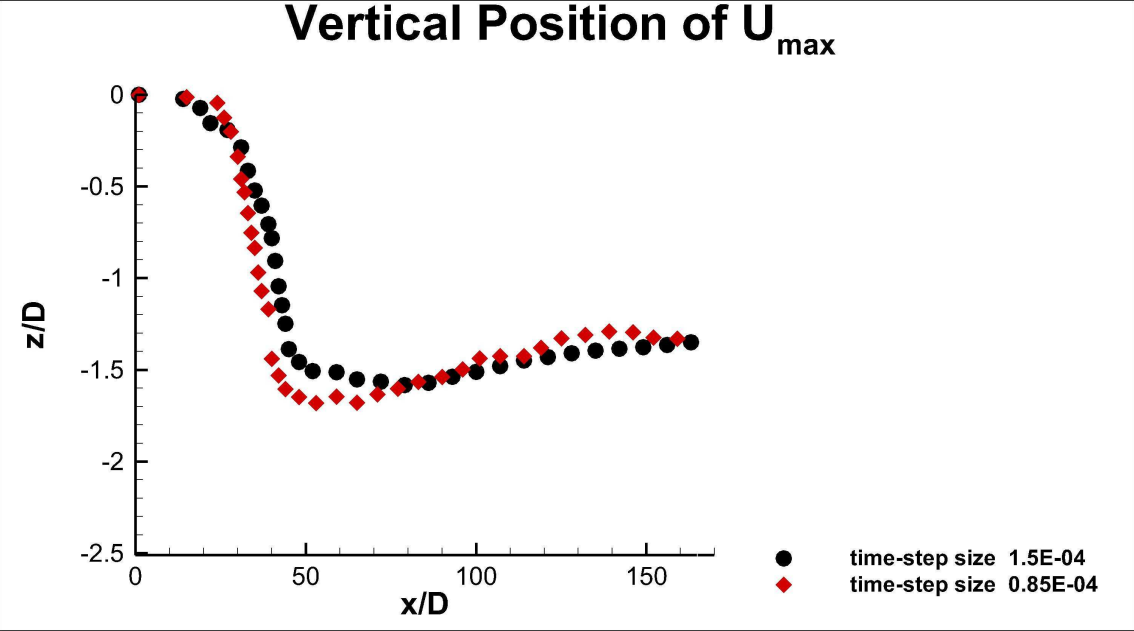


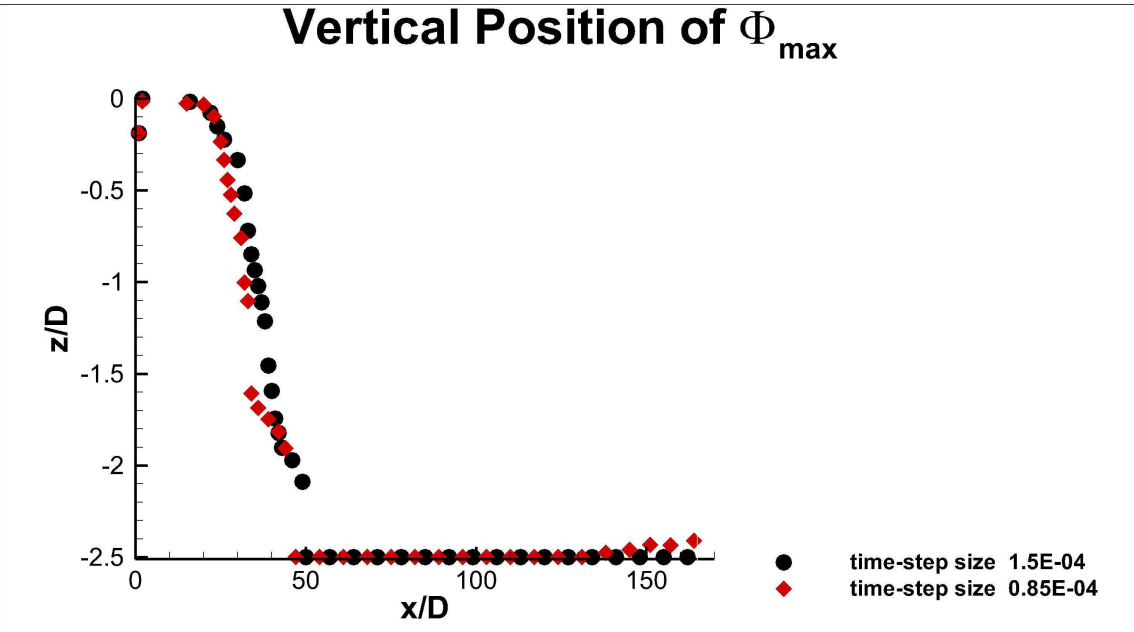
Figure C.3: Streamwise plots of study “Non-Dimensional Wall Distance (Y^+)”.

Study Nr.4: Temporal Resolution

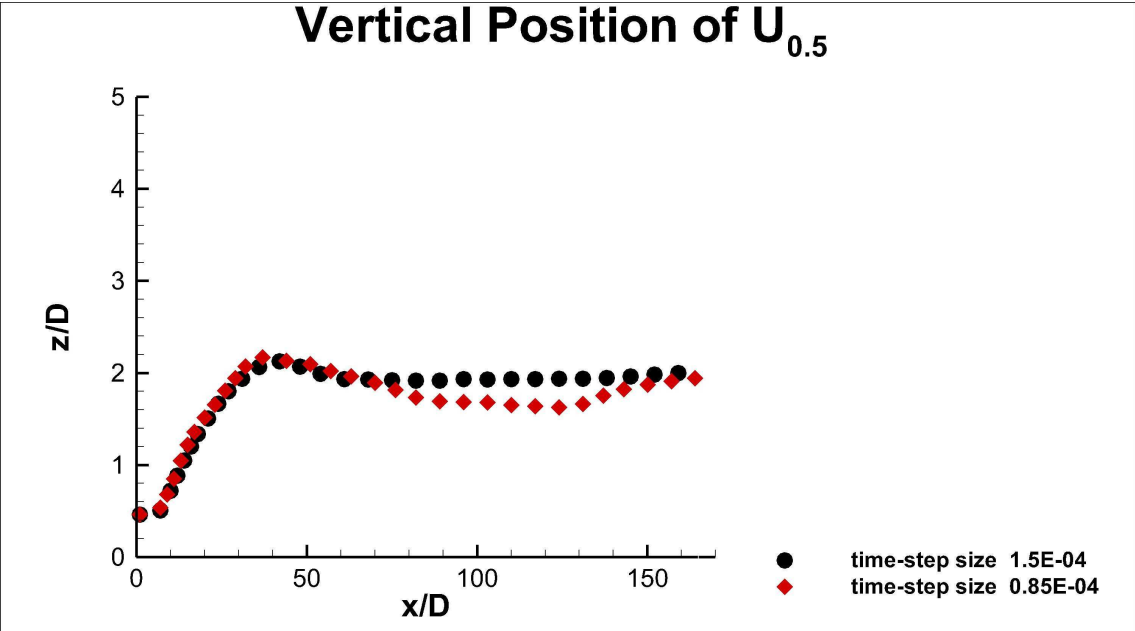




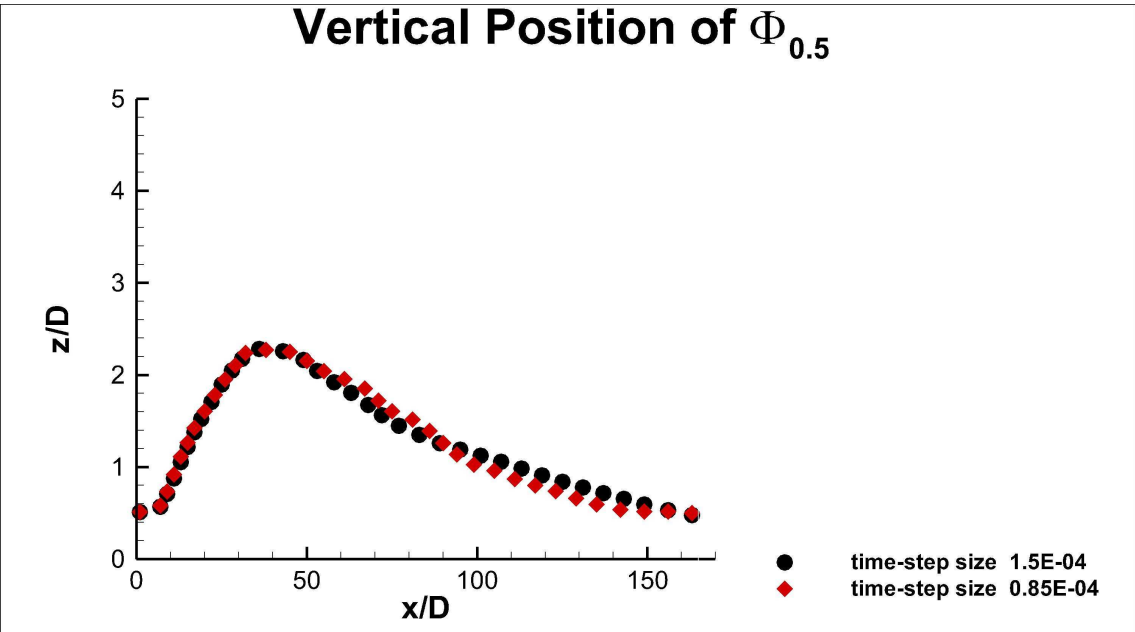
(c)



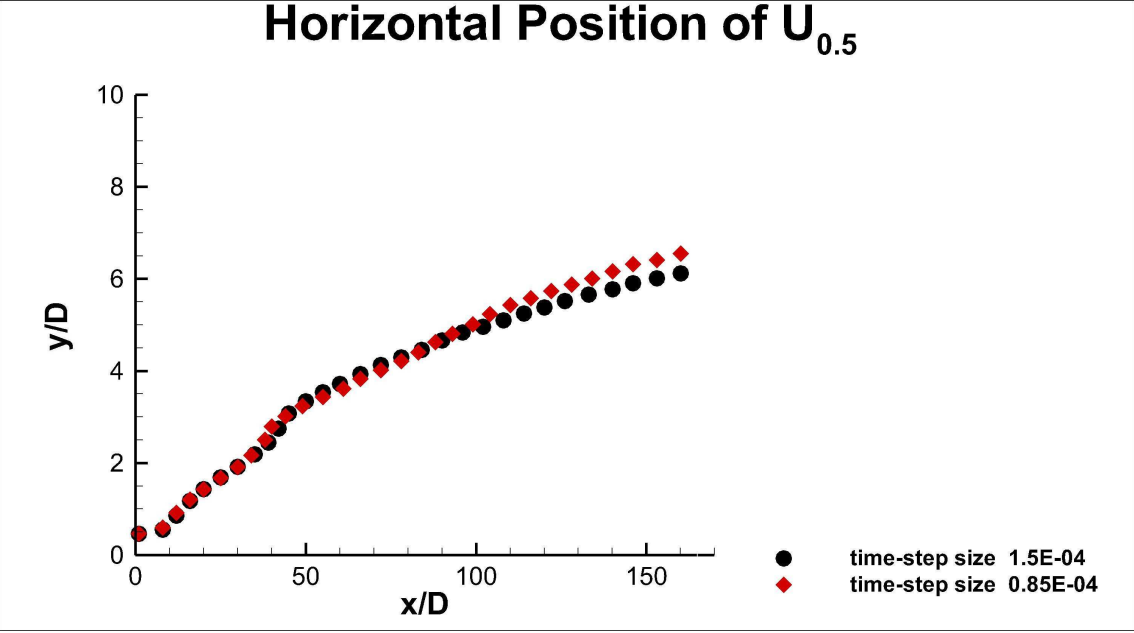
(d)



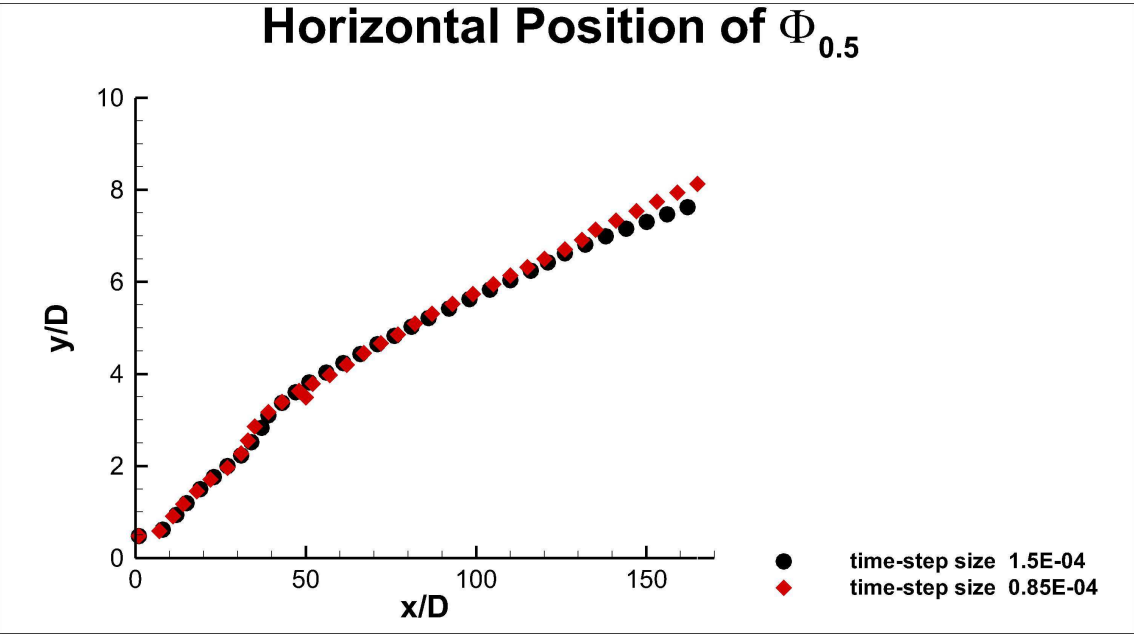
(e)



(f)



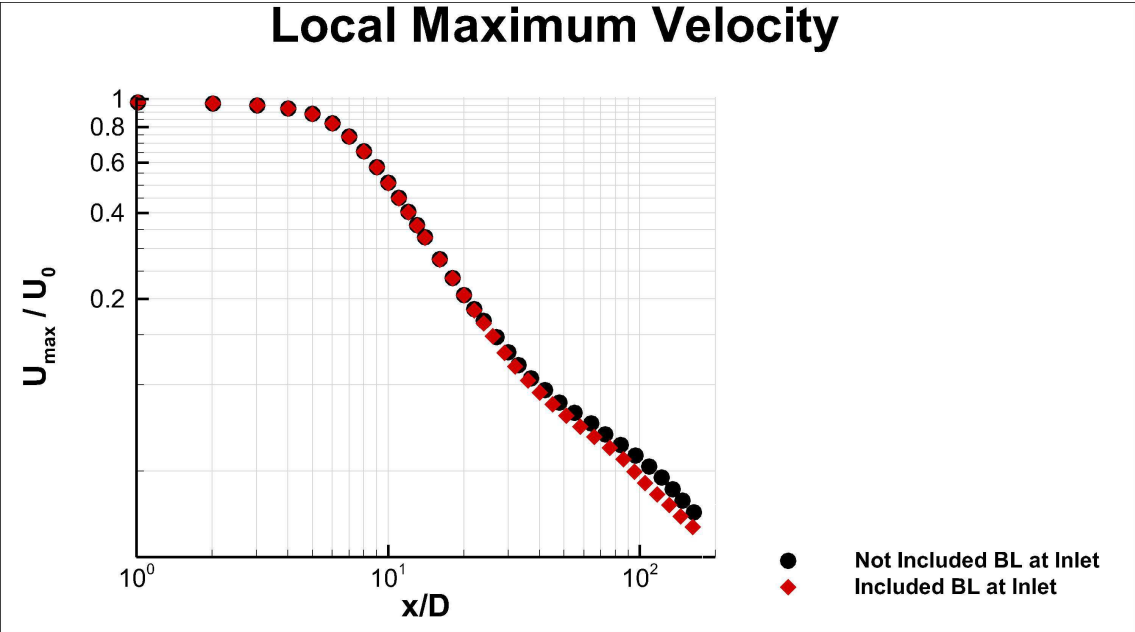
(g)



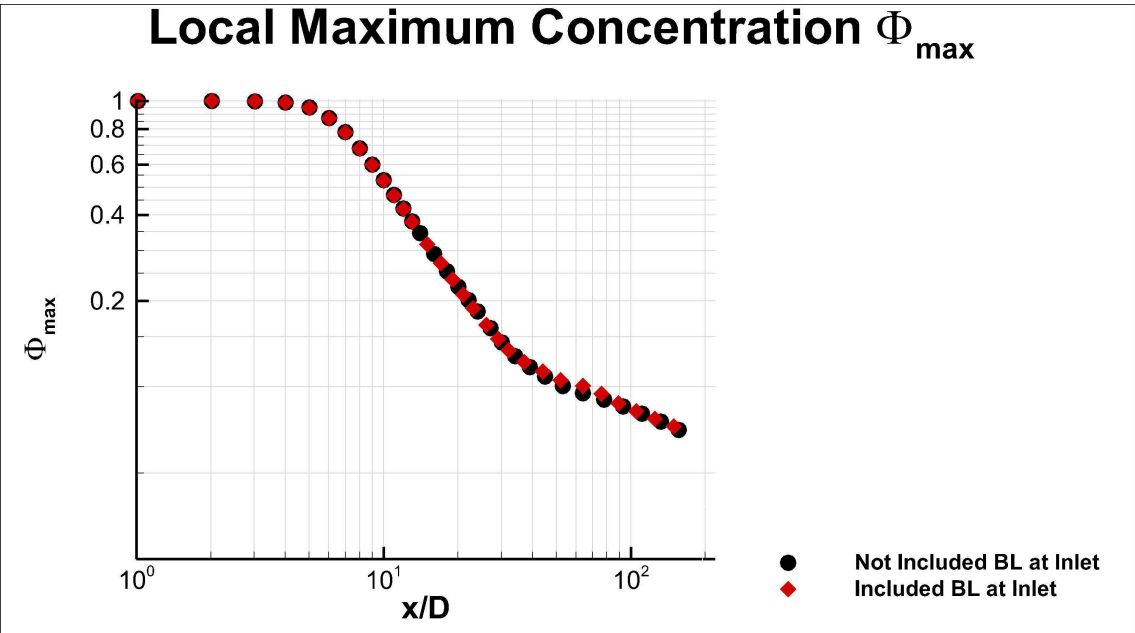
(h)

Figure C.4: Streamwise plots of study “*Temporal Resolution*”.

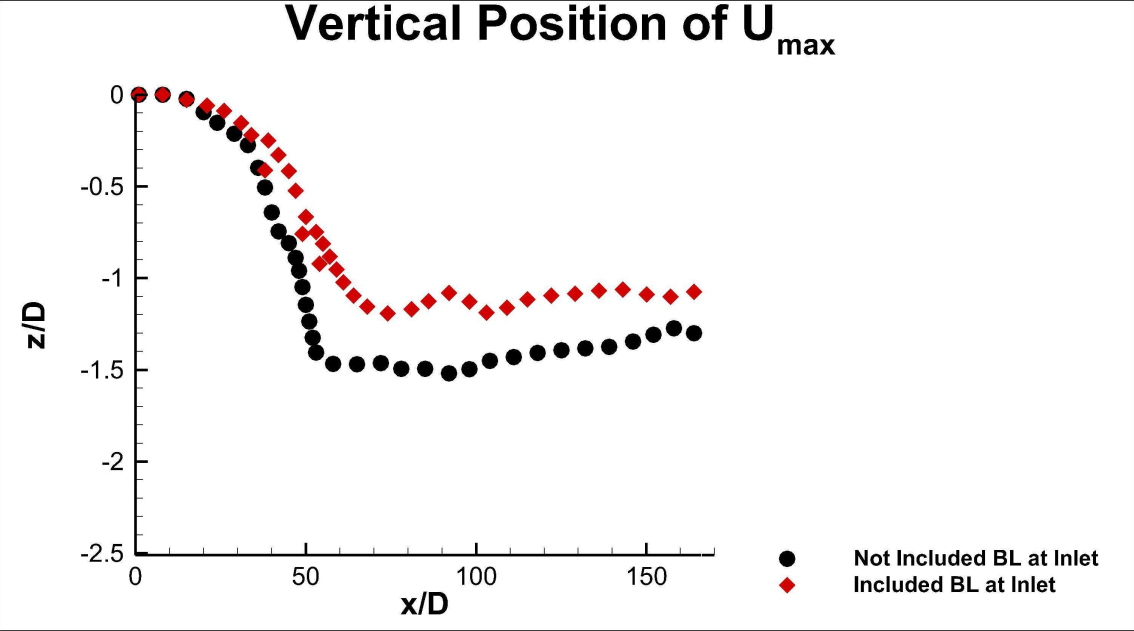
Study Nr.5: Inlet Ground Boundary Layer (H-Type)



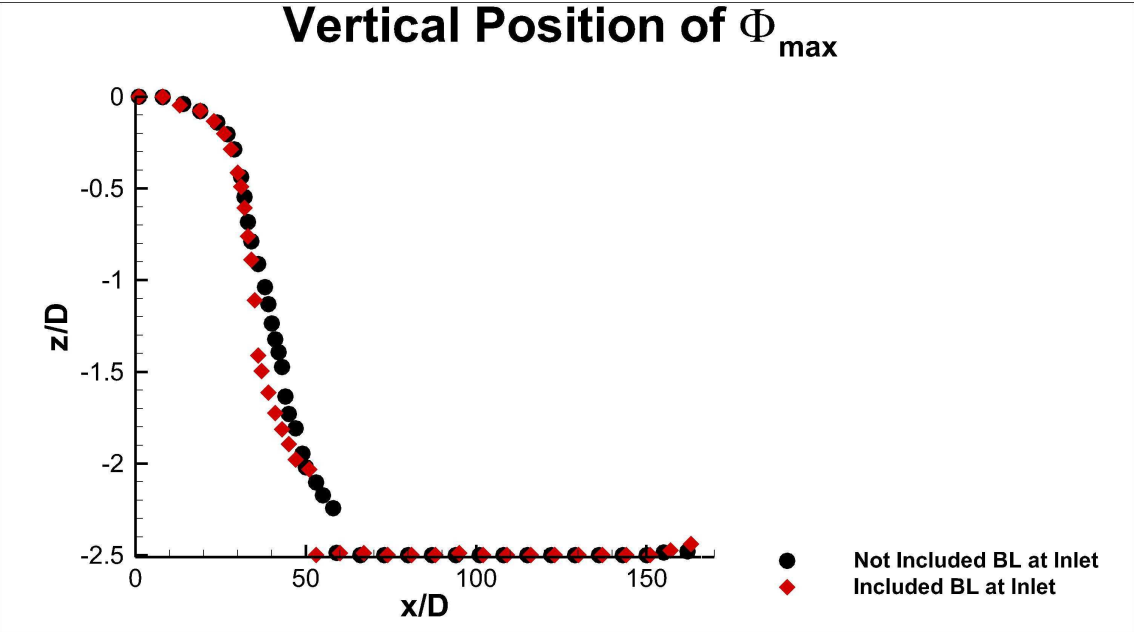
(a)



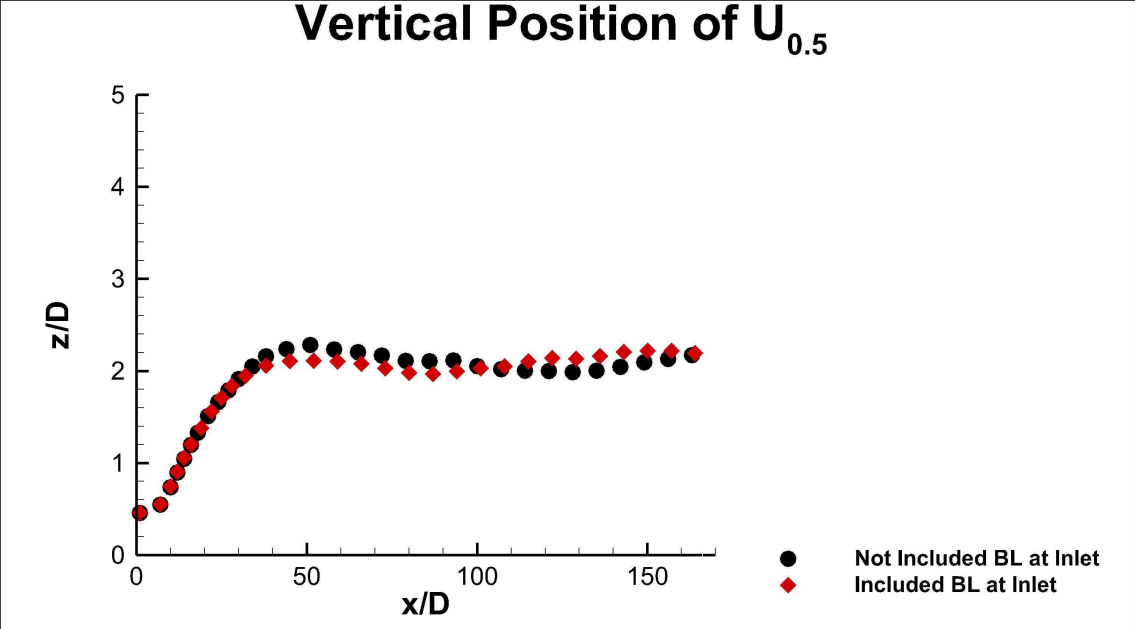
(b)



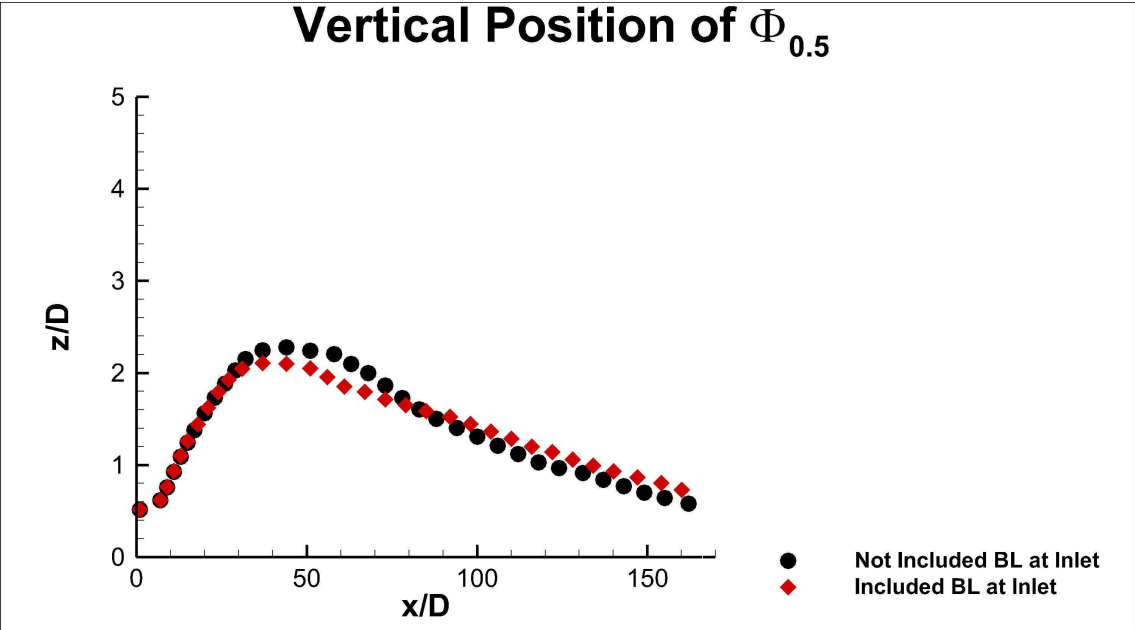
(c)



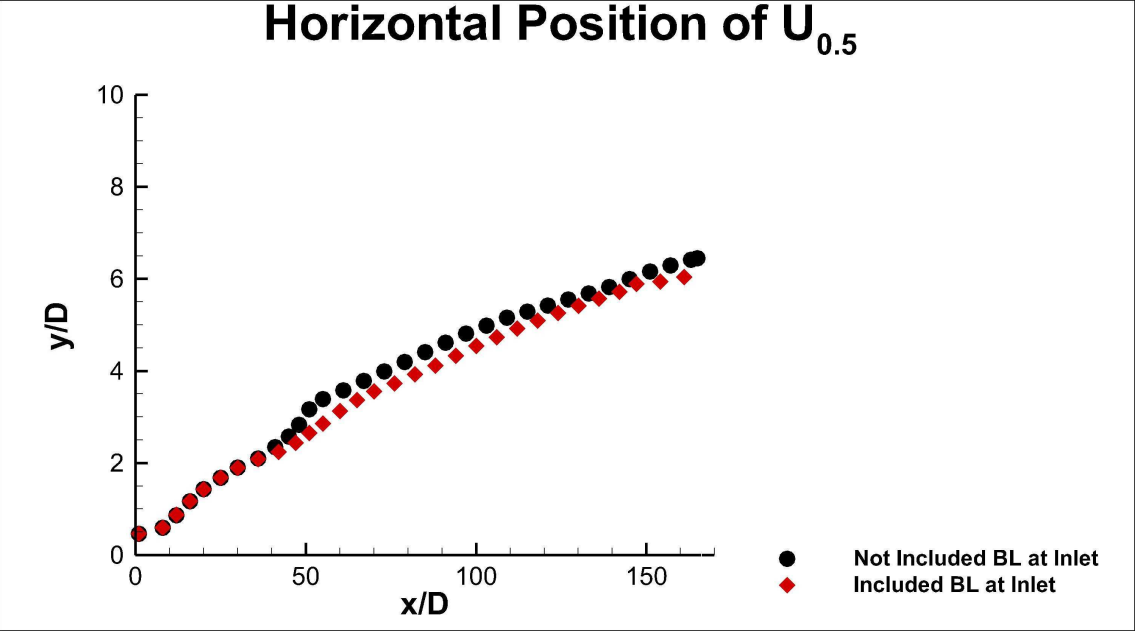
(d)



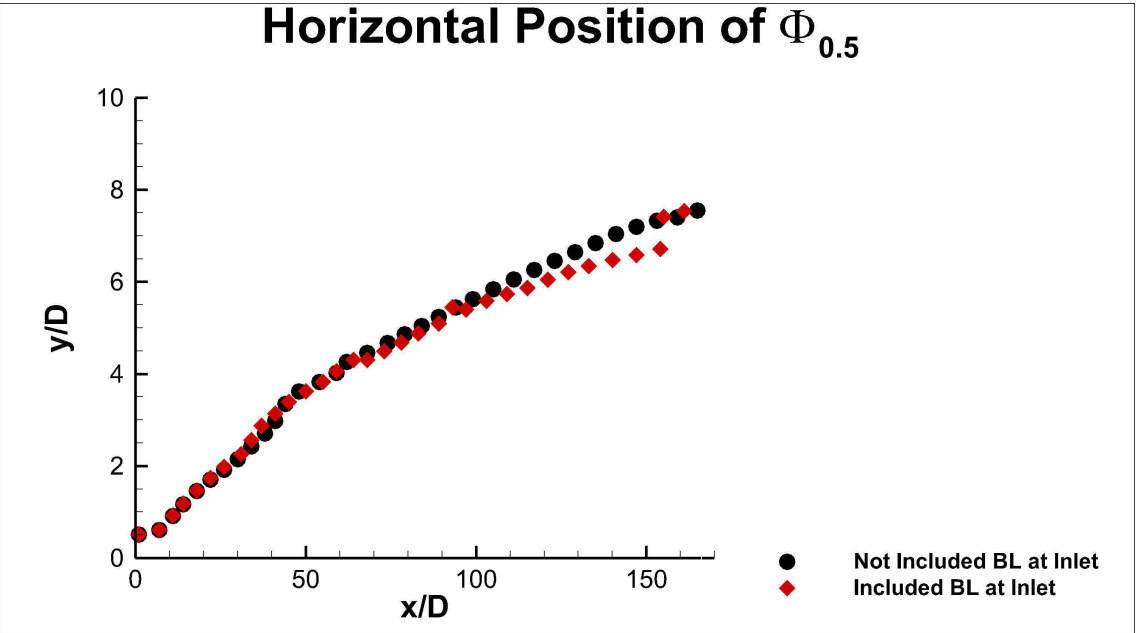
(e)



(f)



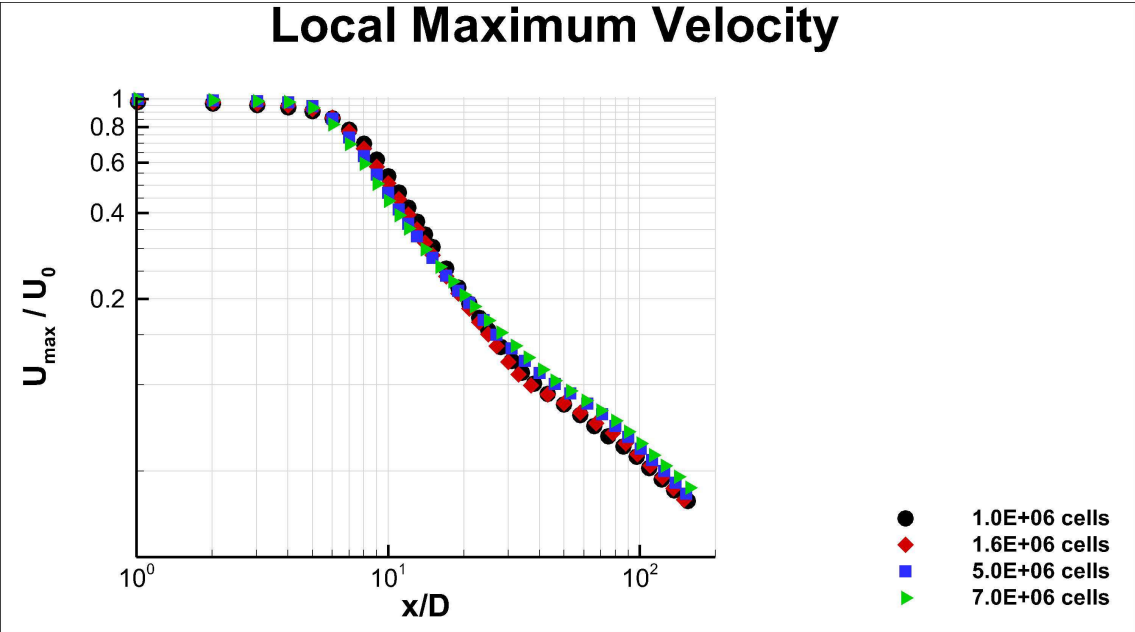
(g)



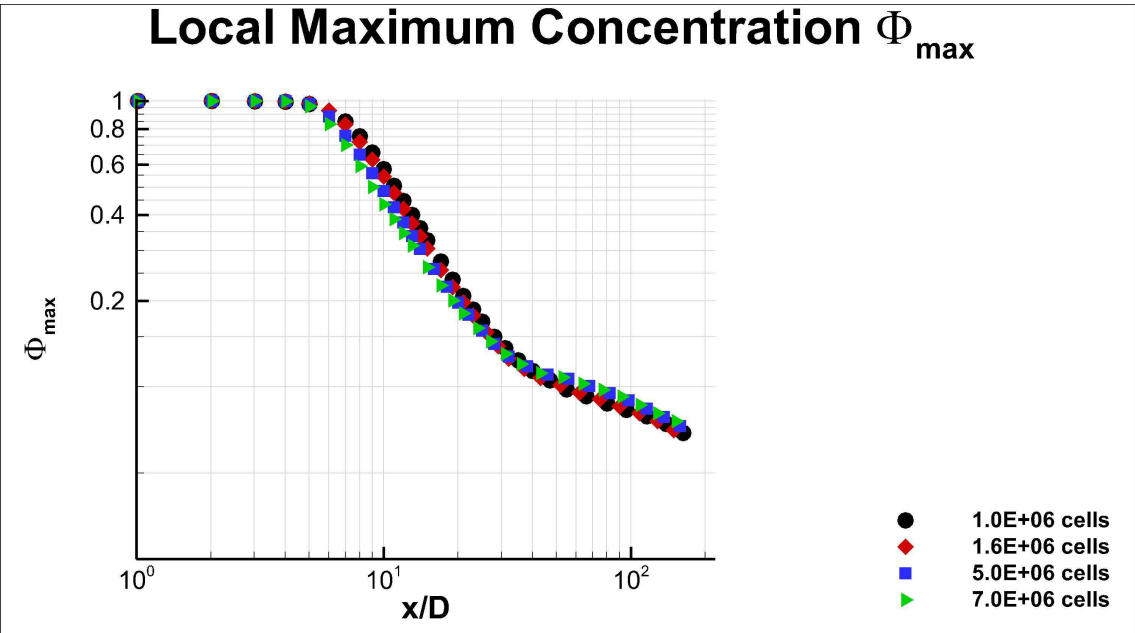
(h)

Figure C.5: Streamwise plots of study “Inlet Ground Boundary Layer (H-Type)”.

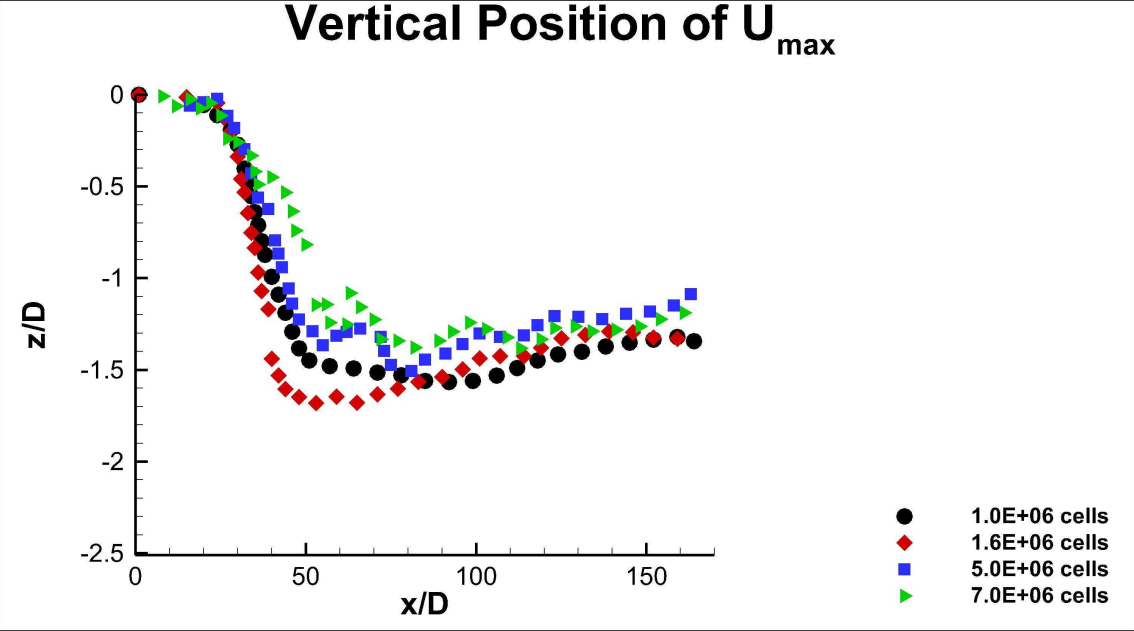
Study Nr.6: H-Type Mesh Density



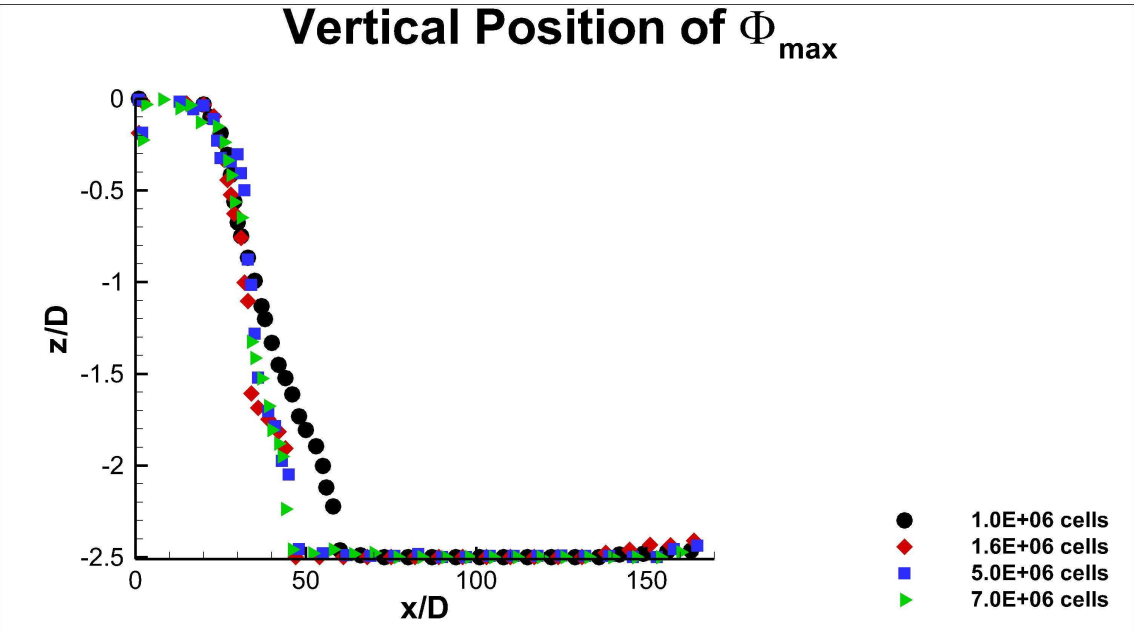
(a)



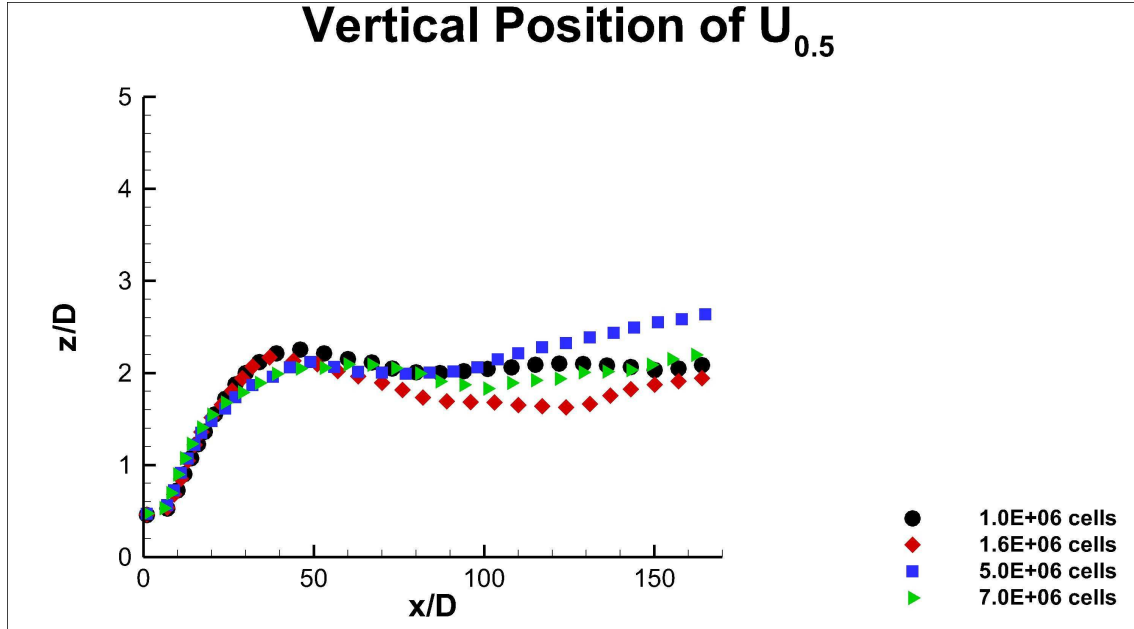
(b)



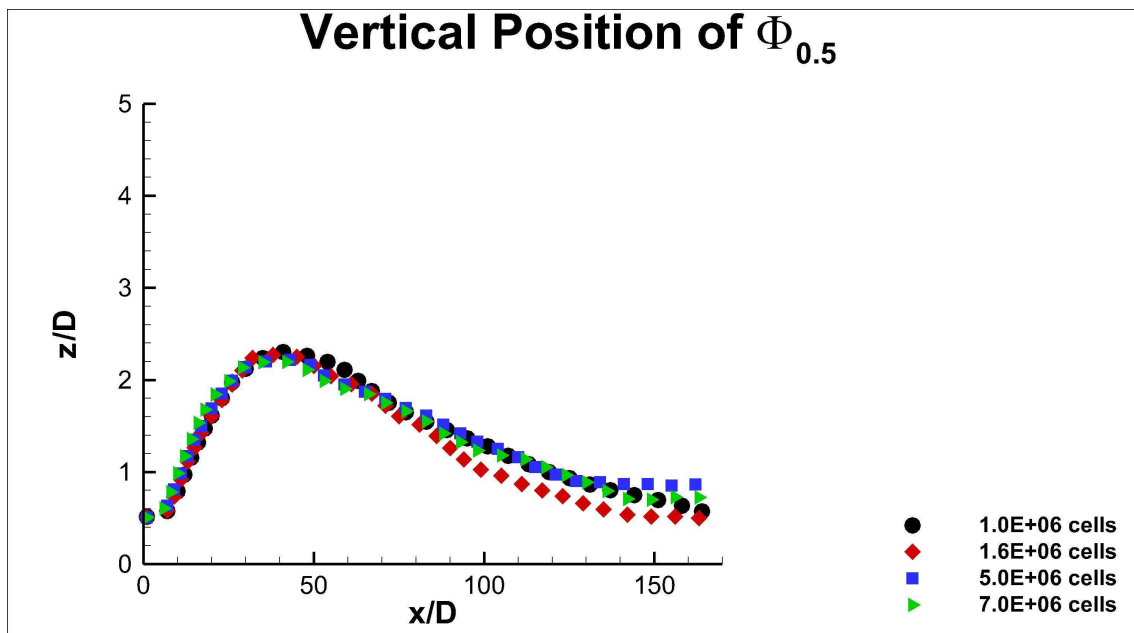
(c)



(d)



(e)



(f)

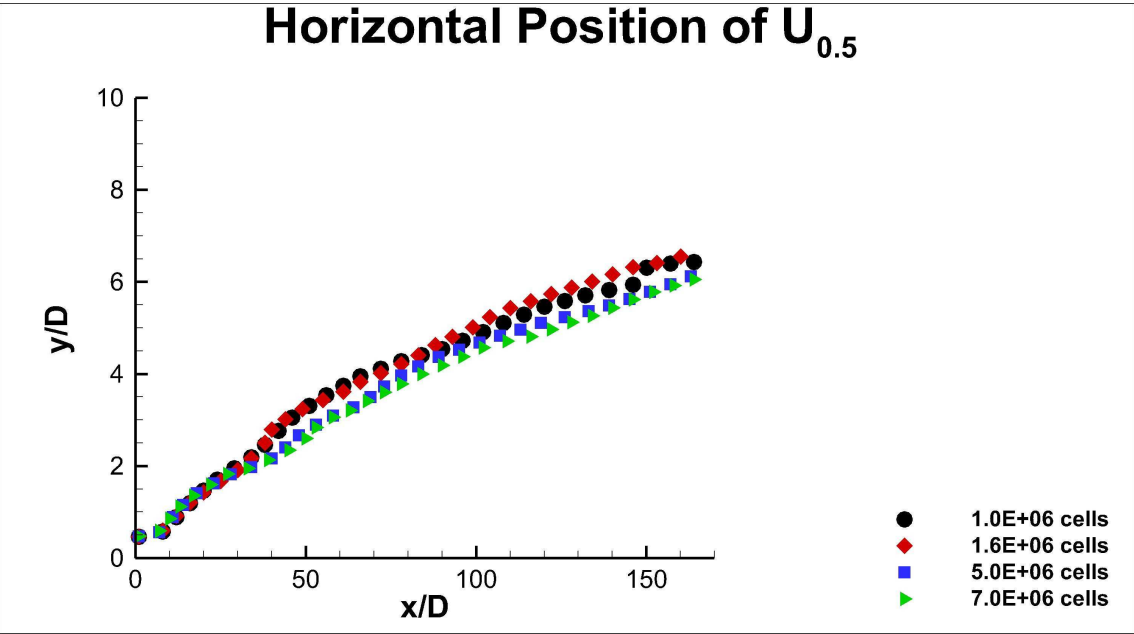
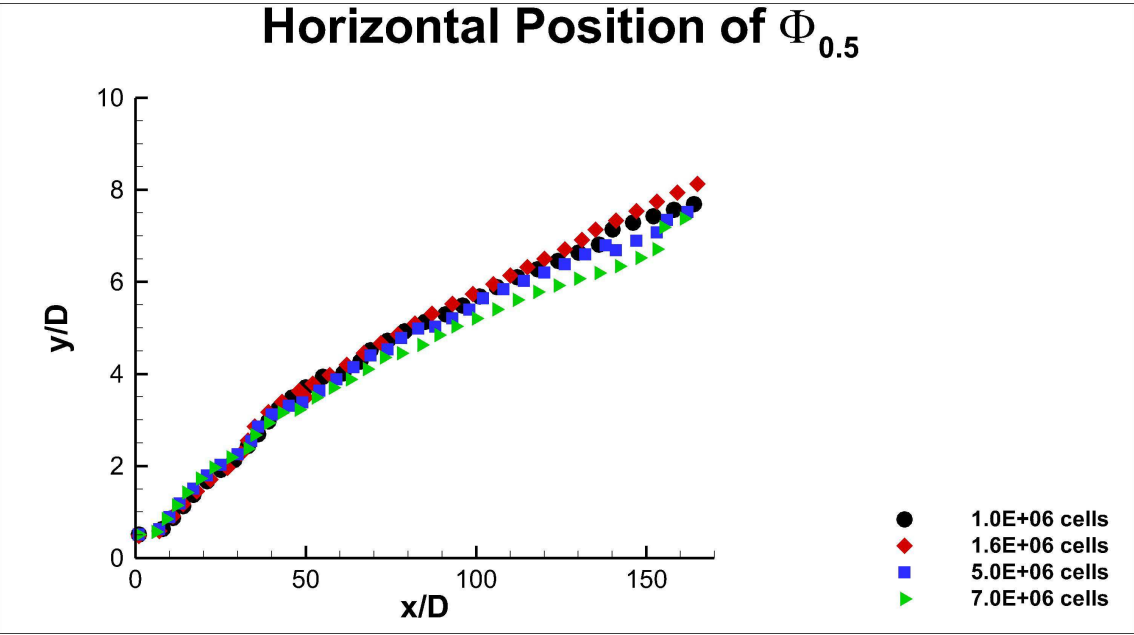
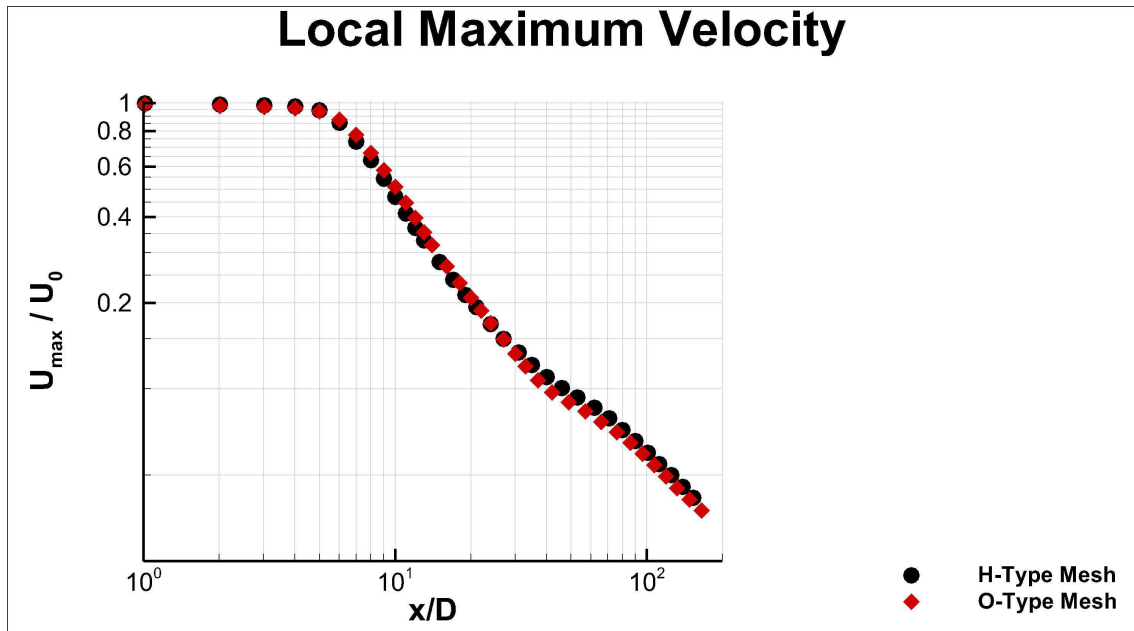
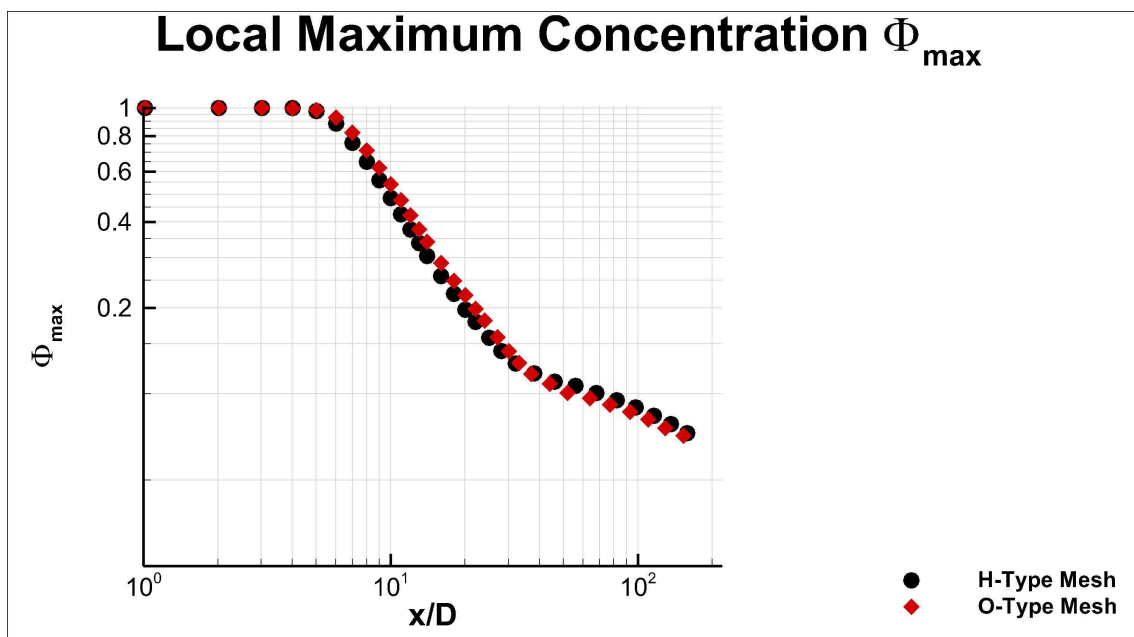
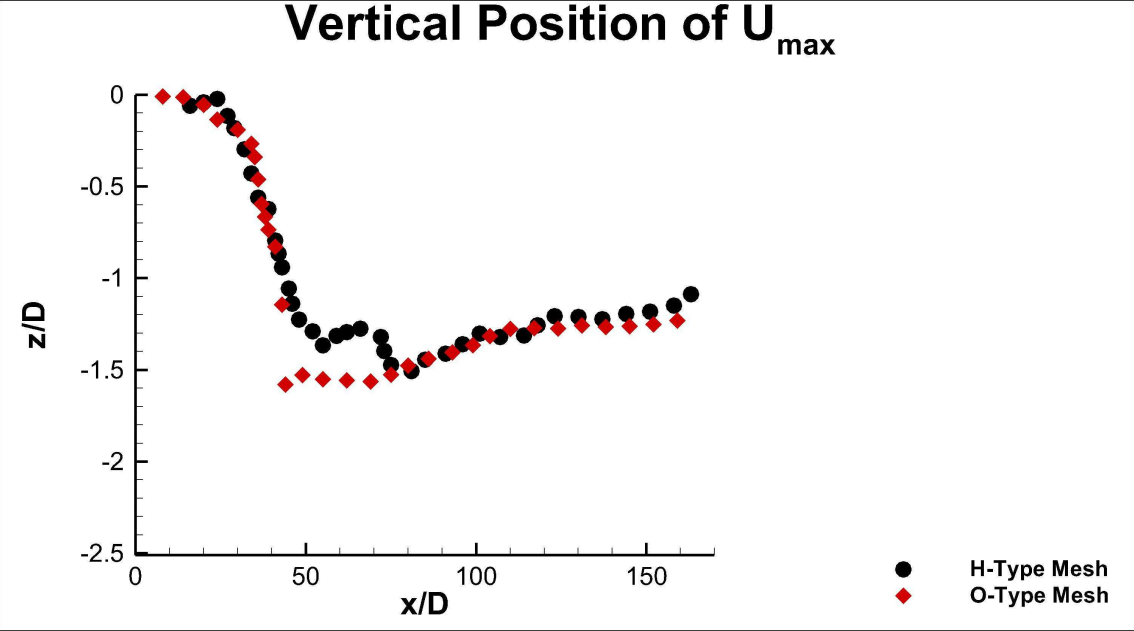
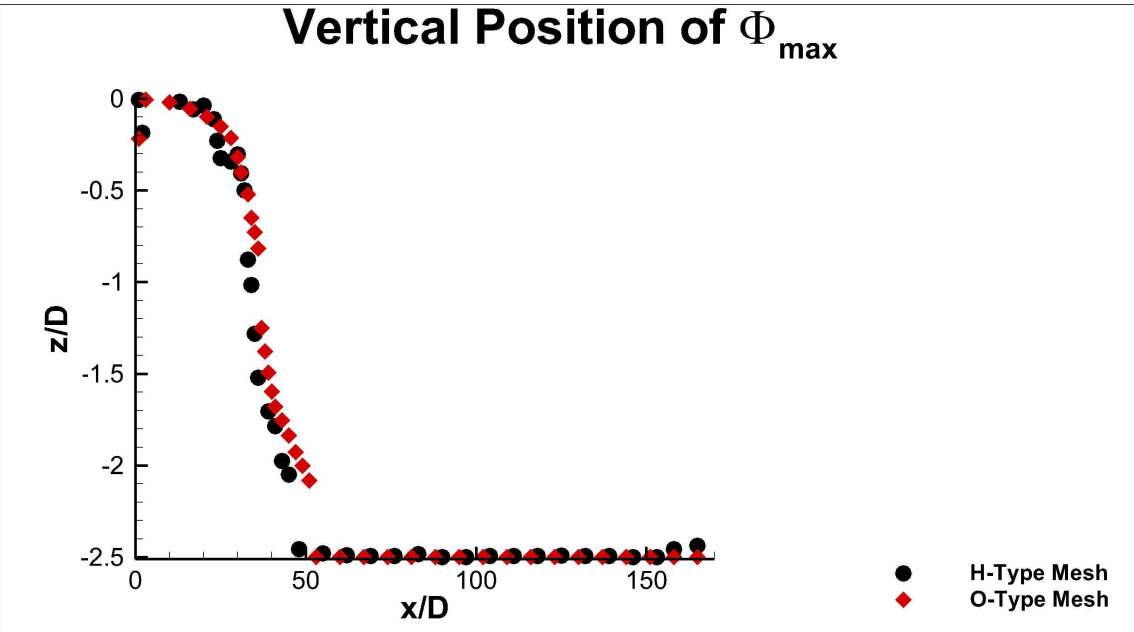
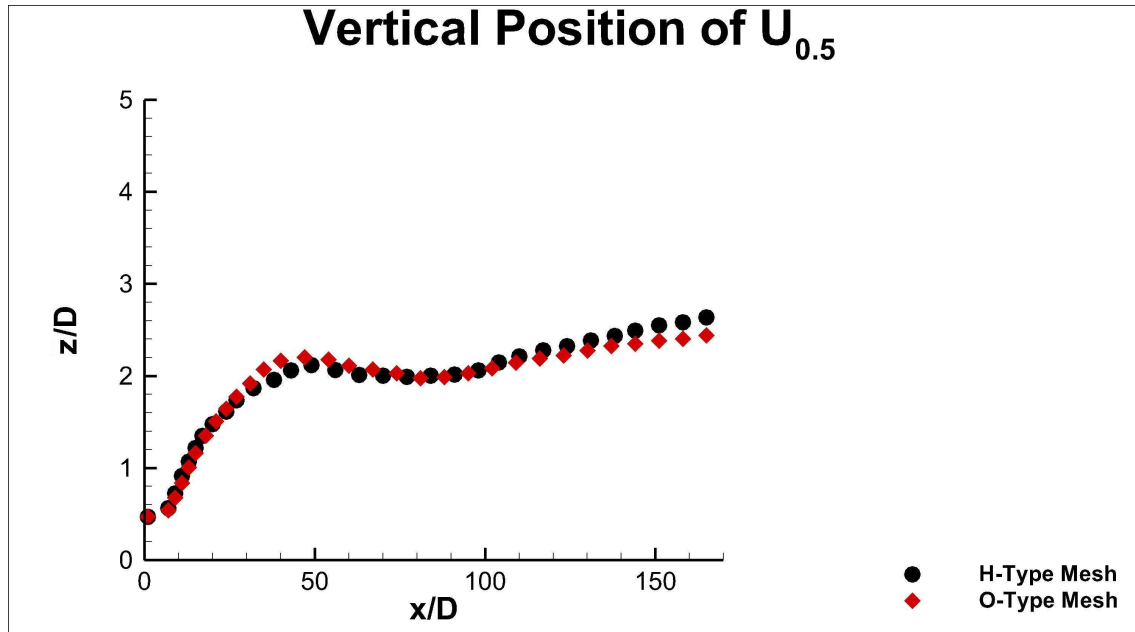


Figure C.6: Streamwise plots of study “H-Type Mesh Density”.

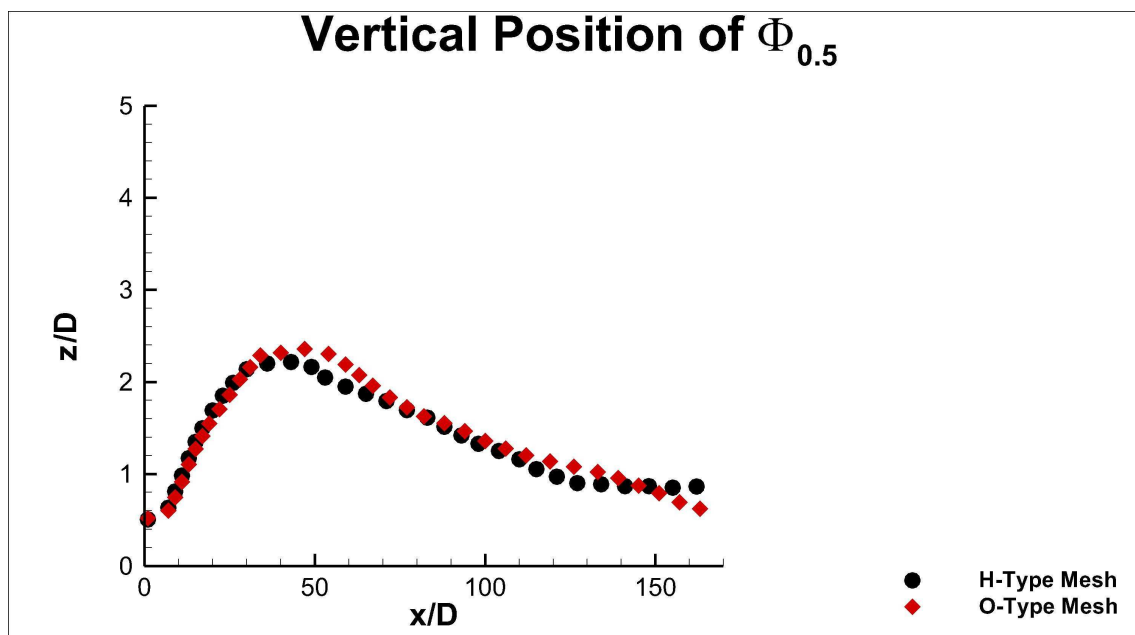
Study Nr.7: H-Type Vs O-Type



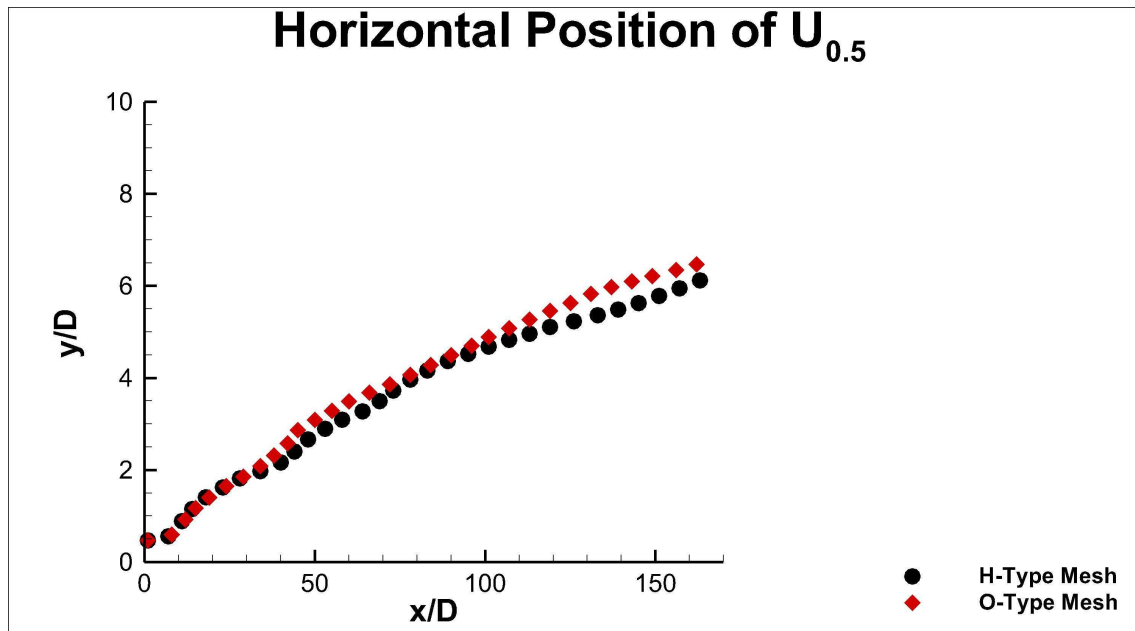




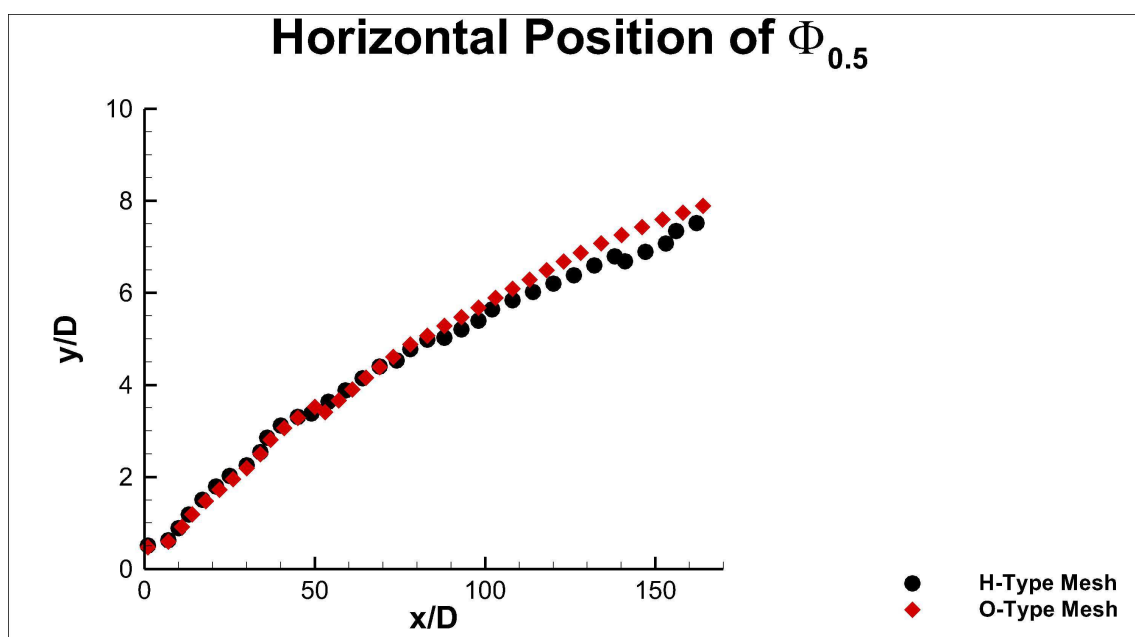
(e)



(f)



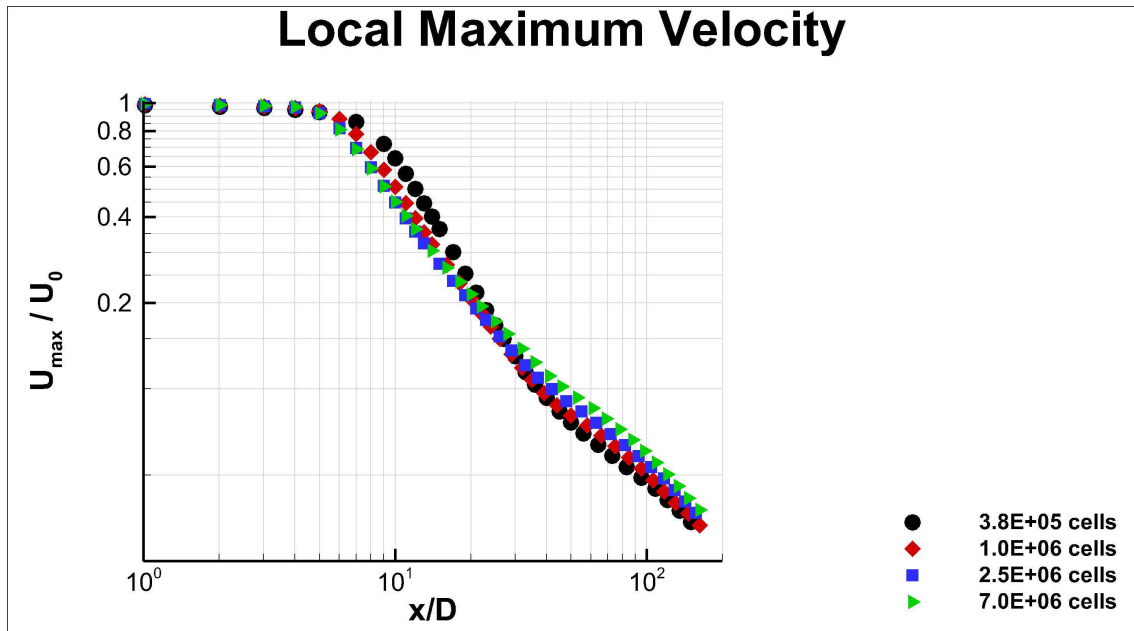
(g)



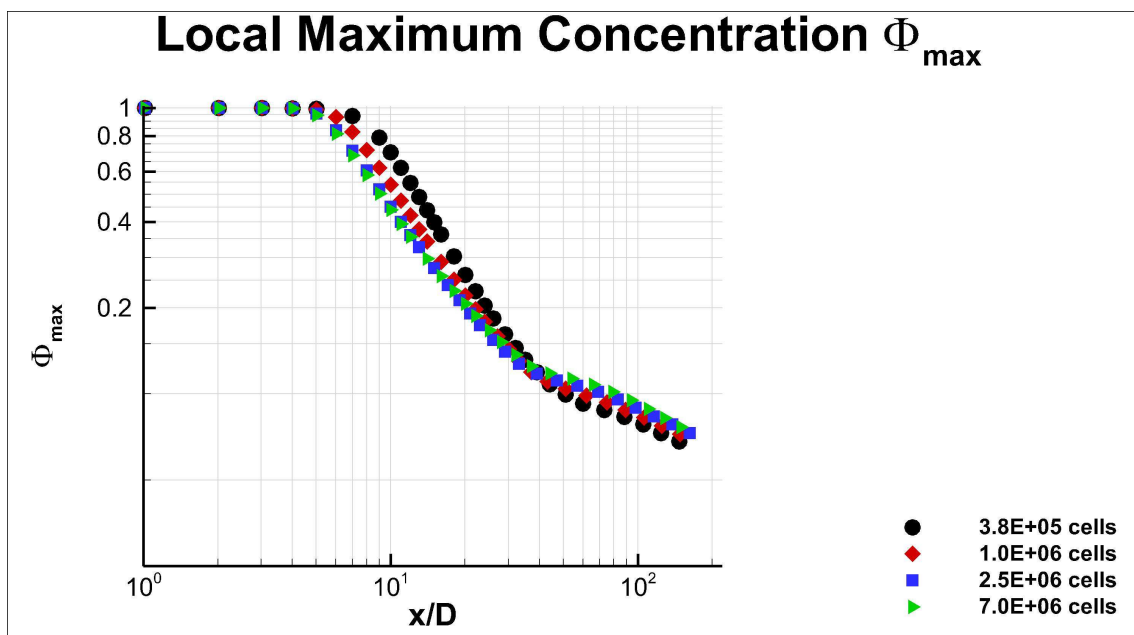
(h)

Figure C.7: Streamwise plots of study “H-Type Vs O-Type”.

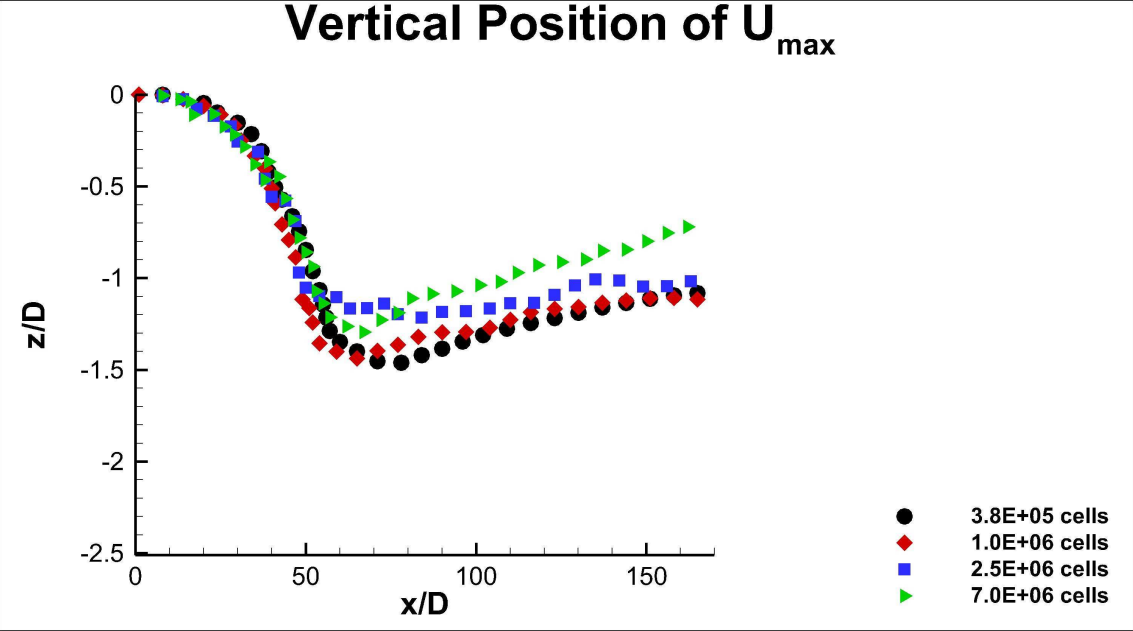
Study Nr.8: O-Type Mesh Density



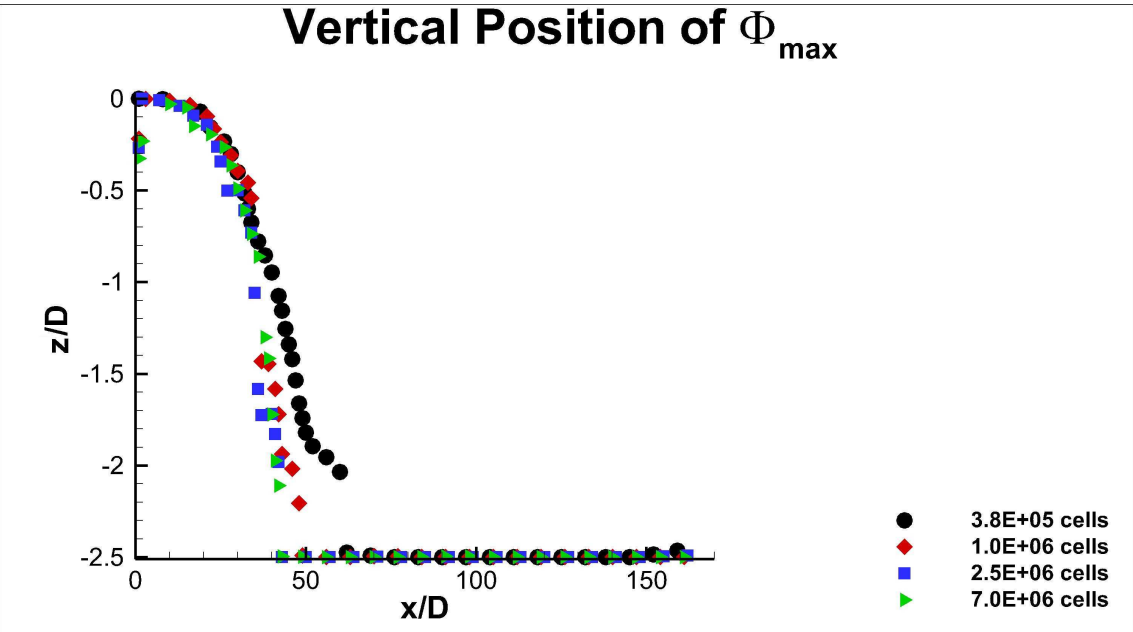
(a)



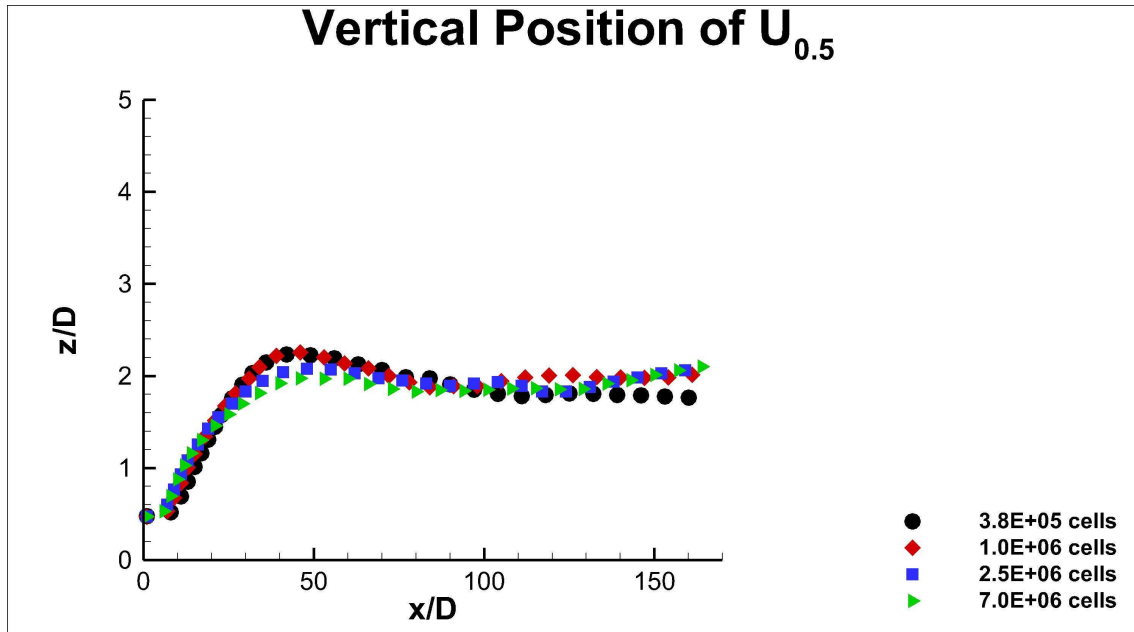
(b)



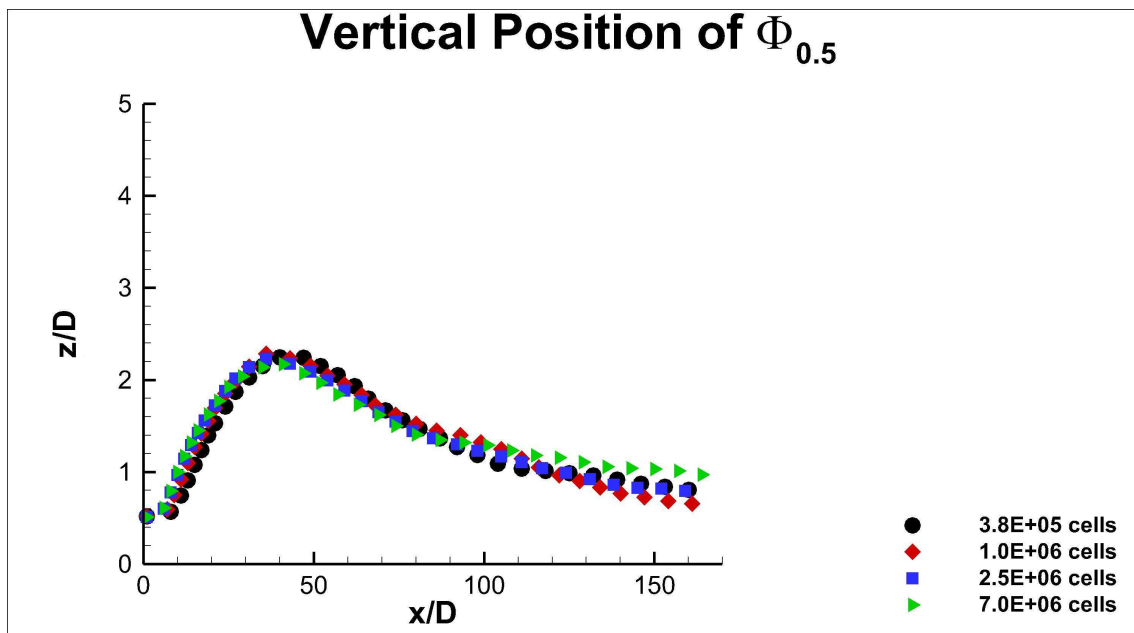
(c)



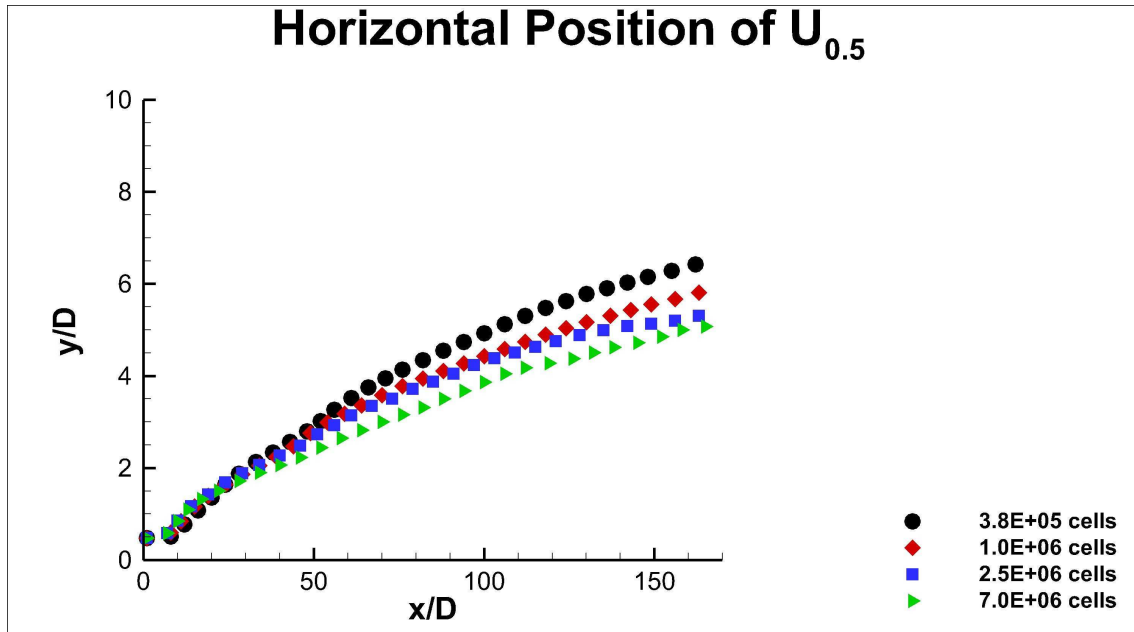
(d)



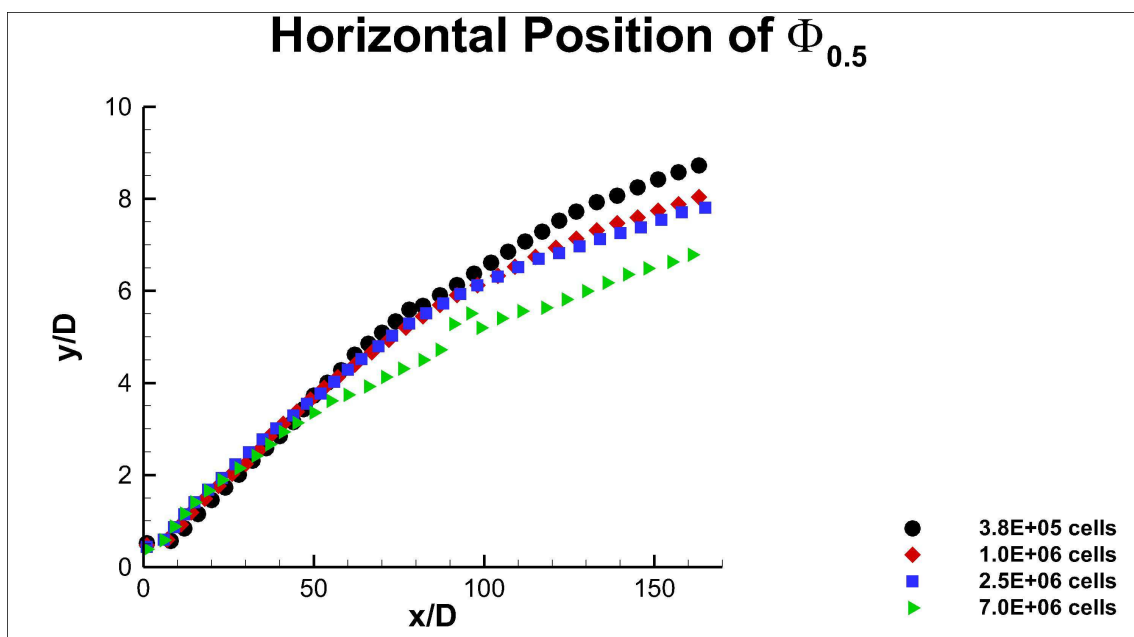
(e)



(f)



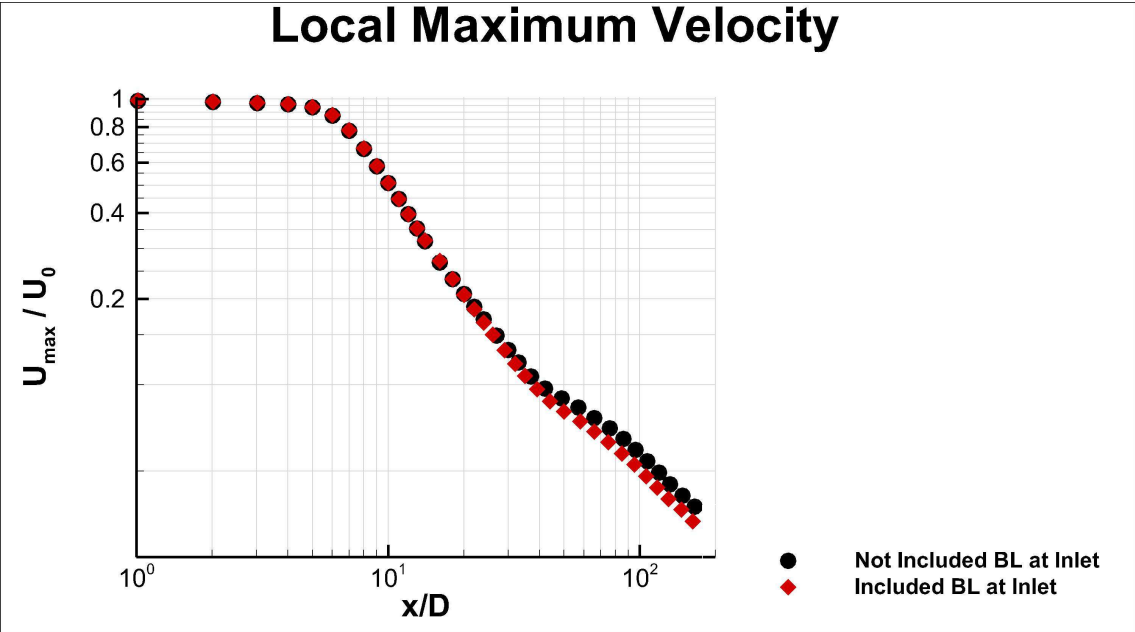
(g)



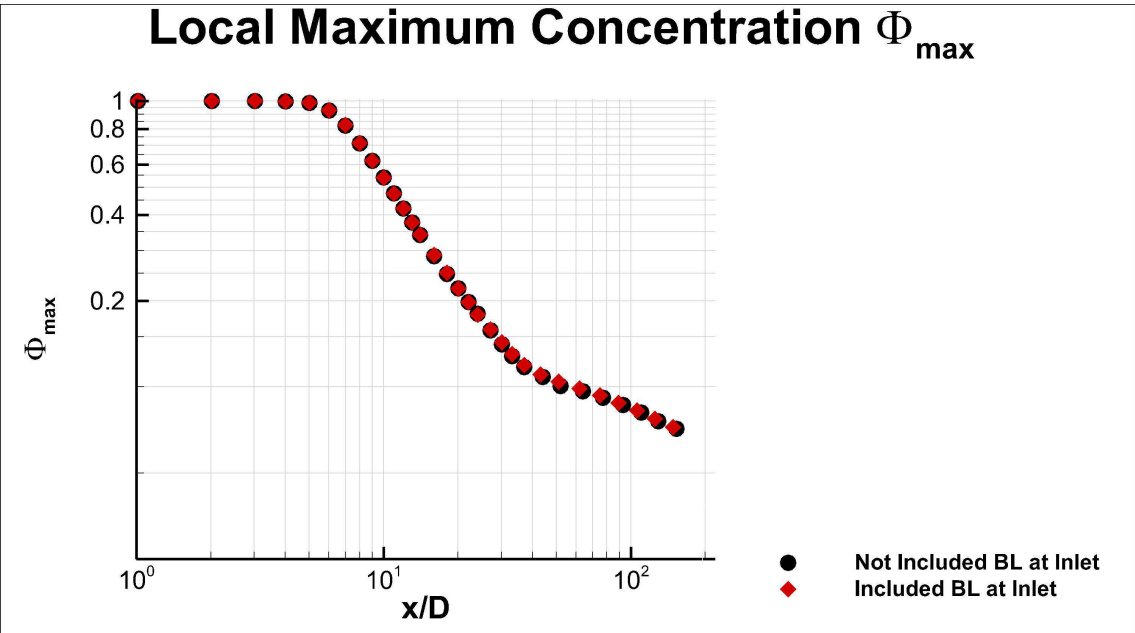
(h)

Figure C.8: Streamwise plots of study “O-Type Mesh Density”.

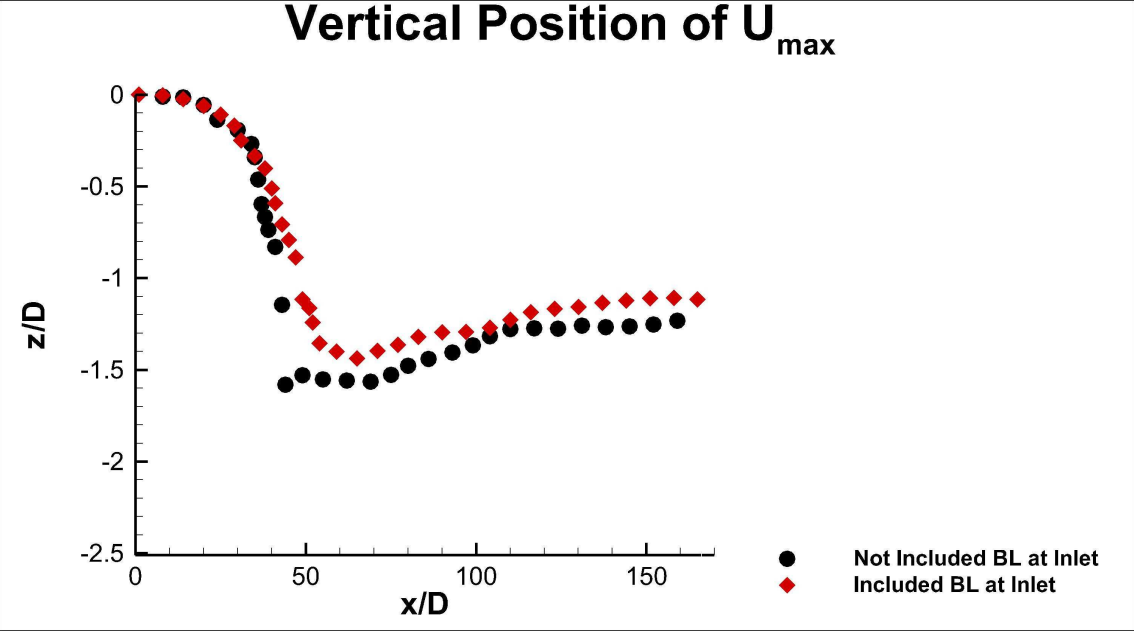
Study Nr.9: Inlet Ground Boundary Layer (O-Type)



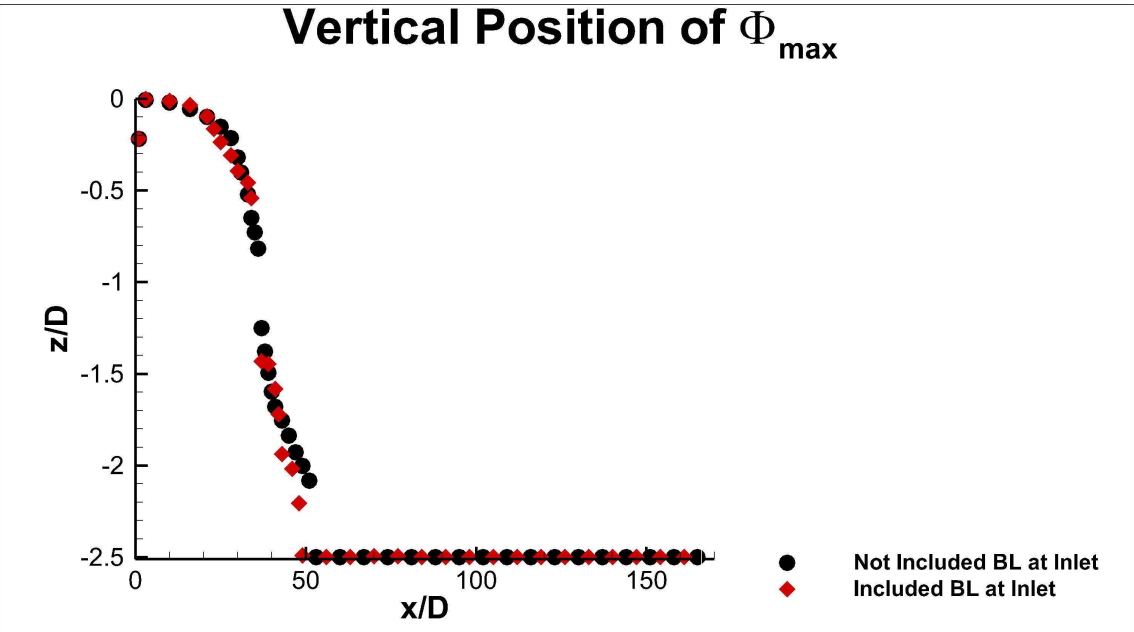
(a)



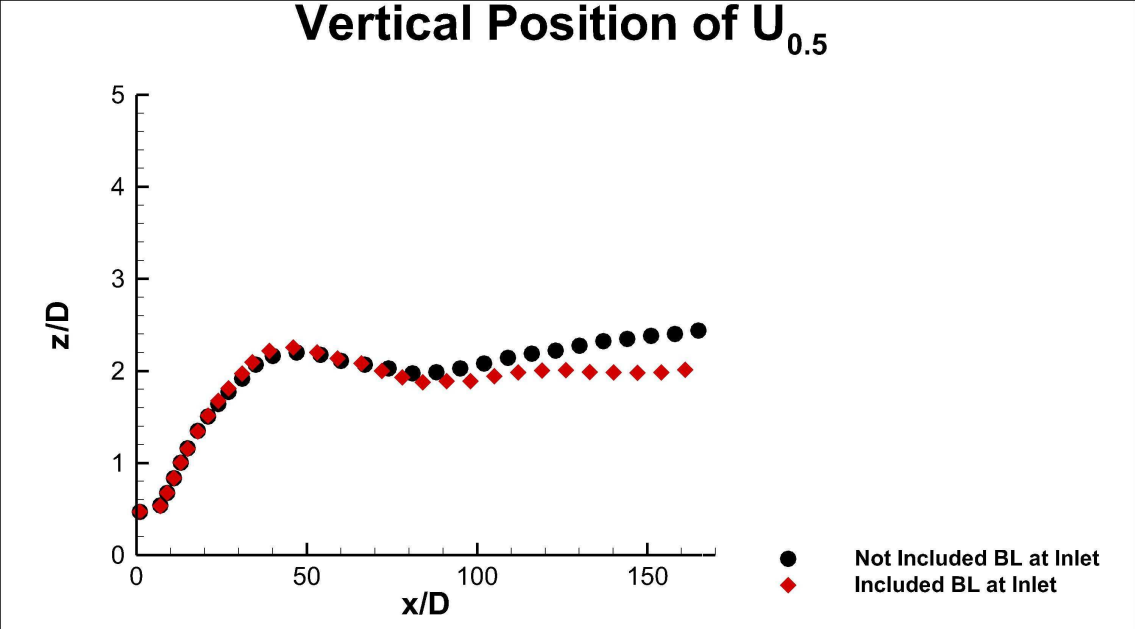
(b)



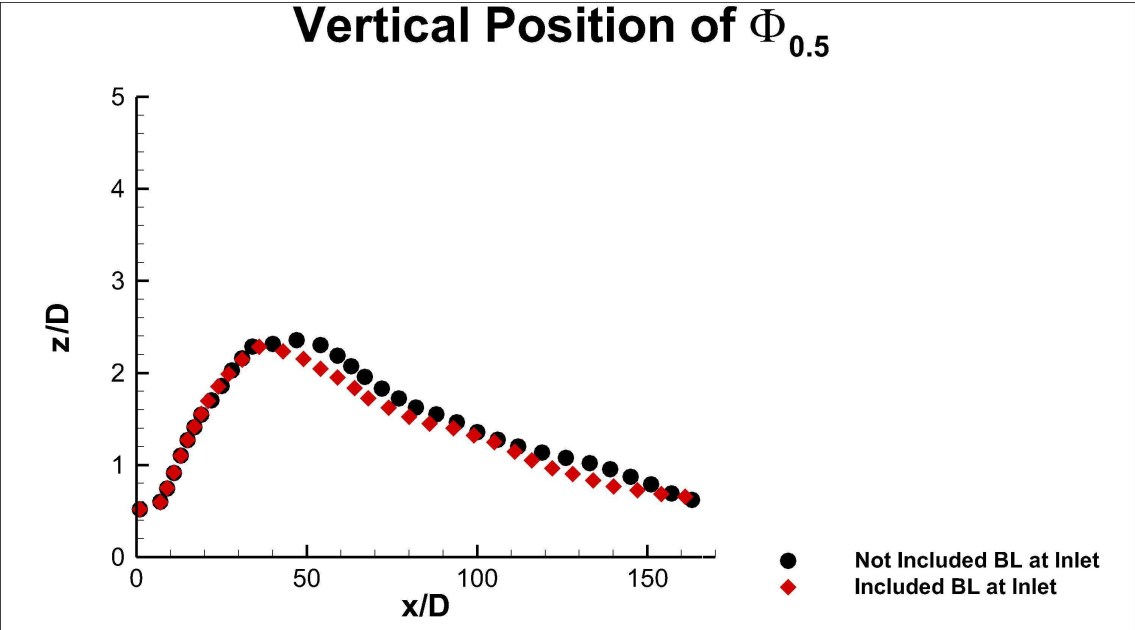
(c)



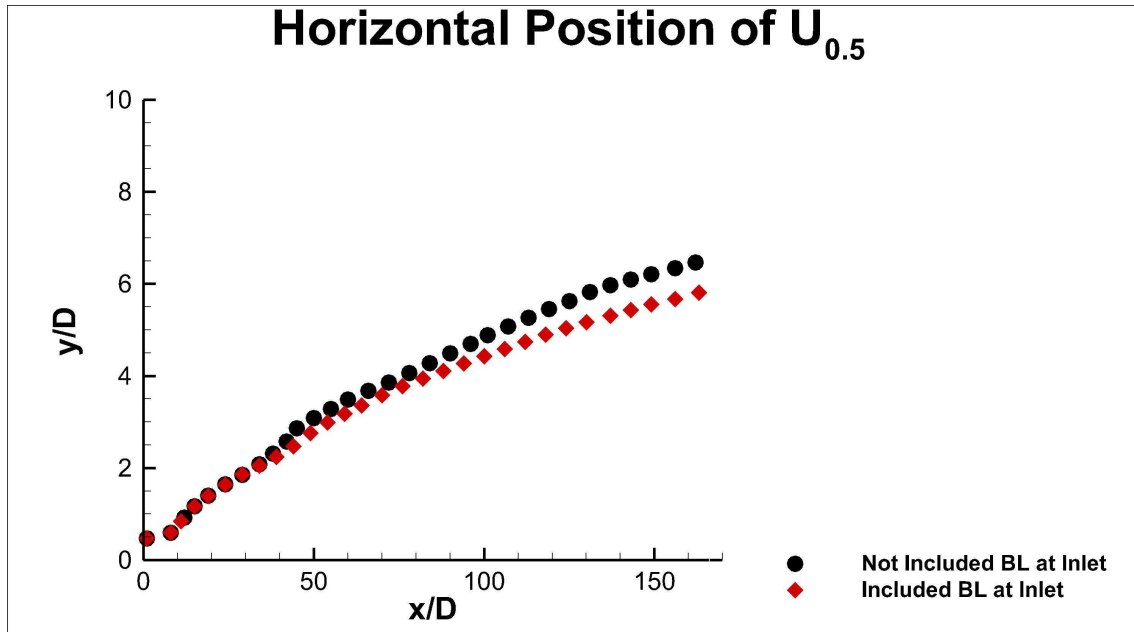
(d)



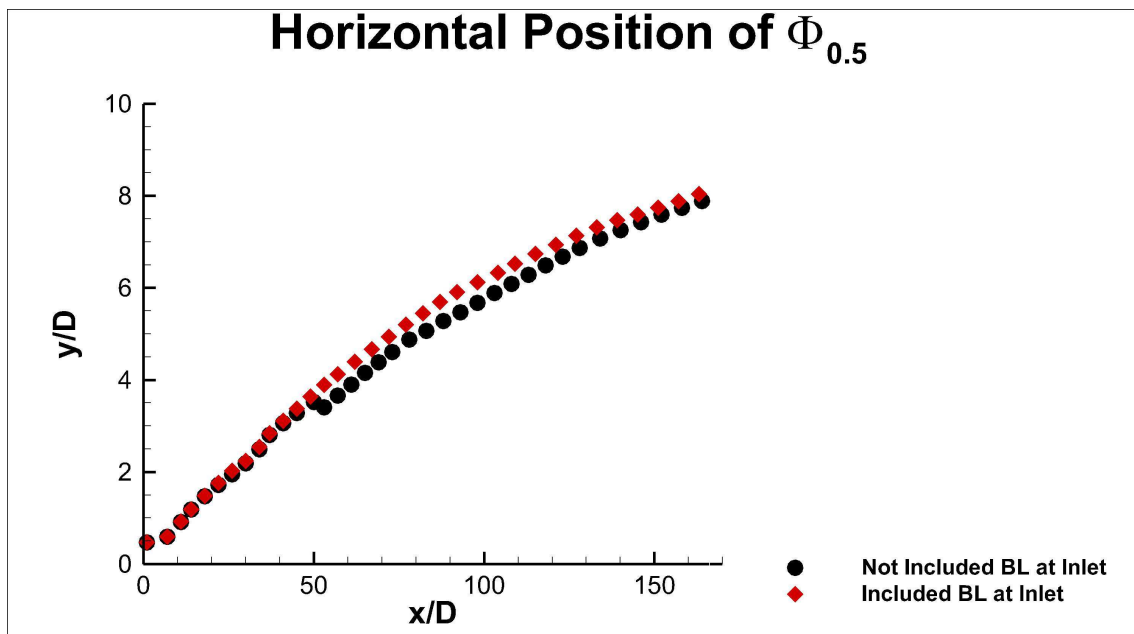
(e)



(f)



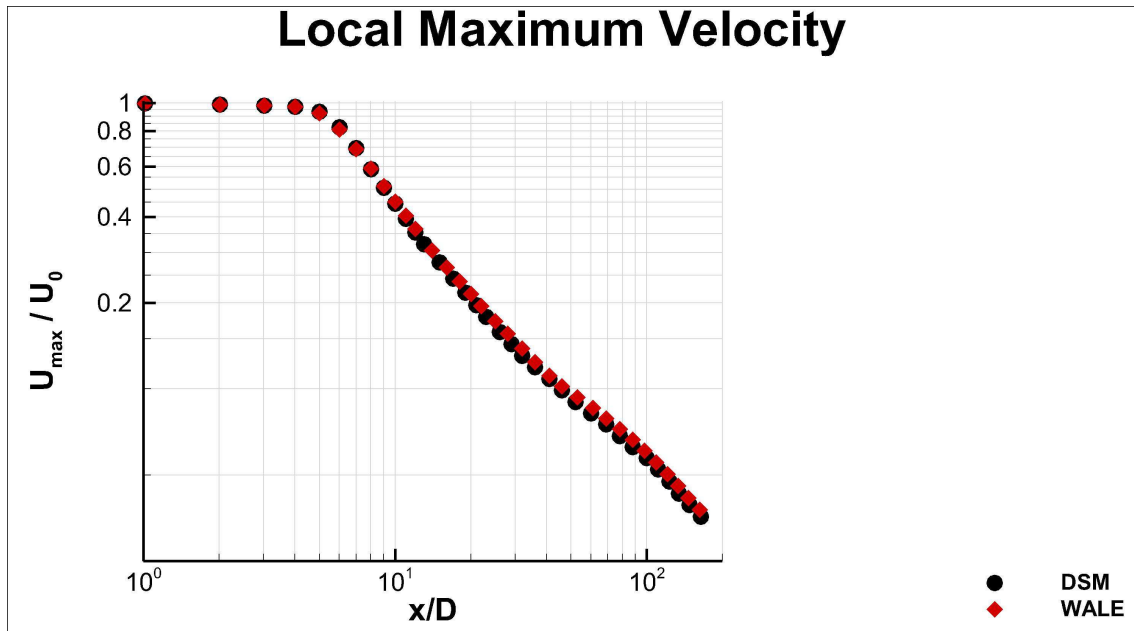
(g)



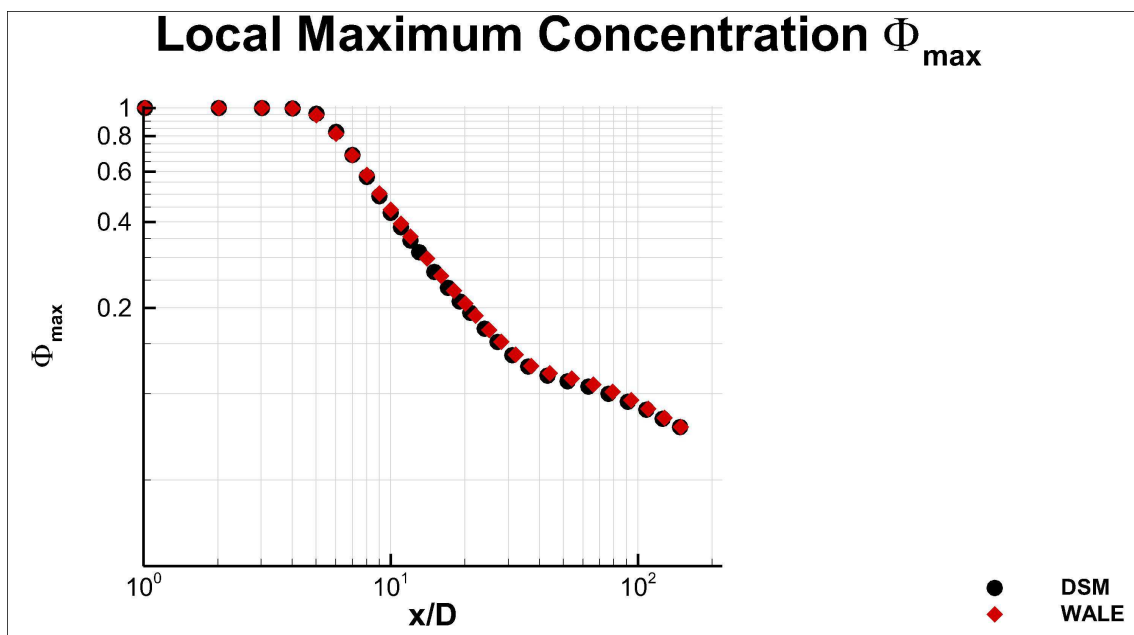
(h)

Figure C.9: Streamwise plots of study “Inlet Ground Boundary Layer (O-Type)”.

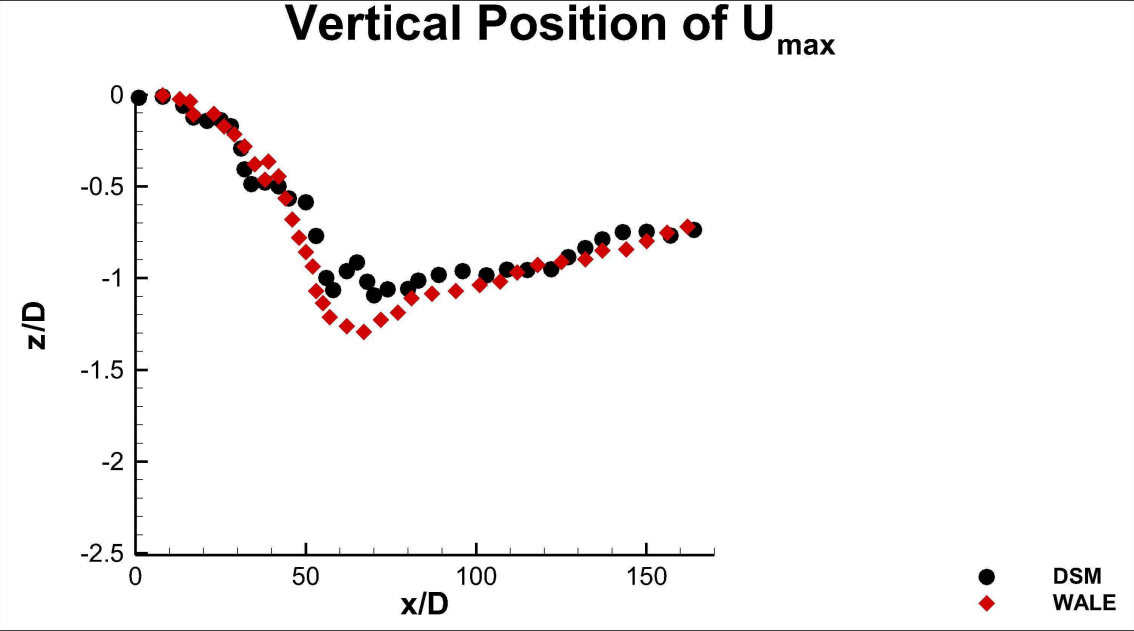
Study Nr.10: SGS Modelling (O-Type)



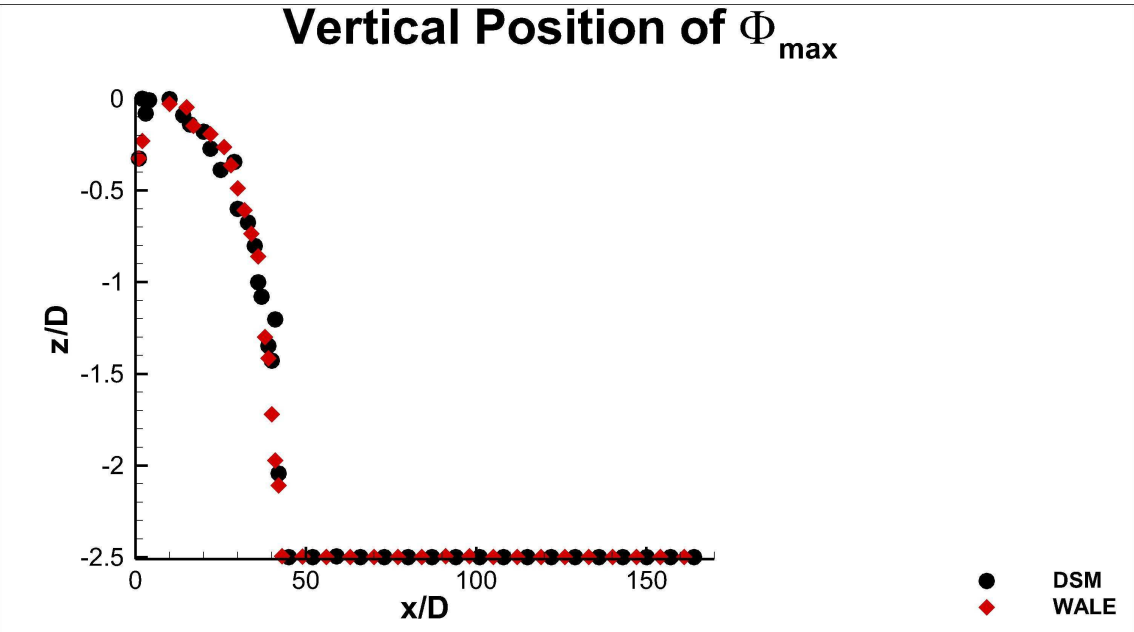
(a)



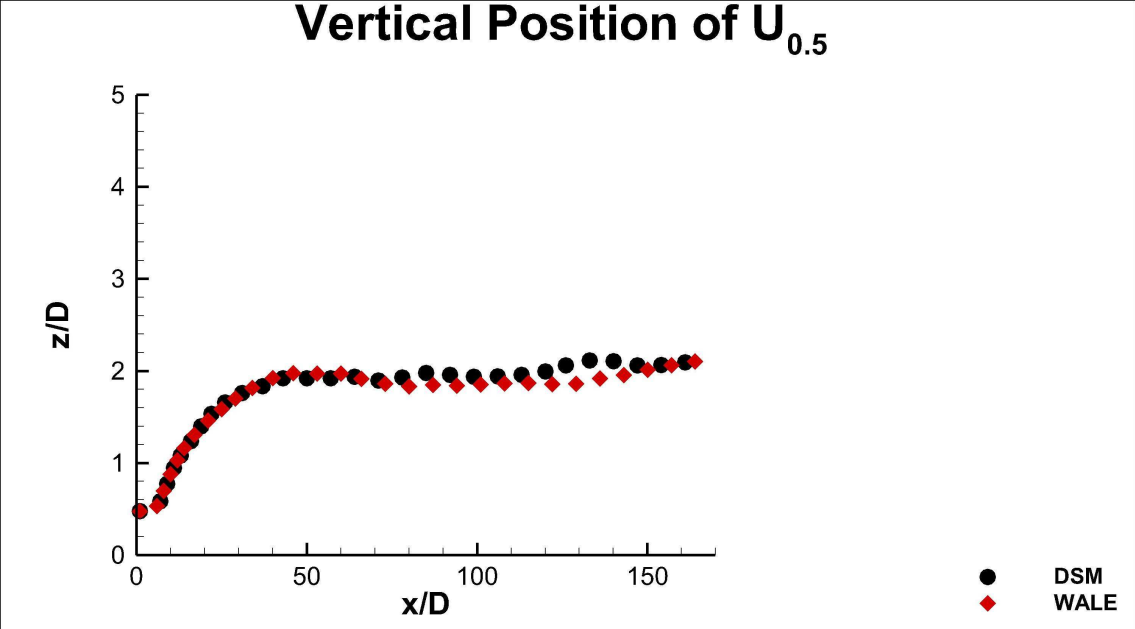
(b)



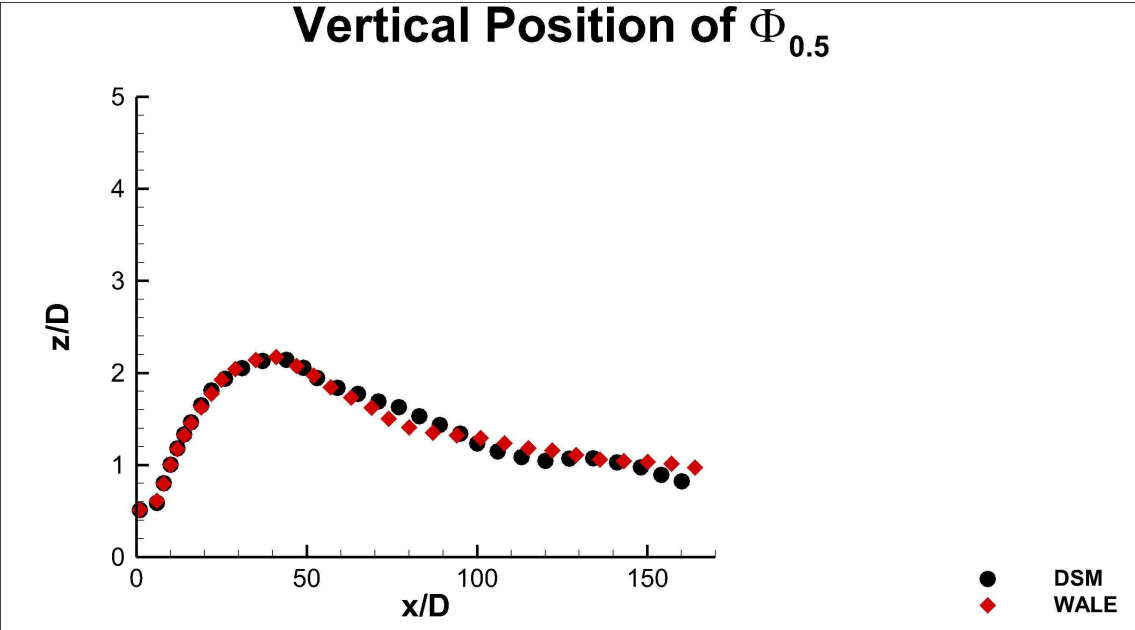
(c)



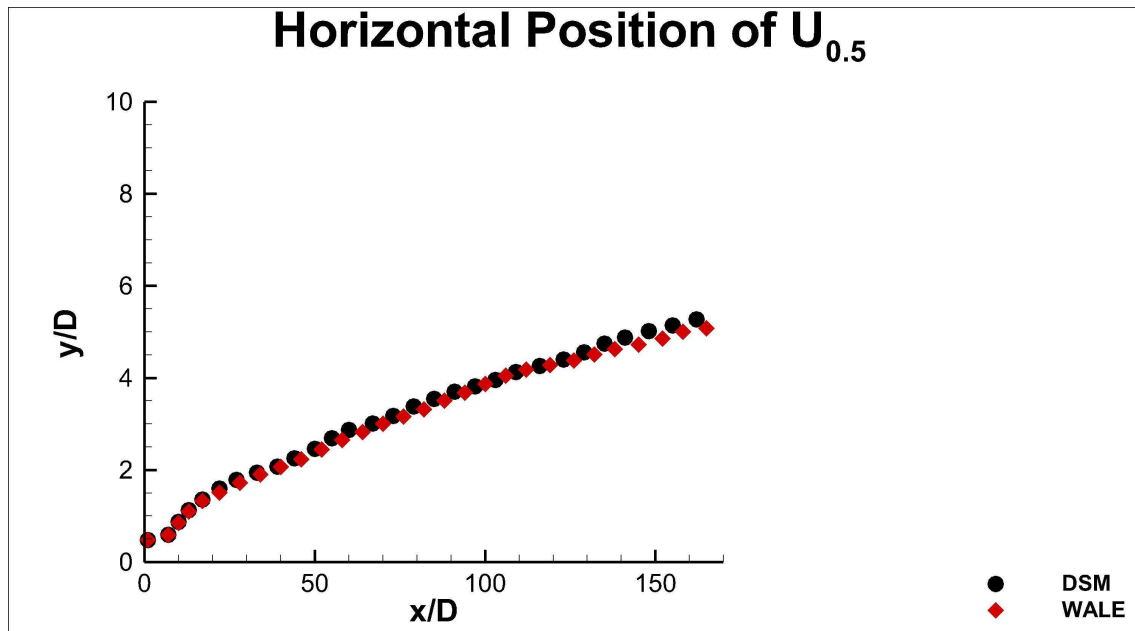
(d)



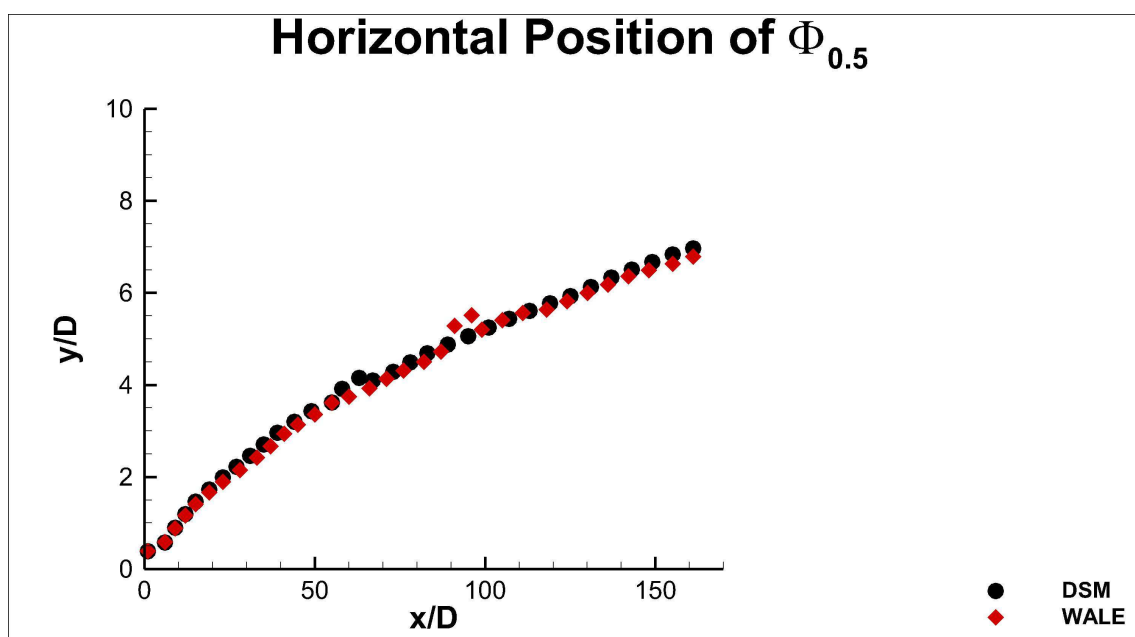
(e)



(f)



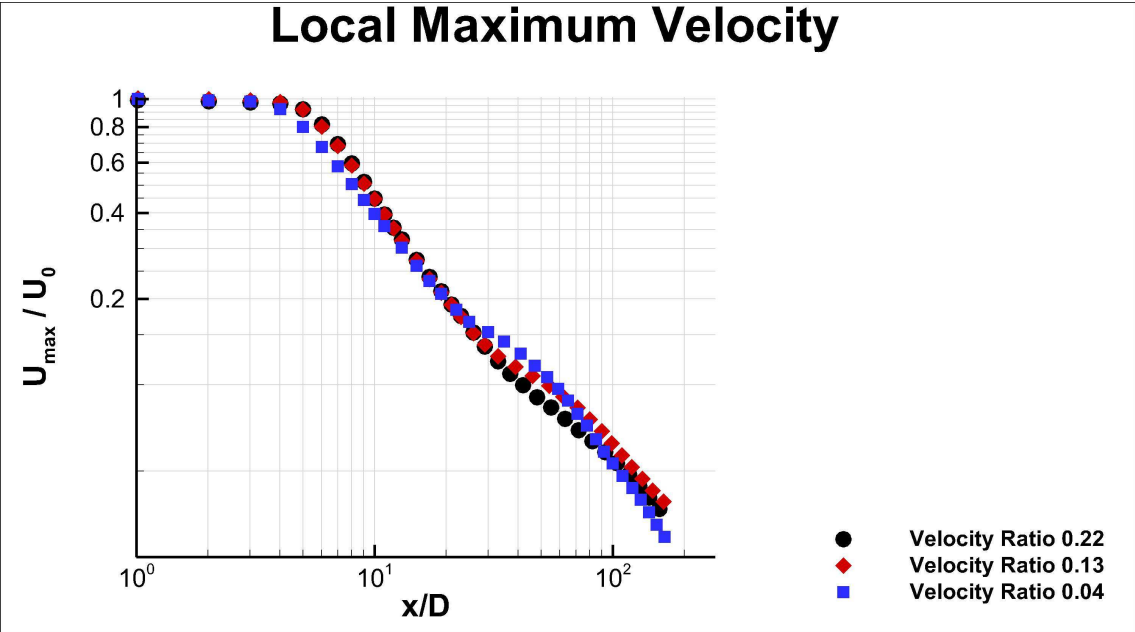
(g)



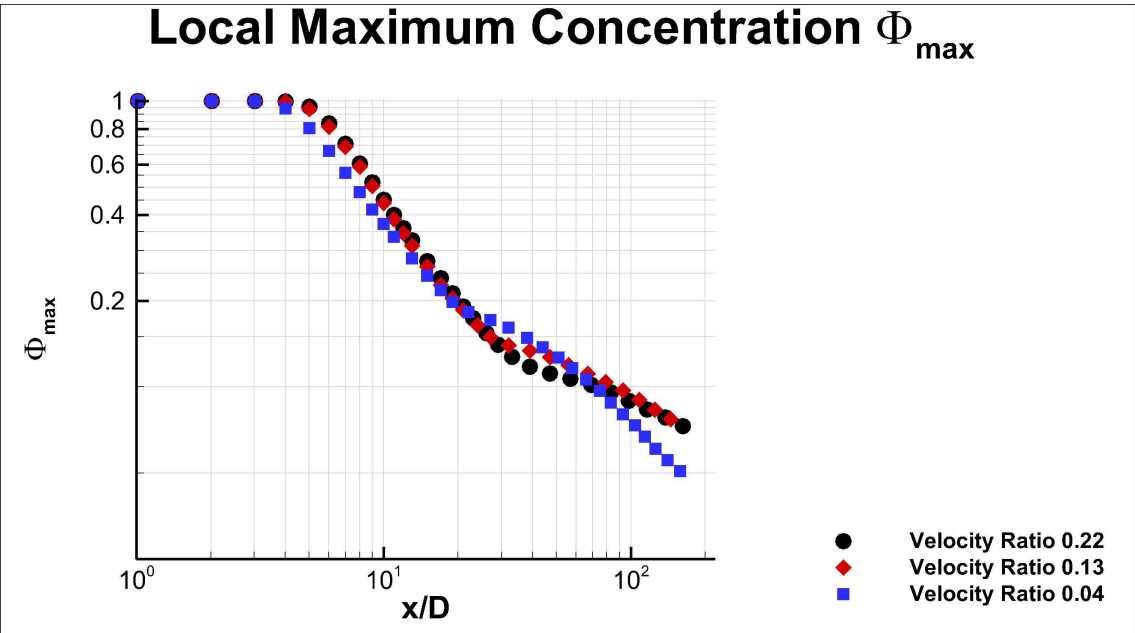
(h)

Figure C.10: Streamwise plots of study “SGS Modelling (O-Type)”.

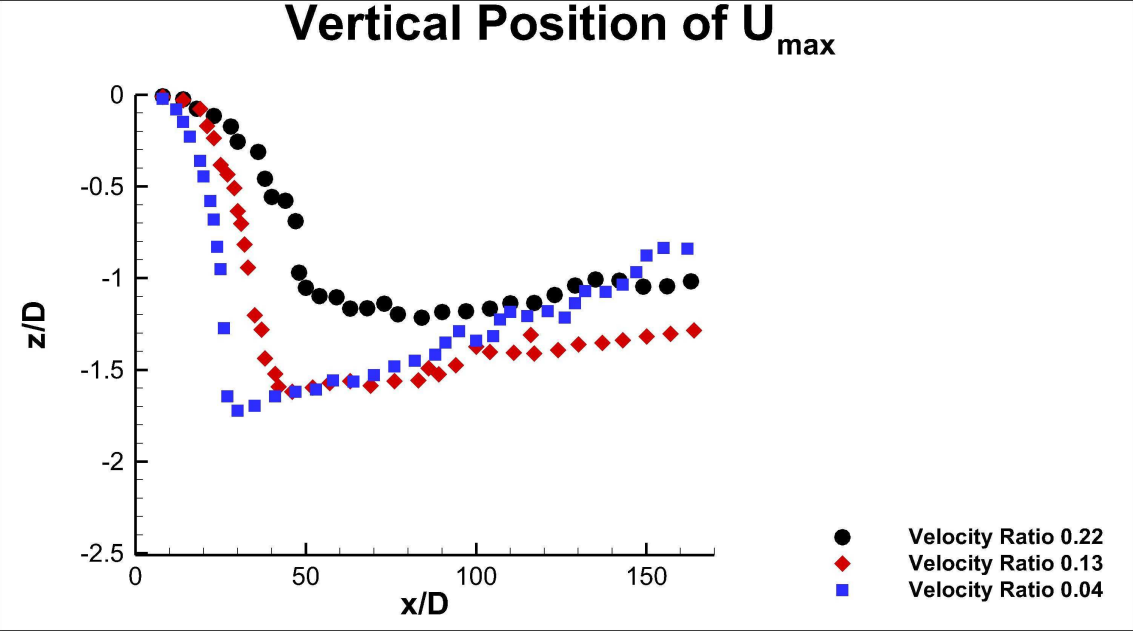
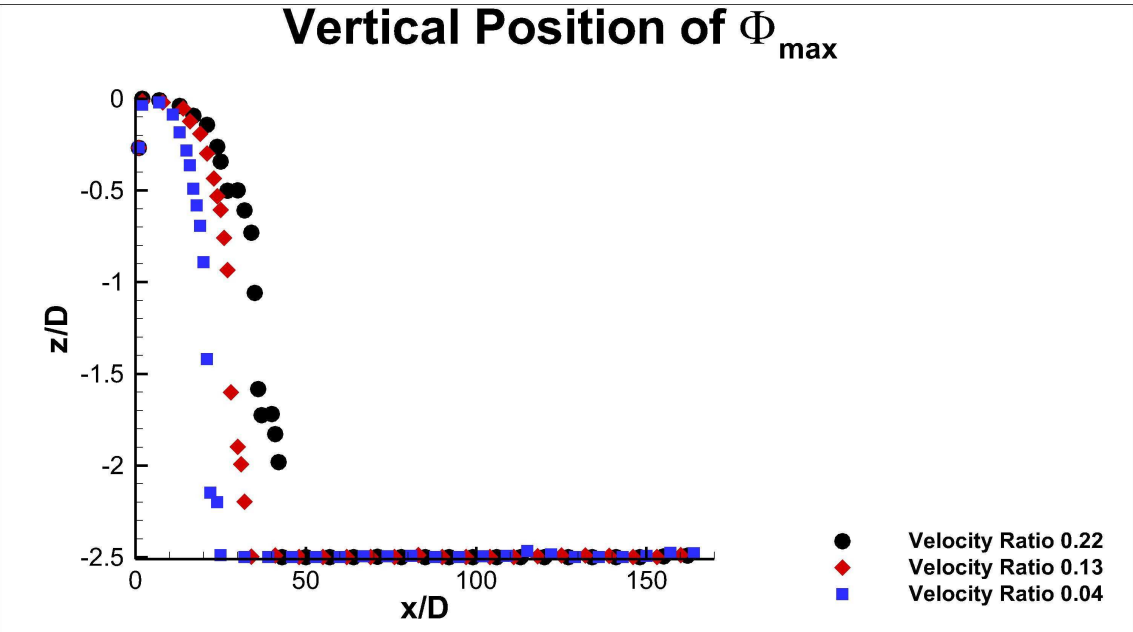
Study Nr.11: Co-Flow Effect

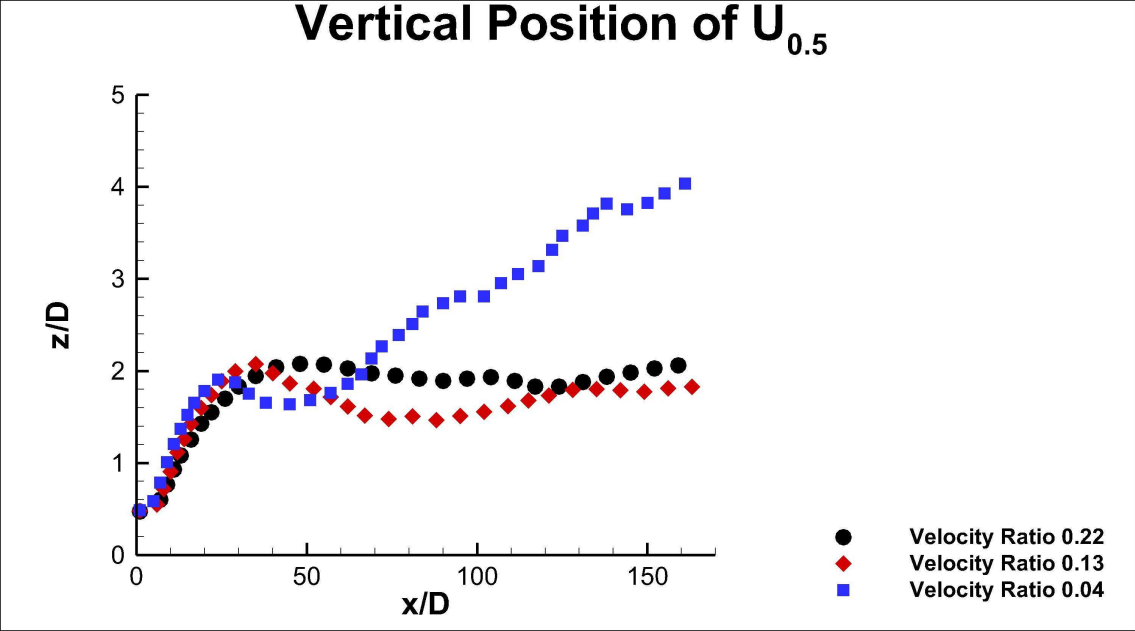


(a)

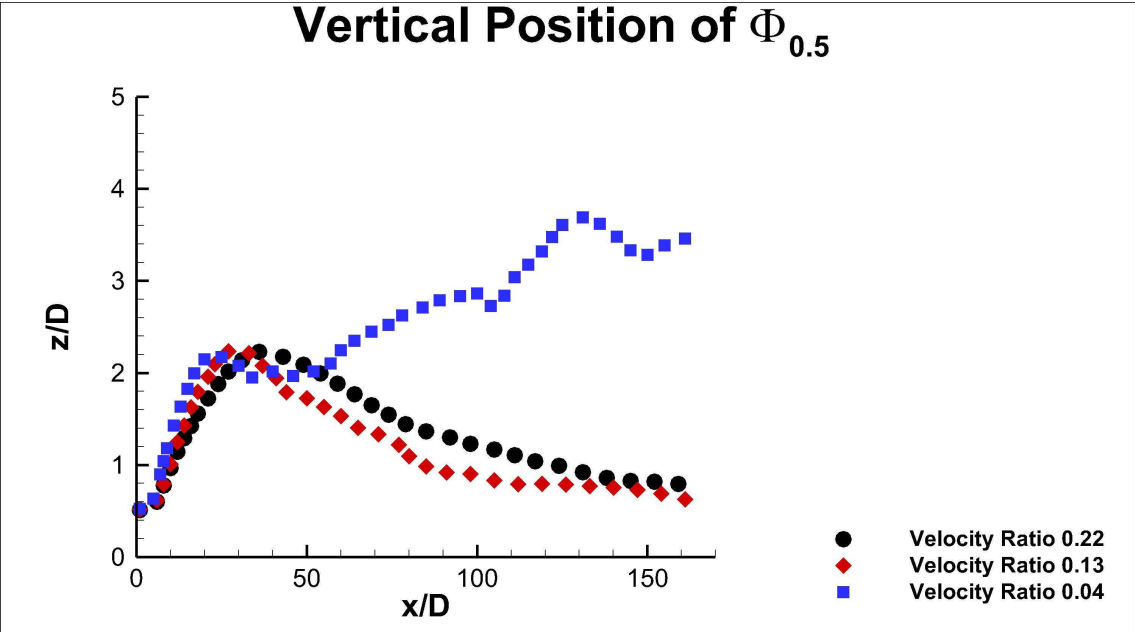


(b)

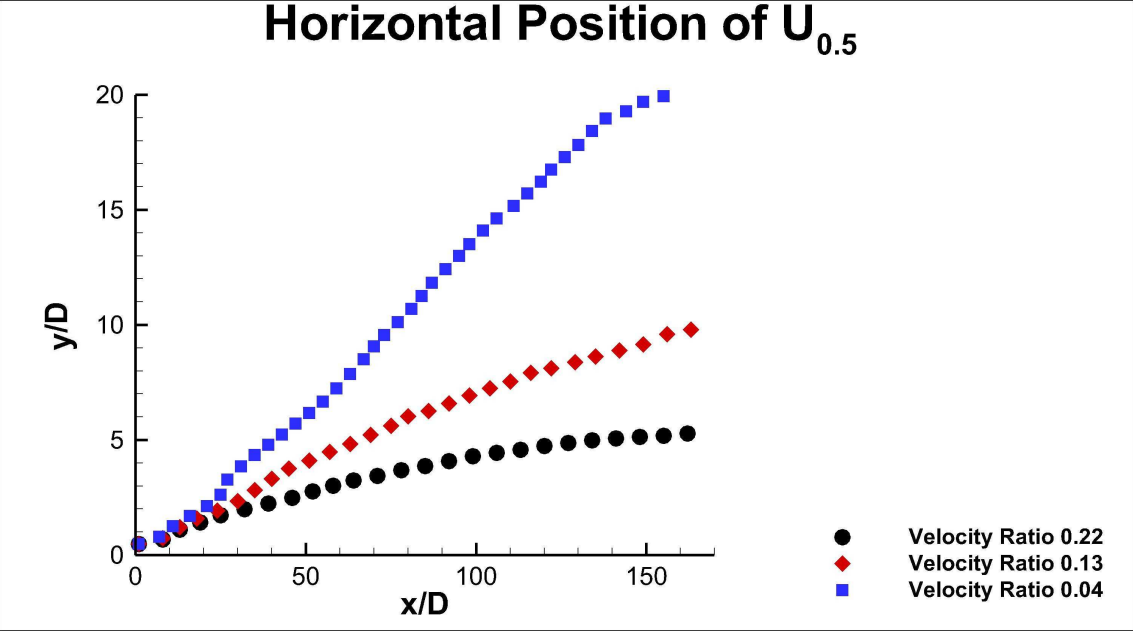




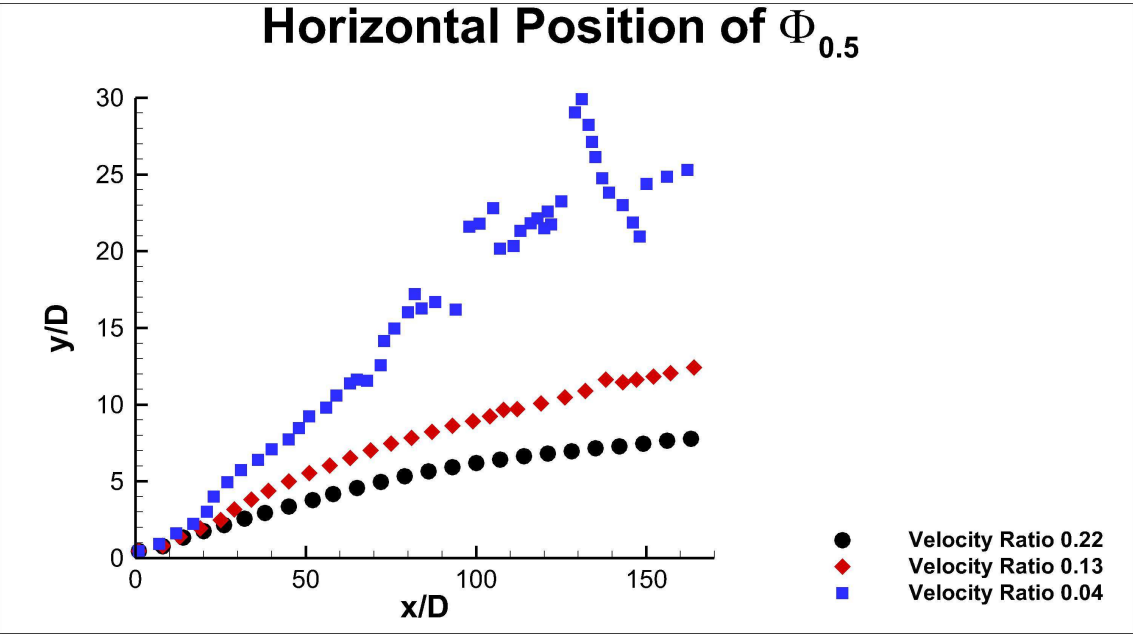
(e)



(f)



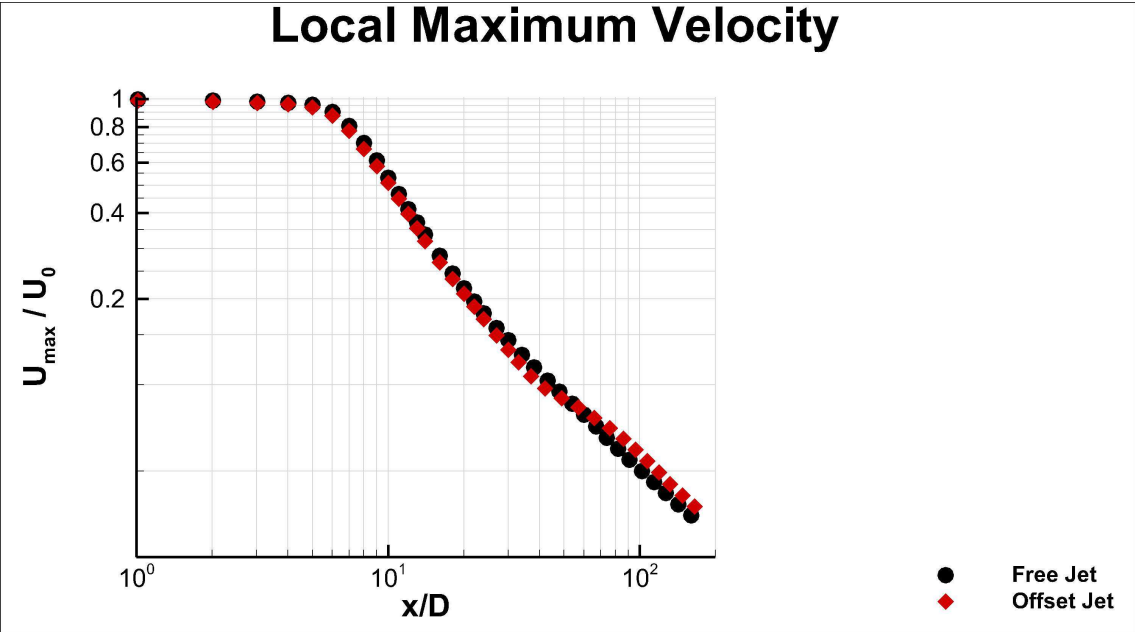
(g)



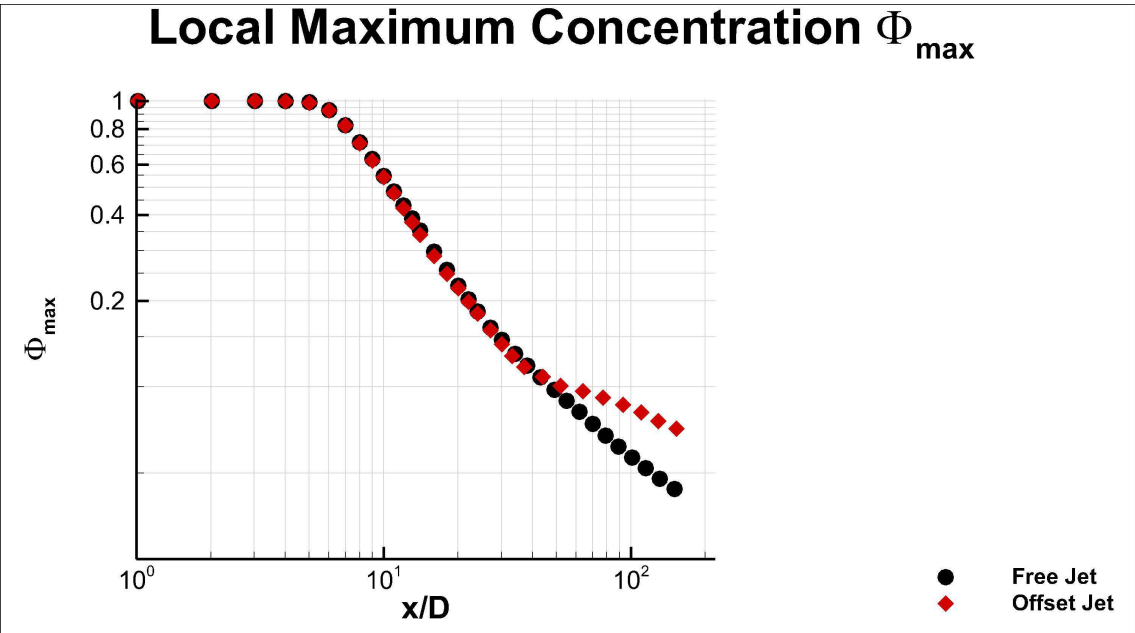
(h)

Figure C.11: Streamwise plots of study “Co-Flow Effect”.

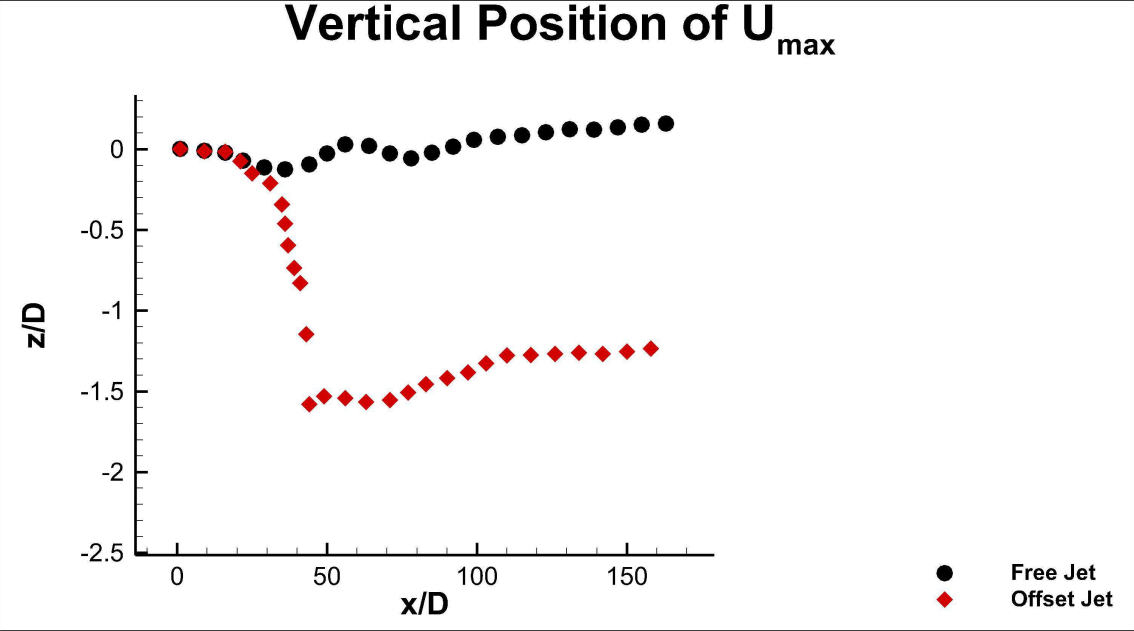
Study Nr.12: Wall Effect



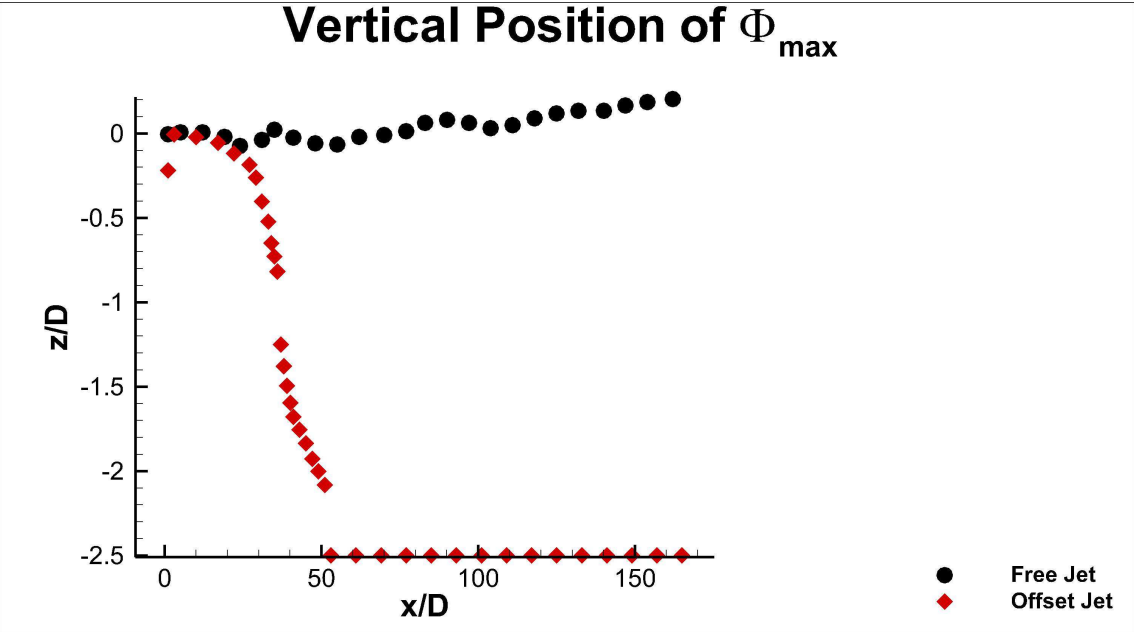
(a)



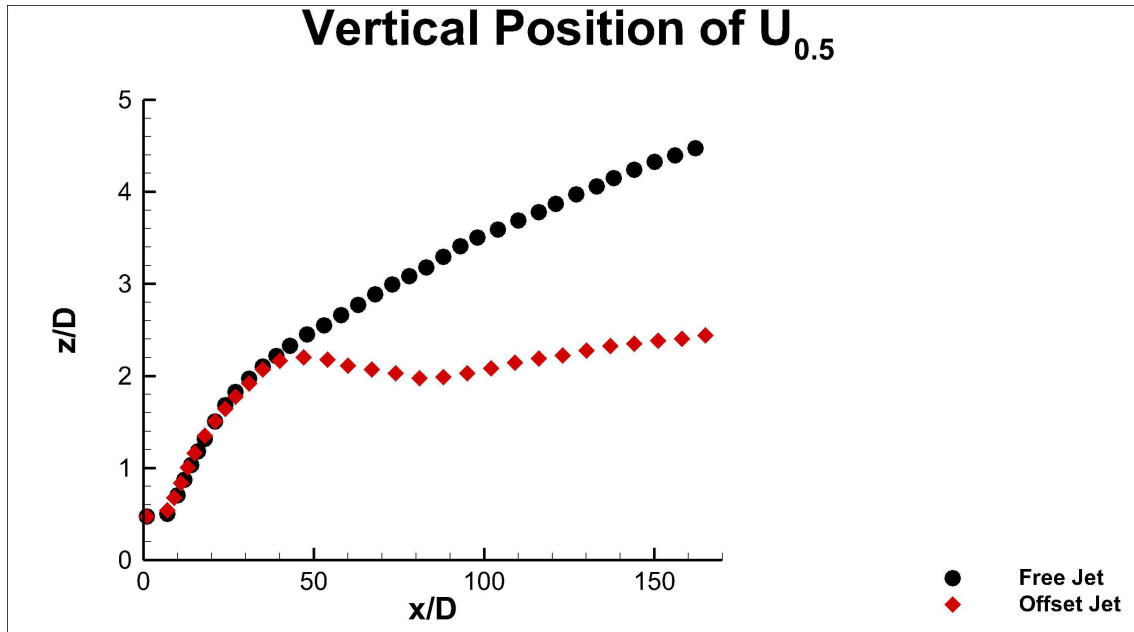
(b)



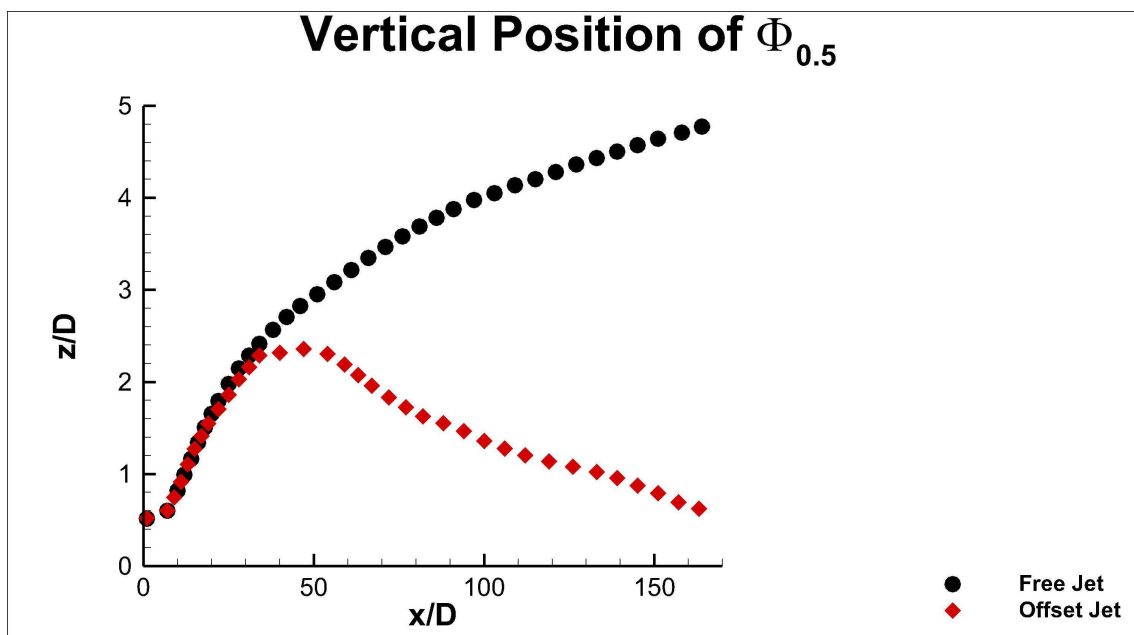
(c)



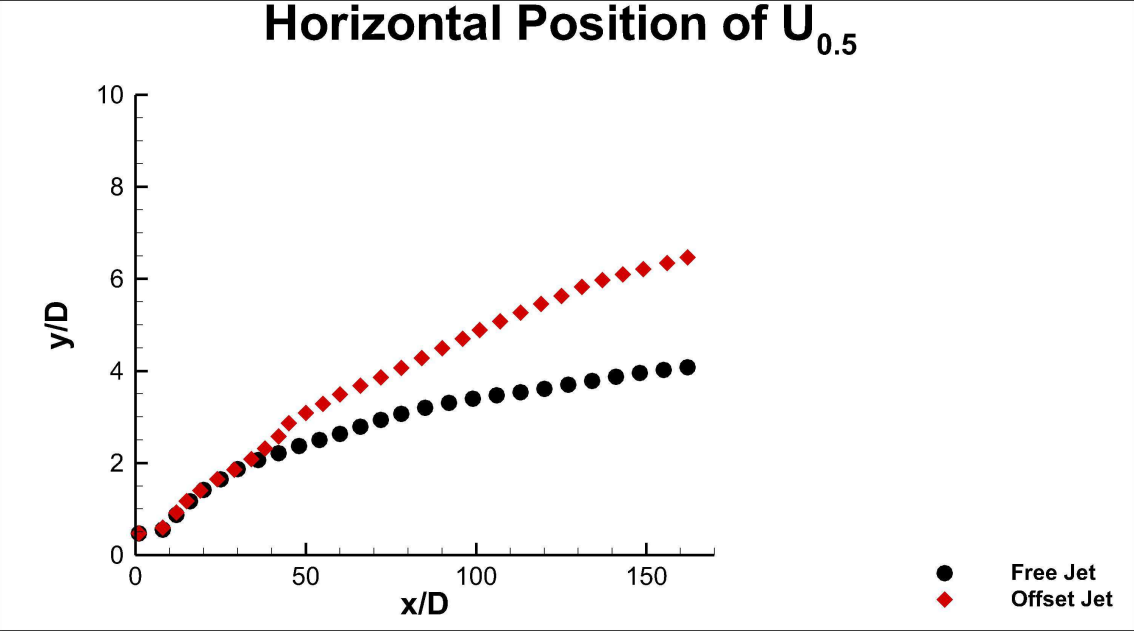
(d)



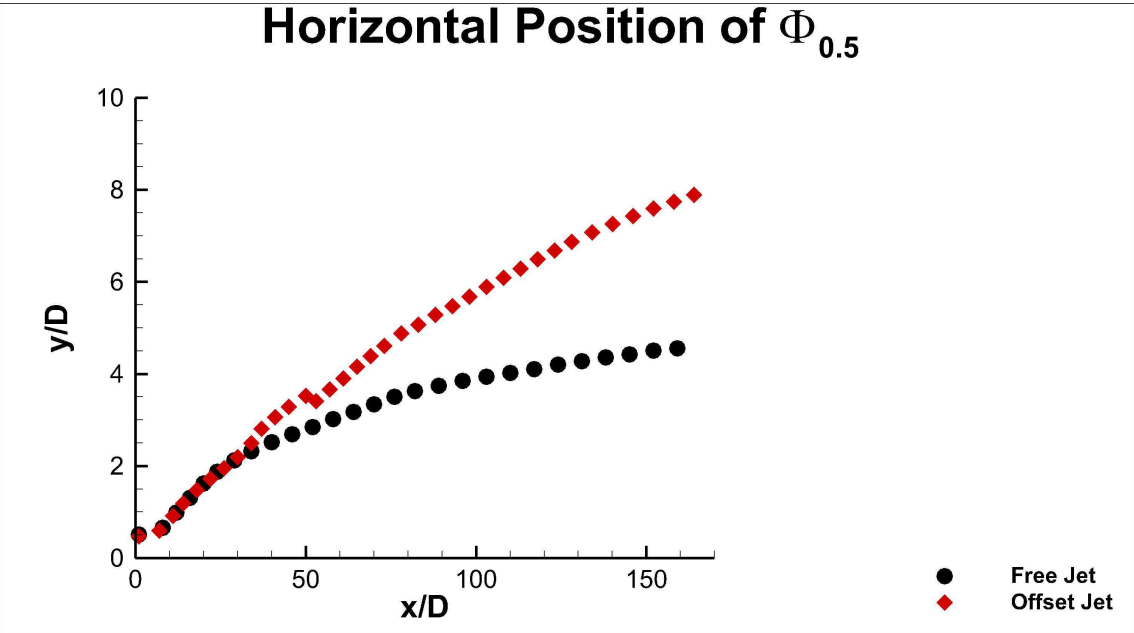
(e)



(f)



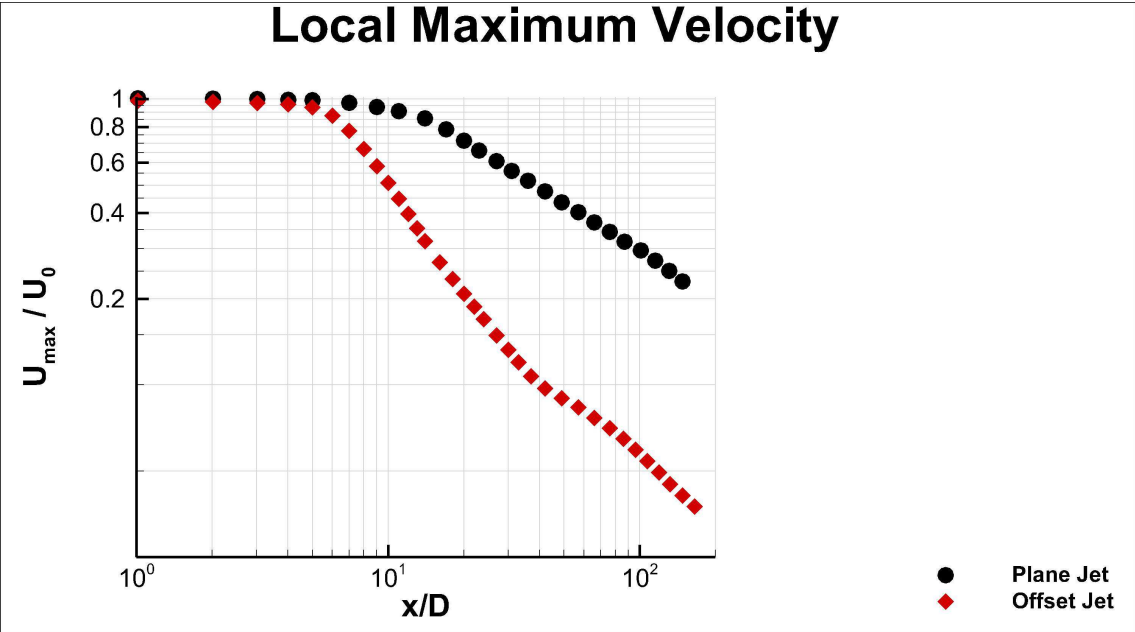
(g)



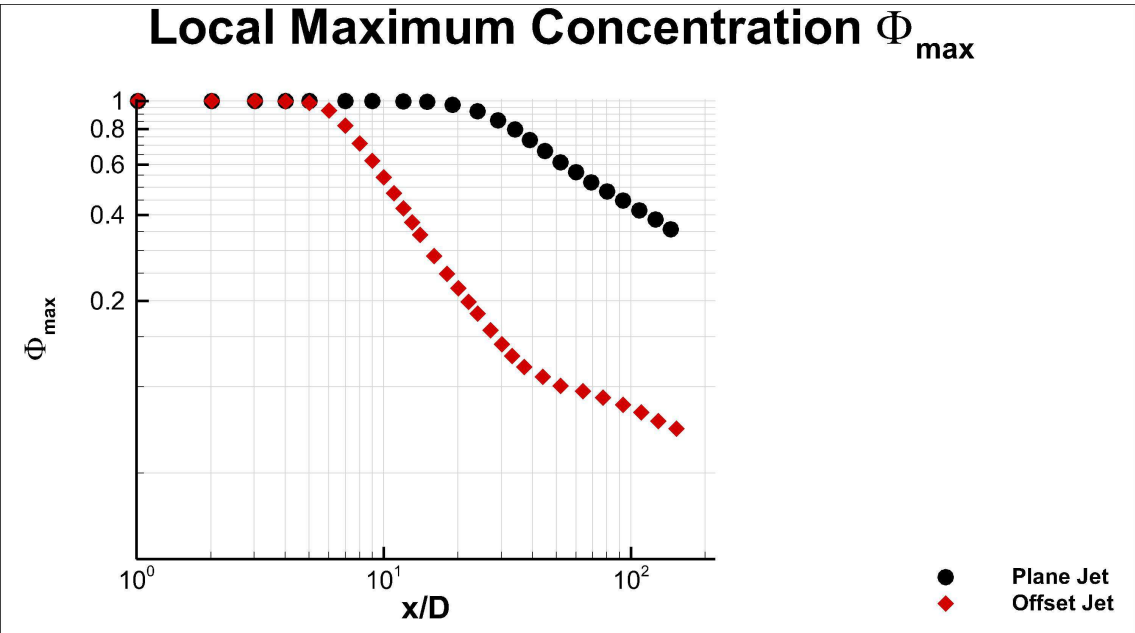
(h)

Figure C.12: Streamwise plots of study “Wall Effect”.

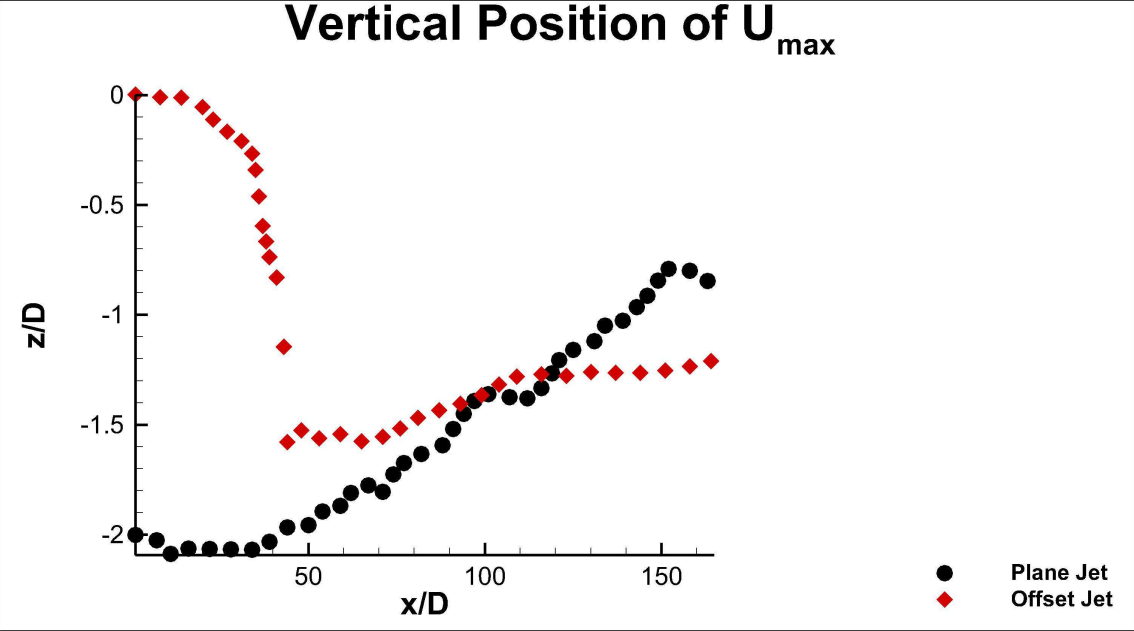
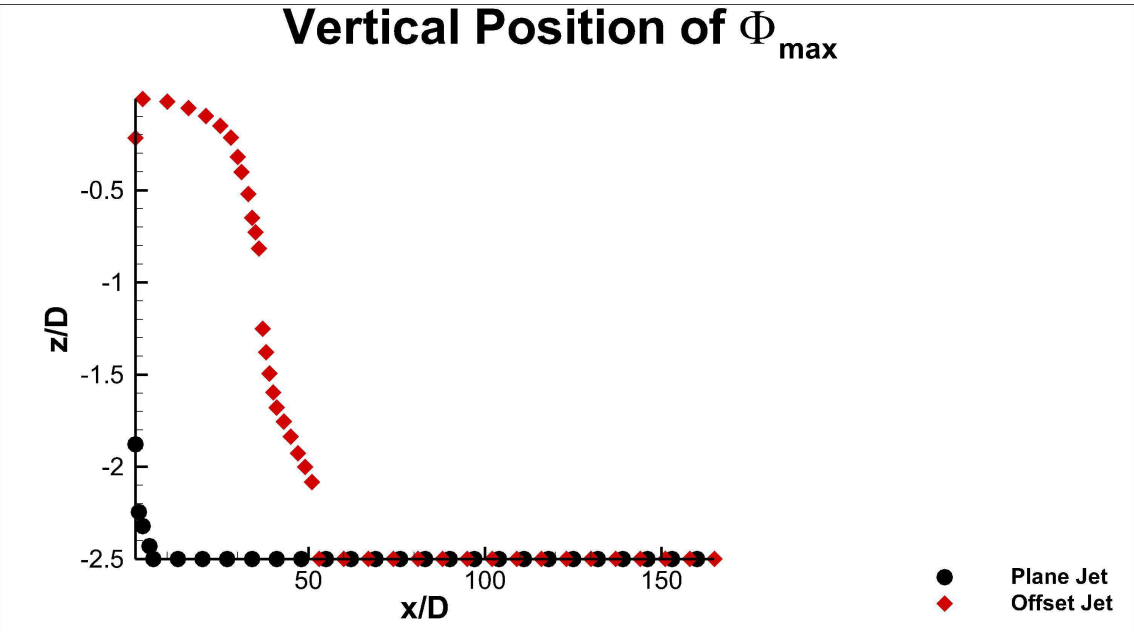
Study Nr.13: Three-Dimensionality Effect



(a)



(b)



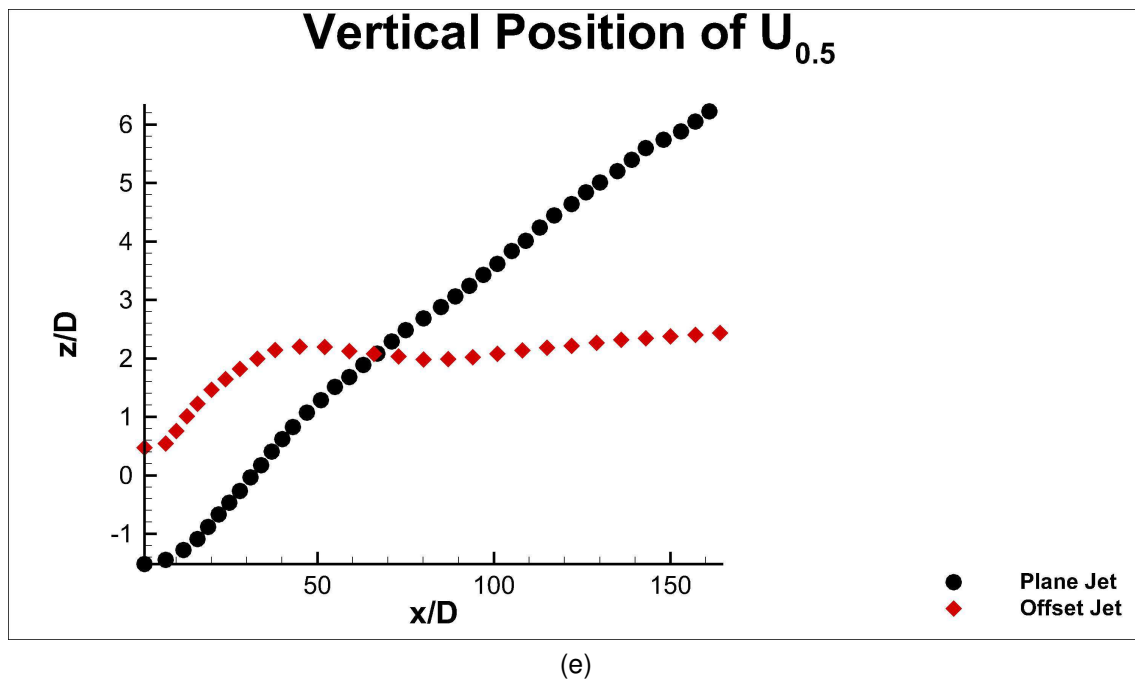


Figure C.13: Streamwise plots of study “*Three-Dimensionality Effect*”.

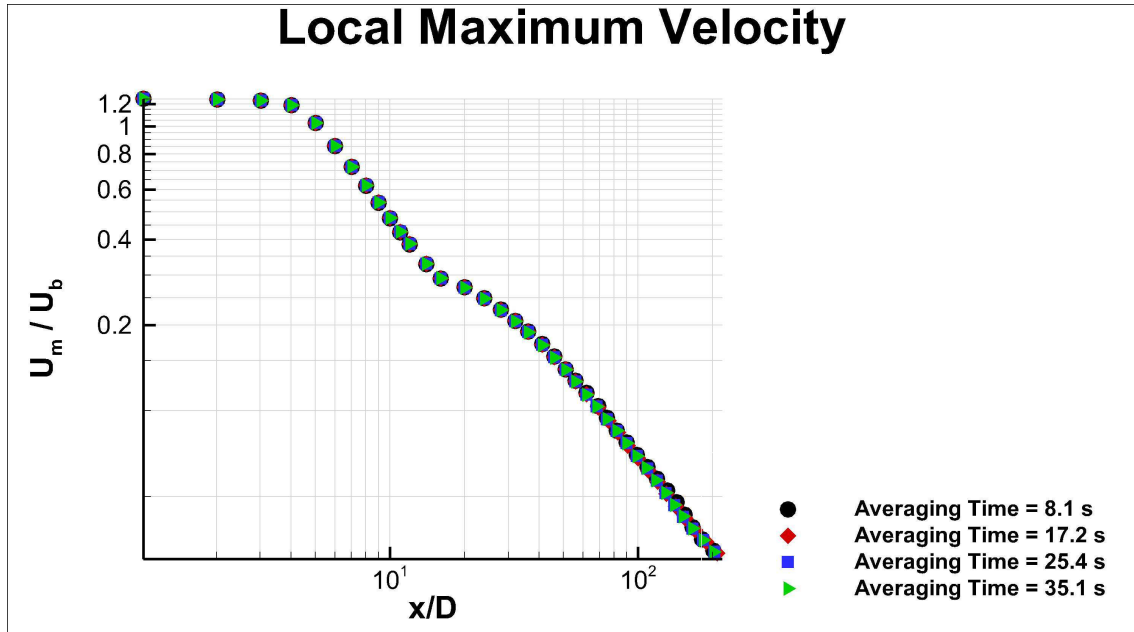
Appendix D

Final Results:

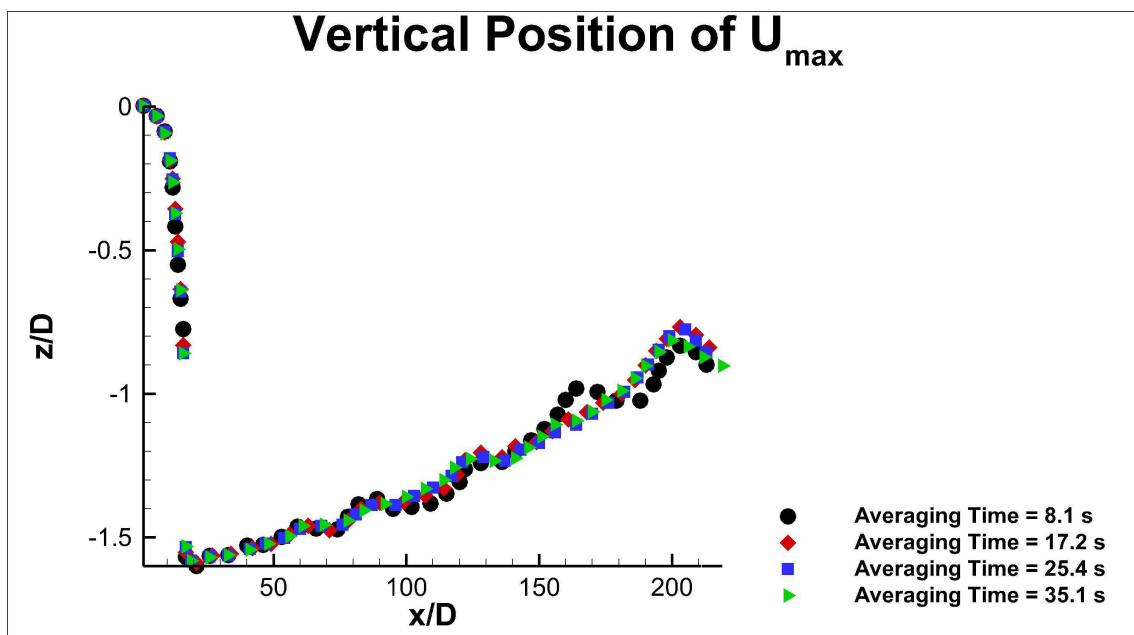
Streamwise Evolution Plots

The present Appendix includes the streamwise evolution mean jet characteristics plots of the final results discussed in chapter 10. This post processing approach is called advanced post-processing method; refer to section 8.1.2 for description of this approach. Description of the investigated setup corresponding to these results is available in section 6.2.

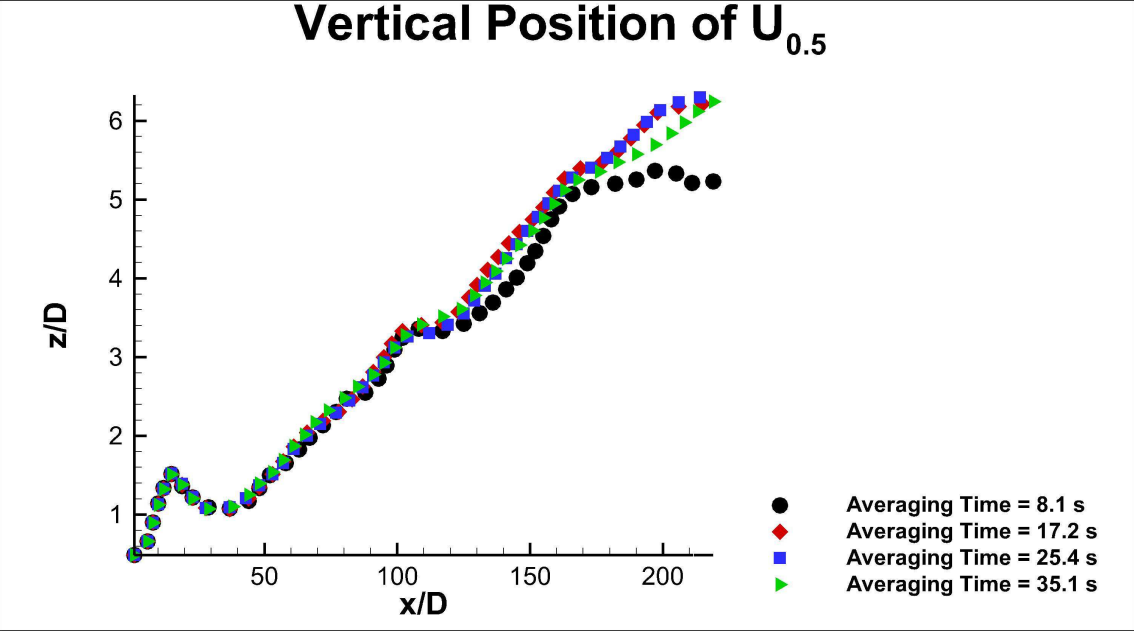
D.1 Temporal Convergence Study



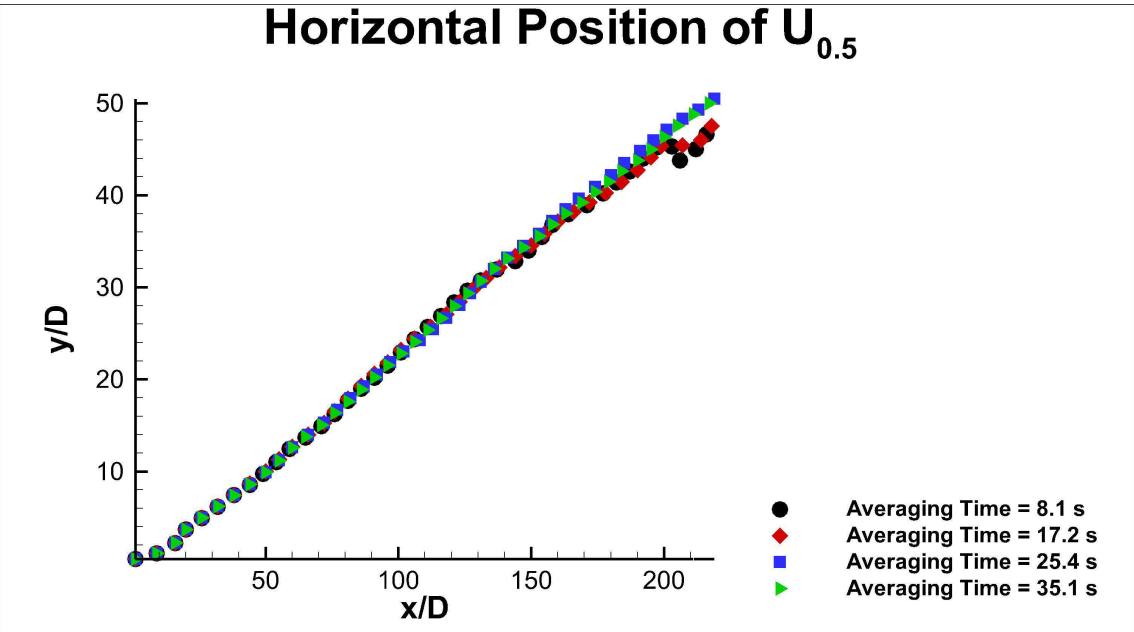
(a)



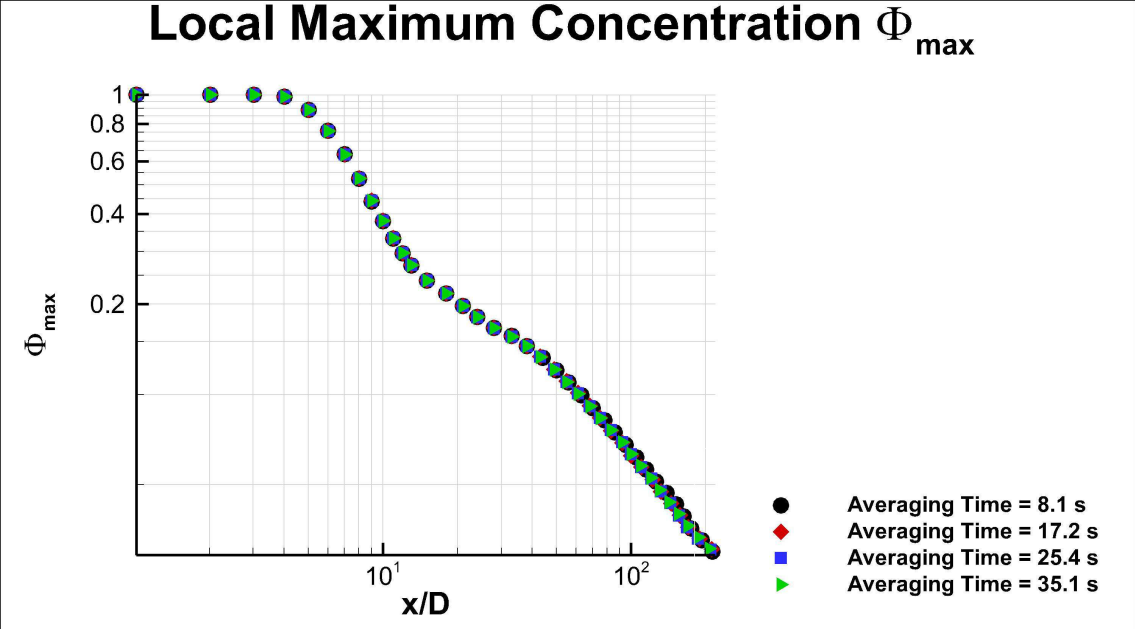
(b)



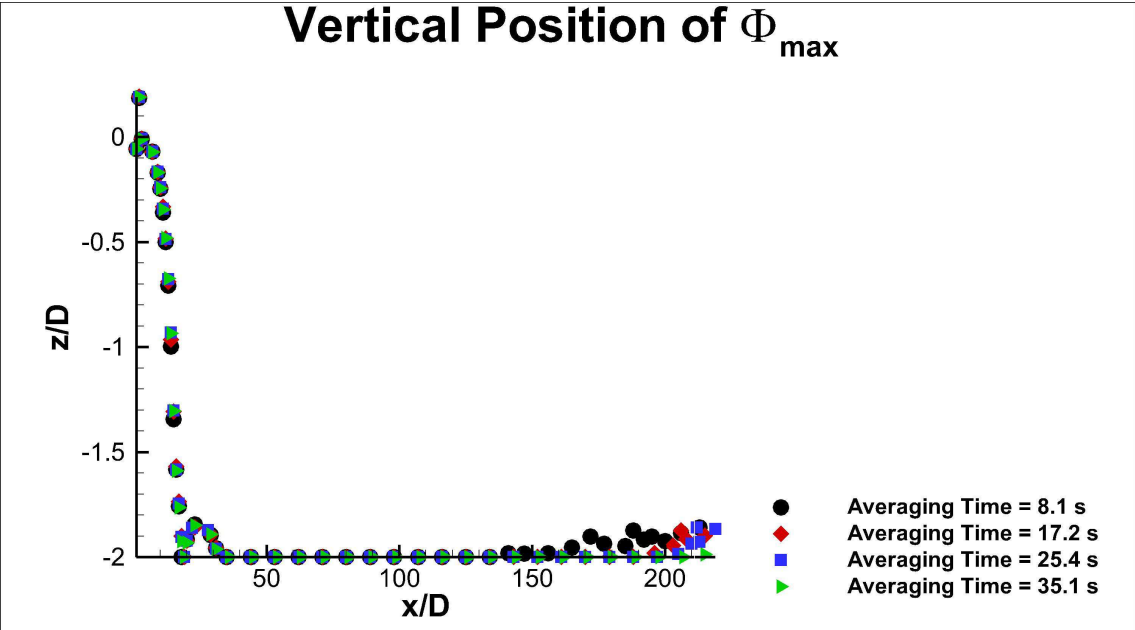
(c)



(d)



(e)



(f)

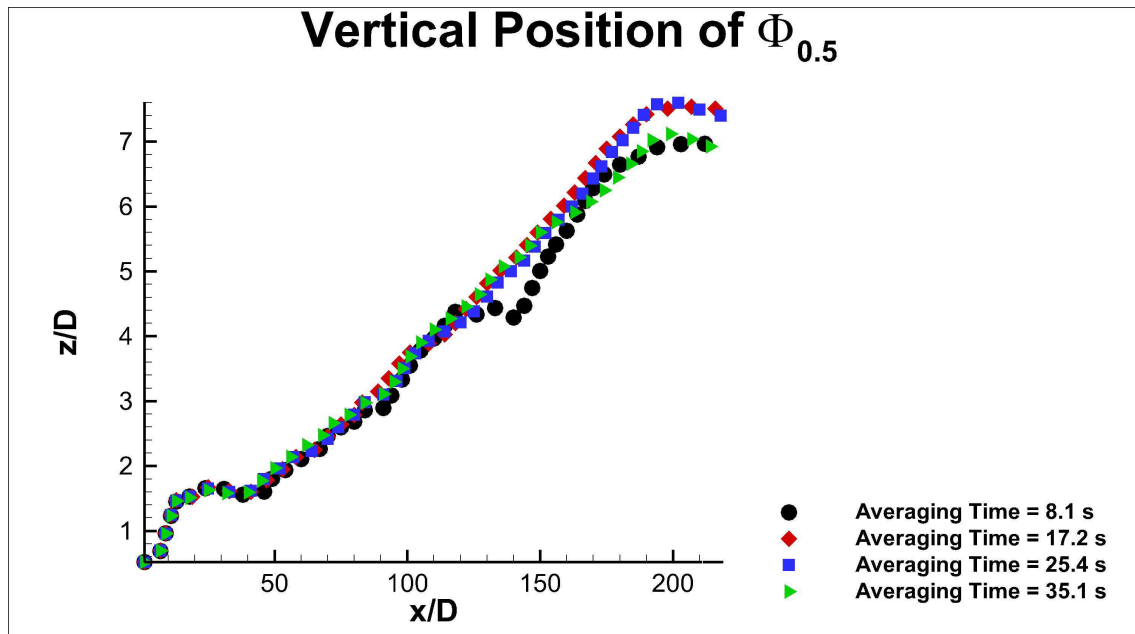
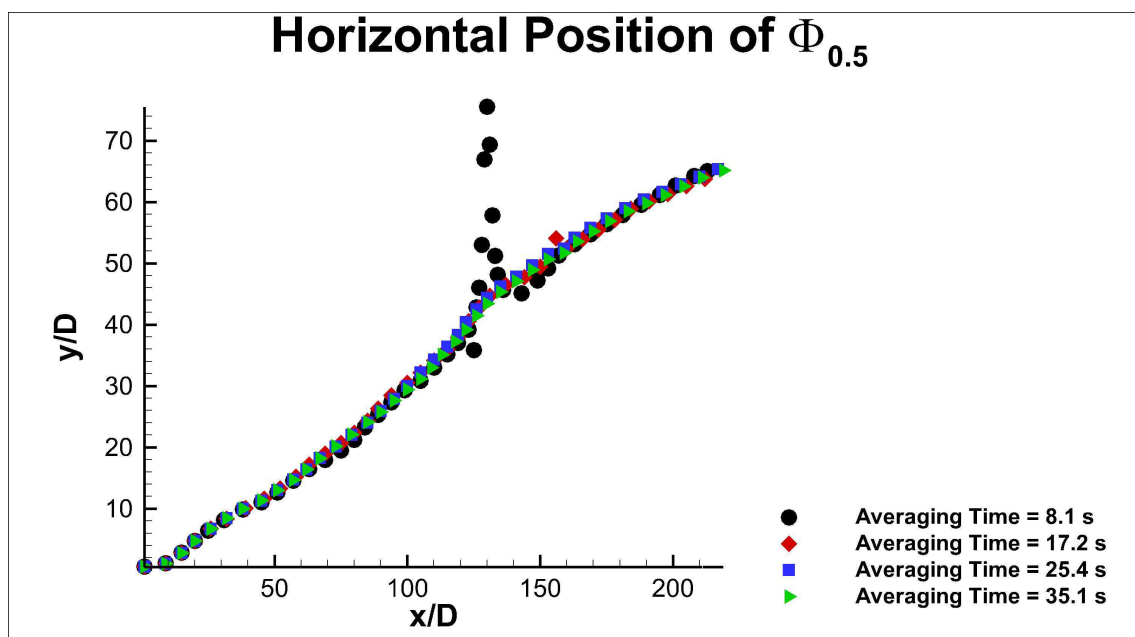
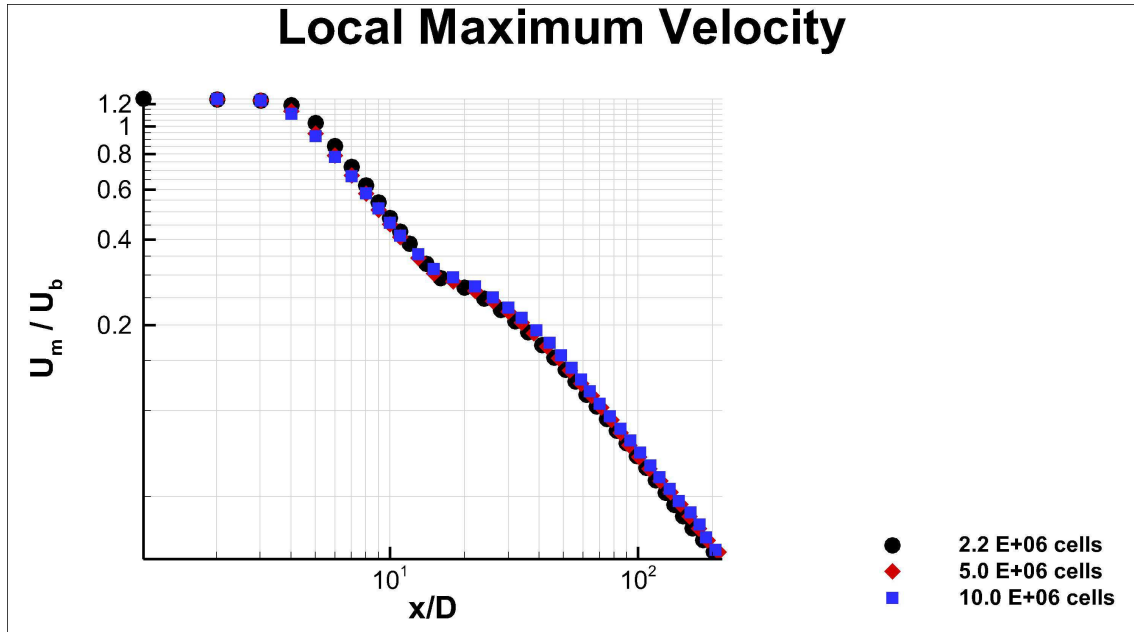
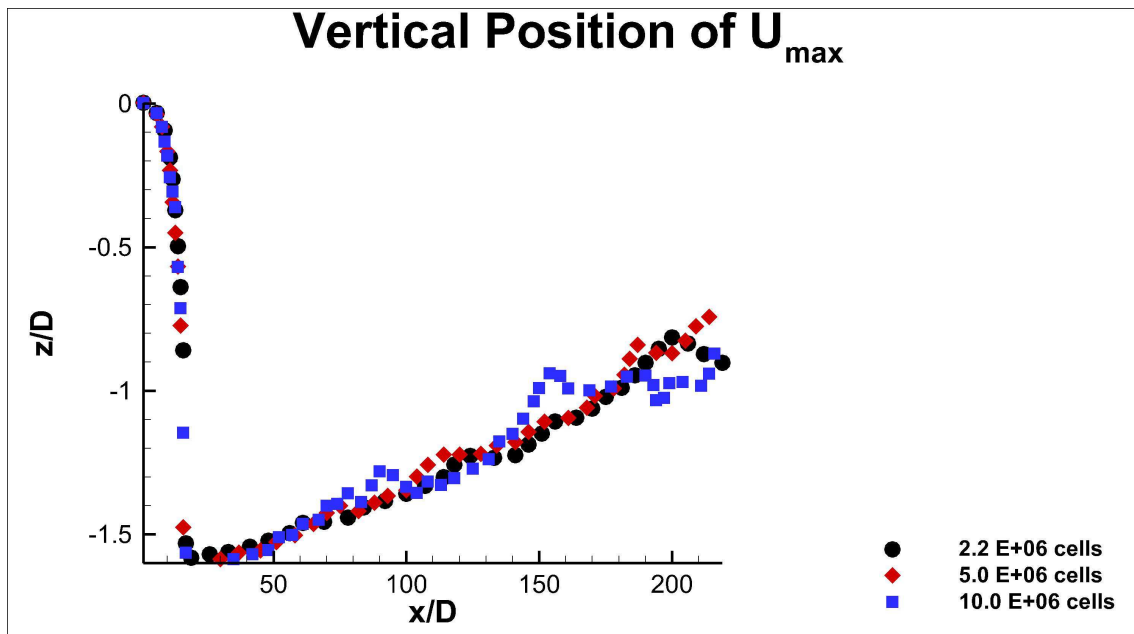


Figure D.1: Streamwise plots of the final results “temporal convergence study”.

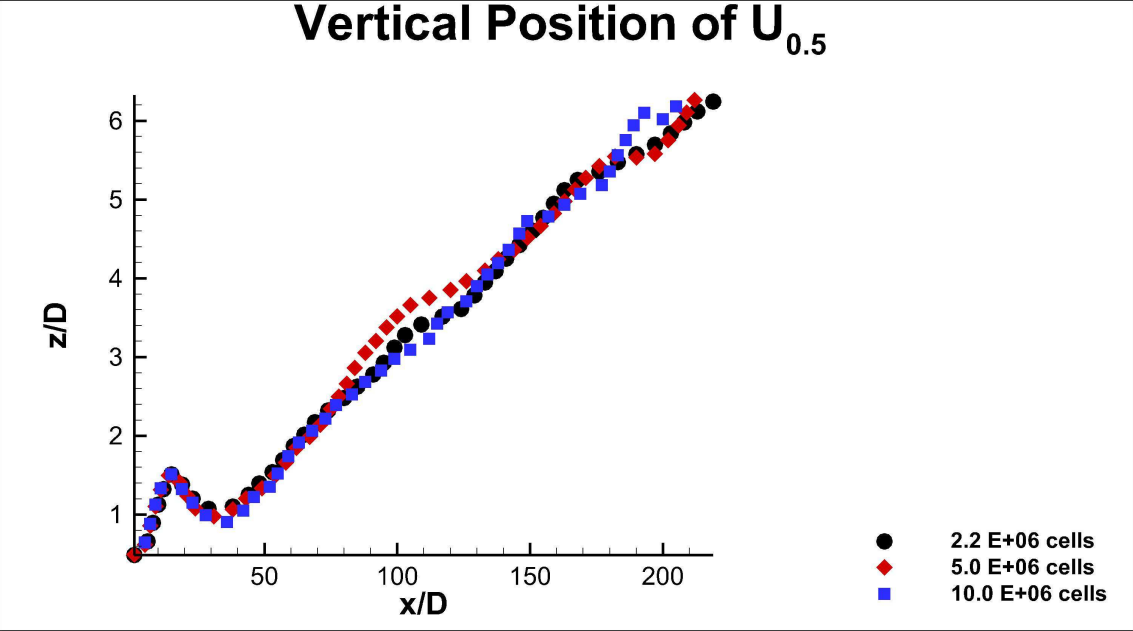
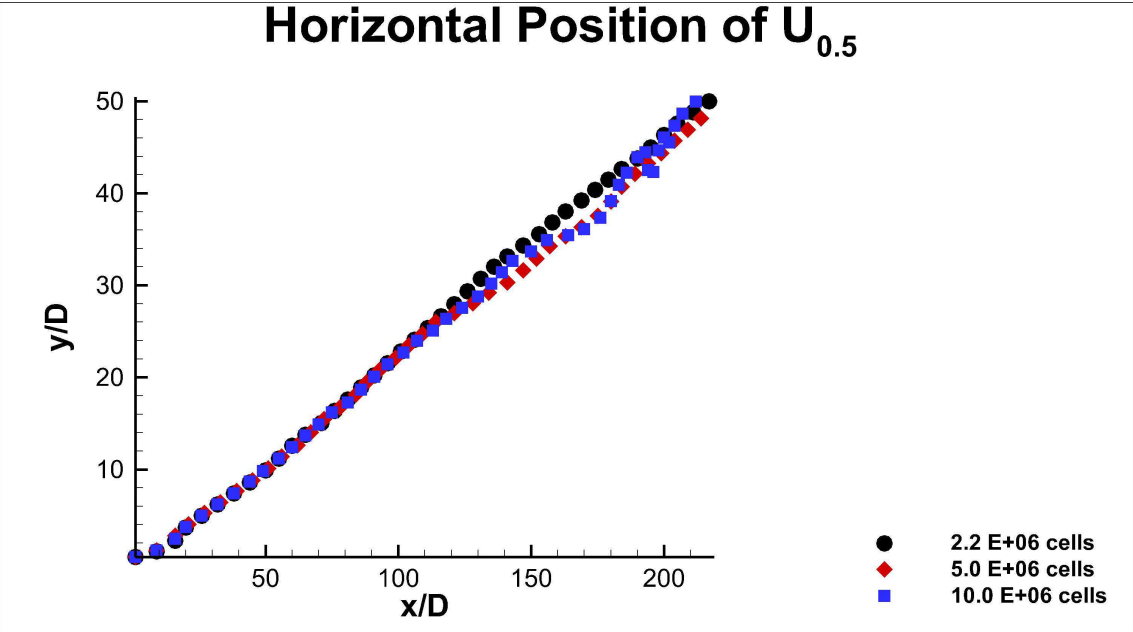
D.2 Grid Convergence Study

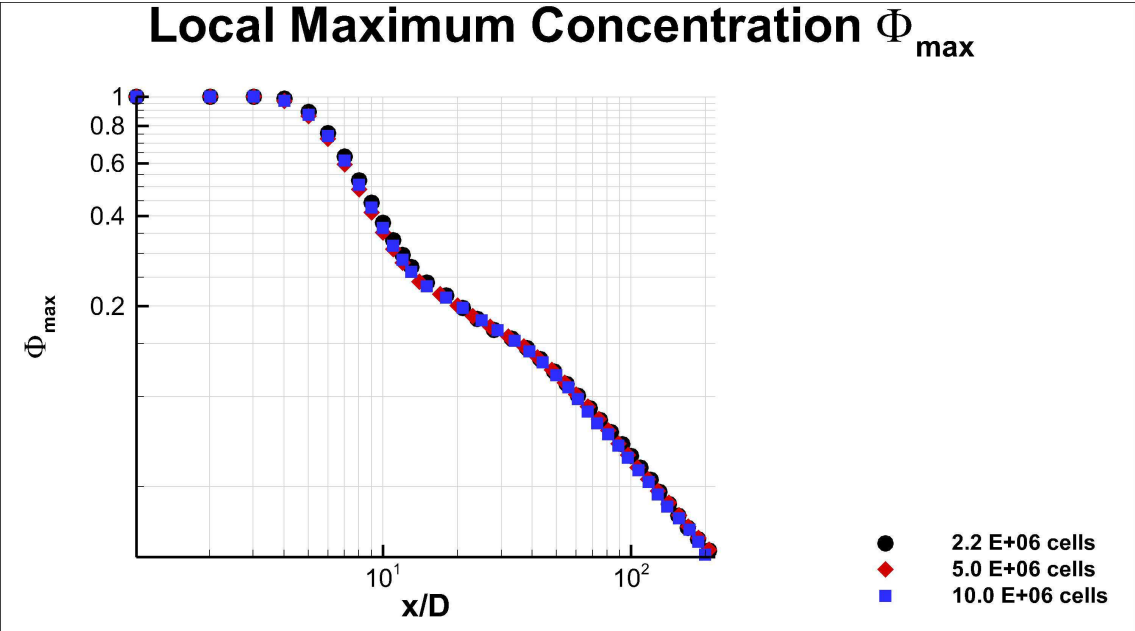


(a)

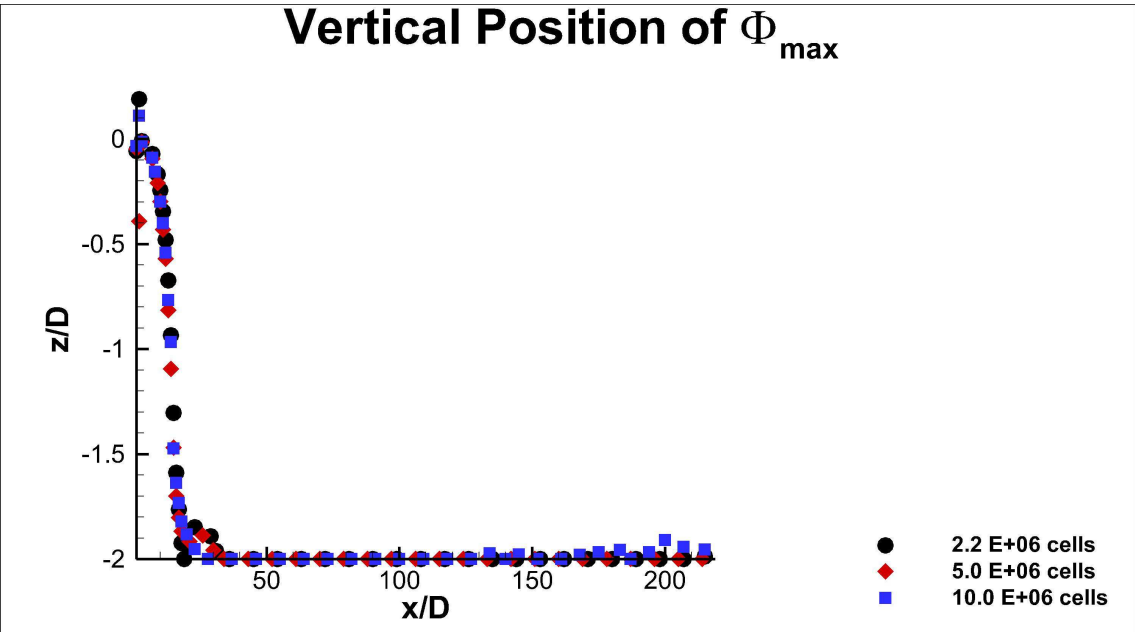


(b)





(e)



(f)

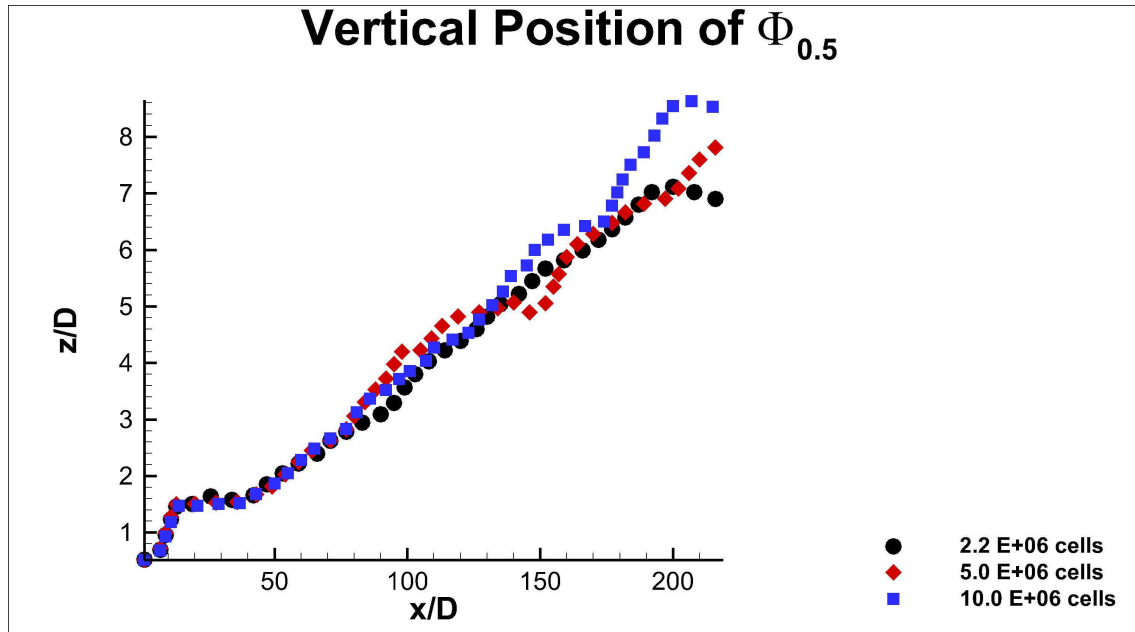
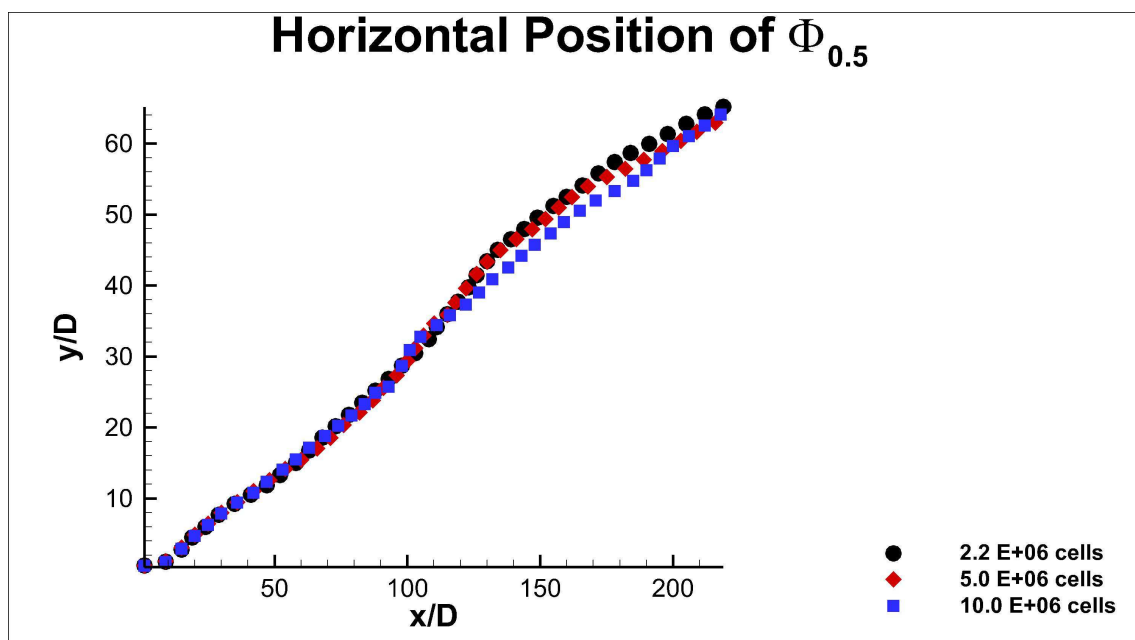
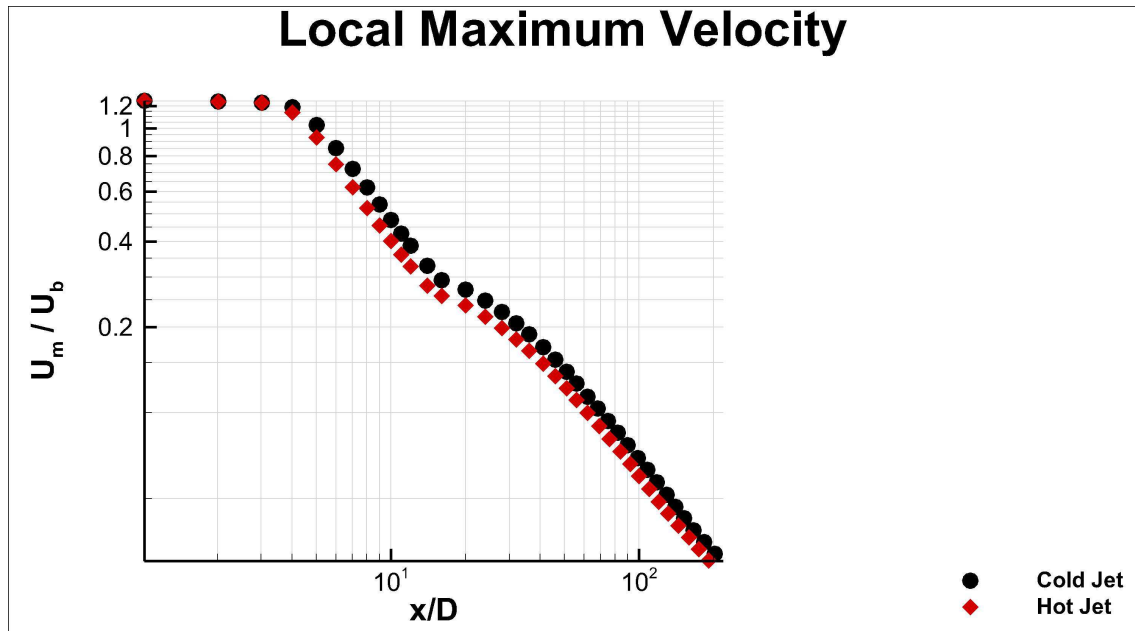
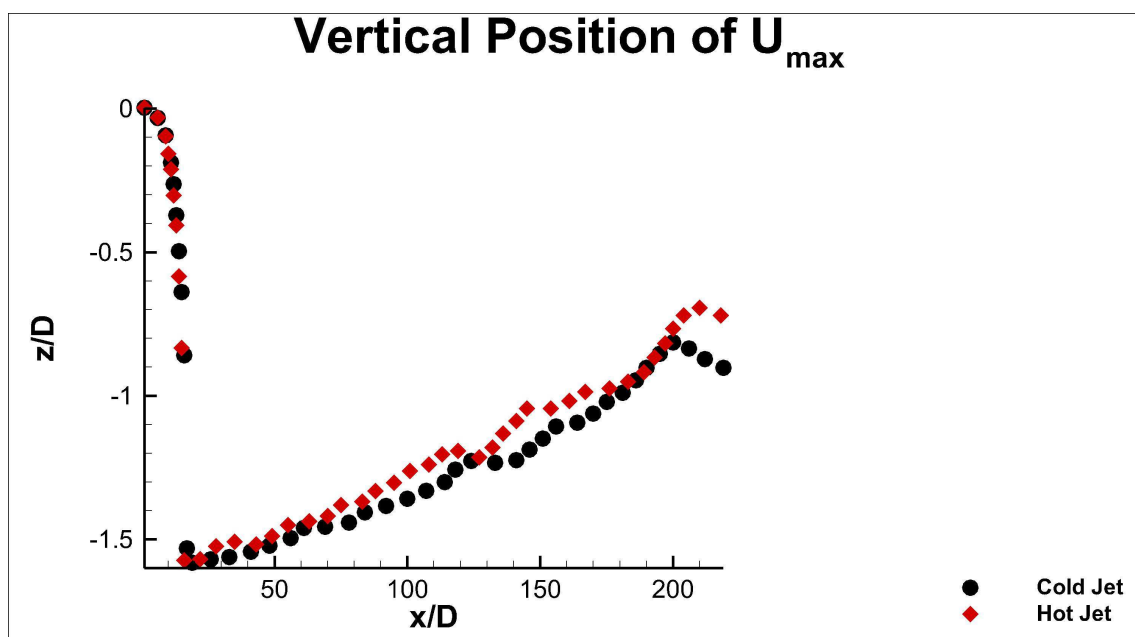


Figure D.2: Streamwise plots of the final results “*grid convergence*” study.

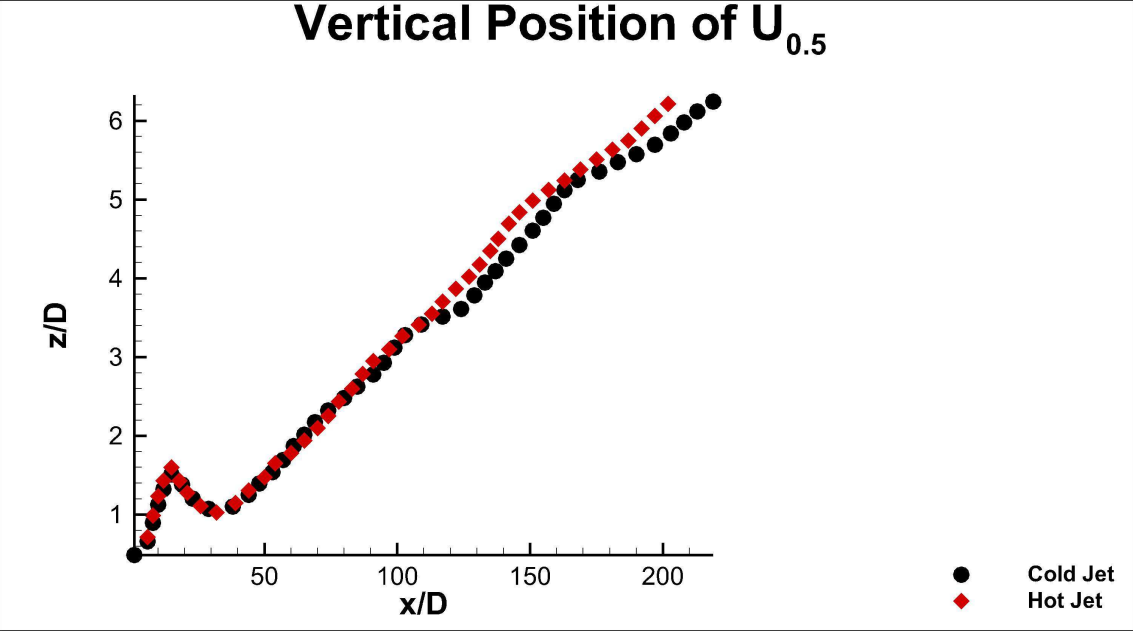
D.3 Buoyancy Effect Study



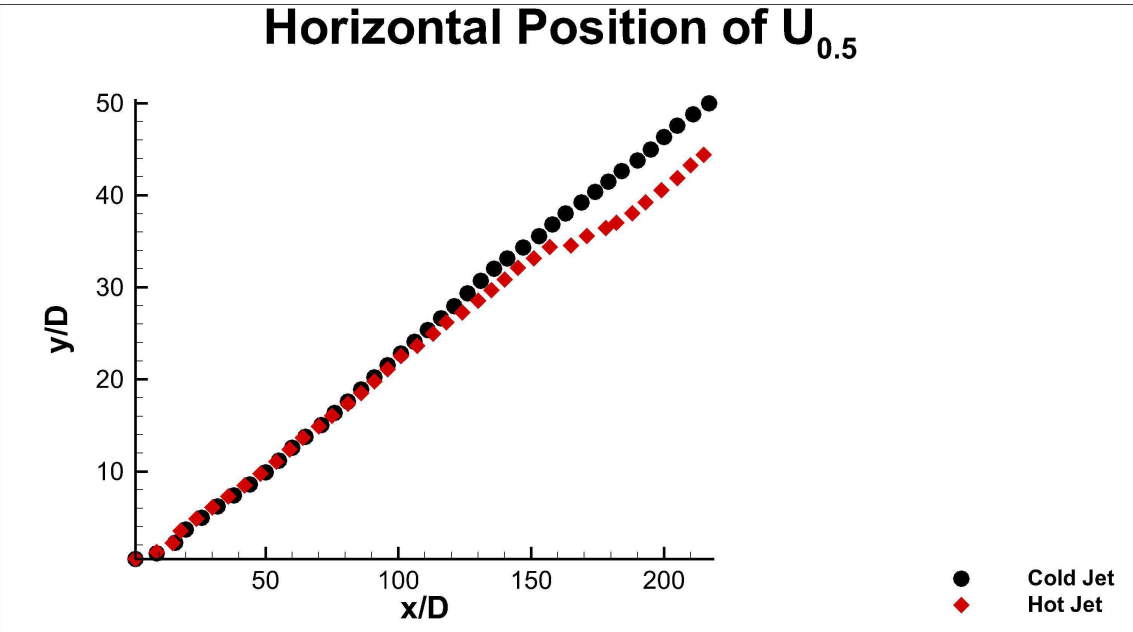
(a)



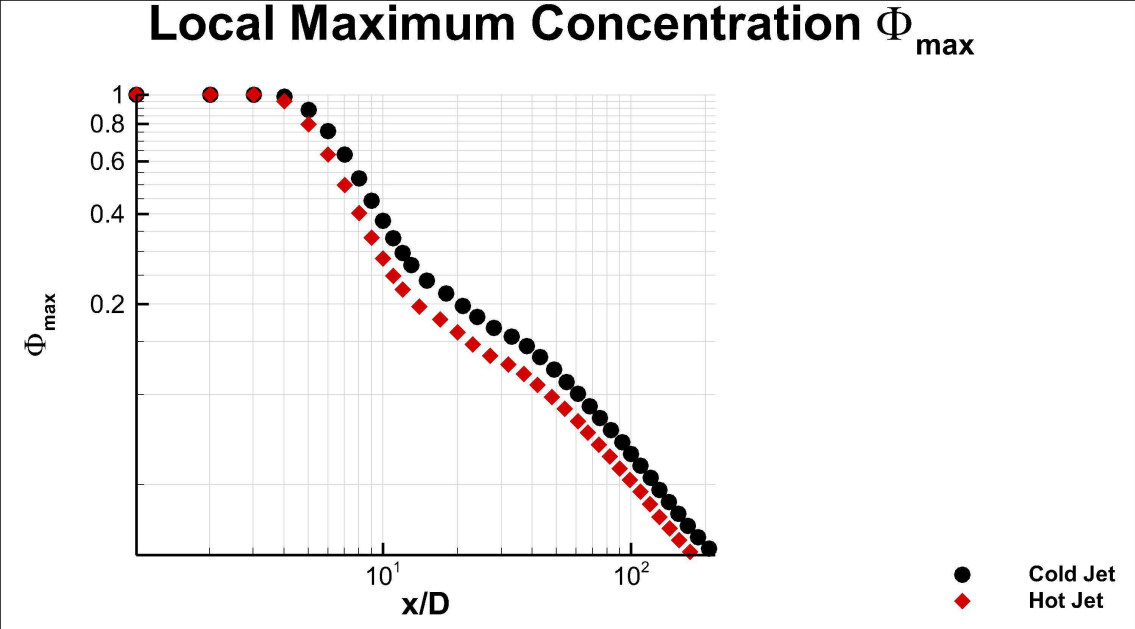
(b)



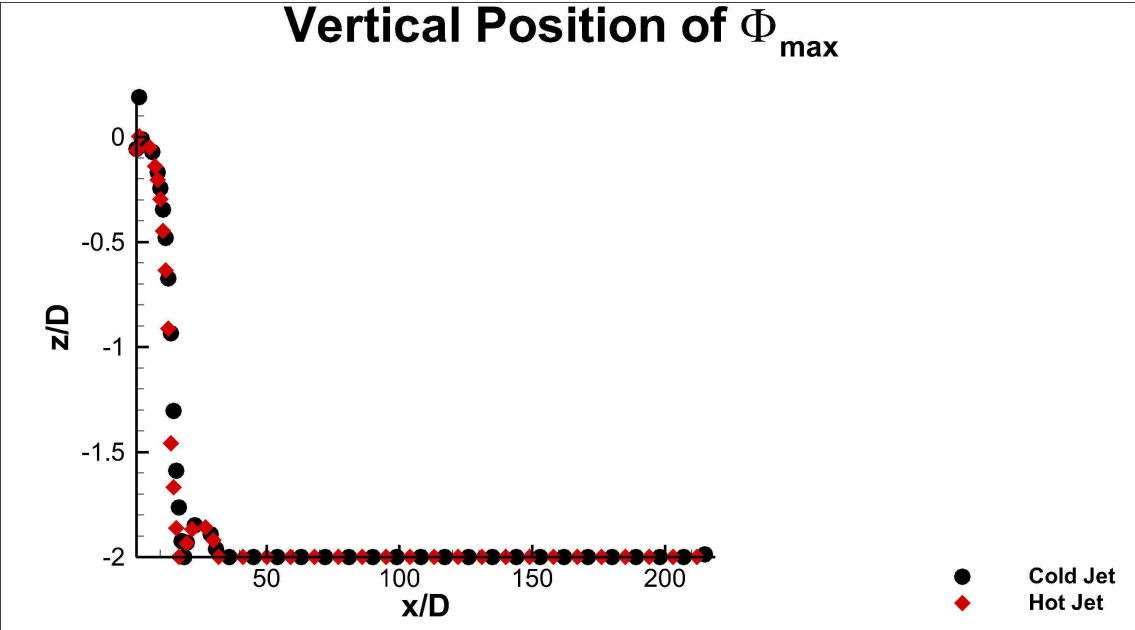
(c)



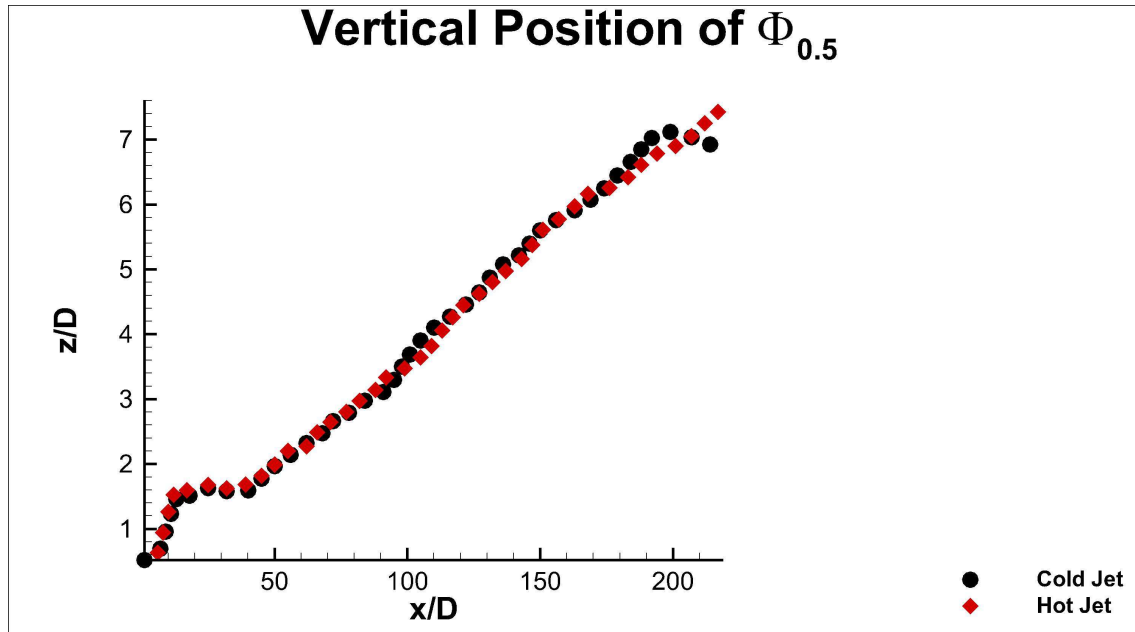
(d)



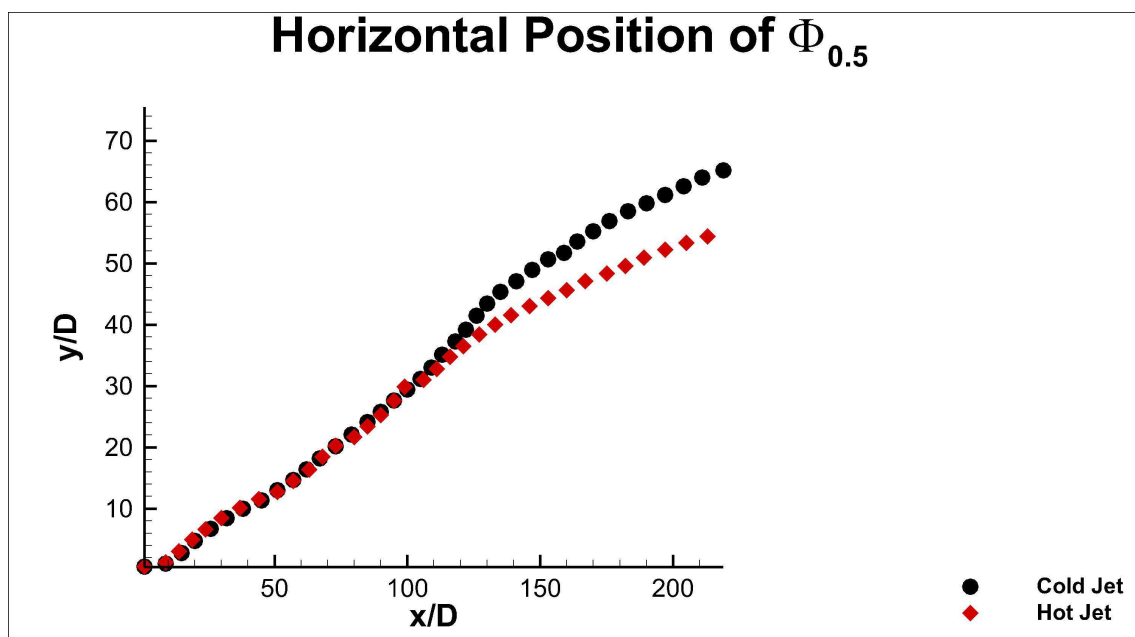
(e)



(f)



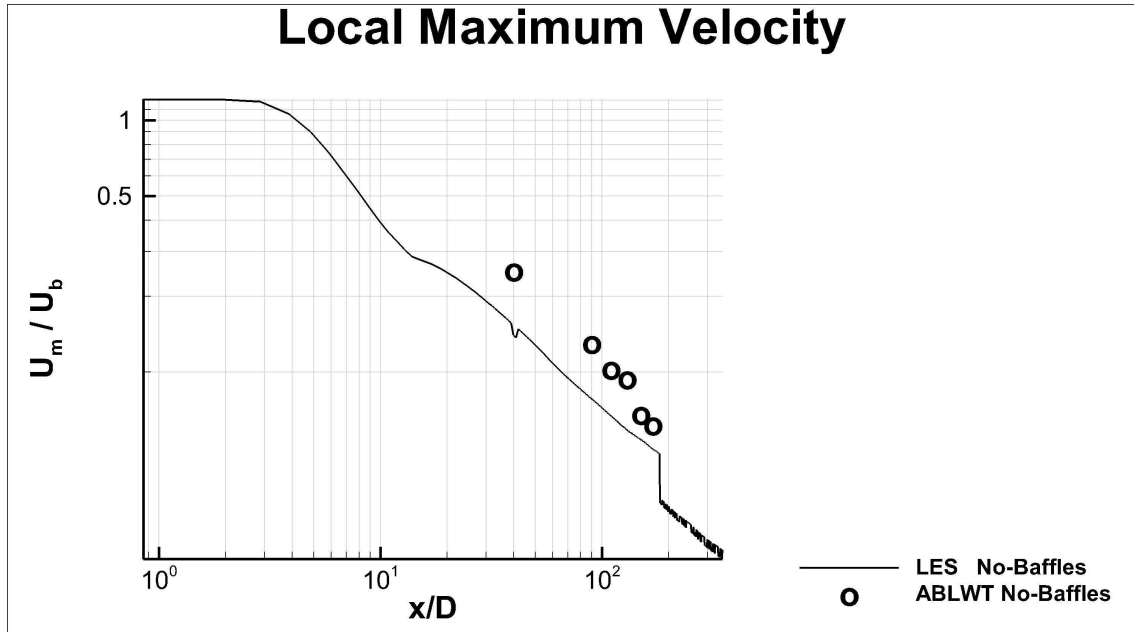
(g)



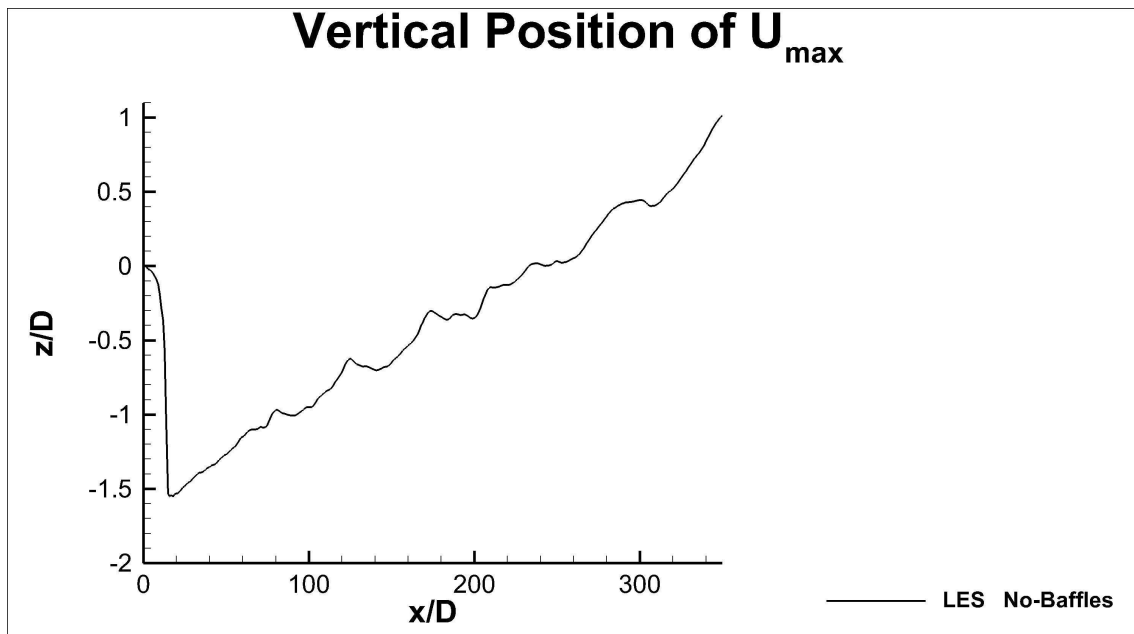
(h)

Figure D.3: Streamwise plots of the “*buoyancy effect*” study.

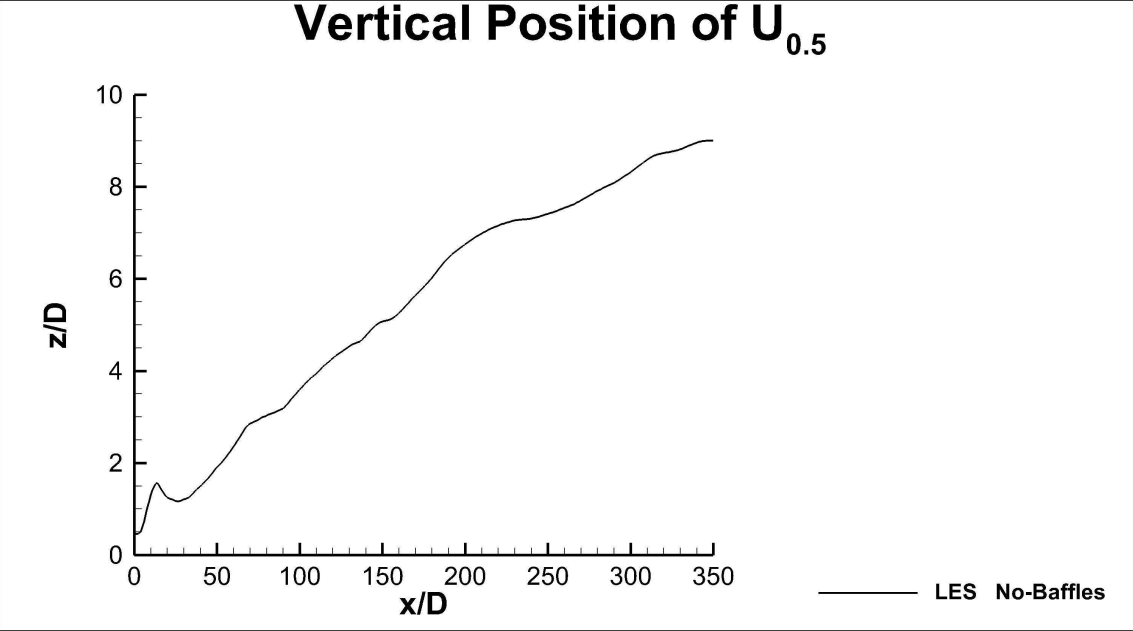
D.4 Hot Unresisted 3DOJ



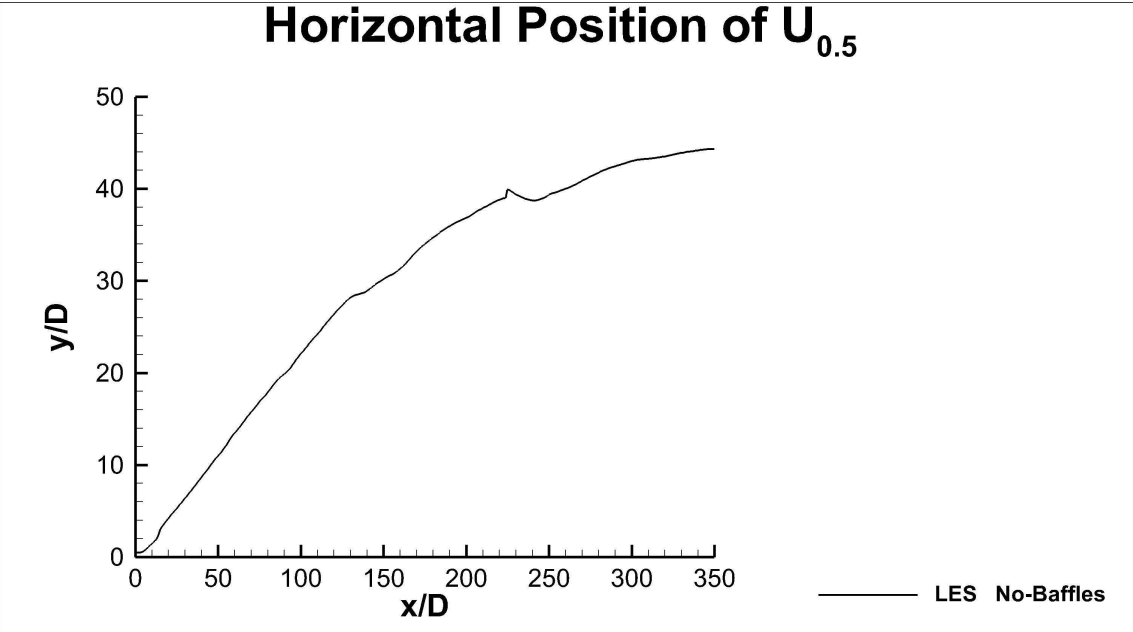
(a)



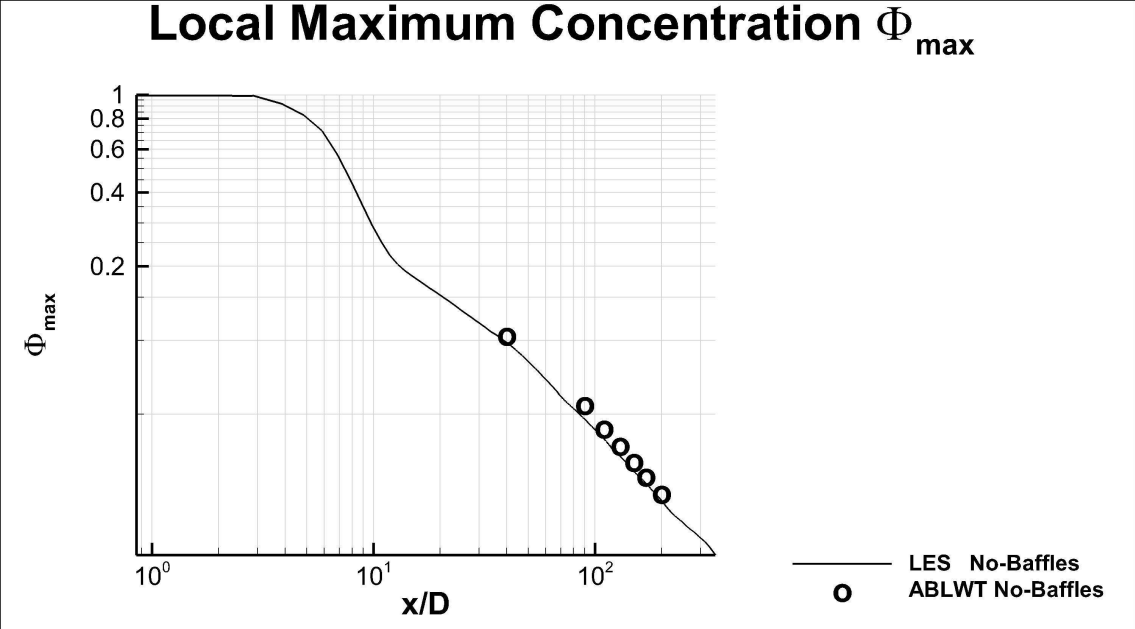
(b)



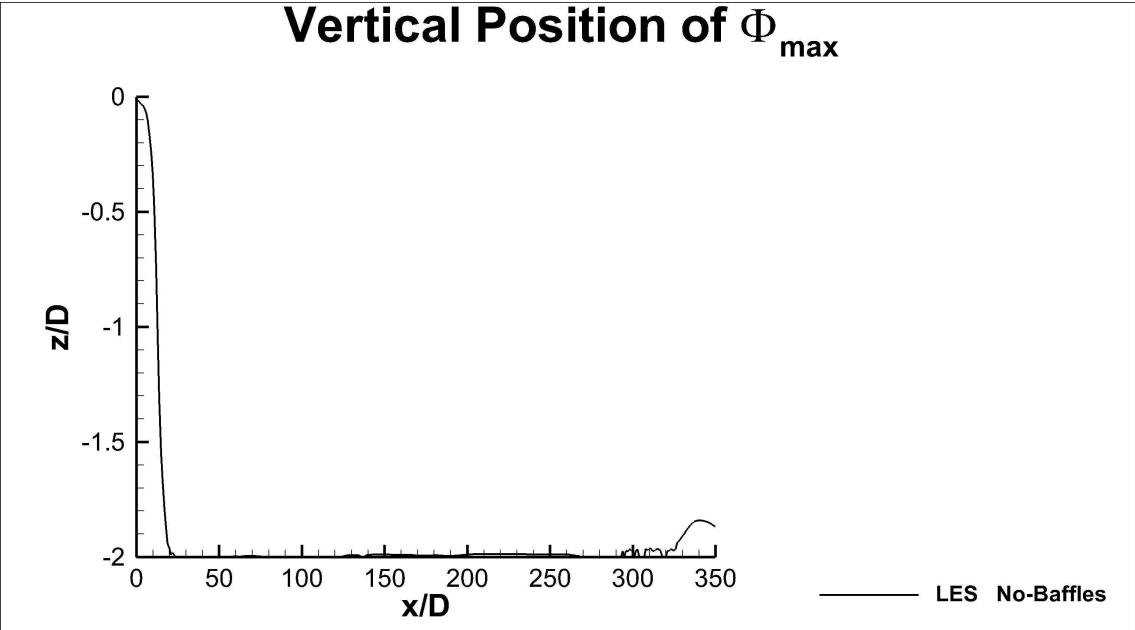
(c)



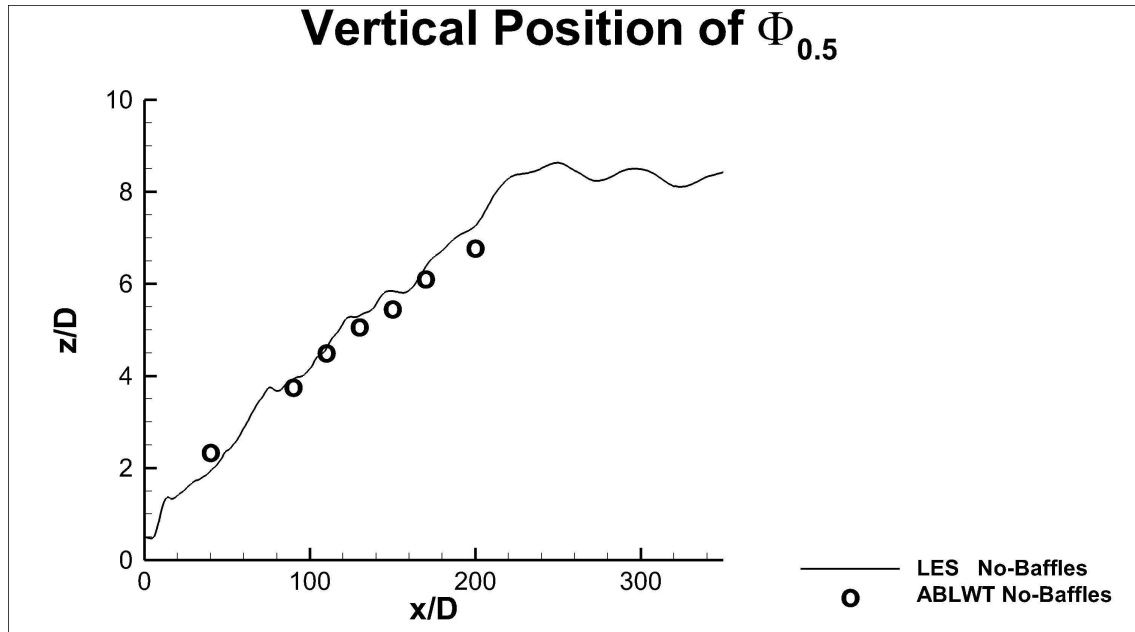
(d)



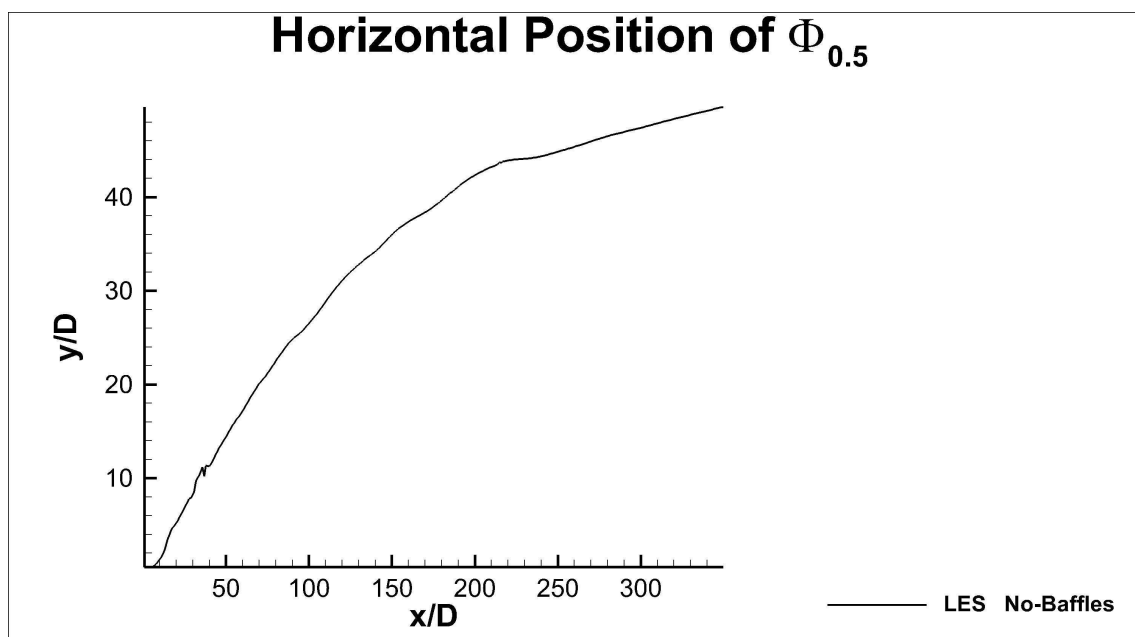
(e)



(f)



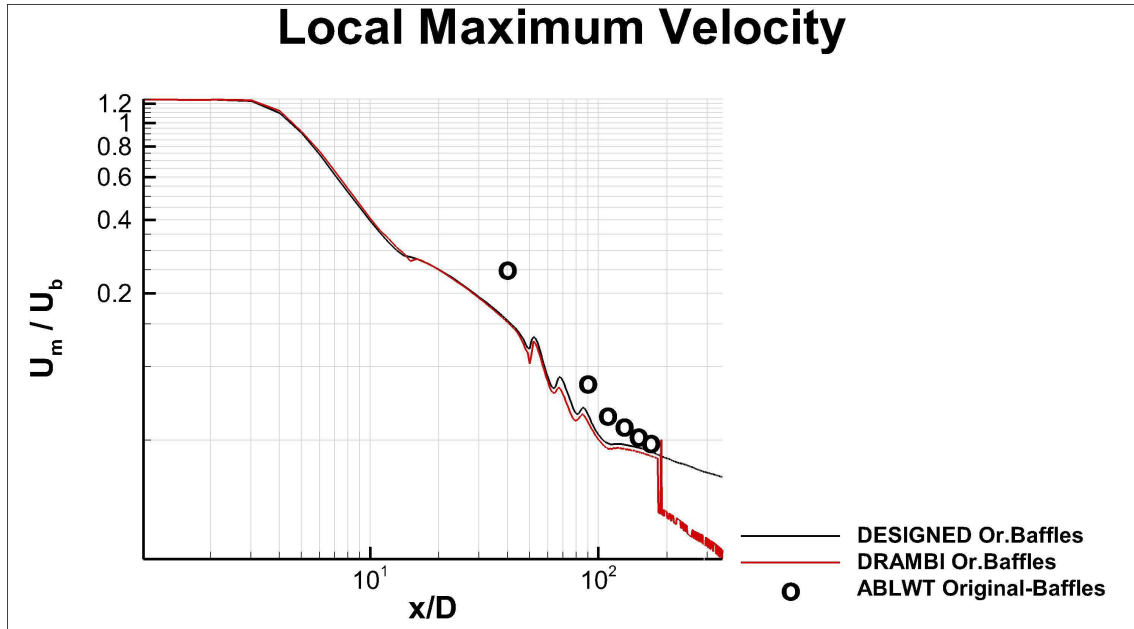
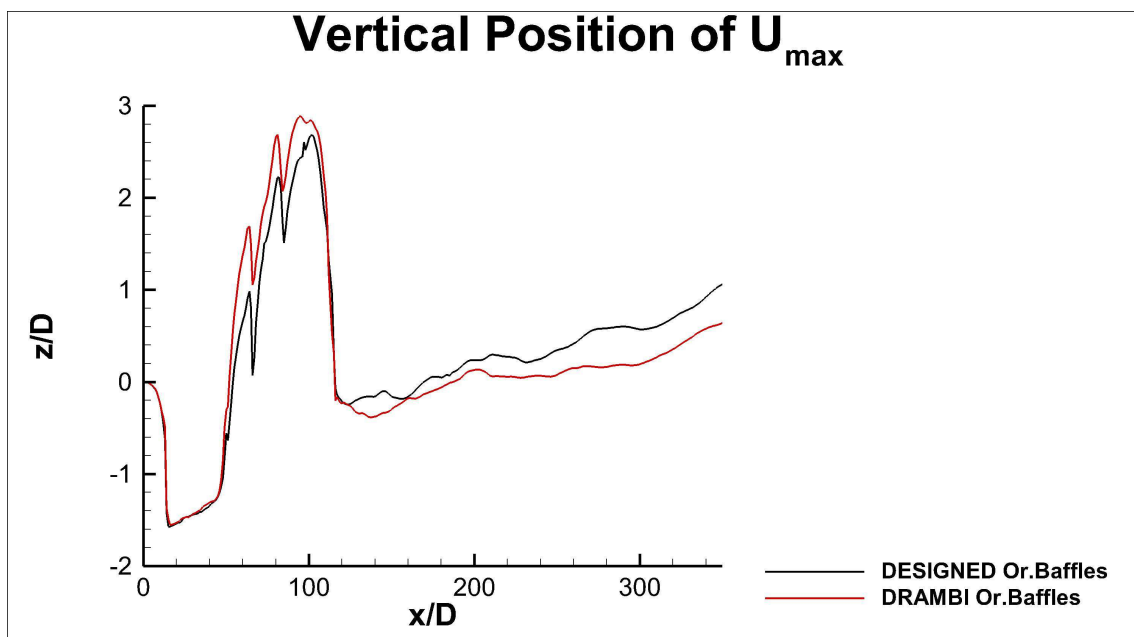
(g)

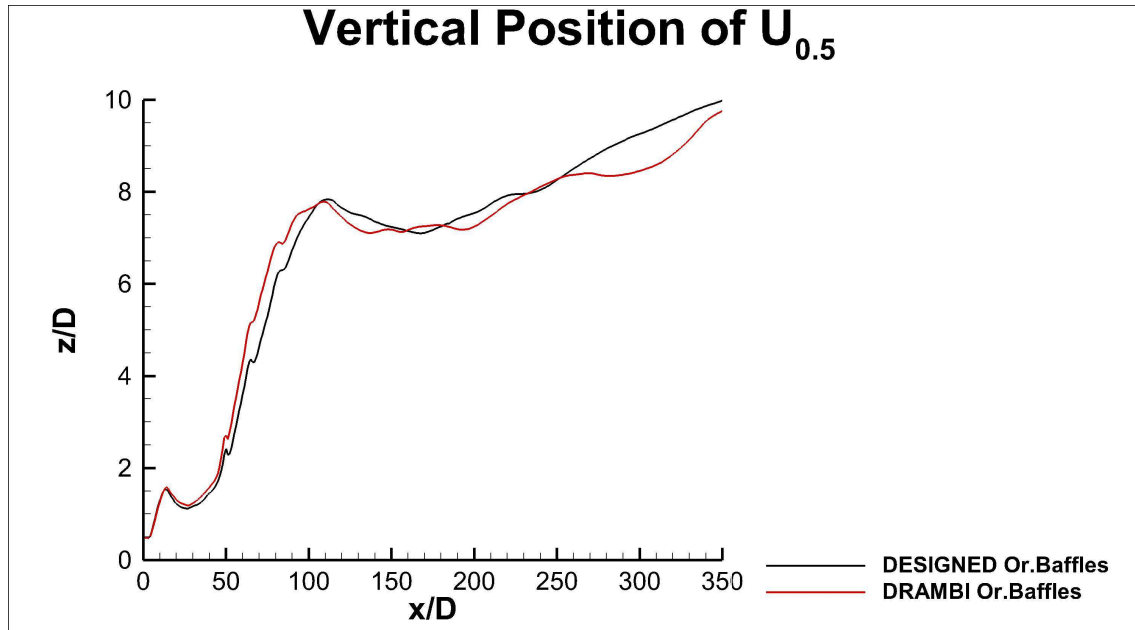


(h)

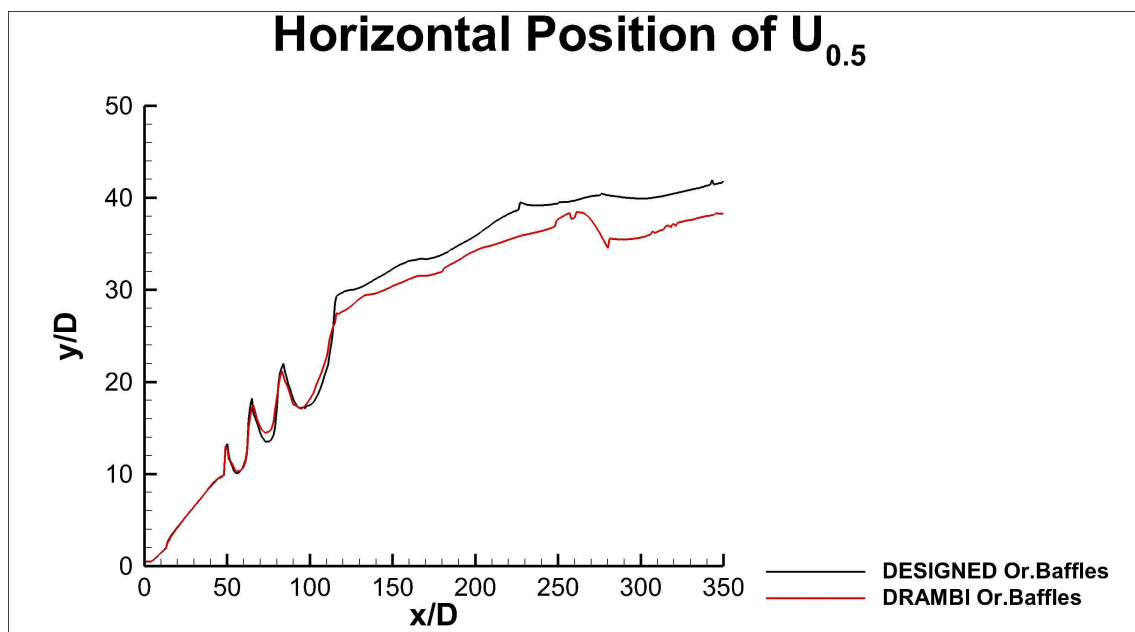
Figure D.4: Streamwise plots of the unresisted jet simulation.

D.5 Original Baffle Arrangement

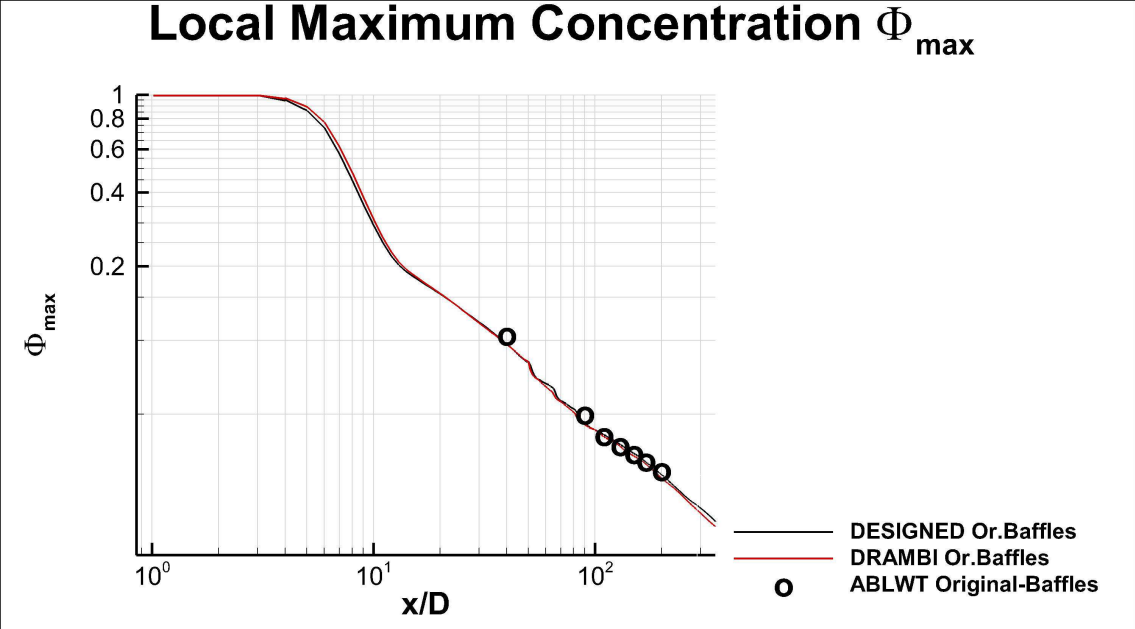




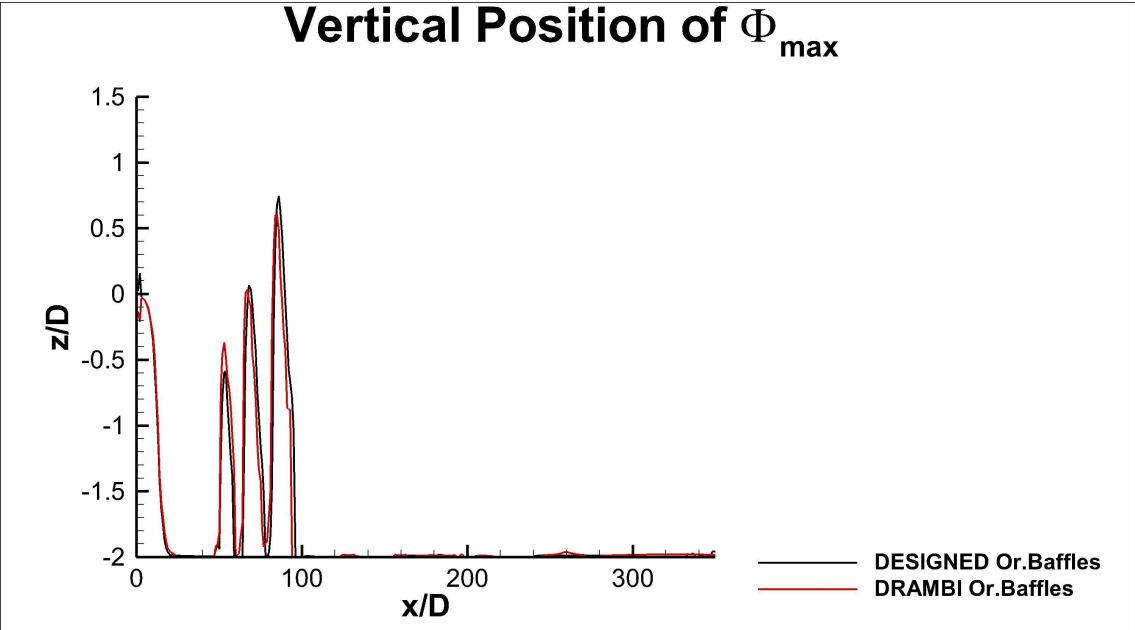
(c)



(d)



(e)



(f)

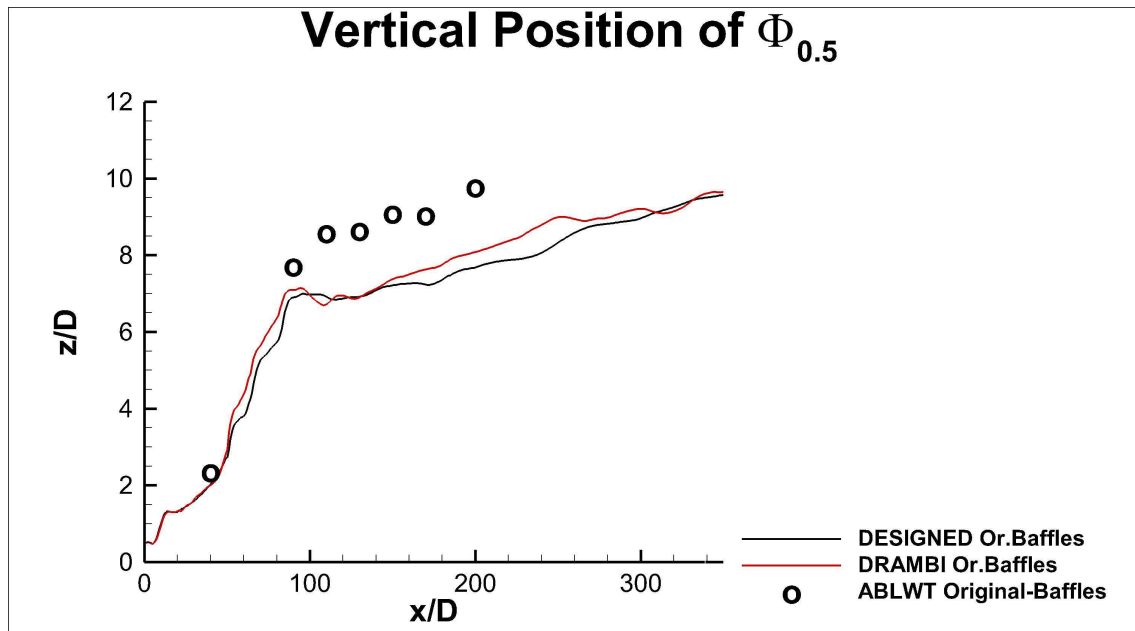
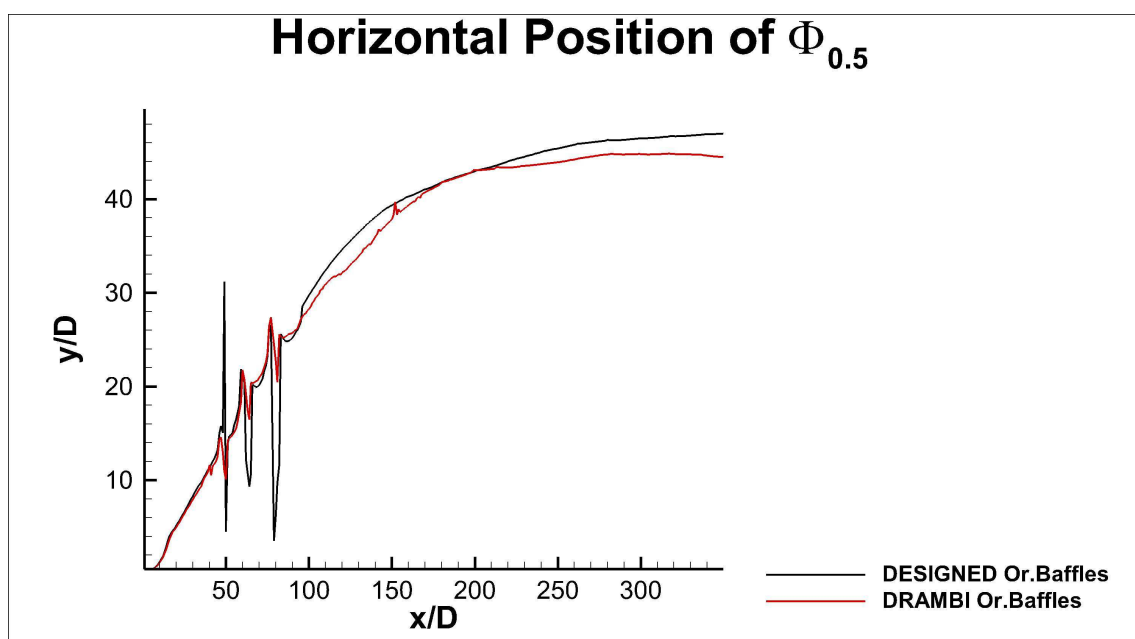
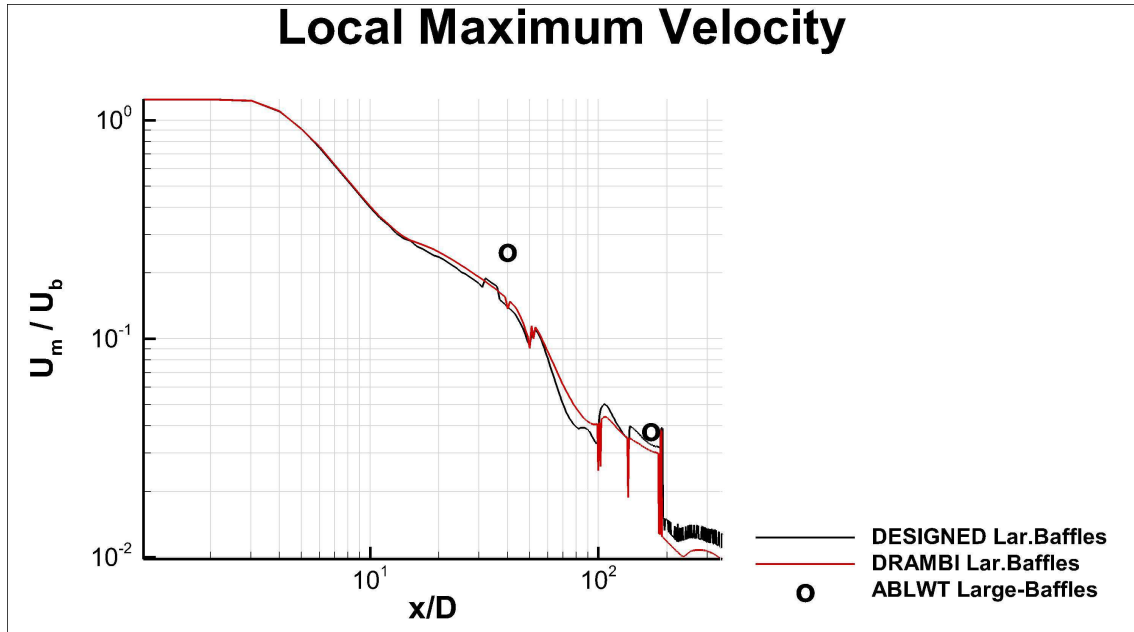
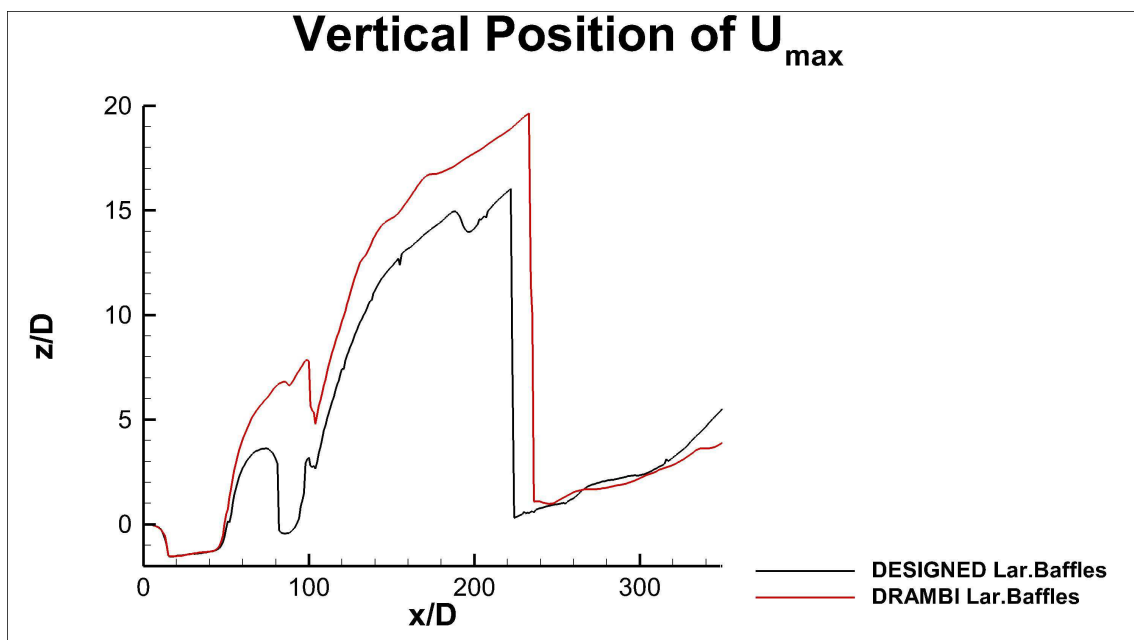
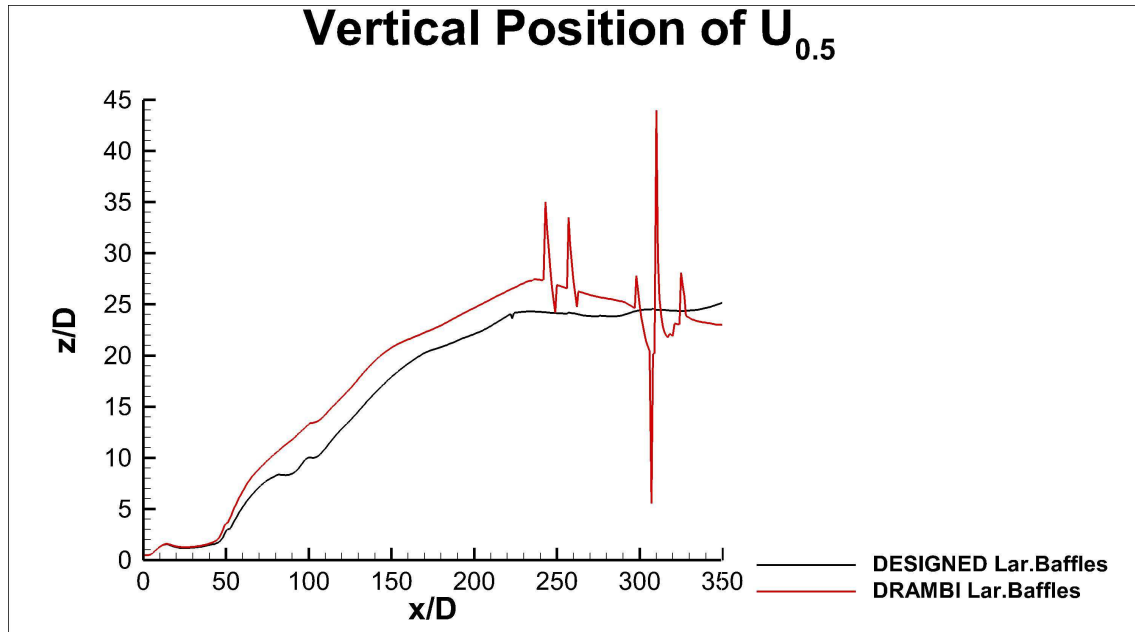
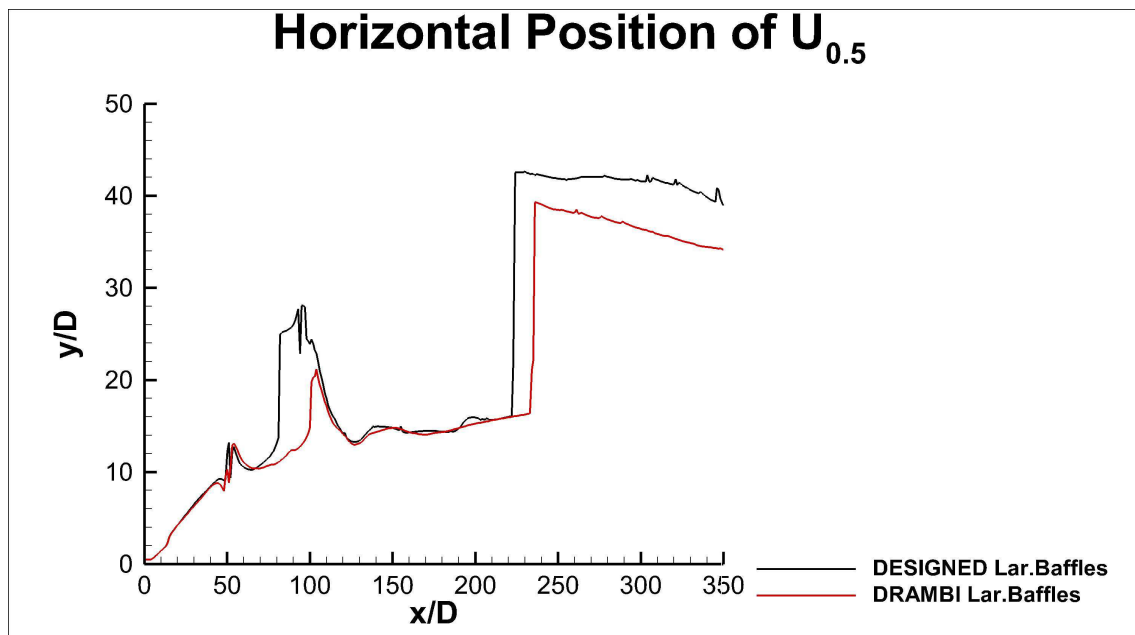
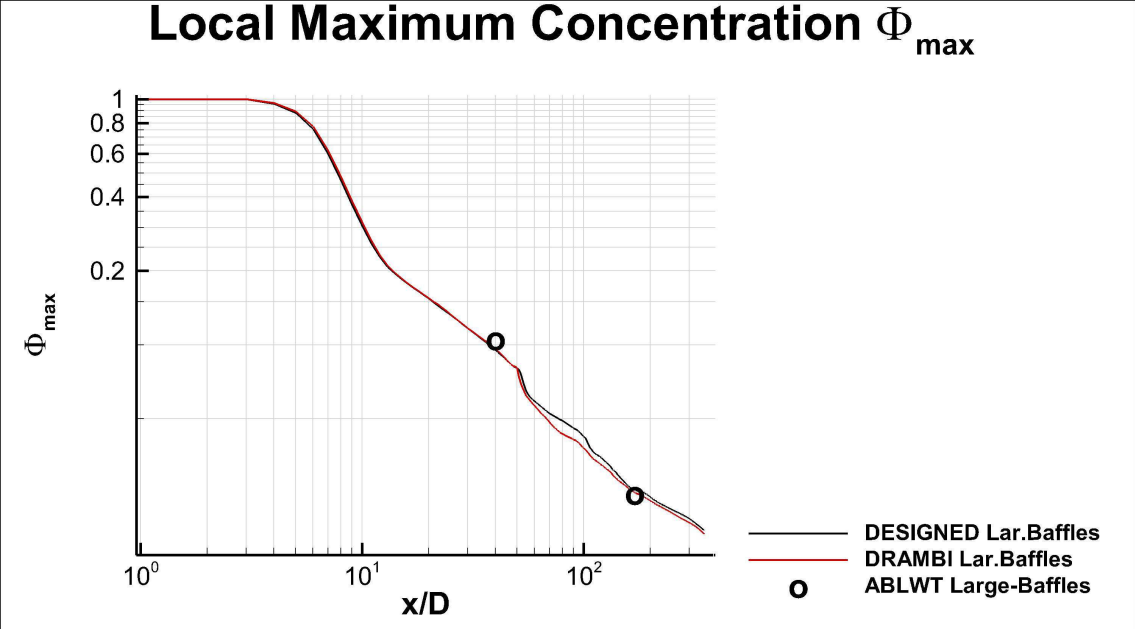


Figure D.5: Streamwise plots of the “ORIGINAL” baffle arrangement.

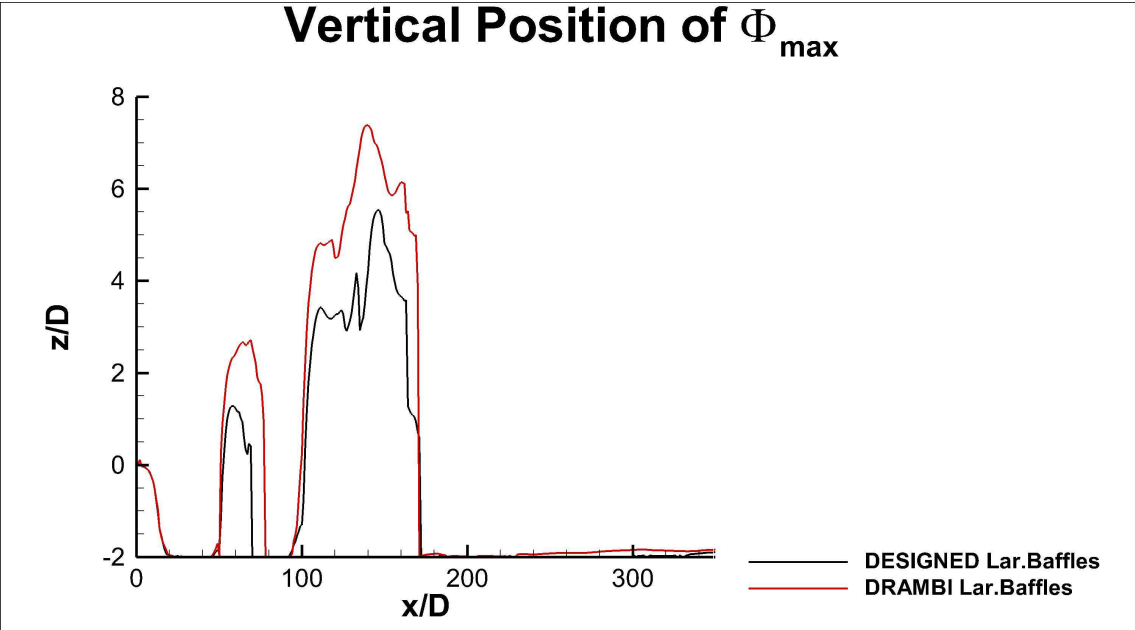
D.6 Large Baffle Arrangement



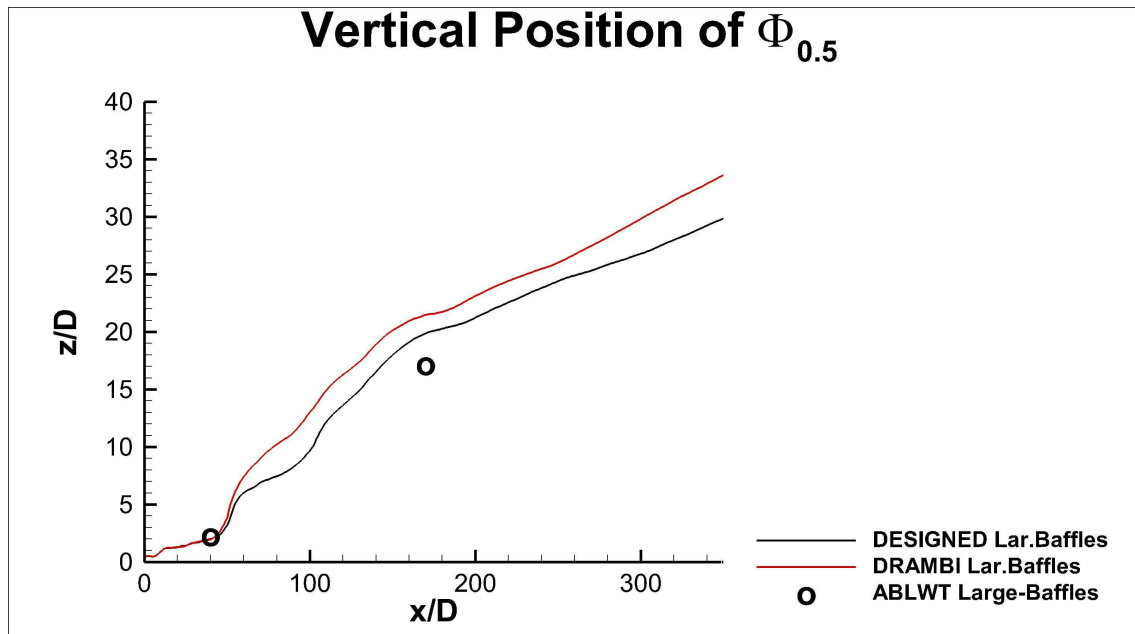




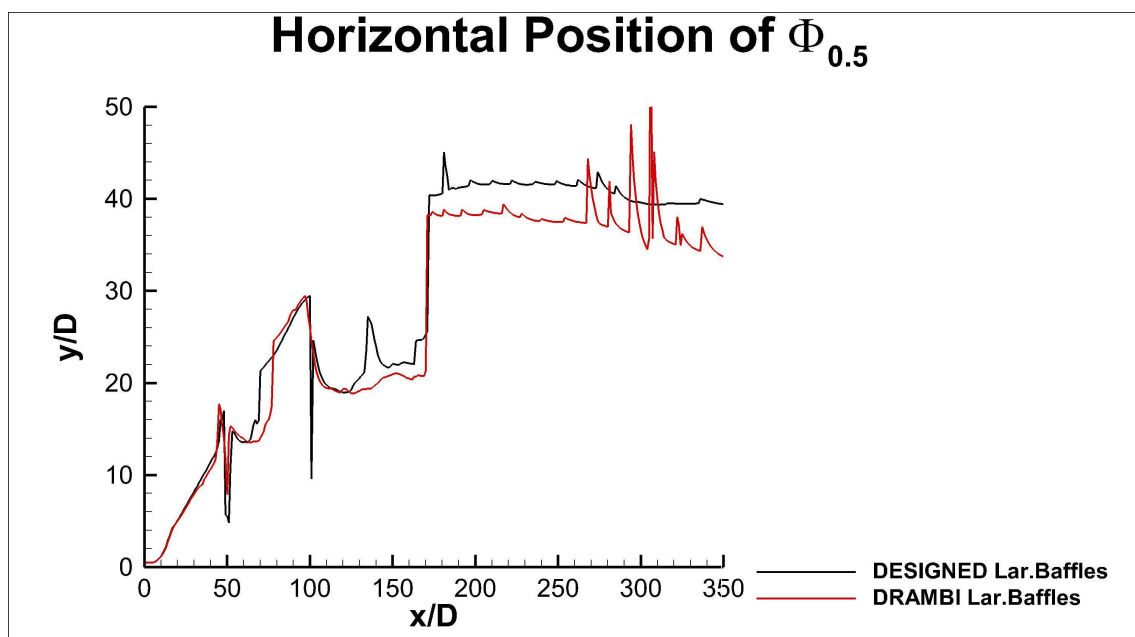
(e)



(f)



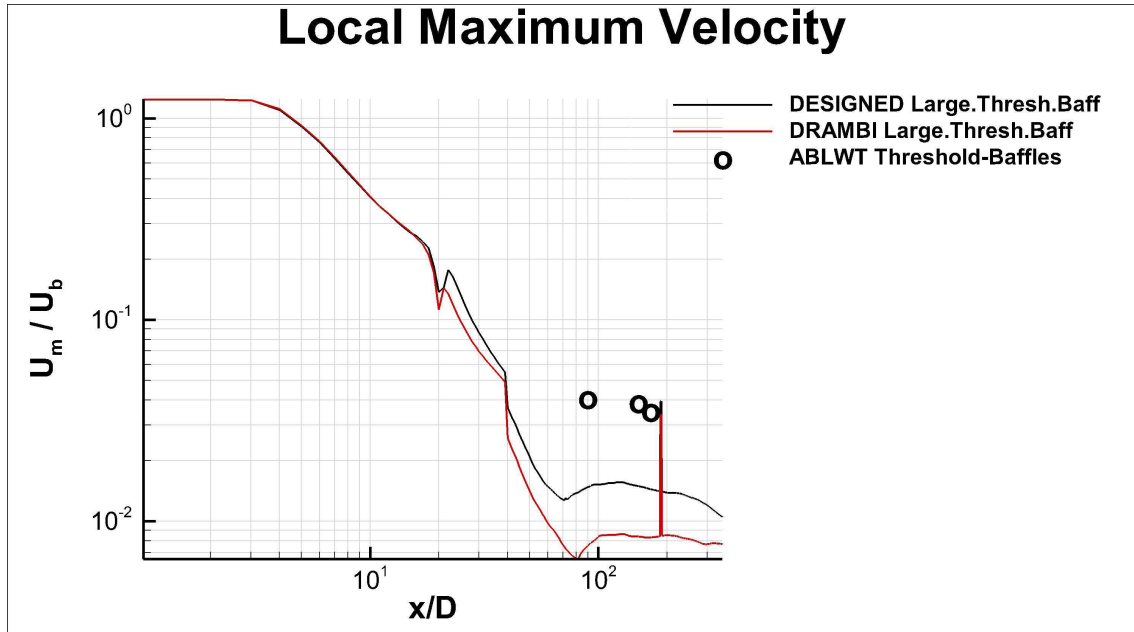
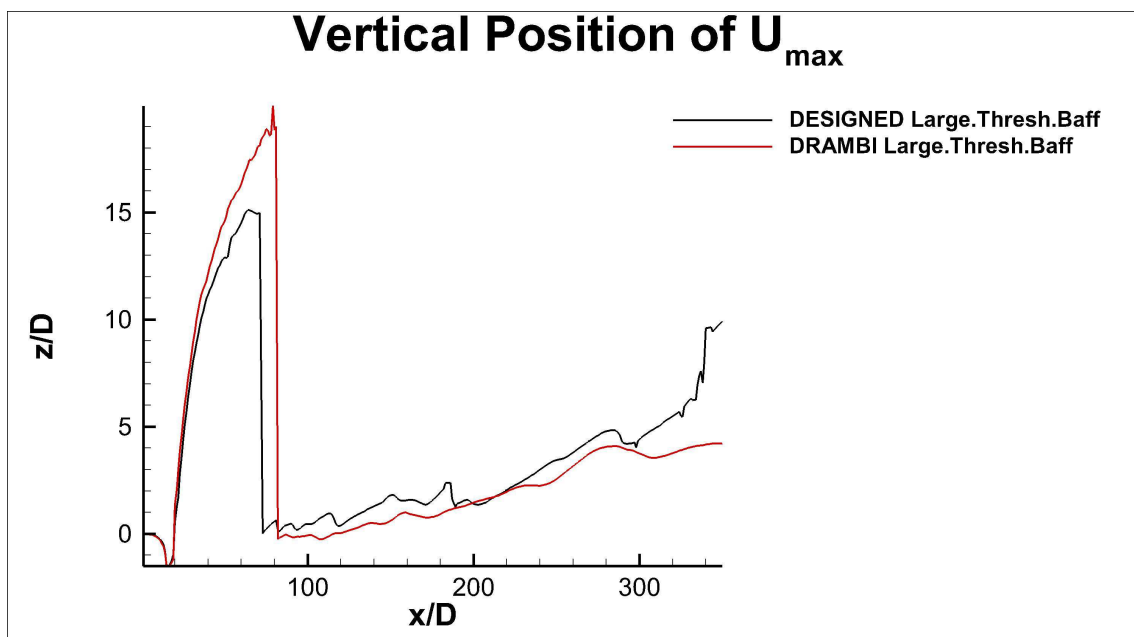
(g)

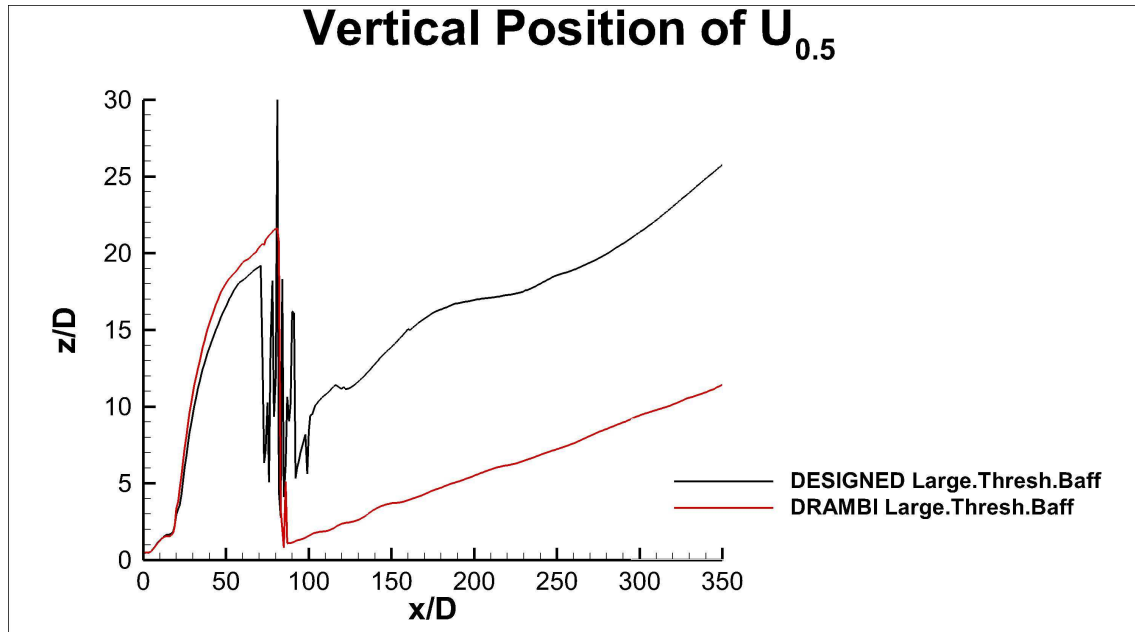


(h)

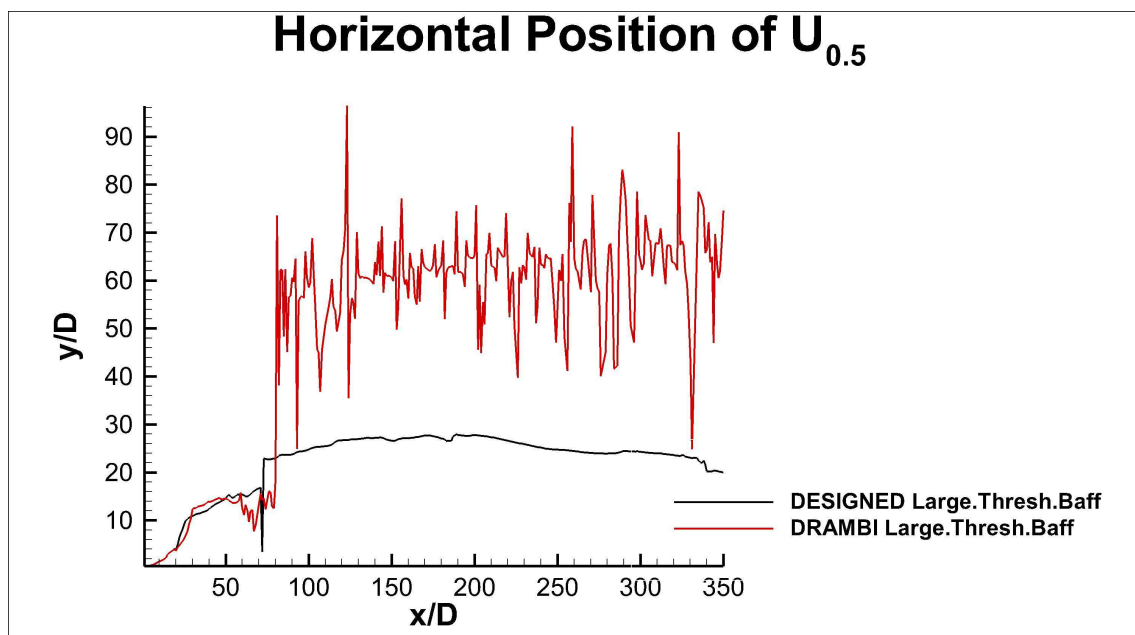
Figure D.6: Streamwise plots of the “*LARGE*” baffle arrangement.

D.7 Large Threshold Baffle Arrangement

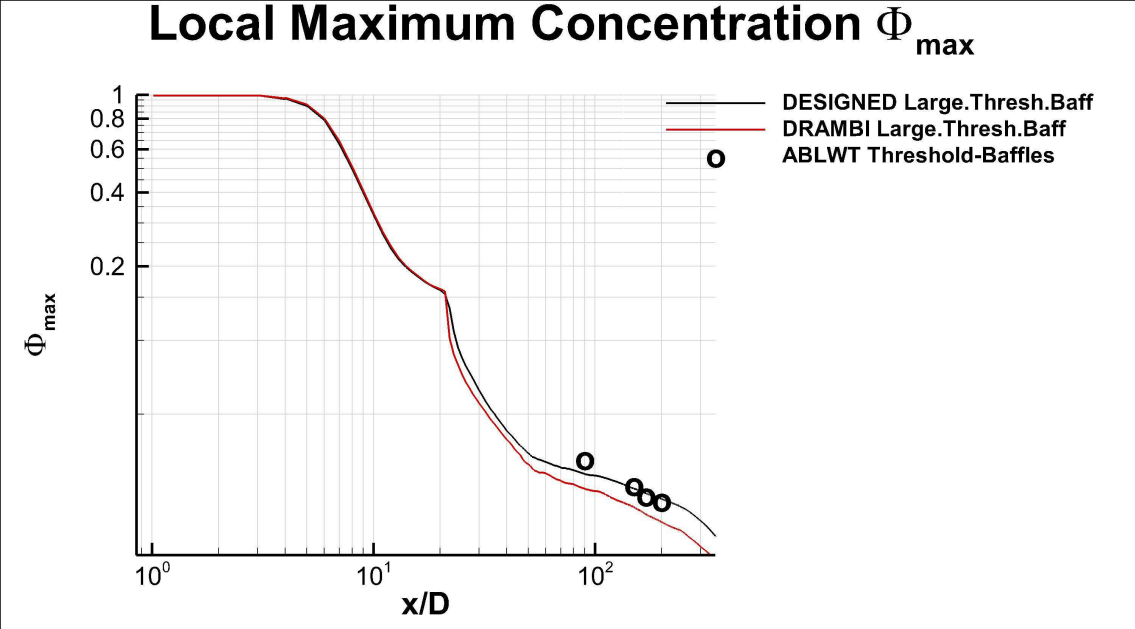




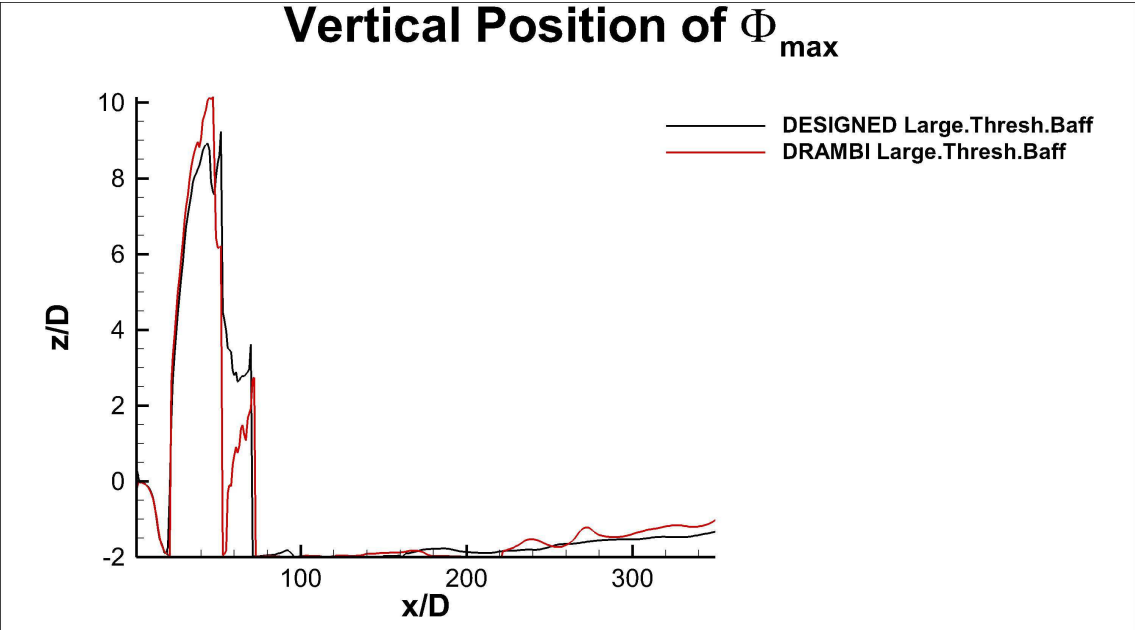
(c)



(d)



(e)



(f)

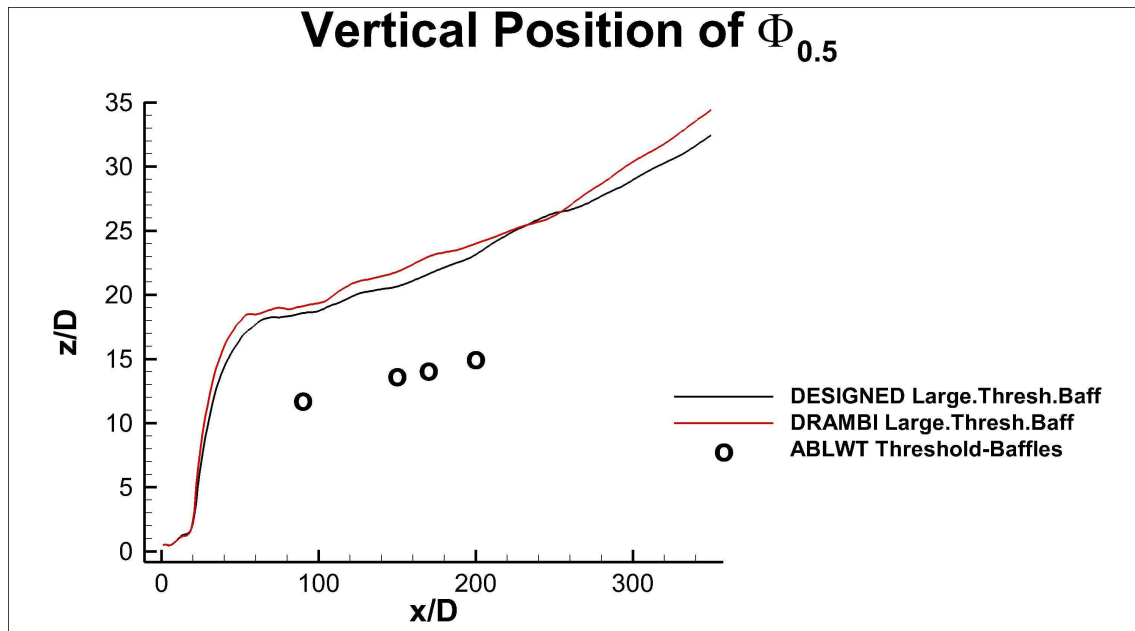
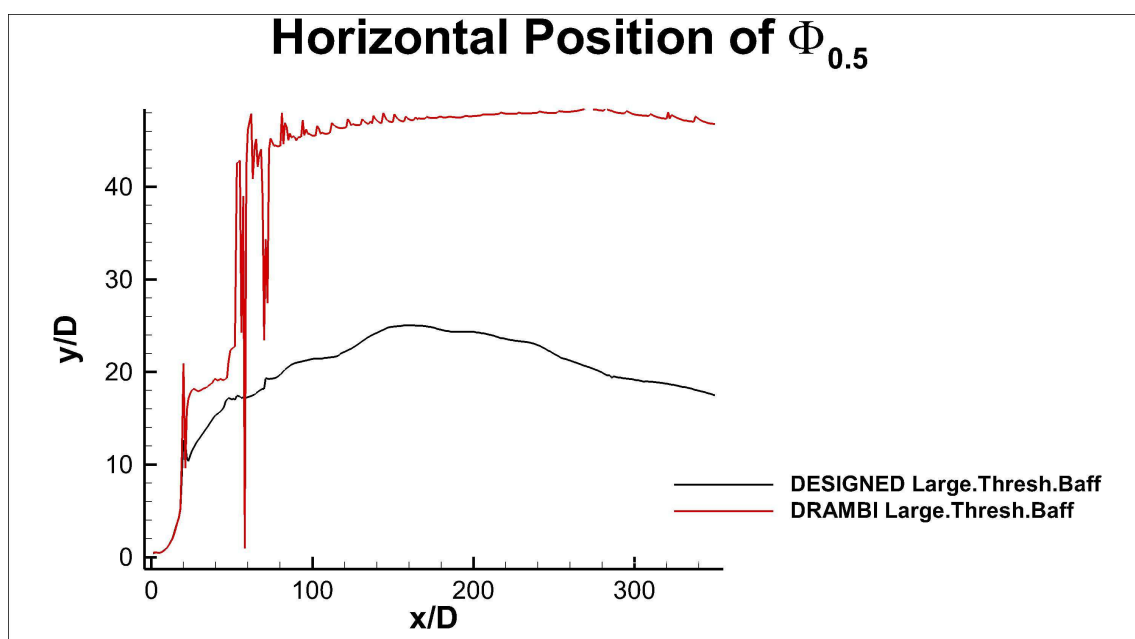
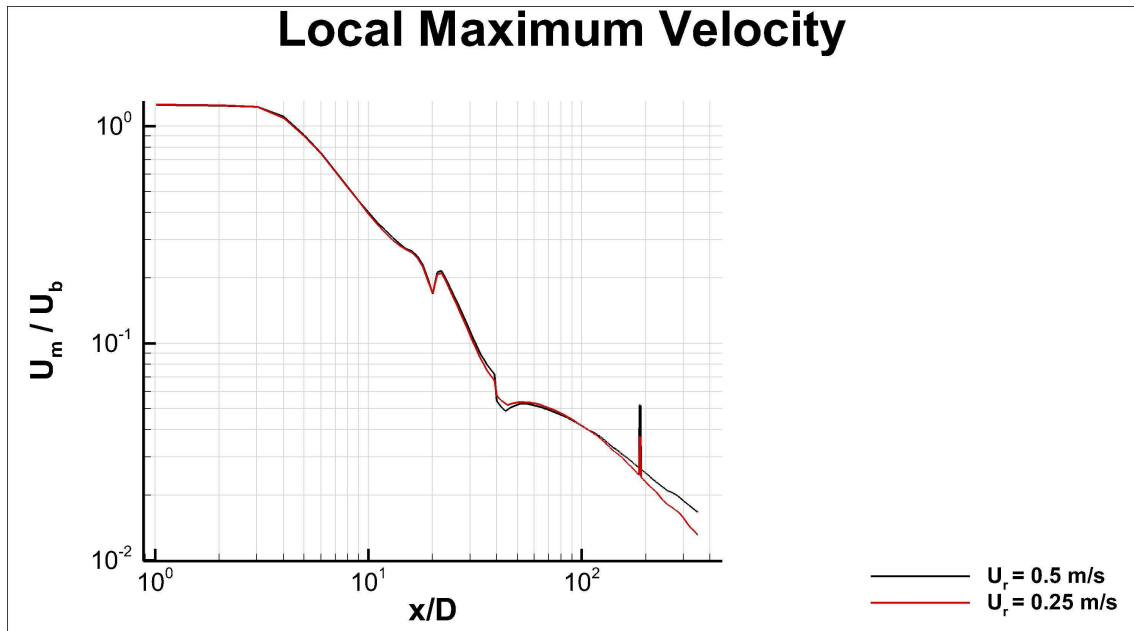
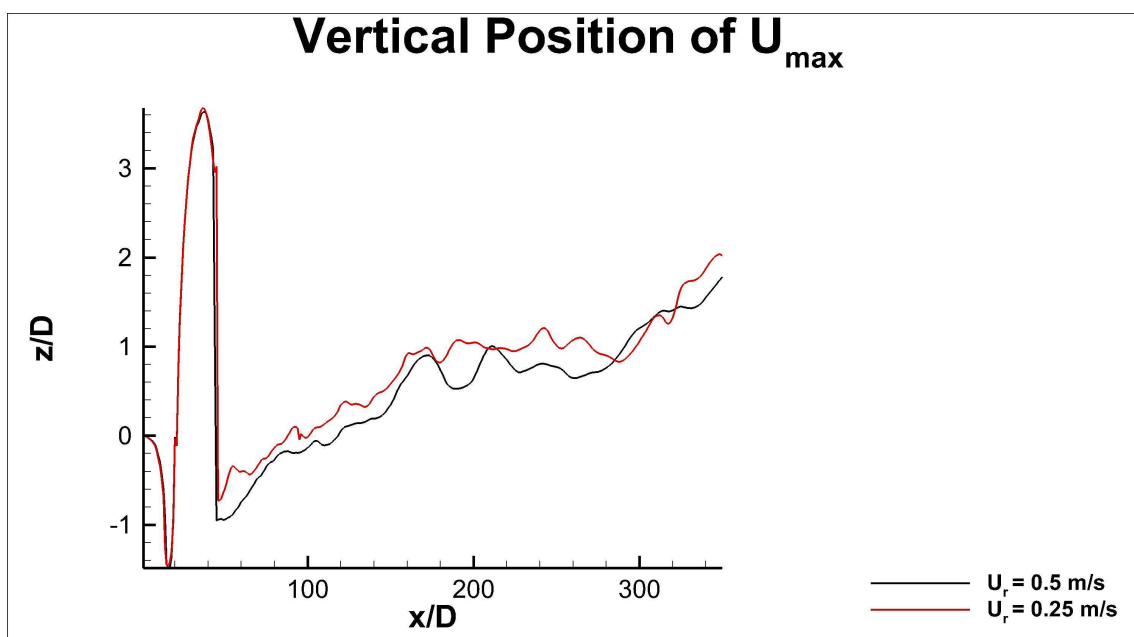


Figure D.7: Streamwise plots of the “*THRESHOLD-LARGE*” baffle arrangement.

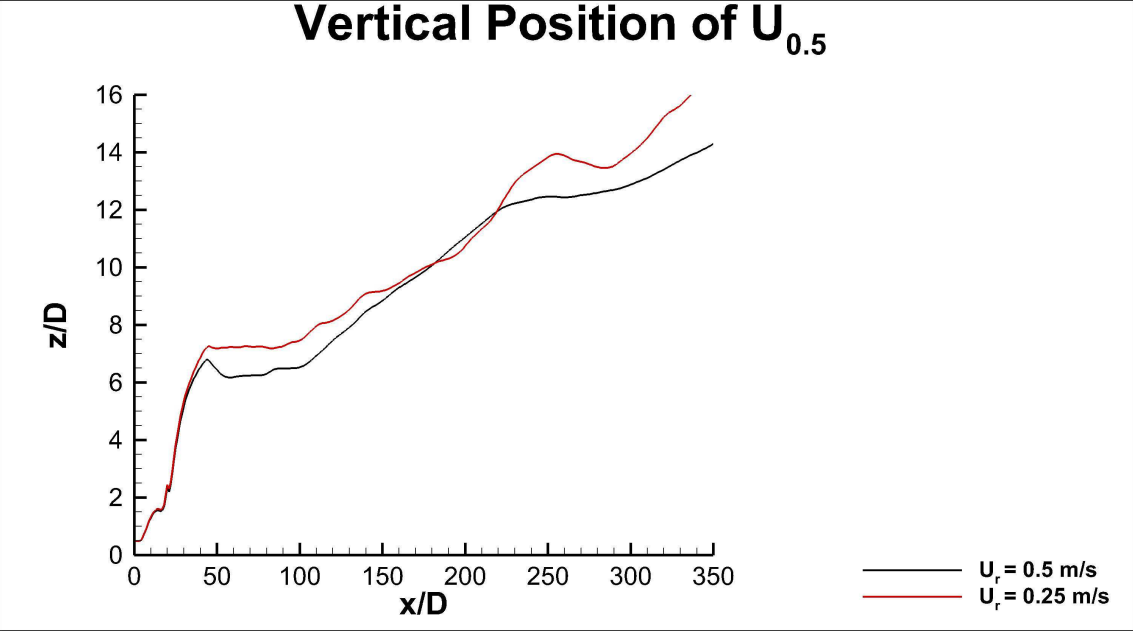
D.8 Co-Flow Effect Study



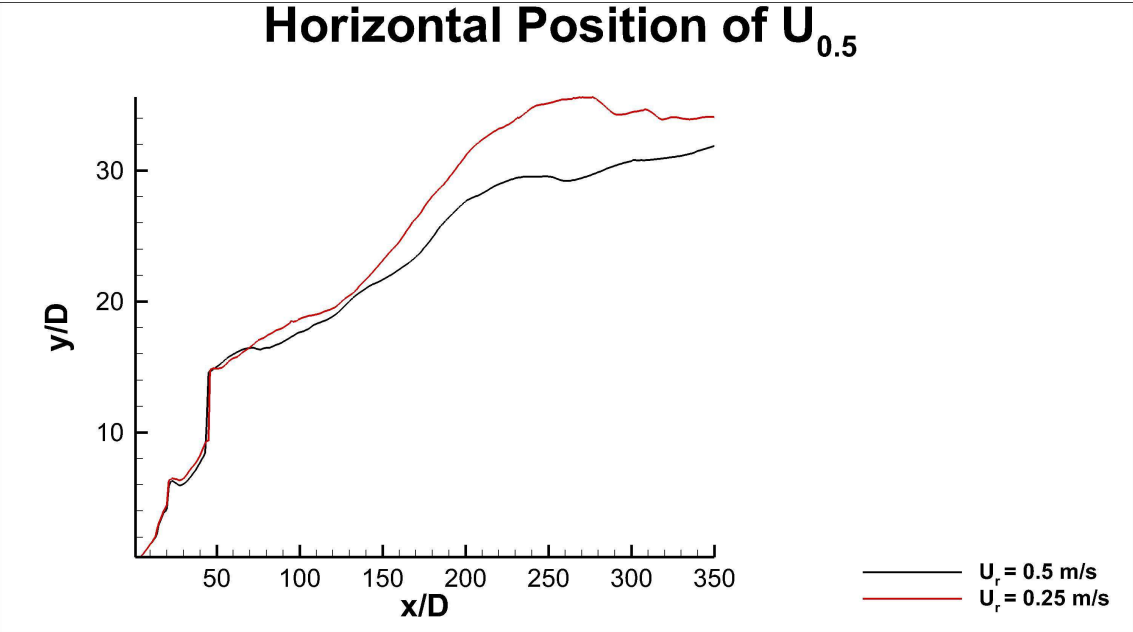
(a)



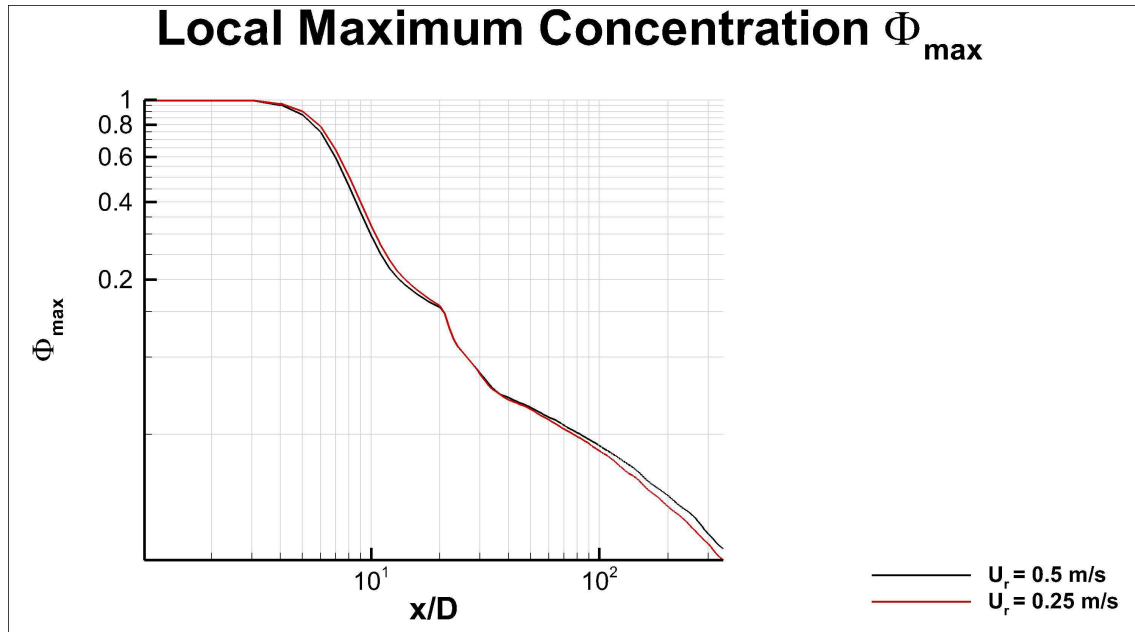
(b)



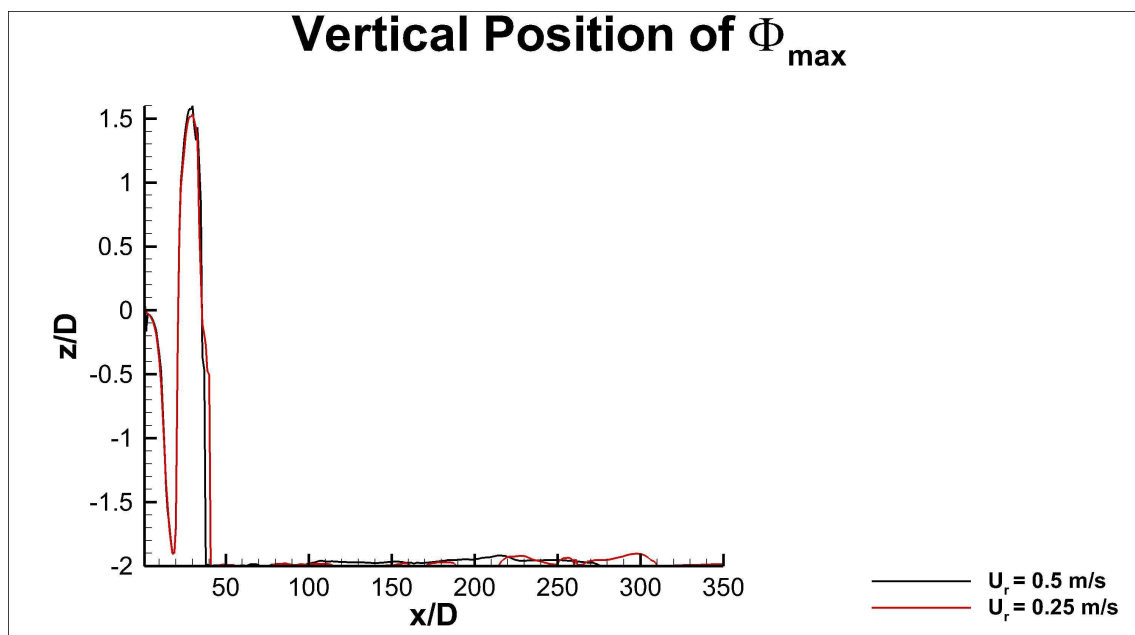
(c)



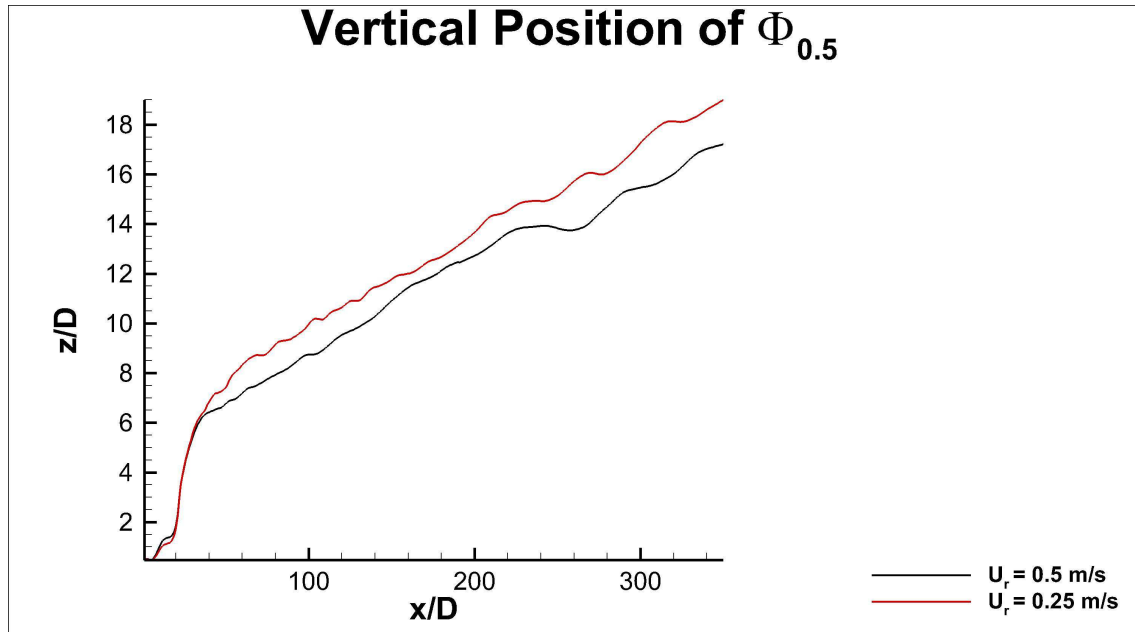
(d)



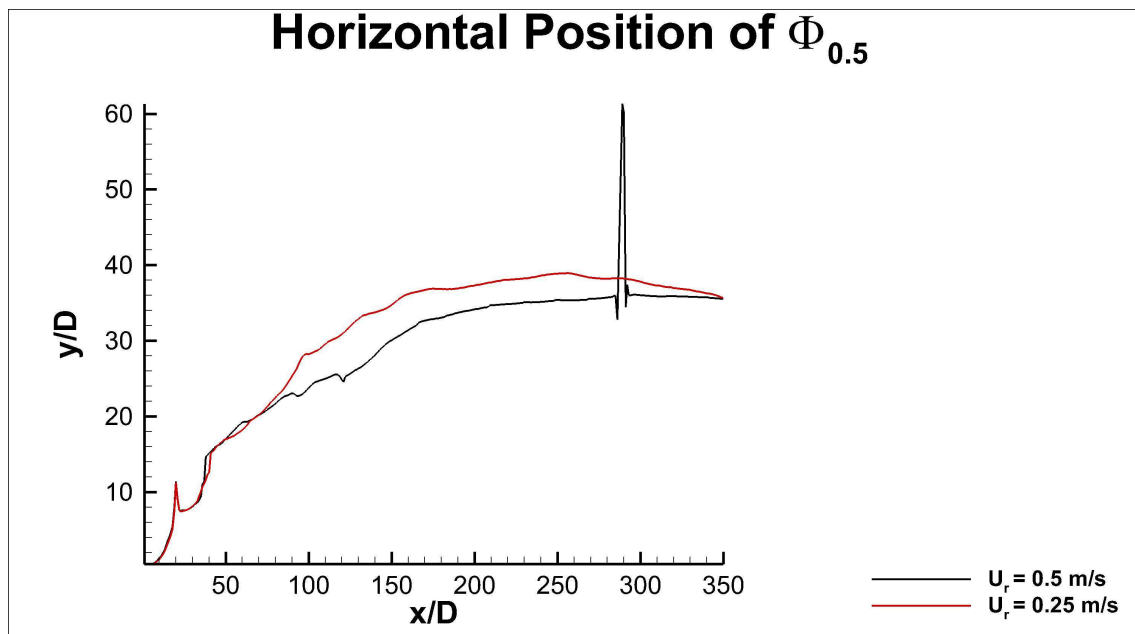
(e)



(f)



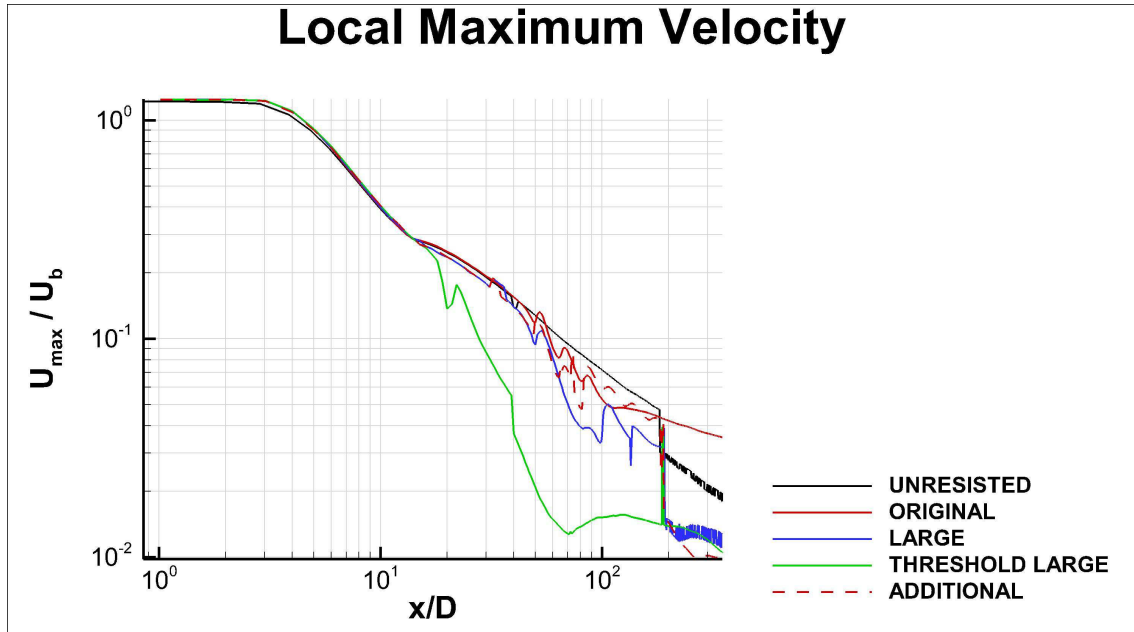
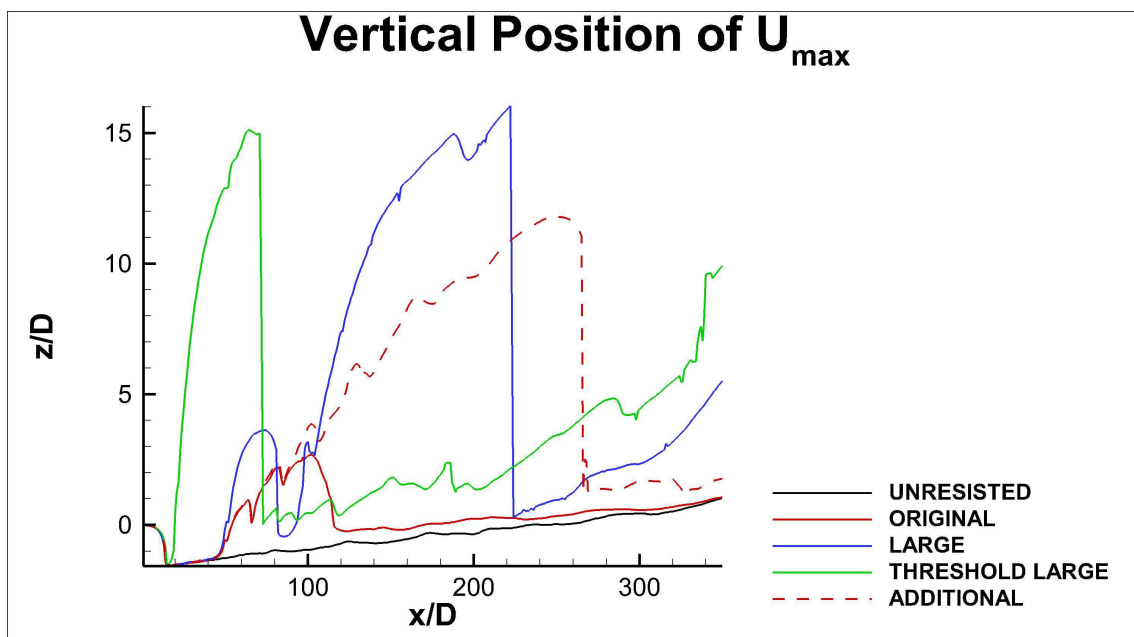
(g)

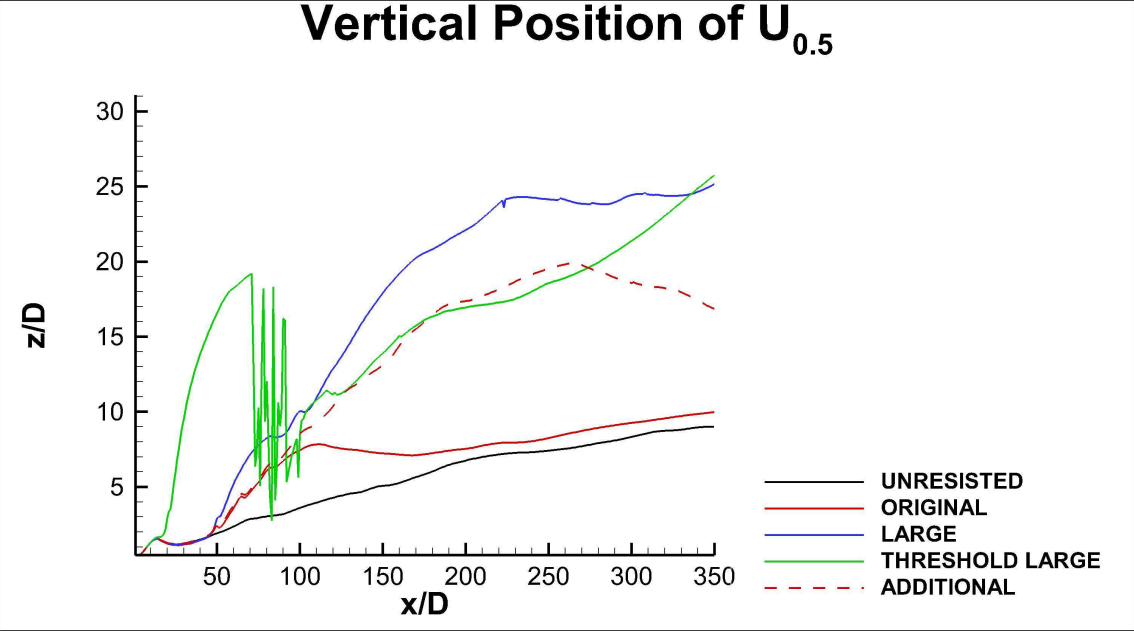


(h)

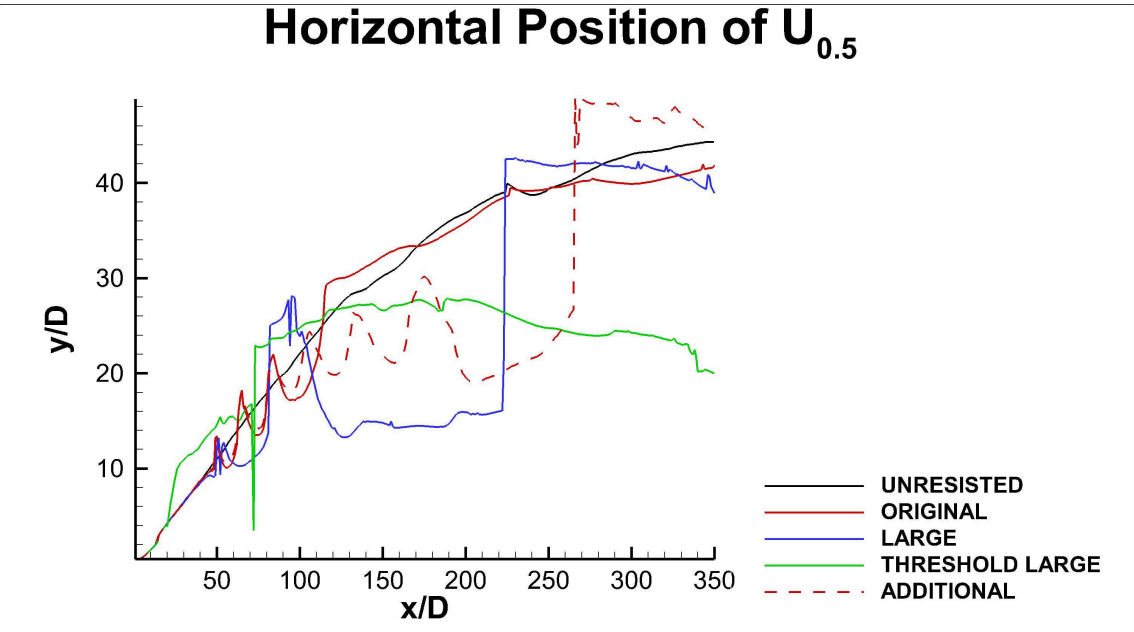
Figure D.8: Streamwise plots of the co-flow effect study.

D.9 Baffle Arrangement

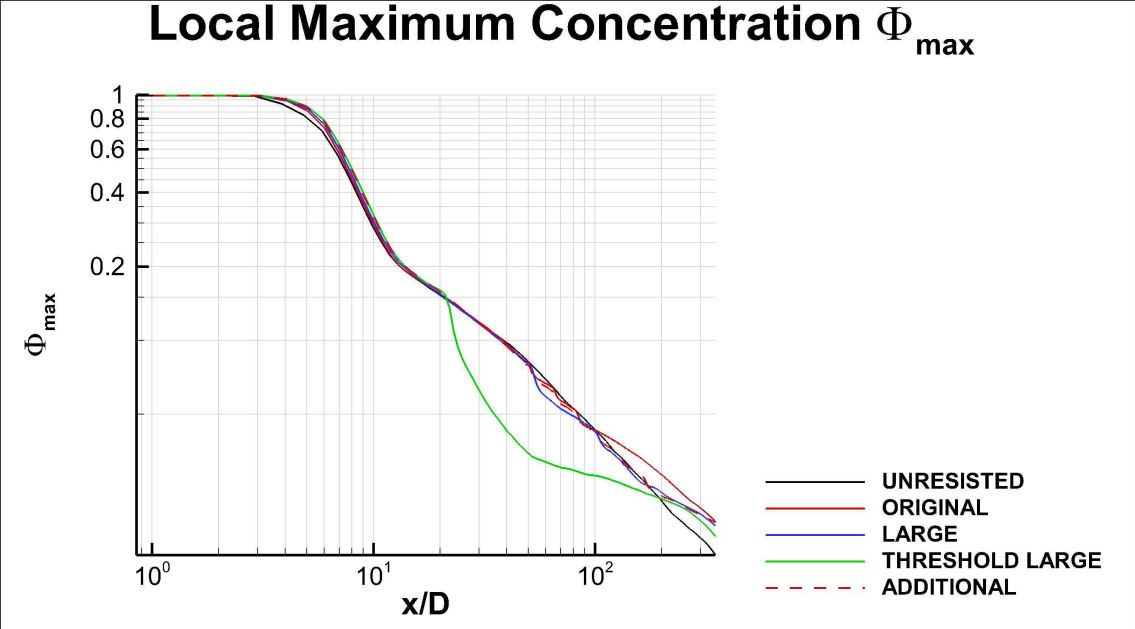




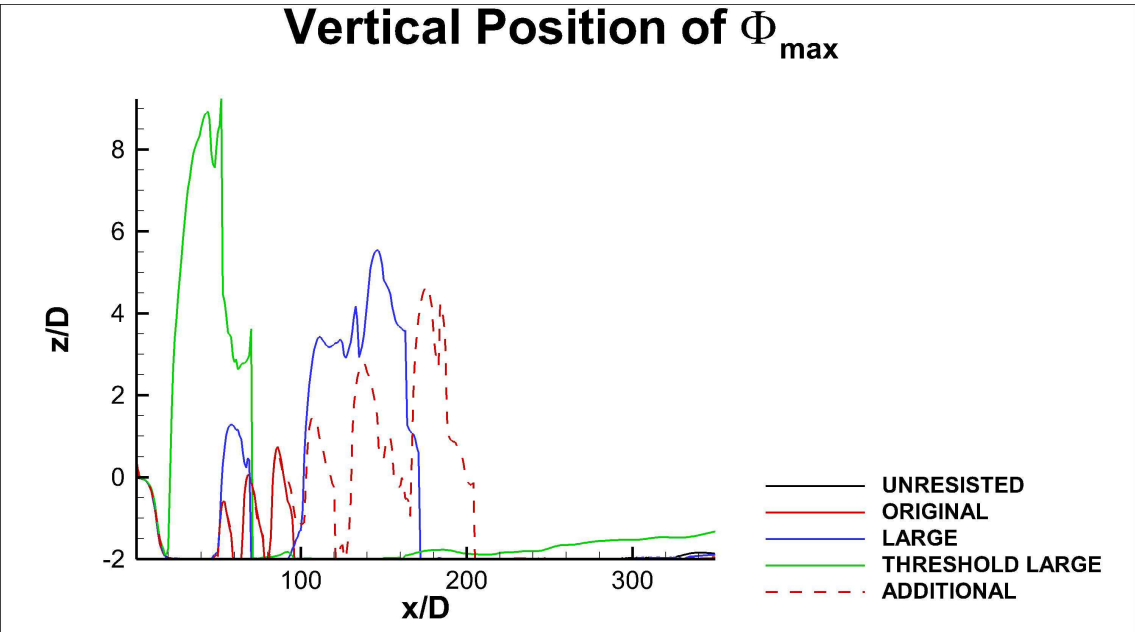
(c)



(d)



(e)



(f)

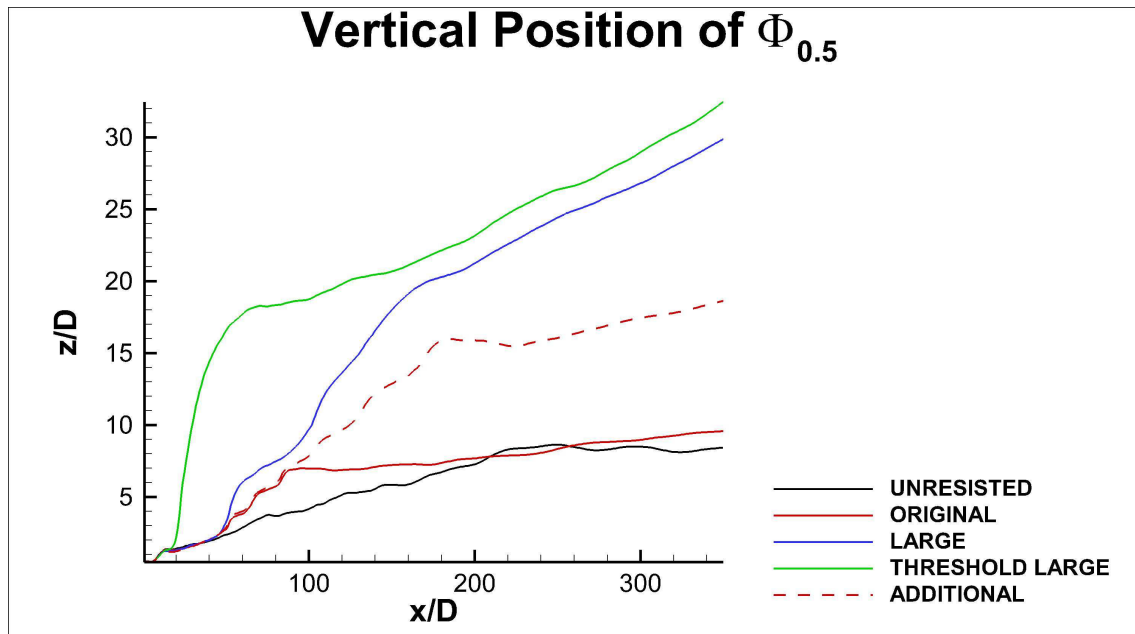
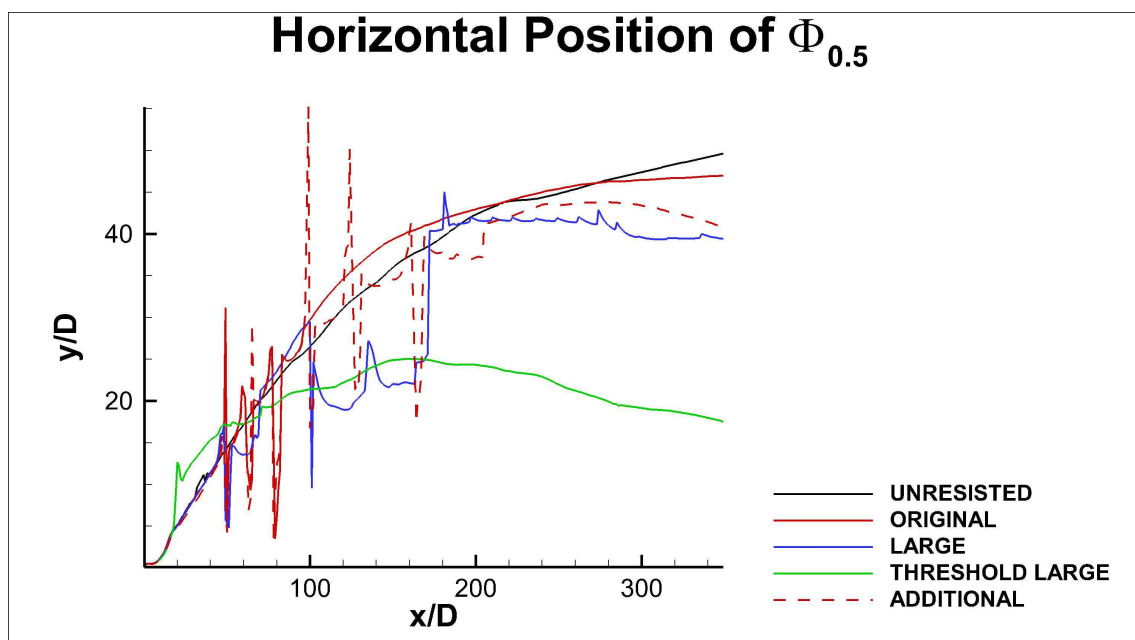


Figure D.9: Streamwise plot comparison between different baffle arrangements.

Appendix E

Final Results:

Vertical Profiles

The present Appendix includes all the final comparisons of the LES results with the wind tunnel measurements in terms of vertical velocity and concentration profiles at certain locations along the centreline. These results are discussed in chapter 10. Description of the investigated setup corresponding to the presented results is available in section 6.2.

E.1 Hot Unresisted 3DOJ

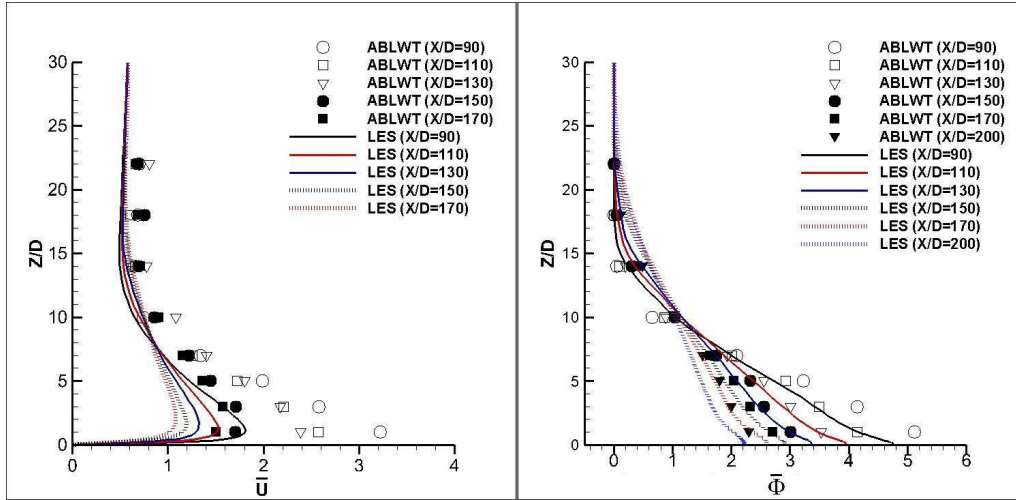


Figure E.1: Vertical profiles of $\bar{U}$ and $\bar{\Phi}$ at the farfield of the “UNRESISTED” 3DOJ. Case 5 results and the ABLWT experimental data are plotted at the stations where available.

E.2 Original Baffle Arrangement

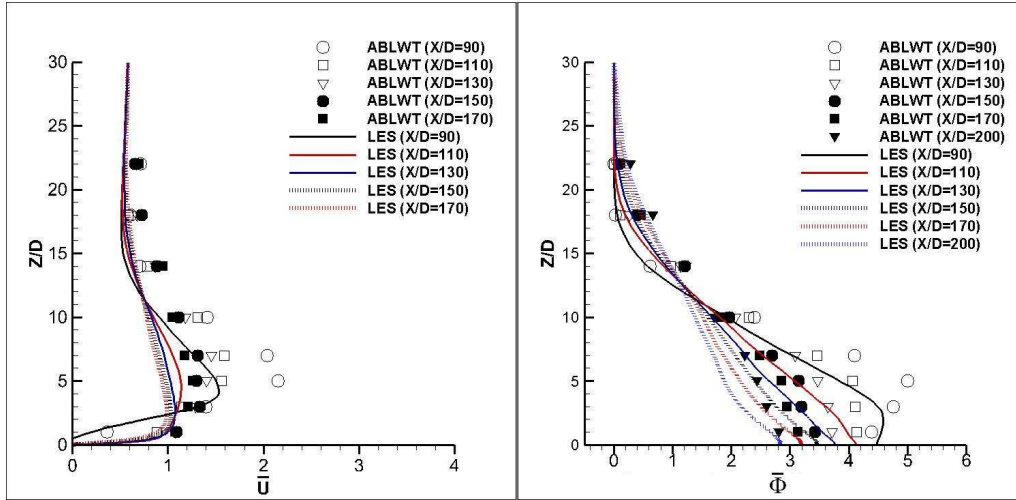


Figure E.2: Repetition of figure E.1 using Case 6 results “*designed baffles*”.

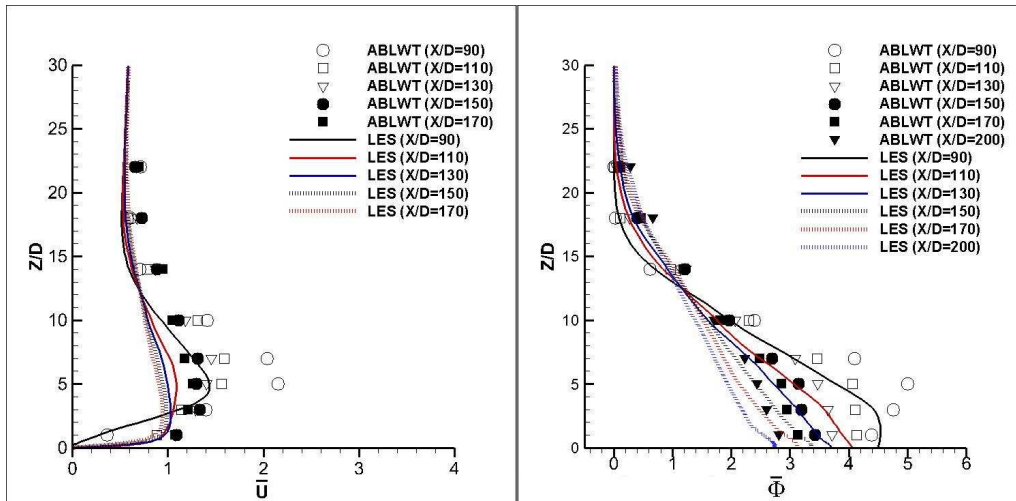


Figure E.3: Repetition of figure E.1 using Case 7 results “*modelled baffles*”.

E.3 Large Baffle Arrangement

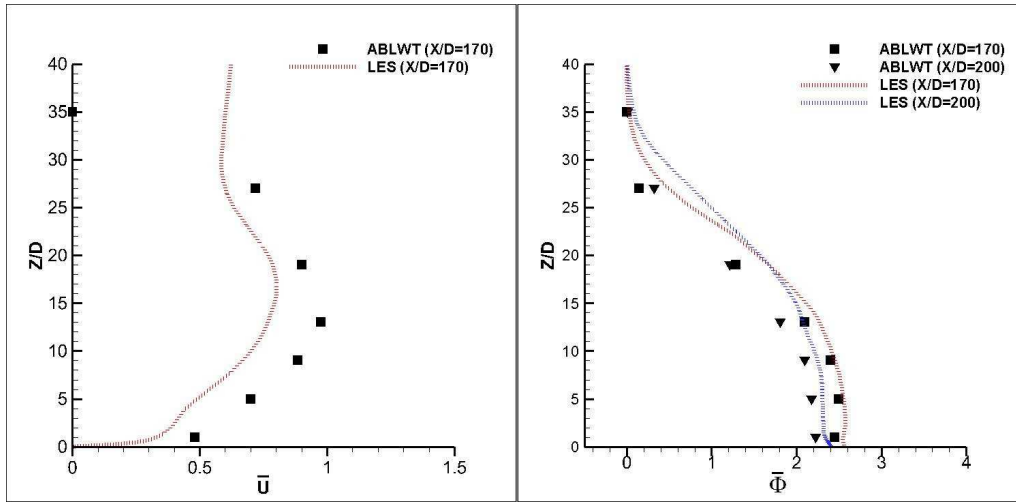


Figure E.4: Repetition of figure E.1 using Case 8 results “*designed baffles*”.

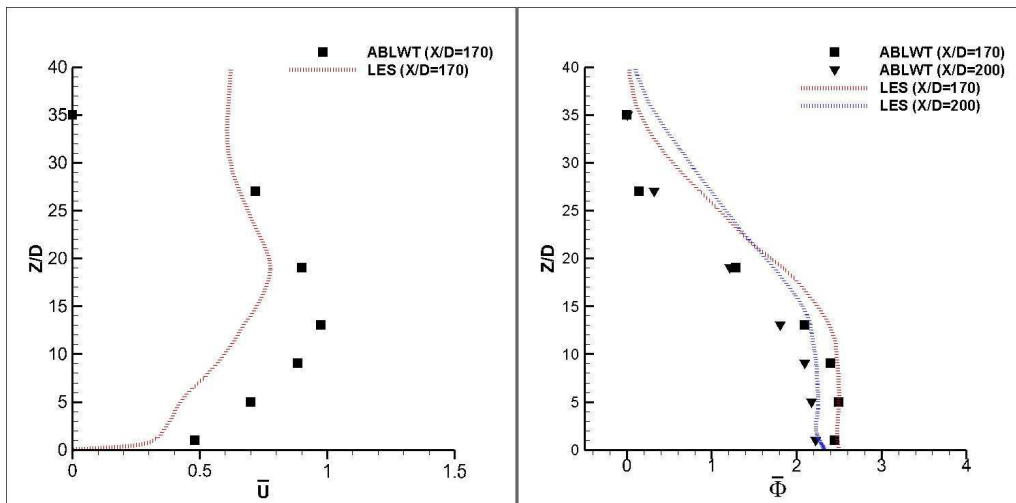


Figure E.5: Repetition of figure E.1 using Case 9 results “*modelled baffles*”.

E.4 Threshold-Large B. Arrangement

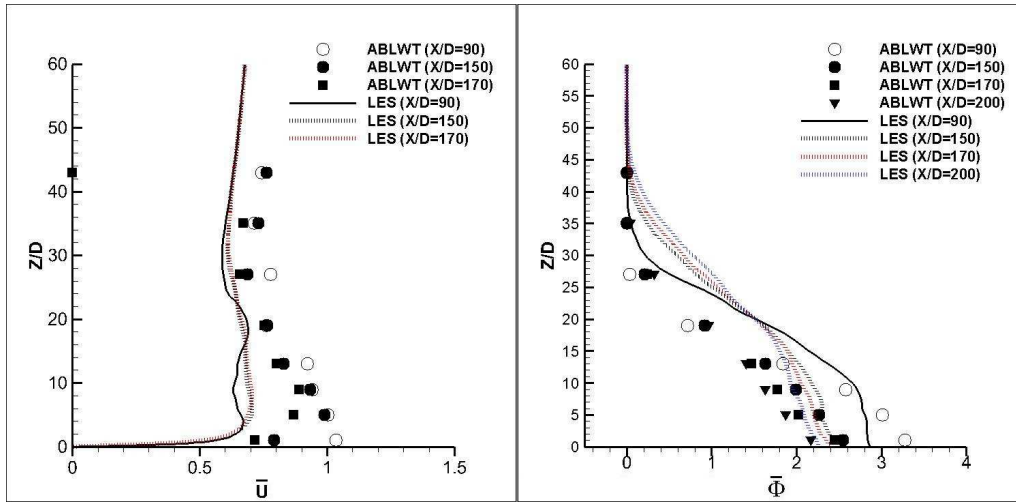


Figure E.6: Repetition of figure E.1 using Case 12 results "designed baffles".

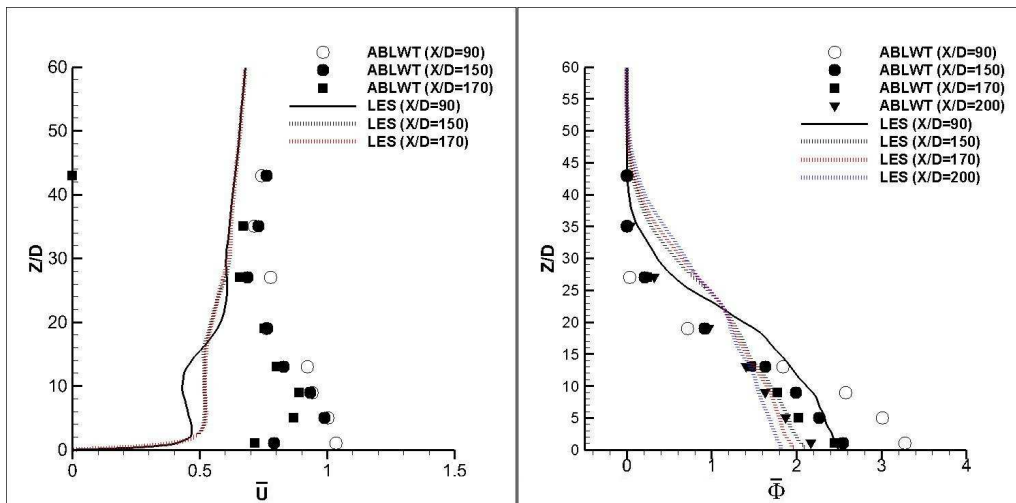


Figure E.7: Repetition of figure E.1 using Case 13 results "modelled baffles".

Appendix F

Final Results:

Axisymmetric-Mirrored Comparisons of Baffle Representation Method

F.1 Designed Vs Modelled Baffles

The present Appendix includes mirrored juxtapositions on the vertical and horizontal planes of numerical results of the baffle simulations using two different methods: a) the conventional approach “designed baffles”, and b) the virtual baffle approach “DRAMBI”. These results are discussed in chapter 10. Description of the investigated setup corresponding to these results is available in section 6.2.

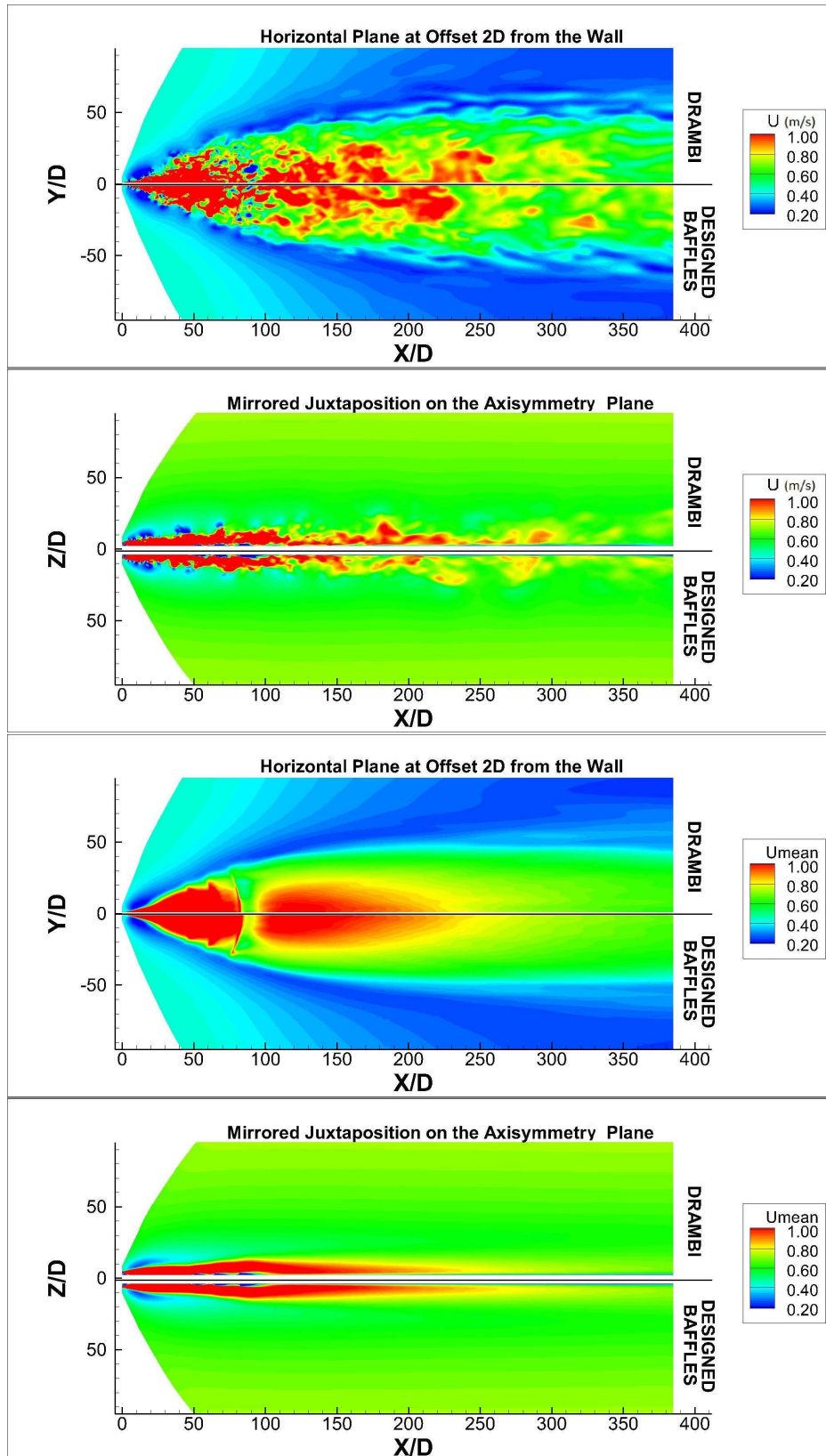


Figure F.1: Comparison of the “*designed baffles*” (Case 6) and the “*modelled baffles*” (Case 7) for the “*ORIGINAL*” baffle arrangement. Image setup repeated from figure 10.7.

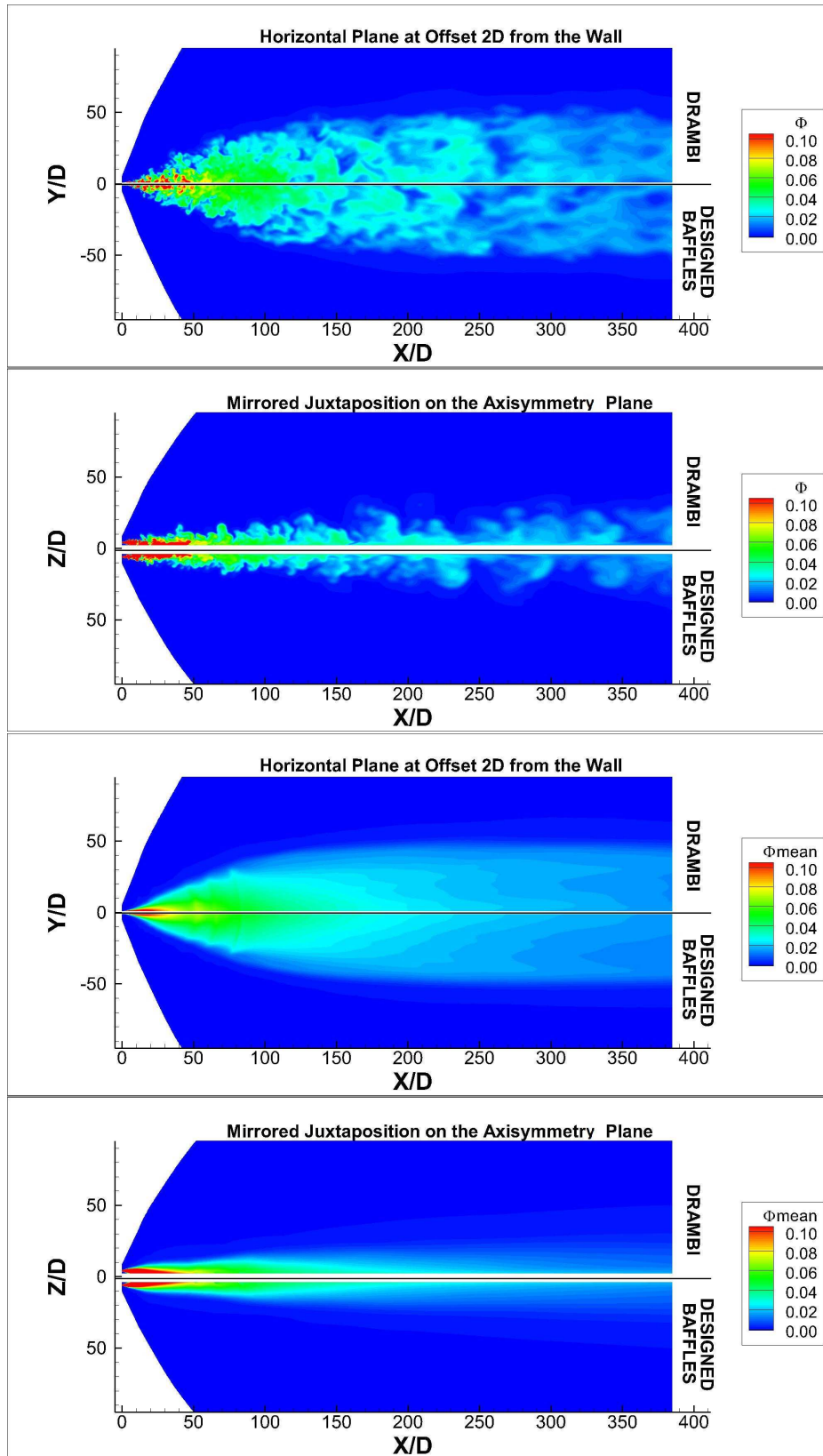


Figure F.2: Comparison of the “*designed baffles*” (Case 6) and the “*modelled baffles*” (Case 7) for the “*ORIGINAL*” baffle arrangement. Image setup repeated from figure 10.8.

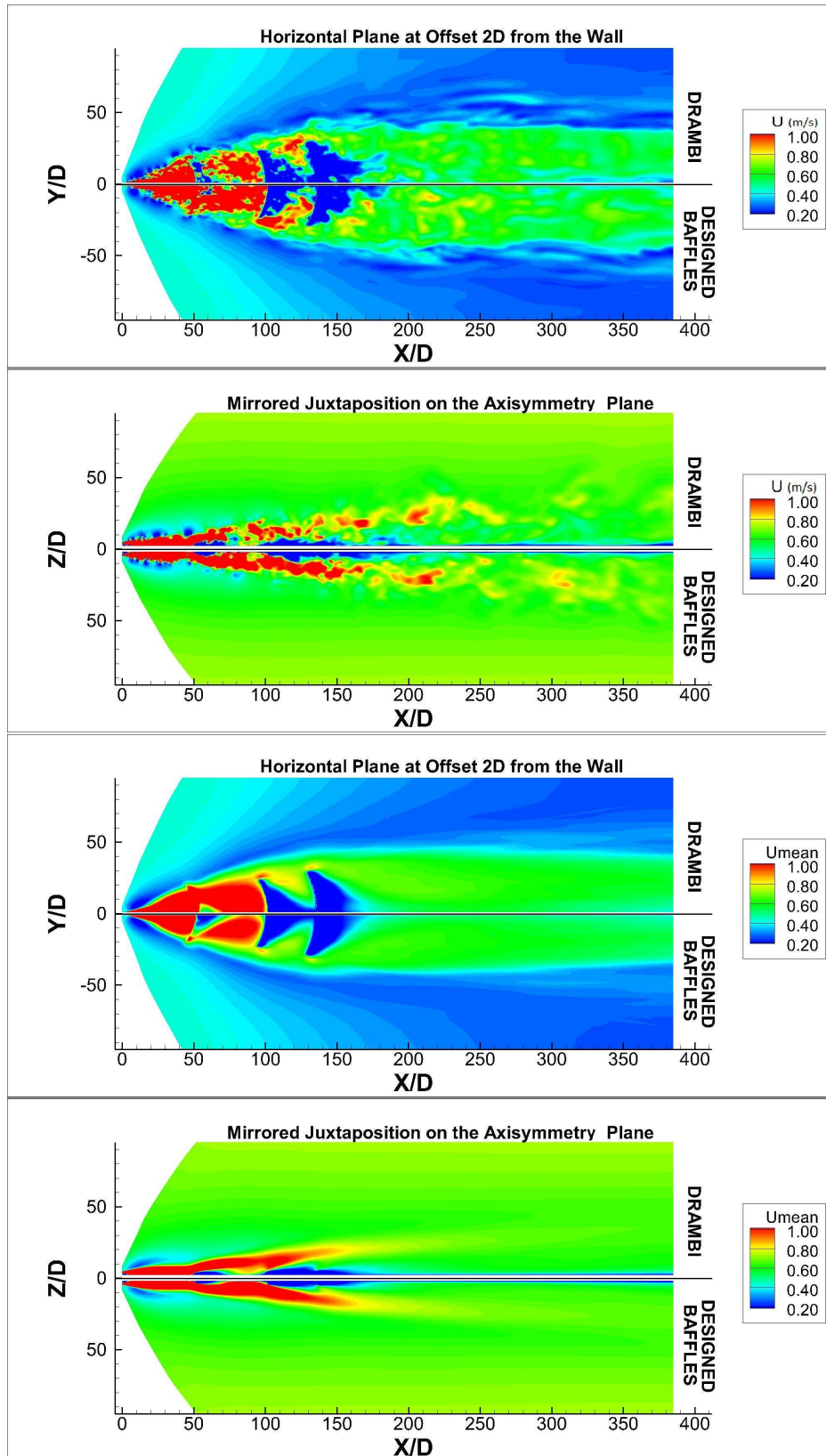


Figure F.3: Comparison of the “*designed baffles*” (Case 8) and the “*modelled baffles*” (Case 9) for the “*LARGE*” baffle arrangement. Image setup repeated from figure 10.7.

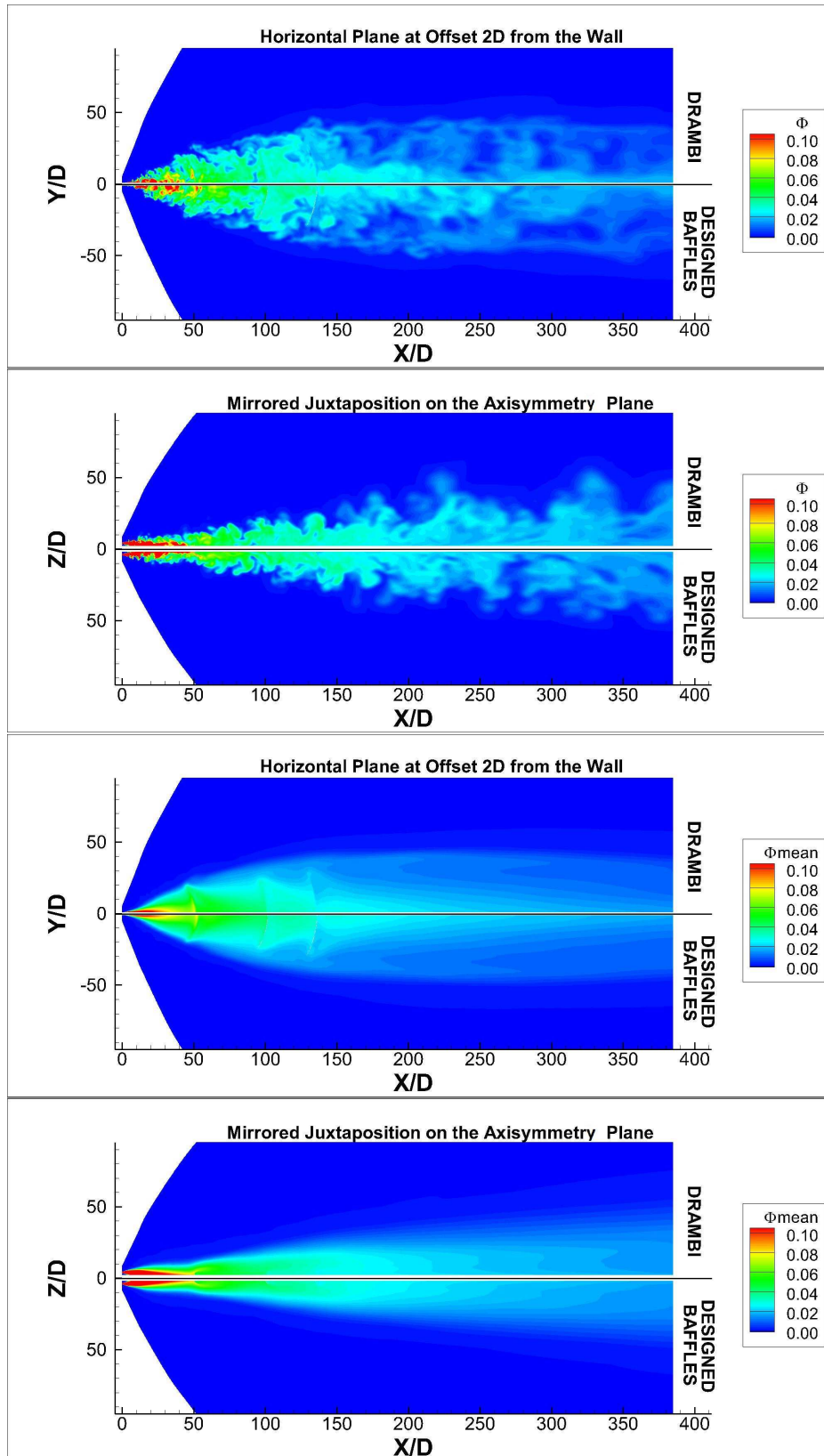


Figure F.4: Comparison of the “*designed baffles*” (Case 8) and the “*modelled baffles*” (Case 9) for the “*LARGE*” baffle arrangement. Image setup repeated from figure 10.8.

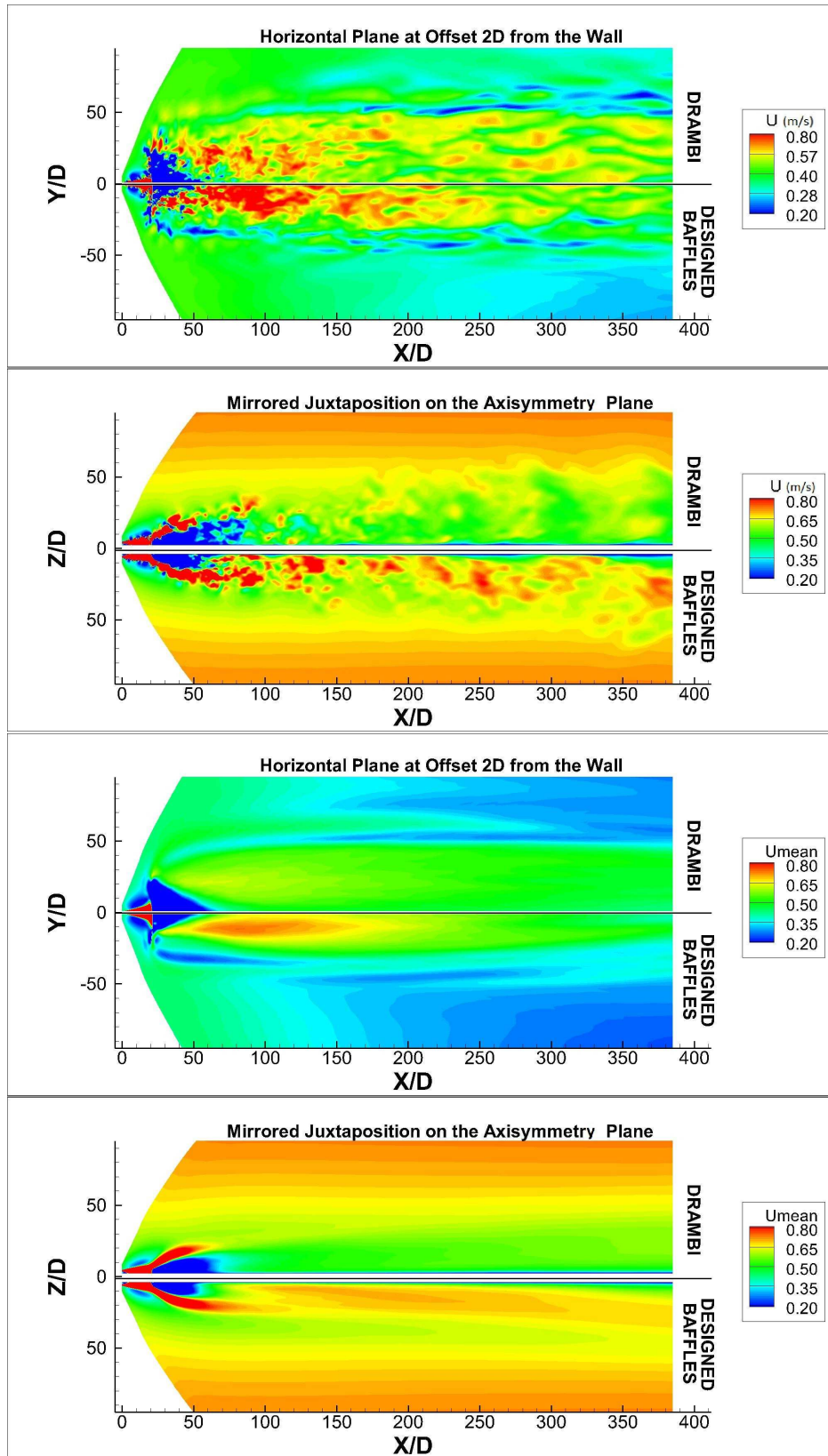


Figure F.5: Comparison of the “*designed baffles*” (Case 12) and the “*modelled baffles*” (Case 13) for the “*THRESHOLD-LARGE*” baffle arrangement. Image setup repeated from figure 10.7.

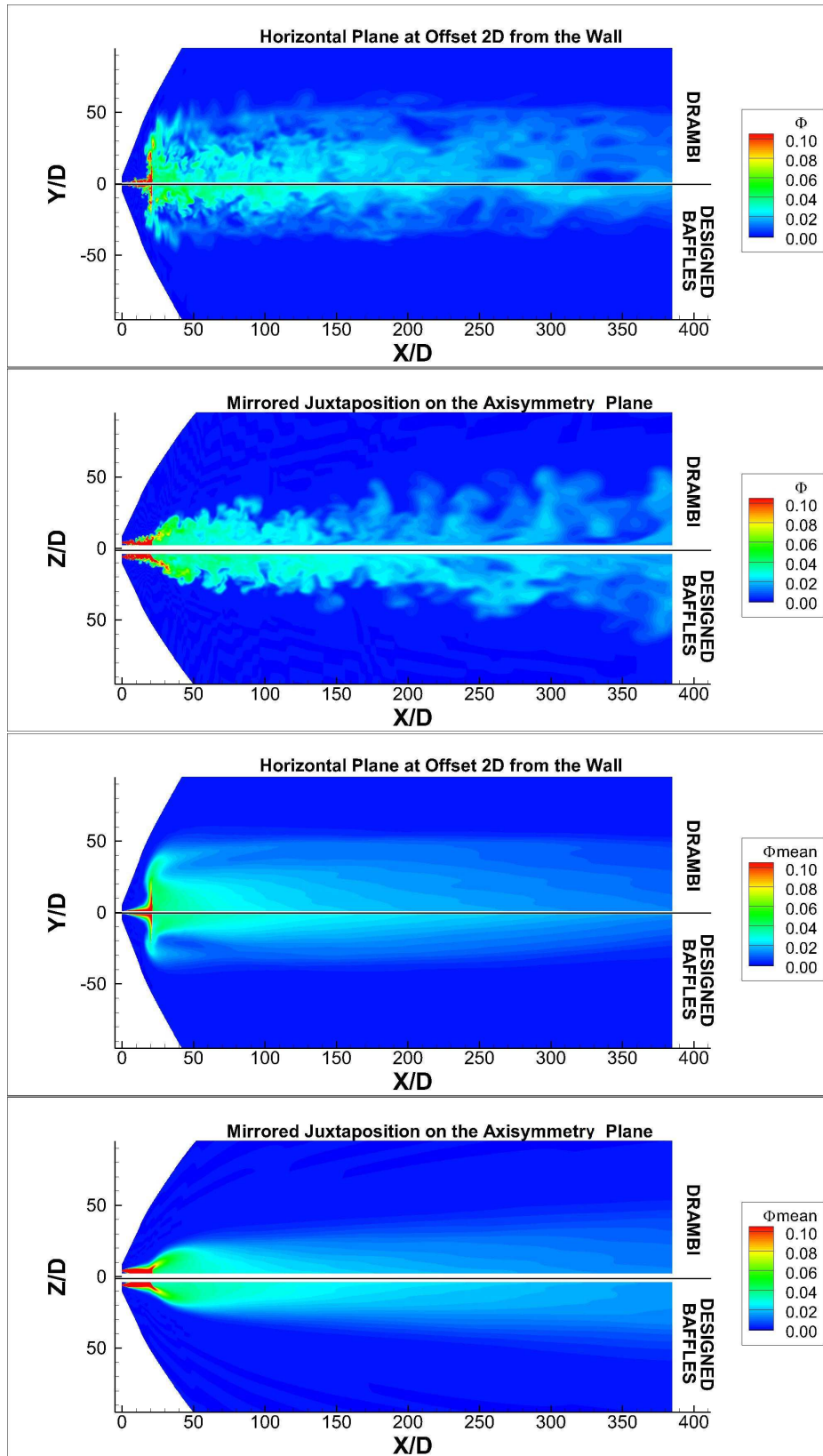


Figure F.6: Comparison of the “*designed baffles*” (Case 12) and the “*modelled baffles*” (Case 13) for the “*THRESHOLD-LARGE*” baffle arrangement. Image setup repeated from figure 10.8.

Appendix G

Code and Scripting

G.1 FLUENT User Defined Functions

The following pages include the complete set of user defined functions used throughout this thesis simulations, as included in the source file to be compiled by FLUENT before the simulation begins. This particular version was used for the simulation of Case 7. Refer to table 10.1 for details of this simulation.

Spanelis_PhD_Thesis_UDF.c

```

#include "udf.h"
int sample_1 = 0 ;
int sample_2 = 0 ;
/*-----*/
/*----- USER INPUT -----*/
/*-----*/

#define NUM_UDM 9
#define PI 3.1415926535897
static int last_ts = -1, Alter1=0, Alter2=0, Alter3=0, PsiID1 = 0, PsiID2 = 0, PsiID3 = 0;
static int DownCount1, DownCount2, DownCount3, UpCount1=0, UpCount2=0, UpCount3=0;
int curr_ts;

#define Jet_Offset 0.012 /* Jet Offaset (m) */
#define Djet 0.006 /* Jet Diameter (m) */
#define Ujet 25.4 /* Jet Bulk Velocity (m/s) */
#define CoFlowAngle (0.0*0.017453292519943) /* Wind Direction (radians)*/
#define ABLControlHEIGHT 0.04 /* Zr (m) */
#define ABLControlVelocity 0.5 /* Ur (m) */
#define Jet_Temperature 412.00 /* (K) */
#define Ambient_Temperature 290.15 /* (K) */

#define angle1 (40*0.017453292519943) /* Baffle-1 Slope (radians)*/
#define angle2 (50*0.017453292519943) /* Baffle-2 Slope (radians)*/
#define angle3 (60*0.017453292519943) /* Baffle-3 Slope (radians)*/

#define B_Centre_1 (20*Djet) /* Baffle-1 Position of the Centre or Radius (m)*/
#define B_Centre_2 (20*Djet) /* Baffle-2 Position of the Centre or Radius (m)*/
#define B_Centre_3 (20*Djet) /* Baffle-3 Position of the Centre or Radius (m)*/

#define Rlmin_D 30.0 /* -----*/
#define Rlmax_D 31.0263 /* */
#define R2min_D 44.42 /* */
#define R2max_D 45.4553 /* */
#define R3min_D 61.7 /* */
#define R3max_D 62.6586 /* */
#define Ylmin_D -19.08 /* */
#define Ylmax_D 19.08 /* Boundaries of Vc in DRAMBI */
#define Y2min_D -22.75 /* */
#define Y2max_D 22.75 /* */
#define Y3min_D -28.35 /* */
#define Y3max_D 28.35 /* */
#define HEIGHT1_D -1.25 /* */
#define HEIGHT2_D -0.9166666 /* */
#define HEIGHT3_D -0.558329 /* -----*/

#define DRAMBI_UO 0.0

#define Fd_Max (1.2 * (Ujet * Ujet) * PI * (Djet * Djet) * 0.25 * 0.5) /* Initial prediction of
the force to help DRA: rho * (U^2) * PI * (D^2)/4 * 50% */
#define Fd_Min (Fd_Max/500)

#define DeltaFMax (Fd_Max/10)
#define DFMin (-1*DeltaFMax)
#define DeltaF0Max (Fd_Max/10)
#define DeltaF0Min (Fd_Max/1000)

real RAD, RADscaled, Factor_bi, THETA;
real Rlmin = Rlmin_D * Djet, Rlmax = Rlmax_D * Djet, R2min = R2min_D * Djet, R2max = R2max_D *
Djet, R3min = R3min_D * Djet, R3max = R3max_D * Djet, Ylmin = Ylmin_D * Djet, Ylmax = Ylmax_D *
Djet, Y2min = Y2min_D * Djet, Y2max = Y2max_D * Djet, Y3min = Y3min_D * Djet, Y3max = Y3max_D *
Djet, HEIGHT1 = HEIGHT1_D * Djet, HEIGHT2 = HEIGHT2_D * Djet, HEIGHT3 = HEIGHT3_D * Djet;
real VC01, VC02, VC03, VC1, VC2, VC3;
real lambdal=1.0, lambda2=1.0, lambda3=1.0, sigma1=0.5, sigma2=0.5, sigma3=0.5;
real Psi1, Psi2, Psi3, Psi1m1=1.0, Psi2m1=1.0, Psi3m1=1.0, Psi1m2=1.0, Psi2m2=1.0, Psi3m2=1.0,
Psi1m3=1.0, Psi2m3=1.0, Psi3m3=1.0, alpha1, alpha2, alpha3;
real DeltaF_1= DeltaFMax, DeltaF_2=DeltaFMax, DeltaF_3=DeltaFMax;
real DeltaF0_1=(DeltaF0Max-DeltaF0Min)/10, DeltaF0_2=(DeltaF0Max-DeltaF0Min)/10, DeltaF0_3=(
DeltaF0Max-DeltaF0Min)/10;
real Fd1=(Fd_Max-Fd_Min)/2, Fd2=(Fd_Max-Fd_Min)/2, Fd3=(Fd_Max-Fd_Min)/2;

```

```

Spanelis_PhD_Thesis_UDF.c

/*=====*/
/*==  DOMAIN INITIALIZATION FUNCTION ==*/
/*=====*/

DEFINE_INIT(my_init_func,d)
{
    real x[ND_ND];
    real z;
    real wall_distance;
    cell_t c;
    Thread *t;

    thread_loop_c(t,d)
    {
        begin_c_loop_all(c,t)
        {
            C_CENTROID(x,c,t);
            z = x[2];
            wall_distance = z+Jet_Offset;
            C_U(c,t) = cos(CoFlowAngle)*ABLControlVelocity*pow((wall_distance/ABLControlHEIGHT),(1.0/7.0));
            C_V(c,t) = -1*sin(CoFlowAngle)*ABLControlVelocity*pow((wall_distance/ABLControlHEIGHT),(1.0/7.0));
        }
        end_c_loop_all(c,t)
    }
}

/*=====*/
/*==  INLET PROFILES  ==*/
/*=====*/

DEFINE_PROFILE(inlet_x_velocity,thread,position)
{
    real R_in = Djet/2;
    real R;
    real x[ND_ND];
    real z;
    real y;
    real wall_distance;
    face_t f;

    begin_f_loop(f,thread)
    {
        F_CENTROID(x,f,thread);
        y = x[1];
        z = x[2];
        wall_distance = z+Jet_Offset;
        R = pow((z*z+y*y),0.5);

        if (R < (R_in*1.01))
        {
            F_PROFILE(f,thread,position) = Ujet*1.28*pow((1.0-R/R_in),(1.0/7.0));
        }
        else
        {
            F_PROFILE(f,thread,position) = cos(CoFlowAngle)*ABLControlVelocity*pow((wall_distance/ABLControlHEIGHT),(1.0/7.0));
        }
    }
    end_f_loop(f,thread)
}

DEFINE_PROFILE(inlet_y_velocity,thread,position)
{
    real R_in = Djet/2;
    real R;
    real x[ND_ND];
    real z;
    real y;
    real wall_distance;
    face_t f;

```

Spanelis_PhD_Thesis_UDF.c

```

begin_f_loop(f,thread)
{
    F_CENTROID(x,f,thread);
    y = x[1];
    z = x[2];
    wall_distance = z+Jet_Offset;
    R = pow((z*z+y*y),0.5);

    if (R < (R_in*1.01))
    {
        F_PROFILE(f,thread,position) = 0.0;
    }
    else
    {
        F_PROFILE(f,thread,position) = -1*sin(CoFlowAngle)*ABLControlVelocity*pow((
            wall_distance/ABLControlHEIGHT), (1.0/7.0));
    }
}
end_f_loop(f,thread)
}

DEFINE_PROFILE(inlet_temperature,thread,position)
{
    real R_in = Djet/2;
    real R;
    real x[ND_ND];
    real z;
    real y;
    face_t f;

    begin_f_loop(f,thread)
    {
        F_CENTROID(x,f,thread);
        y = x[1];
        z = x[2];
        R = pow((z*z+y*y),0.5);

        if (R < (R_in*1.01))
        {
            F_PROFILE(f,thread,position) = Jet_Temperature;
        }
        else
        {
            F_PROFILE(f,thread,position) = Ambient_Temperature;
        }
    }
    end_f_loop(f,thread)
}

DEFINE_PROFILE(scalar_profile,thread,position)
{
    real R_in = Djet/2;
    real R;
    real x[ND_ND];
    real z;
    real y;
    face_t f;

    begin_f_loop(f,thread)
    {
        F_CENTROID(x,f,thread);
        y = x[1];
        z = x[2];
        R = pow((z*z+y*y),0.5);
        if (R < (R_in*1.01))
        {
            F_PROFILE(f,thread,position) = 1.0;
        }
        else
        {

```

```

                                Spanelis_PhD_Thesis_UDF.c

        if (R > (R_in*1.01))
        {
            F_PROFILE(f,thread,position) = 0.0;
        }
        else
        {
            F_PROFILE(f,thread,position) = 0.5;
        }
    }
}
end_f_loop(f,thread)
}

DEFINE_PROFILE(visual_scalar_profile,thread,position)
{
    real R_in = Djet/2;
    real R;
    real x[ND_ND];
    real z;
    real y;
    face_t f;

    begin_f_loop(f,thread)
    {
        F_CENTROID(x,f,thread);
        y = x[1];
        z = x[2];
        R = pow((z*z+y*y),0.5);
        if (z > 0.00002)
        {
            if (R < (R_in*1.01))
            {
                F_PROFILE(f,thread,position) = 2.0;
            }
            else
            {
                if (R > (R_in*1.01))
                {
                    F_PROFILE(f,thread,position) = 0.0;
                }
                else
                {
                    F_PROFILE(f,thread,position) = 1.0;
                }
            }
        }
        else
        {
            if (z < -0.00002)
            {
                F_PROFILE(f,thread,position) = 0.0;
            }
            else
            {
                if (R > (R_in*1.01))
                {
                    F_PROFILE(f,thread,position) = 0.0;
                }
                else
                {
                    F_PROFILE(f,thread,position) = 1.0;
                }
            }
        }
    }
    end_f_loop(f,thread)
}

/*=====*/
/*=      SECURE UDM LOCATIONS      =*/
/*=      ON LOADING                =*/
/*=====*/

```



```

Spanelis_PhD_Thesis_UDF.c

static int udm_offset = UDM_UNRESERVED;
DEFINE_EXECUTE_ON_LOADING(on_loading, libname)
{
    if (udm_offset == UDM_UNRESERVED)
    {
        udm_offset = Reserve_User_Memory_Vars(NUM_UDM);
    }
    if (udm_offset == UDM_UNRESERVED)
    {
        udm_offset = 0;
    }
    # if !RP_NODE
        Message("\nYou need to define up to %d extra UDMs in GUI and then reload current library %s\n",
            NUM_UDM, libname);
        Message("\nPresently udm_offset is set to zero");
    # endif
    }
    else
    {
    # if !RP_NODE
        Message("%d UDMs have been reserved by the current library %s\n", NUM_UDM, libname);
    # endif
    }
    # if !RP_NODE
        Message("\nUDM Offset for Current Loaded Library = %d\n", udm_offset);

        FILE *fp;
        fp = fopen("Baffle1.txt", "a");
        fprintf(fp, "%s %s %s %s %s %s %s\n", "CURRENT_TIME", "Fd1", "Psi1", "sigma1", "lambda1",
            "DeltaF0_1", "DeltaF_1");
        fclose(fp);

        fp = fopen("Baffle2.txt", "a");
        fprintf(fp, "%s %s %s %s %s %s %s\n", "CURRENT_TIME", "Fd2", "Psi2", "sigma2", "lambda2",
            "DeltaF0_2", "DeltaF_2");
        fclose(fp);

        fp = fopen("Baffle3.txt", "a");
        fprintf(fp, "%s %s %s %s %s %s %s\n", "CURRENT_TIME", "Fd3", "Psi3", "sigma3", "lambda3",
            "DeltaF0_3", "DeltaF_3");
        fclose(fp);
    # endif
    }

/*=====*/
/*===          ===*/
/*===  INITIALIZE THE UDMs and UDSs  ===*/
/*=====*/

DEFINE_ON_DEMAND(initialize_fcv)
{
    Domain *d;
    Thread *ct;
    cell_t c;
    d=Get_Domain(1);
    real x[ND_ND];
    VC01 = 0.0;
    VC02 = 0.0;
    VC03 = 0.0;

    thread_loop_c(ct,d)
    {
        begin_c_loop(c,ct)
        C_CENTROID(x,c,ct);

        RAD = sqrt(pow((x[0]-B_Centre_1),2)+(pow(x[1],2))); /* 1st Baffle Radial_Coordinate */
        if ( RAD>=Rlmin && x[0]>=B_Centre_1 && RAD<=Rlmax && x[2]<=HEIGHT1 && x[1]>=Ylmin && x[1]<=
            Ylmax)

```



```

Spanelis_PhD_Thesis_UDF.c

{
    VC01 += C_VOLUME(c,ct);
}

RAD = sqrt(pow((x[0]-B_Centre_2),2)+(pow(x[1],2))); /* 2nd Baffle Radial_Coordinate */
if ( RAD>=R2min && x[0]>=B_Centre_2 && RAD<=R2max && x[2]<=HEIGHT2 && x[1]>=Y2min && x[1]<=
Y2max)
{
    VC02 += C_VOLUME(c,ct);
}

RAD = sqrt(pow((x[0]-B_Centre_3),2)+(pow(x[1],2))); /* 3rd Baffle Radial_Coordinate */
if ( RAD>=R3min && x[0]>=B_Centre_3 && RAD<=R3max && x[2]<=HEIGHT3 && x[1]>=Y3min && x[1]<=
Y3max)
{
    VC03 += C_VOLUME(c,ct);
}

end_c_loop(c,ct)
}
# if RP_NODE /* Perform node synchronized actions here.
               Does nothing in Serial */
VC01 = PRF_GRSUM1(VC01);
VC02 = PRF_GRSUM1(VC02);
VC03 = PRF_GRSUM1(VC03);
# endif

}

DEFINE_ON_DEMAND(initialize_udms)
{
    Domain *d;
    Thread *ct;
    cell_t c;
    d=Get_Domain(1);
    int i;

    for (i=0;i<NUM_UDM;i++)
    {
        thread_loop_c(ct,d)
        {
            begin_c_loop(c,ct)
            {
                C_UDMI(c,ct,udm_offset+i)=0.0;
            }
            end_c_loop(c,ct)
        }
    }

    sample_1 = 0 ;
    sample_2 = 0 ;
}

DEFINE_ON_DEMAND(initialize_level_1)
{
    Domain *d;
    Thread *ct;
    cell_t c;
    d=Get_Domain(1);
    thread_loop_c(ct,d)
    {
        begin_c_loop(c,ct)
        {
            C_UDMI(c,ct,udm_offset+0)=0.0;
            C_UDMI(c,ct,udm_offset+1)=0.0;
            C_UDMI(c,ct,udm_offset+4)=0.0;
            C_UDMI(c,ct,udm_offset+5)=0.0;
        }
        end_c_loop(c,ct)
    }

    sample_1 = 0 ;
}

DEFINE_ON_DEMAND(initialize_level_2)
{

```

```

                                Spanelis_PhD_Thesis_UDF.c

Domain *d;
Thread *ct;
cell_t c;
d=Get_Domain(1);
  thread_loop_c(ct,d)
  {
    begin_c_loop(c,ct)
    {
      C_UDMI(c,ct,udm_offset+2)=0.0;
      C_UDMI(c,ct,udm_offset+3)=0.0;
      C_UDMI(c,ct,udm_offset+6)=0.0;
      C_UDMI(c,ct,udm_offset+7)=0.0;
    }
    end_c_loop(c,ct)
  }
sample_2 = 0 ;
}

/*=====*/
/*==      REPORT COUNTERS TO TERMINAL      ==*/
/*==      AND SAVE TO DATA FILE          ==*/
/*=====*/

DEFINE_EXECUTE_AT_END(report_counters)
{
  #if !RP_NODE    /* SERIAL or HOST */
  Message("=====\n");
  Message("= Sample Size Counted in Level 1 Averaging: %06d ", sample_1); Message(" ==\n");
  Message("=====\n");
  Message("= Sample Size Counted in Level 2 Averaging: %06d ", sample_2); Message(" ==\n");
  Message("=====\n");
  #endif
}

/*-----*/
/*-----LEVEL_1-----*/
/*-----*/
DEFINE_RW_FILE(writer_1,fp)
{
  #if !RP_NODE    /* SERIAL or HOST */
  printf("Writing counter 1 to data file...\n");
  fprintf(fp,"%d",sample_1);
  #endif
}
DEFINE_RW_FILE(reader_1,fp)
{
  #if !RP_NODE
  printf("Reading counter 1 data From data file...\n");
  fscanf(fp,"%d",&sample_1);
  #endif
}
/*-----*/
/*-----LEVEL_2-----*/
/*-----*/
DEFINE_RW_FILE(writer_2,fp)
{
  #if !RP_NODE    /* SERIAL or HOST */
  printf("Writing counter 2 to data file...\n");
  fprintf(fp,"%d",sample_2);
  #endif
}
DEFINE_RW_FILE(reader_2,fp)
{
  #if !RP_NODE
  printf("Reading counter 2 data From data file...\n");
  fscanf(fp,"%d",&sample_2);
  #endif
}

/*=====*/

```

```

                                Spanelis_PhD_Thesis_UDF.c

/*==          CALCULATE AVERAGES          ==*/
/*=====*/

DEFINE_EXECUTE_AT_END(level_1)
{
    Domain *d;
    Thread *ct;
    cell_t c;
    d=Get_Domain(1);
    host_to_node_int_1(sample_1);
    sample_1 = sample_1+1;
    thread_loop_c(ct,d)
    {
        begin_c_loop(c,ct)
        {
            C_UDMI(c,ct,udm_offset) = C_UDSI(c,ct,0)+C_UDMI(c,ct,udm_offset); /* PASSIVE
            SCALAR SUM */
            C_UDMI(c,ct,udm_offset+1) = C_UDMI(c,ct,udm_offset)/sample_1 ; /* PASSIVE
            SCALAR AVERAGE */

            C_UDMI(c,ct,udm_offset+4) = C_UDSI(c,ct,1)+C_UDMI(c,ct,udm_offset+4); /* PASSIVE
            SCALAR 2 SUM */
            C_UDMI(c,ct,udm_offset+5) = C_UDMI(c,ct,udm_offset+4)/sample_1 ; /* PASSIVE
            SCALAR 2 AVERAGE */
        }
        end_c_loop(c,ct)
    }
}

/*=====*/
/*==          CALCULATE HIGHER ORDER STATISTICS          ==*/
/*=====*/
DEFINE_EXECUTE_AT_END(level_2)
{
    Domain *d;
    Thread *ct;
    cell_t c;
    d=Get_Domain(1);
    host_to_node_int_1(sample_2);
    sample_2 = sample_2+1;
    thread_loop_c(ct,d)
    {
        begin_c_loop(c,ct)
        {
            C_UDMI(c,ct,udm_offset+2) = pow(C_UDMI(c,ct,udm_offset+1)-C_UDSI(c,ct,0),2)+C_UDMI(c,
            ct,udm_offset+2) ; /* PASSIVE SCALAR SUM OF FLUCTUATION SQUARES */
            C_UDMI(c,ct,udm_offset+3) = pow(C_UDMI(c,ct,udm_offset+2)/sample_2,0.5) ;
            /* PASSIVE SCALAR RMS */

            C_UDMI(c,ct,udm_offset+6) = pow(C_UDMI(c,ct,udm_offset+5)-C_UDSI(c,ct,1),2)+C_UDMI(c,
            ct,udm_offset+6) ; /* PASSIVE SCALAR 2 SUM OF FLUCTUATION SQUARES */
            C_UDMI(c,ct,udm_offset+7) = pow(C_UDMI(c,ct,udm_offset+6)/sample_2,0.5) ;
            /* PASSIVE SCALAR 2 RMS */
        }
        end_c_loop(c,ct)
    }
}

DEFINE_ADJUST(dragcoefficient,d)
{
    Thread *t;
    cell_t c;
    real sum_x_momentum1 = 0.0 ;
    real sum_x_momentum2 = 0.0 ;
    real sum_x_momentum3 = 0.0 ;
    VC1 = 0.0 ;
    VC2 = 0.0 ;
    VC3 = 0.0 ;
    real x[ND ND];
    real Zscaled = 0.0;

```

Spanelis_PhD_Thesis_UDF.c

```

#if !RP_HOST /* SERIAL or NODE */

thread_loop_c(t,d)
{
    begin_c_loop(c,t)

    C_CENTROID(x,c,t);

    RAD = sqrt(pow((x[0]-B_Centre_1),2)+(pow(x[1],2))); /* 1st Baffle
    Radial Coordinate */
    if ( RAD>=Rlmin && x[0]>=B_Centre_1 && RAD<=Rlmax && x[2]<=HEIGHT1 && x[1]>=Ylmin
    && x[1]<=Ylmax && C_U(c,t)>= DRAMBI_U0 )
    {
        Zscaled = (x[2]+Jet_Offset)/(HEIGHT1+Jet_Offset);
        RADscaled = (RAD-Rlmin)/(Rlmax-Rlmin);
        Factor_bi = ((RADscaled-Zscaled+1)/2);
        sum_x_momentum1 += C_R(c,t) * C_VOLUME(c,t) * C_U(c,t) * Factor_bi;
        VC1 += C_VOLUME(c,t);
    }

    RAD = sqrt(pow((x[0]-B_Centre_2),2)+(pow(x[1],2))); /* 2nd Baffle
    Radial Coordinate */
    if ( RAD>=R2min && x[0]>=B_Centre_2 && RAD<=R2max && x[2]<=HEIGHT2 && x[1]>=Y2min
    && x[1]<=Y2max && C_U(c,t)>= DRAMBI_U0 )
    {
        Zscaled = (x[2]+Jet_Offset)/(HEIGHT2+Jet_Offset);
        RADscaled = (RAD-R2min)/(R2max-R2min);
        Factor_bi = ((RADscaled-Zscaled+1)/2);
        sum_x_momentum2 += C_R(c,t) * C_VOLUME(c,t) * C_U(c,t) * Factor_bi;
        VC2 += C_VOLUME(c,t);
    }

    RAD = sqrt(pow((x[0]-B_Centre_3),2)+(pow(x[1],2))); /* 3rd Baffle
    Radial Coordinate */
    if ( RAD>=R3min && x[0]>=B_Centre_3 && RAD<=R3max && x[2]<=HEIGHT3 && x[1]>=Y3min
    && x[1]<=Y3max && C_U(c,t)>= DRAMBI_U0 )
    {
        Zscaled = (x[2]+Jet_Offset)/(HEIGHT3+Jet_Offset);
        RADscaled = (RAD-R3min)/(R3max-R3min);
        Factor_bi = ((RADscaled-Zscaled+1)/2);
        sum_x_momentum3 += C_R(c,t) * C_VOLUME(c,t) * C_U(c,t) * Factor_bi;
        VC3 += C_VOLUME(c,t);
    }
    end_c_loop(c,t)
}

# if RP_NODE /* Perform node synchronized actions here Does nothing in Serial */
sum_x_momentum1 = PRF_GRSUM1(sum_x_momentum1);
sum_x_momentum2 = PRF_GRSUM1(sum_x_momentum2);
sum_x_momentum3 = PRF_GRSUM1(sum_x_momentum3);
VC1 = PRF_GRSUM1(VC1);
VC2 = PRF_GRSUM1(VC2);
VC3 = PRF_GRSUM1(VC3);
# endif /* RP_NODE */

thread_loop_c(t,d)
{
    begin_c_loop(c,t)

    C_CENTROID(x,c,t);

    RAD = sqrt(pow((x[0]-B_Centre_1),2)+(pow(x[1],2))); /* 1st Baffle
    Radial Coordinate */
    if ( RAD>=Rlmin && x[0]>=B_Centre_1 && RAD<=Rlmax && x[2]<=HEIGHT1 && x[1]>=Ylmin
    && x[1]<=Ylmax && C_U(c,t)>= DRAMBI_U0 )
    {
        Zscaled = (x[2]+Jet_Offset)/(HEIGHT1+Jet_Offset);
        RADscaled = (RAD-Rlmin)/(Rlmax-Rlmin);
        Factor_bi = ((RADscaled-Zscaled+1)/2);
    }
}

```

```

Spanelis_PhD_Thesis_UDF.c

C_UDMI(c,t,udm_offset+8) = C_R(c,t)*C_VOLUME(c,t)*C_U(c,t)*Factor_bi/
sum_x_momentum1;
}

RAD = sqrt(pow((x[0]-B_Centre_2),2)+(pow(x[1],2))); /* 2nd Baffle
Radial_Coordinate */
if ( RAD>=R2min && x[0]>=B_Centre_2 && RAD<=R2max && x[2]<=HEIGHT2 && x[1]>=Y2min
&& x[1]<=Y2max && C_U(c,t)>= DRAMBI_U0 )
{
Zscaled = (x[2]+Jet_Offset)/(HEIGHT2+Jet_Offset);
RADscaled = (RAD-R2min)/(R2max-R2min);
Factor_bi = ((RADscaled-Zscaled+1)/2);
C_UDMI(c,t,udm_offset+8) = C_R(c,t)*C_VOLUME(c,t)*C_U(c,t)*Factor_bi/
sum_x_momentum2;
}

RAD = sqrt(pow((x[0]-B_Centre_3),2)+(pow(x[1],2))); /* 3rd Baffle
Radial_Coordinate */
if (RAD>=R3min && x[0]>=B_Centre_3 && RAD<=R3max && x[2]<=HEIGHT3 && x[1]>=Y3min
&& x[1]<=Y3max && C_U(c,t)>= DRAMBI_U0 )
{
Zscaled = (x[2]+Jet_Offset)/(HEIGHT3+Jet_Offset);
RADscaled = (RAD-R3min)/(R3max-R3min);
Factor_bi = ((RADscaled-Zscaled+1)/2);
C_UDMI(c,t,udm_offset+8) = C_R(c,t)*C_VOLUME(c,t)*C_U(c,t)*Factor_bi/
sum_x_momentum3;
}
end_c_loop(c,t)
}

/*=====
/*===== EVALUATE FORCING (DRA) =====
/*=====

curr_ts = N_TIME;
if ((last_ts != curr_ts) && (CURRENT_TIME > 0.005) )
{
Psi1 = (VC1/(VC01/2))-1.0 ;
Psi2 = (VC2/(VC02/2))-1.0 ;
Psi3 = (VC3/(VC03/2))-1.0 ;

alpha1 = tanh(Psi1*10)*lambda1;
alpha2 = tanh(Psi2*10)*lambda2;
alpha3 = tanh(Psi3*10)*lambda3;

/* ----- Baffle 1 -----*/
if ((Psi1*Psi1) >= 0.0)
{
if ((Psi1*Psi1) >= (Psi1m1*Psi1m1))
{
UpCount1=UpCount1+1;
if ((UpCount1 >= 5) && (Alter1 >= 5)) {DeltaF0_1 = DeltaF0_1*1.05; lambda1 = lambda1*1.05;}
DeltaF_1 = DeltaF0_1*alpha1;
if (PsiID1 == 1) {signal = signal/1.1; }
PsiID1 = 0;
}
else
{
if (((Psi1m1*Psi1m1) <= (Psi1m2*Psi1m2)) && (Alter1 >= 5))
{
DownCount1 = DownCount1+1;
if (DownCount1 == 5) {signal = signal*1.1; }
DeltaF_1 = (-DeltaF0_1)*alpha1*signal;
PsiID1 = 1;
}
else
{
DeltaF_1 = DeltaF_1 ;
PsiID1 = 2;
}
}
}
}
}

```


Spanelis_PhD_Thesis_UDF.c

```

else
{
    Alter1 = Alter1+1;
    DeltaF0_1 = DeltaF0_1/1.05;
    lambda1 = lambda1/1.05;
    DeltaF_1 = DeltaF0_1*alpha1;
    PsiID1 = 3;
    DownCount1 = 0;
    UpCount1 = 0;
}
/* ----- Baffle 2 -----*/
if ((Psi2m1*Psi2) >= 0.0)
{
    if ((Psi2*Psi2) >= (Psi2m1*Psi2m1))
    {
        UpCount2=UpCount2+1;
        if ((UpCount2 >= 5) && (Alter2 >= 5)) {DeltaF0_2 = DeltaF0_2*1.05; lambda2 = lambda2*1.05;}
        DeltaF_2 = DeltaF0_2*alpha2;
        if (PsiID2 == 1) {sigma2 = sigma2/1.1; }
        PsiID2 = 0;
    }
    else
    {
        if (((Psi2m1*Psi2m1) <= (Psi2m2*Psi2m2)) && (Alter2 >= 5))
        {
            DownCount2 = DownCount2+1;
            if (DownCount2 == 5) {sigma2 = sigma2*1.1; }
            DeltaF_2 = (-DeltaF0_2)*alpha2*sigma2;
            PsiID2 = 1;
        }
        else
        {
            DeltaF_2 = DeltaF_2 ;
            PsiID2 = 2;
        }
    }
}
else
{
    Alter2 = Alter2+1;
    DeltaF0_2 = DeltaF0_2/1.05;
    lambda2 = lambda2/1.05;
    DeltaF_2 = DeltaF0_2*alpha2;
    PsiID2 = 3;
    DownCount2 = 0;
    UpCount2 = 0;
}
/* ----- Baffle 3 -----*/
if ((Psi3m1*Psi3) >= 0.0)
{
    if ((Psi3*Psi3) >= (Psi3m1*Psi3m1))
    {
        UpCount3=UpCount3+1;
        if ((UpCount3 >= 5) && (Alter3 >= 5)) {DeltaF0_3 = DeltaF0_3*1.05; lambda3 = lambda3*1.05;}
        DeltaF_3 = DeltaF0_3*alpha3;
        if (PsiID3 == 1) {sigma3 = sigma3/1.1; }
        PsiID3 = 0;
    }
    else
    {
        if (((Psi3m1*Psi3m1) <= (Psi3m2*Psi3m2)) && (Alter3 >= 5))
        {
            DownCount3 = DownCount3+1;
            if (DownCount3 == 5) {sigma3 = sigma3*1.1; }
            DeltaF_3 = (-DeltaF0_3)*alpha3*sigma3;
            PsiID3 = 1;
        }
        else
        {
            DeltaF_3 = DeltaF_3 ;
            PsiID3 = 2;
        }
    }
}
}
else

```

```

                                Spanelis_PhD_Thesis_UDF.c

{
Alter3 = Alter3+1;
DeltaF0_3 = DeltaF0_3/1.05;
lambda3_ = lambda3/1.05;
DeltaF_3 = DeltaF0_3*alpha3;
PsiID3 = 3;
DownCount3 = 0;
UpCount3 = 0;
}
/* -----*/

Fd1 = Fd1+DeltaF_1;
Fd2 = Fd2+DeltaF_2;
Fd3 = Fd3+DeltaF_3;

if (Fd1 >= Fd_Max) {Fd1 = Fd_Max;}
if (Fd2 >= Fd_Max) {Fd2 = Fd_Max;}
if (Fd3 >= Fd_Max) {Fd3 = Fd_Max;}
if (Fd1 <= Fd_Min) {Fd1 = Fd_Min;}
if (Fd2 <= Fd_Min) {Fd2 = Fd_Min;}
if (Fd3 <= Fd_Min) {Fd3 = Fd_Min;}

Psilm3 = Psilm2;
Psi2m3 = Psi2m2;
Psi3m3 = Psi3m2;
Psilm2 = Psilm1;
Psi2m2 = Psi2m1;
Psi3m2 = Psi3m1;
Psilm1 = Psi1;
Psi2m1 = Psi2;
Psi3m1 = Psi3;

last_ts = curr_ts;
}
/*=====
=====*/

#endif /* !RP_HOST */

    node_to_host_real_3(Psi1,Psi2,Psi3);
    node_to_host_real_3(Fd1,Fd2,Fd3);
    node_to_host_real_3(sigma1, sigma2, sigma3);
    node_to_host_real_3(lambda1,lambda2,lambda3);
    node_to_host_real_3(DeltaF0_1,DeltaF0_2,DeltaF0_3);
    node_to_host_real_3(DeltaF_1,DeltaF_2,DeltaF_3);

#if RP_HOST
curr_ts = N_TIME;
if (last_ts != curr_ts)
{
    Message("Psi_1:%f, ", Psi1);
    Message("Psi_2:%f, ", Psi2);
    Message("Psi_3:%f \n", Psi3);
    Message("Fd1:%f, ", Fd1);
    Message("Fd2:%f, ", Fd2);
    Message("Fd3:%f \n", Fd3);
    Message("sigma1:%f, ", sigma1);
    Message("sigma2:%f, ", sigma2);
    Message("sigma3:%f \n", sigma3);
    Message("L_1:%f, ", lambda1);
    Message("L_2:%f, ", lambda2);
    Message("L_3:%f \n", lambda3);
    Message("DFo_1:%f, ", DeltaF0_1);
    Message("DFo_2:%f, ", DeltaF0_2);
    Message("DFo_3:%f \n", DeltaF0_3);
    Message("DF_1:%f, ", DeltaF_1);
    Message("DF_2:%f, ", DeltaF_2);
}
}

```

```

Spanelis_PhD_Thesis_UDF.c

Message("DF_3:%f \n", DeltaF_3);

FILE *fp;
fp = fopen("Baffle1.txt", "a");
fprintf(fp, "%f %f %f %f %f %f %f \n", CURRENT_TIME, Fd1, Psi1, sigma1, lambda1, DeltaF0_1,
DeltaF_1);
fclose(fp);

fp = fopen("Baffle2.txt", "a");
fprintf(fp, "%f %f %f %f %f %f %f \n", CURRENT_TIME, Fd2, Psi2, sigma2, lambda2, DeltaF0_2,
DeltaF_2);
fclose(fp);

fp = fopen("Baffle3.txt", "a");
fprintf(fp, "%f %f %f %f %f %f %f \n", CURRENT_TIME, Fd3, Psi3, sigma3, lambda3, DeltaF0_3,
DeltaF_3);
fclose(fp);

last_ts = curr_ts;
}
# endif /*RPHOST*/
}

DEFINE_SOURCE(xmom_source, c, t, dS, eqn)
{
    real x[ND_ND];
    C_CENTROID(x, c, t);

    #if !RP_HOST
    real SV = 0.0;

    /* 1st baffle array x-momentum */

    RAD = sqrt(pow((x[0]-B_Centre_1),2)+(pow(x[1],2)));
    if ( RAD>=R1min && x[0]>=B_Centre_1 && RAD<=R1max && x[2]<=HEIGHT1 && x[1]>=Y1min && x[1]<=
Y1max && C_U(c,t)>= DRAMBI_U0 )
    {
        SV = -1* Fd1 * C_UDMI(c,t,udm_offset+8) / C_VOLUME(c,t);
        dS[eqn] = 0.0;
        return SV;
    }

    /* 2nd baffle array x-momentum */

    RAD = sqrt(pow((x[0]-B_Centre_2),2)+(pow(x[1],2)));
    if ( RAD>=R2min && x[0]>=B_Centre_2 && RAD<=R2max && x[2]<=HEIGHT2 && x[1]>=Y2min && x[1]<=
Y2max && C_U(c,t)>= DRAMBI_U0 )
    {
        SV = -1* Fd2 * C_UDMI(c,t,udm_offset+8) / C_VOLUME(c,t);
        dS[eqn] = 0.0;
        return SV;
    }

    /* 3rd baffle array x-momentum */

    RAD = sqrt(pow((x[0]-B_Centre_3),2)+(pow(x[1],2)));
    if (RAD>=R3min && x[0]>=B_Centre_3 && RAD<=R3max && x[2]<=HEIGHT3 && x[1]>=Y3min && x[1]<=
Y3max && C_U(c,t)>= DRAMBI_U0 )
    {
        SV = -1* Fd3 * C_UDMI(c,t,udm_offset+8) / C_VOLUME(c,t);
        dS[eqn] = 0.0;
        return SV;
    }
    else

```



```

Spanelis_PhD_Thesis_UDF.c

{
    SV = 0.0;
    dS[eqn] = 0.0;
    return SV;
}
#endif
}

/* ===== */
/*== Evaluate Source Terms and Derivatives ==*/
/*== Based on the Waighting Factors ==*/
/* ===== */

DEFINE_SOURCE(zmom_source, c, t, dS, eqn)
{
    real x[ND_ND];
    C_CENTROID(x,c,t);

    #if !RP_HOST
    real SV = 0.0;

    /* 1st baffle array z-momentum */

    RAD = sqrt(pow((x[0]-B_Centre_1),2)+(pow(x[1],2)));
    if ( RAD>=R1min && x[0]>=B_Centre_1 && RAD<=R1max && x[2]<=HEIGHT1 && x[1]>=Y1min && x[1]<=
    Y1max && Psi1<=0.1 && C_U(c,t)>= DRAMBI_U0 )
    {
        THETA = asin(x[1]/RAD);
        SV = Psi1 * Fd1/cos(THETA) * tan(angle1) * C_UDMI(c,t,udm_offset+8) / C_VOLUME(c,t);
        dS[eqn] = 0.0;
        return SV;
    }

    /* 2nd baffle array z-momentum */

    RAD = sqrt(pow((x[0]-B_Centre_2),2)+(pow(x[1],2)));
    if ( RAD>=R2min && x[0]>=B_Centre_2 && RAD<=R2max && x[2]<=HEIGHT2 && x[1]>=Y2min && x[1]<=
    Y2max && Psi2<=0.1 && C_U(c,t)>= DRAMBI_U0 )
    {
        THETA = asin(x[1]/RAD);
        SV = Psi2 * Fd2/cos(THETA) * tan(angle2) * C_UDMI(c,t,udm_offset+8) / C_VOLUME(c,t);
        dS[eqn] = 0.0;
        return SV;
    }

    /* 3rd baffle array z-momentum */

    RAD = sqrt(pow((x[0]-B_Centre_3),2)+(pow(x[1],2)));
    if (RAD>=R3min && x[0]>=B_Centre_3 && RAD<=R3max && x[2]<=HEIGHT3 && x[1]>=Y3min && x[1]<=
    Y3max && Psi3<=0.1 && C_U(c,t)>= DRAMBI_U0 )
    {
        THETA = asin(x[1]/RAD);
        SV = Psi3 * Fd3/cos(THETA) * tan(angle3) * C_UDMI(c,t,udm_offset+8) / C_VOLUME(c,t);
        dS[eqn] = 0.0;
        return SV;
    }
    else
    {
        SV = 0.0;
        dS[eqn] = 0.0;
        return SV;
    }
}
#endif

}

DEFINE_SOURCE(ymom_source, c, t, dS, eqn)
{

```

```

Spanelis_PhD_Thesis_UDF.c

real x[ND_ND];
real Zscaled = 0.0;
C_CENTROID(x,c,t);

#ifdef RP_HOST
real SV = 0.0;

/* 1st baffle array y-momentum */

RAD = sqrt(pow((x[0]-B_Centre_1),2)+(pow(x[1],2)));
if ( RAD>=Rlmin && x[0]>=B_Centre_1 && RAD<=Rlmax && x[2]<=HEIGHT1 && x[1]>=Ylmin && x[1]<=
Ylmax && C_U(c,t)>= DRAMBI_U0 && Psi1<=0.1 )
{
THETA = asin(x[1]/RAD);
SV = -1 * Fd1 * tan(THETA) * C_UDMI(c,t,udm_offset+8) / C_VOLUME(c,t);
dS[eqn] = 0.0;
return SV;
}

/* 2nd baffle array y-momentum */

RAD = sqrt(pow((x[0]-B_Centre_2),2)+(pow(x[1],2)));
if ( RAD>=R2min && x[0]>=B_Centre_2 && RAD<=R2max && x[2]<=HEIGHT2 && x[1]>=Y2min && x[1]<=
Y2max && C_U(c,t)>= DRAMBI_U0 && Psi2<=0.1 )
{
THETA = asin(x[1]/RAD);
SV = -1* Fd2 * tan(THETA) * C_UDMI(c,t,udm_offset+8) / C_VOLUME(c,t);
dS[eqn] = 0.0;
return SV;
}

/* 3rd baffle array y-momentum */

RAD = sqrt(pow((x[0]-B_Centre_3),2)+(pow(x[1],2)));
if (RAD>=R3min && x[0]>=B_Centre_3 && RAD<=R3max && x[2]<=HEIGHT3 && x[1]>=Y3min && x[1]<=
Y3max && C_U(c,t)>= DRAMBI_U0 && Psi3<=0.1 )
{
THETA = asin(x[1]/RAD);
SV = -1* Fd3 * tan(THETA) * C_UDMI(c,t,udm_offset+8) / C_VOLUME(c,t);
dS[eqn] = 0.0;
return SV;
}
else
{
SV = 0.0;
dS[eqn] = 0.0;
return SV;
}
#endif
}

```

G.2 Tecplot Post-Processing Script

The present code is written in the Python-based scripting language of TecPlot 360. It performs volumetric operations within the three dimensional computational domain of *LES* in order to obtain the streamwise evolution of the mean flow characteristics. Explanation of this method, referred as advanced post-processing method, is presented in section 8.1.2.

```

Advanced_Post_Processing.mcr

#!MC 1200
# Created by Tecplot 360 build 12.0.0.3116

#          $!Loop 9          #multi-case post processing#

$!VarSet |CaseNr| = "7"
$!VarSet |XbyDSkip| = 1

$!NEWLAYOUT

# _____ ( USER INPUTS ) _____ #

$!VarSet |JetClearence| = 2                # in Jet Diameters
$!VarSet |NozzleSize| = 0.006              # in meters
$!VarSet |DomainSize| = 350                # in Jet Diameters
$!VarSet |MeshStaggeringOffset| = 0.015    # in Jet Diameters
$!VarSet |Ur| = 0.50                       # in meters per second
$!VarSet |Zr| = 0.04                       # in Jet Diameters
$!VarSet |ListOfVars| = "X" "Y" "Z" "x/D" "y/D" "z/D" "U" "V" "W" "URms" "VRms"
"WRms" "UMax" "UMaxPos" "UHalf" "UHalfPosVER" "UHalfPosHOR" "Phi" "PhiRms"
"PhiMax" "PhiMaxPos" "PhiHalfPosVER" "PhiHalfPosHOR" "UScaledJetDimension"
"PhiScaledJetDimension" "U/UMax" "V/UMax" "W/UMax" "Phi/PhiMax" "URms/UMax"
"VRms/UMax" "WRms/UMax" "PhiRms/PhiMax"
#-----#
#          $!VarSet |CaseNr| = (100+|Loop|)          #multi-case post processing#
#          $!VarSet |XbyDSkip| = 1                  #multi-case post processing#
#-----#

$!If |INBATCHMODE| == 0

#          $!PROMPTFORTEXTSTRING |CaseNr|
#          INSTRUCTIONS = "Type The Nr of the Case to Process"

#          $!PROMPTFORTEXTSTRING |XbyDSkip|
#          INSTRUCTIONS = "Ener Step Size (in x/D units)"

$!EndIf
$!VarSet |MFBD| =
'W:\ASTRAL_BACKUP\POST_PROCESSING\4_ABLWT\1_DATA_STORAGE\Case_|CaseNr|'
$!VarSet |WIND| = '{Uf}' = |Ur|*(((z|+|JetClearence|*|NozzleSize|+0.00001)
/|Zr|)**(1.0/7.0))'
$!VarSet |NrStations| = (int(|DomainSize|/|XbyDSkip|))

# _____ ( Load FLUENT data ) _____ #

$!READDATASET "STANDARDSYNTAX" "1.0" "LoadOption" "CaseAndData"
"FILENAME CaseFile" "|MFBD|/1_FLUENT_RESULTS/Case_.cas" "FILENAME DataFile"
"|MFBD|/1_FLUENT_RESULTS/Case_.dat" "GridZones" "CellsAndBoundaries"
"IncludeParticleData" "No" "AllPolyZones" "No" "AverageToNodes" "No"
"SaveUncompressedFiles" "No"
DATASETREADER = 'Fluent Data Loader'
#-----#

$!VarSet |InitNumVar| = |NumVars|

# _____ ( PUT NON DIMENTIONAL 3D AXES ) _____ #

$!ALTERDATA EQUATION = '{x/D}={x}/|NozzleSize|'
$!GETVARNUMBYNAME |NDX|
NAME = "x/D"

$!ALTERDATA EQUATION = '{y/D}={y}/|NozzleSize|'
$!GETVARNUMBYNAME |NDY|
NAME = "y/D"

$!ALTERDATA EQUATION = '{z/D}={z}/|NozzleSize|'
$!GETVARNUMBYNAME |NDZ|
NAME = "z/D"

$!THREEDAXIS XDETAIL{VARNUM = |NDX|}
$!THREEDAXIS YDETAIL{VARNUM = |NDY|}
$!THREEDAXIS ZDETAIL{VARNUM = |NDZ|}

```

```

Advanced_Post_Processing.mcr

$!RemoveVar |NDX|
$!RemoveVar |NDY|
$!RemoveVar |NDZ|
#-----#

# ( INITIALIZE VARIABLES ) #

$!ALTERDATA EQUATION = '{V} = {Mean Y Velocity} '
$!ALTERDATA EQUATION = '{Vrms} = {RMS Y Velocity}'
$!ALTERDATA EQUATION = '{W} = {Mean Z Velocity} '
$!ALTERDATA EQUATION = '{Wrms} = {RMS Z Velocity}'
$!ALTERDATA EQUATION = '|WIND|'
$!ALTERDATA EQUATION = '{Utest} = {Mean X Velocity} - {Uf}'
$!ALTERDATA EQUATION = '{U} = {Mean X Velocity}'
$!ALTERDATA EQUATION = '{Urms} = {RMS X Velocity}'
$!ALTERDATA EQUATION = '{UMax} = 0.0'
$!ALTERDATA EQUATION = '{UMaxPos} = 0.0'
$!ALTERDATA EQUATION = '{UHalf} = 0.0'
$!ALTERDATA EQUATION = '{UHalfPosVER} = 0.0'
$!ALTERDATA EQUATION = '{UHalfPosHOR} = 0.0'
$!ALTERDATA EQUATION = '{Phi} = {UDM-2}'
$!ALTERDATA EQUATION = '{PhiRms} = {UDM-4}'
$!ALTERDATA EQUATION = '{PhiMax} = 0.0'
$!ALTERDATA EQUATION = '{PhiMaxPos} = 0.0'
$!ALTERDATA EQUATION = '{PhiHalf} = 0.0'
$!ALTERDATA EQUATION = '{PhiHalfPosVER} = 0.0'
$!ALTERDATA EQUATION = '{PhiHalfPosHOR} = 0.0'
$!ALTERDATA EQUATION = '{UScaledJetDimension} = 0 '
$!ALTERDATA EQUATION = '{PhiScaledJetDimension} = 0 '
$!ALTERDATA EQUATION = '{U/UMax} = 0 '
$!ALTERDATA EQUATION = '{V/UMax} = 0 '
$!ALTERDATA EQUATION = '{W/UMax} = 0 '
$!ALTERDATA EQUATION = '{Phi/PhiMax} = 0 '
$!ALTERDATA EQUATION = '{URms/UMax} = 0 '
$!ALTERDATA EQUATION = '{VRms/UMax} = 0 '
$!ALTERDATA EQUATION = '{WRms/UMax} = 0 '
$!ALTERDATA EQUATION = '{PhiRms/PhiMax} = 0 '
$!DELETEVARS [4-|InitNumVar|]
#-----#

# ( EXTRACT PLANES FROM VOLUME ) #

$!VarSet |InitNrOfZones| = |NumZones|

$!GLOBALTHREED SLICE{NORMAL{X = 1}}
$!GLOBALTHREED SLICE{NORMAL{Y = 0}}
$!GLOBALTHREED SLICE{NORMAL{Z = 0}}

$!VarSet |Station| = 0
$!Loop |NrStations|
$!VarSet |Station| += |XbyDSkip|
$!VarSet |StationOffset| = (|MeshStaggeringOffset|+|Station|)

$!GLOBALTHREED SLICE{ORIGIN{X = |StationOffset|}}
$!CREATESLICEZONEFROMPLANE
  SLICESOURCE = VOLUMEZONES
  FORCEEXTRACTIONTOSINGLE Zone = YES
$!VarSet |ZoneNrStation| = (|Loop|+|InitNrOfZones|)
$!RENAMEDATASETZONE Zone = |ZoneNrStation| NAME = 'Case_|CaseNr|_Plane at
x/D = |Station|'
$!EndLoop

$!DELETEZONES [1-|InitNrOfZones|]

$!RemoveVar |Station|
$!RemoveVar |StationOffset|
$!RemoveVar |ZoneNrStation|
$!RemoveVar |InitNrOfZones|

#-----#

# ( Here Get The Variables' Positions ) #

$!GETVARNUMBYNAME |VarNumUf| NAME = "Uf"

```



```

Advanced_Post_Processing.mcr

$!GETVARNUMBYNAME |VarNumU| NAME = "U"
$!GETVARNUMBYNAME |VarNumUtest| NAME = "Utest"
$!GETVARNUMBYNAME |VarNumUMax| NAME = "UMax"
$!GETVARNUMBYNAME |VarNumUMaxPos| NAME = "UMaxPos"
$!GETVARNUMBYNAME |VarNumUHalfPosVER| NAME = "UHalfPosVER"
$!GETVARNUMBYNAME |VarNumUHalfPosHOR| NAME = "UHalfPosHOR"
$!GETVARNUMBYNAME |VarNumPhi| NAME = "Phi"
$!GETVARNUMBYNAME |VarNumPhiMax| NAME = "PhiMax"
$!GETVARNUMBYNAME |VarNumPhiMaxPos| NAME = "PhiMaxPos"
$!GETVARNUMBYNAME |VarNumPhiHalfPosVER| NAME = "PhiHalfPosVER"
$!GETVARNUMBYNAME |VarNumPhiHalfPosHOR| NAME = "PhiHalfPosHOR"
$!GETVARNUMBYNAME |VarNumX| NAME = "X"
$!GETVARNUMBYNAME |VarNumY| NAME = "Y"
$!GETVARNUMBYNAME |VarNumZ| NAME = "Z"
$!GETVARNUMBYNAME |VarNumXByD| NAME = "x/D"
#-----#
$!WRITEDATASET "MFBDD/2__SLICES/NrStations|_N-D_ANALYSIS_PLANES.plt"
  INCLUDETEXT = NO
  INCLUDEGEOM = NO
  INCLUDECUSTOMLABELS = NO
  ASSOCIATELAYOUTWITHDATAFILE = NO
  BINARY = YES
  USEPOINTFORMAT = NO
  PRECISION = 9
  TECPLOTVERSIONTOWRITE = TECPLOTCURRENT

#_____ ( PUT NON DIMENSIONAL 3D AXES ) _____#

$!GETVARNUMBYNAME |NDX|
  NAME = "x/D"
$!GETVARNUMBYNAME |NDY|
  NAME = "y/D"
$!GETVARNUMBYNAME |NDZ|
  NAME = "z/D"
$!THREEDAXIS XDETAIL{VARNUM = |NDX|}
$!THREEDAXIS YDETAIL{VARNUM = |NDY|}
$!THREEDAXIS ZDETAIL{VARNUM = |NDZ|}

$!RemoveVar |NDX|
$!RemoveVar |NDY|
$!RemoveVar |NDZ|

#_____ ( EXTRACT VERTICAL CURVES FROM THE PLANES ) _____#

$!VarSet |InitNrOfZones| = |NumZones|
$!VarSet |Station| = 0

$!GLOBALTHREED SLICE{NORMAL{X = 0}}
$!GLOBALTHREED SLICE{NORMAL{Y = 1}}
$!GLOBALTHREED SLICE{NORMAL{Z = 0}}

$!Loop |NrStations|
  $!VarSet |Station| += |XbyDSkip|
  $!ACTIVEFIELDMAPS = [|Loop|]
  $!GLOBALTHREED SLICE{ORIGIN{Y = |MeshStaggeringOffset|}}
  $!CREATESLICEZONEFROMPLANE SLICESOURCE = SURFACEZONES
    FORCEEXTRACTIONTOSINGLE Zone = YES COPYCELLCENTEREDVALUES = NO
  $!VarSet |ZoneNrStation| = (|Loop|+|InitNrOfZones|)
  $!RENAMEDATASETZONE Zone = |ZoneNrStation|
    NAME = 'Case_|CaseNr|_VERTICAL x/D = |Station|'
$!EndLoop

$!DELETEZONES [1-|InitNrOfZones|]

$!RemoveVar |Station|
$!RemoveVar |ZoneNrStation|
$!RemoveVar |InitNrOfZones|

#=====#
#===== U MAX =====#
#=====#

$!PLOTTYPE = XYLINE
$!Loop |NrStations|

```

```

Advanced_Post_Processing.mcr

$!VarSet |VerticalZoneNR| = |Loop|
$!LINEMAP [1] ASSIGN{ Zone = |VerticalZoneNR|}

# _____ ( Find the UMax Indices ) _____ #
$!VarSet |1stMax| = 0.0
$!VarSet |2ndMax| = 0.0
$!VarSet |3rdMax| = 0.0
$!VarSet |4thMax| = 0.0
$!VarSet |IndexAt1stMax| = 0.0
$!VarSet |IndexAt2ndMax| = 0.0
$!VarSet |IndexAt3rdMax| = 0.0
$!VarSet |IndexAt4thMax| = 0.0
$!Loop |MaxI|

$!GetFieldValue |Profile|
Zone = |VerticalZoneNR| Var = |VarNumUtest| Index = |Loop|

$!If |Profile| > |1stMax|
$!VarSet |4thMax| = |3rdMax|
$!VarSet |3rdMax| = |2ndMax|
$!VarSet |2ndMax| = |1stMax|
$!VarSet |1stMax| = |Profile|
$!VarSet |IndexAt4thMax| = |IndexAt3rdMax|
$!VarSet |IndexAt3rdMax| = |IndexAt2ndMax|
$!VarSet |IndexAt2ndMax| = |IndexAt1stMax|
$!VarSet |IndexAt1stMax| = |Loop|
$!EndIf

$!If |Profile| > |2ndMax|
$!If |Profile| != |1stMax|
$!VarSet |4thMax| = |3rdMax|
$!VarSet |3rdMax| = |2ndMax|
$!VarSet |2ndMax| = |Profile|
$!VarSet |IndexAt4thMax| = |IndexAt3rdMax|
$!VarSet |IndexAt3rdMax| = |IndexAt2ndMax|
$!VarSet |IndexAt2ndMax| = |Loop|
$!EndIf
$!EndIf

$!If |Profile| > |3rdMax|
$!If |Profile| != |1stMax|
$!If |Profile| != |2ndMax|
$!VarSet |4thMax| = |3rdMax|
$!VarSet |3rdMax| = |Profile|
$!VarSet |IndexAt4thMax| = |IndexAt3rdMax|
$!VarSet |IndexAt3rdMax| = |Loop|
$!EndIf
$!EndIf
$!EndIf

$!If |Profile| > |4thMax|
$!If |Profile| != |1stMax|
$!If |Profile| != |2ndMax|
$!If |Profile| != |3rdMax|
$!VarSet |4thMax| = |Profile|
$!VarSet |IndexAt4thMax| = |Loop|
$!EndIf
$!EndIf
$!EndIf
$!EndIf

$!EndLoop
#-----#

# _____ | Perform 2nd Order Polynomial Interpolation to _____ #
# _____ | Identify the Location and Magnitude of a variable | _____ #

# _____ INTERPOLATED VARIABLE _____ #

$!VarSet |VARIABLE| = |VarNumUtest|
#-----#

$!VarSet |I1| = |IndexAt2ndMax|
$!VarSet |Imax| = |IndexAt1stMax|

```

Advanced_Post_Processing.mcr

```

$!VarSet |I2| = |IndexAt3rdMax|
$!VarSet |I3| = |IndexAt4thMax|

$!GetFieldValue |U1|
Zone = |VerticalZoneNR|      Var = |VARIABLE|      Index = |I1|
$!GetFieldValue |Umax|
Zone = |VerticalZoneNR|      Var = |VARIABLE|      Index = |Imax|
$!GetFieldValue |U2|
Zone = |VerticalZoneNR|      Var = |VARIABLE|      Index = |I2|
$!GetFieldValue |U3|
Zone = |VerticalZoneNR|      Var = |VARIABLE|      Index = |I3|

$!GetFieldValue |X1|
Zone = |VerticalZoneNR|      Var = |VarNumZ|      Index = |I1|
$!GetFieldValue |Xmax|
Zone = |VerticalZoneNR|      Var = |VarNumZ|      Index = |Imax|
$!GetFieldValue |X2|
Zone = |VerticalZoneNR|      Var = |VarNumZ|      Index = |I2|
$!GetFieldValue |X3|
Zone = |VerticalZoneNR|      Var = |VarNumZ|      Index = |I3|

$!If |Xmax| >= |X1|
$!If |Xmax| >= |X2|
$!VarSet |U2| = |U3|
$!VarSet |X2| = |X3|
$!EndIf
$!EndIf

$!If |Xmax| <= |X1|
$!If |Xmax| <= |X2|
$!VarSet |U2| = |U3|
$!VarSet |X2| = |X3|
$!EndIf
$!EndIf

$!VarSet |ALPHA| = (((|Umax|-|U1|)-(|Xmax|-|X1|)*(|Umax|-|U2|)/(|Xmax|-|X2|)) /
((|Xmax|*|Xmax|-|X1|*|X1|)-(|Xmax|-|X1|)*(|Xmax|*|Xmax|-|X2|*|X2|)
/(|Xmax|-|X2|)))
$!VarSet |BETA| = (((|Umax|-|U2|)-|ALPHA|*(|Xmax|*|Xmax|-|X2|*|X2|))
/(|Xmax|-|X2|))
$!VarSet |GAMMA| = (|U1|-|ALPHA|*|X1|*|X1|-|BETA|*|X1|)

$!ALTERDATA [|VerticalZoneNR|]
EQUATION = '{UMaxPos} = -1*|BETA|/(2*|ALPHA|)'
$!ALTERDATA [|VerticalZoneNR|]
EQUATION = '{UMax} = |ALPHA|*{UMaxPos}*{UMaxPos} + |BETA|*{UMaxPos} + |GAMMA|'

$!If |X2| <= |X1|
$!VarSet |min| = |X2|
$!VarSet |max| = |X1|
$!If (-1*|BETA|/(2*|ALPHA|)) >= |max|
$!ALTERDATA [|VerticalZoneNR|] EQUATION = '{UMaxPos} = |Xmax|'
$!ALTERDATA [|VerticalZoneNR|] EQUATION = '{UMax} = |Umax|'
$!EndIf
$!If (-1*|BETA|/(2*|ALPHA|)) <= |min|
$!ALTERDATA [|VerticalZoneNR|] EQUATION = '{UMaxPos} = |Xmax|'
$!ALTERDATA [|VerticalZoneNR|] EQUATION = '{UMax} = |Umax|'
$!EndIf
$!EndIf

$!If |X1| <= |X2|
$!VarSet |min| = |X1|
$!VarSet |max| = |X2|
$!If (-1*|BETA|/(2*|ALPHA|)) >= |max|
$!ALTERDATA [|VerticalZoneNR|] EQUATION = '{UMaxPos} = |Xmax|'
$!ALTERDATA [|VerticalZoneNR|] EQUATION = '{UMax} = |Umax|'
$!EndIf
$!If (-1*|BETA|/(2*|ALPHA|)) <= |min|
$!ALTERDATA [|VerticalZoneNR|] EQUATION = '{UMaxPos} = |Xmax|'
$!ALTERDATA [|VerticalZoneNR|] EQUATION = '{UMax} = |Umax|'
$!EndIf
$!EndIf

```


Advanced_Post_Processing.mcr

```

$!RemoveVar |1stMax|
$!RemoveVar |2ndMax|
$!RemoveVar |3rdMax|
$!RemoveVar |4thMax|
$!RemoveVar |IndexAt1stMax|
$!RemoveVar |IndexAt2ndMax|
$!RemoveVar |IndexAt3rdMax|
$!RemoveVar |IndexAt4thMax|
$!RemoveVar |I1|
$!RemoveVar |Imax|
$!RemoveVar |I2|
$!RemoveVar |U1|
$!RemoveVar |Umax|
$!RemoveVar |U2|
$!RemoveVar |X1|
$!RemoveVar |Xmax|
$!RemoveVar |X2|
$!RemoveVar |X3|
$!RemoveVar |ALPHA|
$!RemoveVar |BETA|
$!RemoveVar |GAMMA|
$!RemoveVar |VARIABLE|
#-----#
$!EndLoop

$!PLOTTYPE = XYLINE
#=====#
#=====Phi MAX=====#
#=====#

$!Loop |NrStations|
$!VarSet |VerticalZoneNR| = |Loop|
$!LINEMAP [1] ASSIGN{ Zone = |VerticalZoneNR|}

# ( Find the PhiMax Indices ) #

$!VarSet |1stMax| = -1.0
$!VarSet |2ndMax| = -1.0
$!VarSet |3rdMax| = -1.0
$!VarSet |4thMax| = -1.0
$!VarSet |IndexAt1stMax| = 0
$!VarSet |IndexAt2ndMax| = 0
$!VarSet |IndexAt3rdMax| = 0
$!VarSet |IndexAt4thMax| = 0
$!VarSet |PhiMax| = 0.0

$!Loop |MaxI|

$!GetFieldValue |Profile|
Zone = |VerticalZoneNR| Var = |VarNumPhi| Index = |Loop|
$!GetFieldValue |Z|
Zone = |VerticalZoneNR| Var = |VarNumZ| Index = |Loop|

$!If |Profile| > |1stMax|
$!VarSet |4thMax| = |3rdMax|
$!VarSet |3rdMax| = |2ndMax|
$!VarSet |2ndMax| = |1stMax|
$!VarSet |1stMax| = |Profile|
$!VarSet |IndexAt4thMax| = |IndexAt3rdMax|
$!VarSet |IndexAt3rdMax| = |IndexAt2ndMax|
$!VarSet |IndexAt2ndMax| = |IndexAt1stMax|
$!VarSet |IndexAt1stMax| = |Loop|
$!EndIf

$!If |Profile| > |2ndMax|
$!If |Profile| != |1stMax|
$!VarSet |4thMax| = |3rdMax|
$!VarSet |3rdMax| = |2ndMax|
$!VarSet |2ndMax| = |Profile|
$!VarSet |IndexAt4thMax| = |IndexAt3rdMax|
$!VarSet |IndexAt3rdMax| = |IndexAt2ndMax|
$!VarSet |IndexAt2ndMax| = |Loop|
$!EndIf
$!EndIf

```

```

Advanced_Post_Processing.mcr

$!If |Profile| > |3rdMax|
$!If |Profile| != |1stMax|
$!If |Profile| != |2ndMax|
$!VarSet |4thMax| = |3rdMax|
$!VarSet |3rdMax| = |Profile|
$!VarSet |IndexAt4thMax| = |IndexAt3rdMax|
$!VarSet |IndexAt3rdMax| = |Loop|
$!EndIf
$!EndIf
$!EndIf

$!If |Profile| > |4thMax|
$!If |Profile| != |1stMax|
$!If |Profile| != |2ndMax|
$!If |Profile| != |3rdMax|
$!VarSet |4thMax| = |Profile|
$!VarSet |IndexAt4thMax| = |Loop|
$!EndIf
$!EndIf
$!EndIf
$!EndIf

$!EndLoop
#-----#

# _____| Perform 2nd Order Polynomial Interpolation to _____#
# _____| Identify the Location and Magnitude of a variable | _____#

# _____ INTERPOLATED VARIABLE _____#

$!VarSet |VARIABLE| = |VarNumPhi|
#-----#

$!VarSet |I1| = |IndexAt2ndMax|
$!VarSet |Imax| = |IndexAt1stMax|
$!VarSet |I2| = |IndexAt3rdMax|
$!VarSet |I3| = |IndexAt4thMax|

$!GetFieldValue |Phi1|
Zone = |VerticalZoneNR| Var = |VARIABLE| Index = |I1|
$!GetFieldValue |Phimax|
Zone = |VerticalZoneNR| Var = |VARIABLE| Index = |Imax|
$!GetFieldValue |Phi2|
Zone = |VerticalZoneNR| Var = |VARIABLE| Index = |I2|
$!GetFieldValue |Phi3|
Zone = |VerticalZoneNR| Var = |VARIABLE| Index = |I3|

$!GetFieldValue |X1|
Zone = |VerticalZoneNR| Var = |VarNumZ| Index = |I1|
$!GetFieldValue |Xmax|
Zone = |VerticalZoneNR| Var = |VarNumZ| Index = |Imax|
$!GetFieldValue |X2|
Zone = |VerticalZoneNR| Var = |VarNumZ| Index = |I2|
$!GetFieldValue |X3|
Zone = |VerticalZoneNR| Var = |VarNumZ| Index = |I3|

$!If |Xmax| >= |X1|
$!If |Xmax| >= |X2|
$!VarSet |Phi2| = |Phi3|
$!VarSet |X2| = |X3|
$!EndIf
$!EndIf

$!If |Xmax| <= |X1|
$!If |Xmax| <= |X2|
$!VarSet |Phi2| = |Phi3|
$!VarSet |X2| = |X3|
$!EndIf
$!EndIf

$!VarSet |ALPHA| = (((|Phimax|-|Phi1|)-(|Xmax|-|X1|)*(|Phimax|-|Phi2|)
/ (|Xmax|-|X2|)) / ((|Xmax|*|Xmax|-|X1|*|X1|)-((|Xmax|-|X1|)
* (|Xmax|*|Xmax|-|X2|*|X2|) / (|Xmax|-|X2|))))
$!VarSet |BETA| = (((|Phimax|-|Phi2|)-|ALPHA|*(|Xmax|*|Xmax|-|X2|*|X2|))
/ (|Xmax|-|X2|)) )

```

```

Advanced_Post_Processing.mcr

$!VarSet |GAMMA| = (|Phi1|-|ALPHA|*|X1|*|X1|-|BETA|*|X1| )

$!ALTERDATA [|VerticalZoneNR|]
EQUATION = '{PhiMaxPos} = -1*|BETA|/(2*|ALPHA|)'
$!ALTERDATA [|VerticalZoneNR|]
EQUATION = '{PhiMax} = |ALPHA|*{PhiMaxPos}*{PhiMaxPos} + |BETA|*{PhiMaxPos} +
|GAMMA| '

$!If |X2| <= |X1|
  $!VarSet |min| = |X2|
  $!VarSet |max| = |X1|
  $!If (-1*|BETA|/(2*|ALPHA|)) >= |max|
    $!ALTERDATA [|VerticalZoneNR|] EQUATION = '{PhiMaxPos} = |Xmax|'
    $!ALTERDATA [|VerticalZoneNR|] EQUATION = '{PhiMax} = |Phimax|'
  $!EndIf
  $!If (-1*|BETA|/(2*|ALPHA|)) <= |min|
    $!ALTERDATA [|VerticalZoneNR|] EQUATION = '{PhiMaxPos} = |Xmax|'
    $!ALTERDATA [|VerticalZoneNR|] EQUATION = '{PhiMax} = |Phimax|'
  $!EndIf
$!EndIf

$!If |X1| <= |X2|
  $!VarSet |min| = |X1|
  $!VarSet |max| = |X2|
  $!If (-1*|BETA|/(2*|ALPHA|)) >= |max|
    $!ALTERDATA [|VerticalZoneNR|] EQUATION = '{PhiMaxPos} = |Xmax|'
    $!ALTERDATA [|VerticalZoneNR|] EQUATION = '{PhiMax} = |Phimax|'
  $!EndIf
  $!If (-1*|BETA|/(2*|ALPHA|)) <= |min|
    $!ALTERDATA [|VerticalZoneNR|] EQUATION = '{PhiMaxPos} = |Xmax|'
    $!ALTERDATA [|VerticalZoneNR|] EQUATION = '{PhiMax} = |Phimax|'
  $!EndIf
$!EndIf

$!RemoveVar |1stMax|
$!RemoveVar |2ndMax|
$!RemoveVar |3rdMax|
$!RemoveVar |4thMax|
$!RemoveVar |IndexAt1stMax|
$!RemoveVar |IndexAt2ndMax|
$!RemoveVar |IndexAt3rdMax|
$!RemoveVar |IndexAt4thMax|
$!RemoveVar |I1|
$!RemoveVar |Imax|
$!RemoveVar |I2|
$!RemoveVar |Phi1|
$!RemoveVar |Phimax|
$!RemoveVar |Phi2|
$!RemoveVar |X1|
$!RemoveVar |Xmax|
$!RemoveVar |X2|
$!RemoveVar |X3|
$!RemoveVar |ALPHA|
$!RemoveVar |BETA|
$!RemoveVar |GAMMA|
$!RemoveVar |VARIABLE|
#-----#
$!EndLoop

#=====
#===== U Half VERTICAL =====
#=====
$!ALTERDATA EQUATION = '{UHalf} = {UMax} / 2.0 '
$!GETVARNUMBYNAME |VarNumUHalf| NAME = "UHalf"

$!Loop |NrStations|
  $!VarSet |VerticalZoneNR| = |Loop|
  $!LINEMAP [1] ASSIGN{ Zone = |VerticalZoneNR|}
  #_____ ( Find UHalf Closest Index ) _____#

$!VarSet |1stMax| = 0.0
$!VarSet |2ndMax| = 0.0
$!VarSet |3rdMax| = 0.0
$!VarSet |IndexAt1stMax| = 0.0

```

```

Advanced_Post_Processing.mcr

$!VarSet |IndexAt2ndMax| = 0.0
$!VarSet |IndexAt3rdMax| = 0.0

$!GetFieldValue |UMaxPos|
Zone = |VerticalZoneNR|
1
Var = |VarNumUMaxPos|
Index =

$!GetFieldValue |UHalf|
Zone = |VerticalZoneNR|
1
Var = |VarNumUHalf|
Index =

$!Loop |MaxI|

$!GetFieldValue |Utest|
Zone = |VerticalZoneNR|
|Loop|
Var = |VarNumUtest|
Index =

$!GetFieldValue |Z|
Zone = |VerticalZoneNR|
|Loop|
Var = |VarNumZ|
Index =

$!VarSet |Profile| = (1/(abs(|UHalf| - |Utest|)))

$!If (|Z| - |UMaxPos|) >= 0

$!If |Profile| > |1stMax|
$!VarSet |3rdMax| = |2ndMax|
$!VarSet |2ndMax| = |1stMax|
$!VarSet |1stMax| = |Profile|
$!VarSet |IndexAt3rdMax| = |IndexAt2ndMax|
$!VarSet |IndexAt2ndMax| = |IndexAt1stMax|
$!VarSet |IndexAt1stMax| = |Loop|
$!EndIf

$!If |Profile| > |2ndMax|
$!If |Profile| != |1stMax|
$!VarSet |3rdMax| = |2ndMax|
$!VarSet |2ndMax| = |Profile|
$!VarSet |IndexAt3rdMax| = |IndexAt2ndMax|
$!VarSet |IndexAt2ndMax| = |Loop|
$!EndIf
$!EndIf

$!If |Profile| > |3rdMax|
$!If |Profile| != |1stMax|
$!If |Profile| != |2ndMax|
$!VarSet |3rdMax| = |Profile|
$!VarSet |IndexAt3rdMax| = |Loop|
$!EndIf
$!EndIf
$!EndIf

$!EndLoop

#-----#
# | | Perform a Linear Interpolation to | | #
# | | Identify the Real Location of UHalf | | #
#-----#

# _____ INTERPOLATED VARIABLE _____ #

$!VarSet |VARIABLE| = |VarNumUtest|
#-----#

$!VarSet |I1| = |IndexAt1stMax|
$!VarSet |I2| = |IndexAt2ndMax|

$!GetFieldValue |U1|
Zone = |VerticalZoneNR|
Var = |VARIABLE|
Index = |I1|
$!GetFieldValue |Umax|
Zone = |VerticalZoneNR|
Var = |VarNumUHalf|
Index = |I1|
$!GetFieldValue |U2|
Zone = |VerticalZoneNR|
Var = |VARIABLE|
Index = |I2|
$!GetFieldValue |X1|
Zone = |VerticalZoneNR|
Var = |VarNumZ|
Index = |I1|
$!GetFieldValue |X2|
Zone = |VerticalZoneNR|
Var = |VarNumZ|
Index = |I2|

```



```

Advanced_Post_Processing.mcr

$!VarSet |Xmax| = ( |X1|+ (|X2|-|X1|)/(|U2|-|U1|)*(|Umax|-|U1|) )

$!ALTERDATA [|VerticalZoneNR|] EQUATION = '{UHalfPosVER} = |Xmax| '
$!ALTERDATA [|VerticalZoneNR|] EQUATION = '{UScaledJetDimension} = ({Z} -
{UMaxPos}) / ({UHalfPosVER} - {UMaxPos}) '

$!GetFieldValue |UHalfPosVER|
Zone = |VerticalZoneNR| Var = |VarNumUHalfPosVER| Index =
1

$!If |UHalfPosVER| <= |UMaxPos|
$!ALTERDATA [|VerticalZoneNR|]
EQUATION = '{UHalfPosVER} = (|X1|+|X2|)/2 '
$!ALTERDATA [|VerticalZoneNR|]
EQUATION = '{UScaledJetDimension} = ({Z} - {UMaxPos}) / ({UHalfPosVER} -
{UMaxPos}) '
$!EndIf

$!RemoveVar |1stMax|
$!RemoveVar |2ndMax|
$!RemoveVar |3rdMax|
$!RemoveVar |IndexAt1stMax|
$!RemoveVar |IndexAt2ndMax|
$!RemoveVar |IndexAt3rdMax|
$!RemoveVar |I1|
$!RemoveVar |I2|
$!RemoveVar |U1|
$!RemoveVar |Umax|
$!RemoveVar |U2|
$!RemoveVar |X1|
$!RemoveVar |Xmax|
$!RemoveVar |X2|
#-----#

$!EndLoop

#=====
#-----#
$!ALTERDATA EQUATION = '{PhiHalf} = {PhiMax} / 2.0 '
$!GETVARNUMBYNAME |VarNumPhiHalf| NAME = "PhiHalf"

$!Loop |NrStations|
$!VarSet |VerticalZoneNR| = |Loop|
$!LINEMAP [1] ASSIGN{ Zone = |VerticalZoneNR|}

#-----# ( Find UHalf Closest Index ) -----#

$!VarSet |1stMax| = 0.0
$!VarSet |2ndMax| = 0.0
$!VarSet |3rdMax| = 0.0
$!VarSet |IndexAt1stMax| = 0.0
$!VarSet |IndexAt2ndMax| = 0.0
$!VarSet |IndexAt3rdMax| = 0.0

$!Loop |MaxI|

$!GetFieldValue |Phi|
Zone = |VerticalZoneNR| Var = |VarNumPhi| Index = |Loop|
$!GetFieldValue |PhiMaxPos|
Zone = |VerticalZoneNR| Var = |VarNumPhiMaxPos| Index = |Loop|
$!GetFieldValue |PhiHalf|
Zone = |VerticalZoneNR| Var = |VarNumPhiHalf| Index = |Loop|
$!GetFieldValue |Z|
Zone = |VerticalZoneNR| Var = |VarNumZ| Index = |Loop|

$!VarSet |Profile| = (1/(abs(|PhiHalf| - |Phi|)))

$!If (|Z| - |PhiMaxPos|) >= 0.00001

$!If |Profile| > |1stMax|
$!VarSet |3rdMax| = |2ndMax|

```

```

Advanced_Post_Processing.mcr

$!VarSet |2ndMax| = |1stMax|
$!VarSet |1stMax| = |Profile|
$!VarSet |IndexAt3rdMax| = |IndexAt2ndMax|
$!VarSet |IndexAt2ndMax| = |IndexAt1stMax|
$!VarSet |IndexAt1stMax| = |Loop|
$!EndIf

$!If |Profile| > |2ndMax|
  $!If |Profile| != |1stMax|
    $!VarSet |3rdMax| = |2ndMax|
    $!VarSet |2ndMax| = |Profile|
    $!VarSet |IndexAt3rdMax| = |IndexAt2ndMax|
    $!VarSet |IndexAt2ndMax| = |Loop|
  $!EndIf
$!EndIf

$!If |Profile| > |3rdMax|
  $!If |Profile| != |1stMax|
    $!If |Profile| != |2ndMax|
      $!VarSet |3rdMax| = |Profile|
      $!VarSet |IndexAt3rdMax| = |Loop|
    $!EndIf
  $!EndIf
$!EndIf

$!EndIf
$!EndLoop
#-----#

# _____ | Perform a Linear Interpolation to _____ #
# _____ | Identify the Real Location of PhiHalf | _____ #

# _____ INTERPOLATED VARIABLE _____ #

$!VarSet |VARIABLE| = |VarNumPhi|
#-----#

$!VarSet |I1| = |IndexAt1stMax|
$!VarSet |I2| = |IndexAt2ndMax|

$!GetFieldValue |Phi1|
Zone = |VerticalZoneNR|
|I1| Var = |VARIABLE| Index =
$!GetFieldValue |Phimax|
Zone = |VerticalZoneNR|
1 Var = |VarNumPhiHalf| Index =
$!GetFieldValue |Phi2|
Zone = |VerticalZoneNR|
|I2| Var = |VARIABLE| Index =

$!GetFieldValue |X1|
Zone = |VerticalZoneNR|
|I1| Var = |VarNumZ| Index =
$!GetFieldValue |X2|
Zone = |VerticalZoneNR|
|I2| Var = |VarNumZ| Index =

$!VarSet |Xmax| = ( |X1| + (|X2| - |X1|) / (|Phi2| - |Phi1|) * (|Phimax| - |Phi1|) )

$!ALTERDATA [|VerticalZoneNR|]
EQUATION = '{PhiHalfPosVER} = |Xmax| '
$!ALTERDATA [|VerticalZoneNR|]
EQUATION = '{PhiScaledJetDimension} = ({Z} - {PhiMaxPos}) / ({PhiHalfPosVER} - {PhiMaxPos}) '

$!RemoveVar |1stMax|
$!RemoveVar |2ndMax|
$!RemoveVar |3rdMax|
$!RemoveVar |IndexAt1stMax|
$!RemoveVar |IndexAt2ndMax|
$!RemoveVar |IndexAt3rdMax|
$!RemoveVar |I1|
$!RemoveVar |I2|
$!RemoveVar |Phi1|
$!RemoveVar |Phimax|

```

```

Advanced_Post_Processing.mcr

$!RemoveVar |Phi2|
$!RemoveVar |X1|
$!RemoveVar |Xmax|
$!RemoveVar |X2|

$!EndLoop

=====
$!PLOTTYPE = CARTESIAN3D
=====

# _____ ( EXTRACT U-HORIZONTAL CURVES ) _____ #

$!READDATASET "'|MFBD|/2__SLICES/|NrStations|_N-D_ANALYSIS_PLANES.plt" '
  READDATAOPTION = APPEND
  RESETSTYLE = NO
  INCLUDETEXT = NO
  INCLUDEGEOM = NO
  INCLUDECUSTOMLABELS = NO
  VARLOADMODE = BYNAME
  ASSIGNSTRANDIDS = YES

$!Loop |NrStations|
  $!VarSet |VerticalZoneNR| = |Loop|
  $!VarSet |2DPlaneZoneNR| = (|Loop|+|NrStations|)
  $!VarSet |UHorizontalZoneNR| = (|Loop|+2*|NrStations|)

  $!LINEMAP [1] ASSIGN{ Zone = |VerticalZoneNR|}

  $!GLOBALTHREED SLICE{NORMAL{X = 0}}
  $!GLOBALTHREED SLICE{NORMAL{Y = 0}}
  $!GLOBALTHREED SLICE{NORMAL{Z = 1}}

  $!GetFieldValue |UMaxPos|
    Zone = |Loop|          Var = |VarNumUMaxPos|          Index = 1

  $!VarSet |Station| += |XbyDSkip|
  $!ACTIVEFIELDMAPI = [|2DPlaneZoneNR|]

  $!VarSet |UMaxPosByD| = (|UMaxPos|/|NozzleSize|)
  $!GLOBALTHREED SLICE{ORIGIN{Z = |UMaxPosByD| }}

  $!CREATESLICEZONEFROMPLANE
  SLICESOURCE = SURFACEZONES
  FORCEEXTRACTIONTOSINGLE Zone = YES
  COPYCELLCENTEREDVALUES = NO

  $!RENAMEDATASETZONE
    Zone =|UHorizontalZoneNR|
    NAME = 'Case_|CaseNr|_U-HORIZONTAL x/D = |Station|'
$!EndLoop

$!RemoveVar |VerticalZoneNR|
$!RemoveVar |2DPlaneZoneNR|
$!RemoveVar |UHorizontalZoneNR|
$!RemoveVar |UMaxPosByD|
$!RemoveVar |Station|
#-----#
  $!VarSet |StartDelZones| = (|NrStations|+1)
  $!VarSet |EndDelZones| = (2*|NrStations|)
$!DELETEZONES [|StartDelZones|-|EndDelZones|]
$!RemoveVar |StartDelZones|
$!RemoveVar |EndDelZones|

$!PLOTTYPE = XYLINE

$!Loop |NrStations|
  $!VarSet |VerticalZoneNR| = |Loop|
  $!VarSet |UHorizontalZoneNR| = (|Loop|+|NrStations|)
  $!ACTIVEFIELDMAPI = [1]
  $!LINEMAP [1] ASSIGN{ Zone = |UHorizontalZoneNR|}
# _____ ( Find UHalf Closest Index ) _____ #

```

```

Advanced_Post_Processing.mcr

$!VarSet |1stMax| = 0.0
$!VarSet |2ndMax| = 0.0
$!VarSet |3rdMax| = 0.0
$!VarSet |IndexAt1stMax| = 0.0
$!VarSet |IndexAt2ndMax| = 0.0
$!VarSet |IndexAt3rdMax| = 0.0

$!GetFieldValue |UMaxPos|
Zone = |VerticalZoneNR|      Var = |VarNumUMaxPos|      Index = 1
$!GetFieldValue |UMax|
Zone = |VerticalZoneNR|      Var = |VarNumUMax|      Index = 1
$!GetFieldValue |PhiMax|
Zone = |VerticalZoneNR|      Var = |VarNumPhiMax|      Index = 1
$!GetFieldValue |UHalf|
Zone = |VerticalZoneNR|      Var = |VarNumUHalf|      Index = 1

$!ALTERDATA [|UHorizontalZoneNR|] EQUATION = '{UMax} = |UMax| '
$!ALTERDATA [|UHorizontalZoneNR|] EQUATION = '{UMaxPos} = |UMaxPos| '
$!ALTERDATA [|UHorizontalZoneNR|] EQUATION = '{PhiMax} = |PhiMax| '
$!ALTERDATA [|UHorizontalZoneNR|] EQUATION = '{UHalf} = |UHalf| '

$!Loop |MaxI|

$!GetFieldValue |Utest|
Zone = |UHorizontalZoneNR|    Var = |VarNumUtest|      Index = |Loop|
$!GetFieldValue |UMaxPos|
Zone = |UHorizontalZoneNR|    Var = |VarNumUMaxPos|    Index = |Loop|
$!GetFieldValue |UHalf|
Zone = |UHorizontalZoneNR|    Var = |VarNumUHalf|      Index = |Loop|
$!GetFieldValue |Y|
Zone = |UHorizontalZoneNR|    Var = |VarNumY|          Index = |Loop|

$!VarSet |Profile| = (1/(abs(|UHalf| - |Utest|)))
$!If |Y| >= 0

$!If |Profile| > |1stMax|
$!VarSet |3rdMax| = |2ndMax|
$!VarSet |2ndMax| = |1stMax|
$!VarSet |1stMax| = |Profile|
$!VarSet |IndexAt3rdMax| = |IndexAt2ndMax|
$!VarSet |IndexAt2ndMax| = |IndexAt1stMax|
$!VarSet |IndexAt1stMax| = |Loop|
$!EndIf

$!If |Profile| > |2ndMax|
$!If |Profile| != |1stMax|
$!VarSet |3rdMax| = |2ndMax|
$!VarSet |2ndMax| = |Profile|
$!VarSet |IndexAt3rdMax| = |IndexAt2ndMax|
$!VarSet |IndexAt2ndMax| = |Loop|
$!EndIf
$!EndIf

$!If |Profile| > |3rdMax|
$!If |Profile| != |1stMax|
$!If |Profile| != |2ndMax|
$!VarSet |3rdMax| = |Profile|
$!VarSet |IndexAt3rdMax| = |Loop|
$!EndIf
$!EndIf
$!EndIf

$!EndLoop

# _____ | Perform a Linear Interpolation to _____ #
# _____ | Identify the Real Location of UHalf | _____ #

# _____ INTERPOLATED VARIABLE _____ #

$!VarSet |VARIABLE| = |VarNumUtest|
#-----#
$!VarSet |I1| = |IndexAt1stMax|
$!VarSet |I2| = |IndexAt2ndMax|

```



```

Advanced_Post_Processing.mcr

$!GetFieldValue |U1|
Zone = |UHorizontalZoneNR|          Var = |VARIABLE|          Index = |I1|
$!GetFieldValue |Umax|
Zone = |UHorizontalZoneNR|          Var = |VarNumUHalf|      Index = 1
$!GetFieldValue |U2|
Zone = |UHorizontalZoneNR|          Var = |VARIABLE|          Index = |I2|
$!GetFieldValue |X1|
Zone = |UHorizontalZoneNR|          Var = |VarNumY|          Index = |I1|
$!GetFieldValue |X2|
Zone = |UHorizontalZoneNR|          Var = |VarNumY|          Index = |I2|

$!VarSet |Xmax| = ( |X1|+(|X2|-|X1|)/(|U2|-|U1|)*(|Umax|-|U1|) )

$!ALTERDATA [|UHorizontalZoneNR|] EQUATION = '{UHalfPosHOR} = |Xmax| '
$!ALTERDATA [|UHorizontalZoneNR|] EQUATION = '{UScaledJetDimension} = {Y} / {UHalfPosHOR} '

$!RemoveVar |VerticalZoneNR|
$!RemoveVar |UHorizontalZoneNR|
$!RemoveVar |1stMax|
$!RemoveVar |2ndMax|
$!RemoveVar |3rdMax|
$!RemoveVar |IndexAt1stMax|
$!RemoveVar |IndexAt2ndMax|
$!RemoveVar |IndexAt3rdMax|
$!RemoveVar |I1|
$!RemoveVar |I2|
$!RemoveVar |U1|
$!RemoveVar |Umax|
$!RemoveVar |U2|
$!RemoveVar |X1|
$!RemoveVar |Xmax|
$!RemoveVar |X2|
#-----#
$!EndLoop

=====
$!PLOTTYPE = CARTESIAN3D
=====

# _____ ( EXTRACT Phi-HORIZONTAL CURVES ) _____ #
# _____ #
$!READDATASET "'|MFB|/2__SLICES/|NrStations|_N-D_ANALYSIS_PLANES.plt" '
READDATAOPTION = APPEND
RESETSTYLE = NO
INCLUDETEXT = NO
INCLUDEGEOM = NO
INCLUDECUSTOMLABELS = NO
VARLOADMODE = BYNAME
ASSIGNSTRANDIDS = YES

$!Loop |NrStations|
$!VarSet |VerticalZoneNR| = |Loop|
$!VarSet |UHorizontalZoneNR| = (|Loop|+|NrStations|)
$!VarSet |2DPlaneZoneNR| = (|Loop|+2*|NrStations|)
$!VarSet |PhiHorizontalZoneNR| = (|Loop|+3*|NrStations|)

$!LINEMAP [1] ASSIGN{ Zone = |VerticalZoneNR|}

$!GLOBALTHREED SLICE{NORMAL{X = 0}}
$!GLOBALTHREED SLICE{NORMAL{Y = 0}}
$!GLOBALTHREED SLICE{NORMAL{Z = 1}}

$!GetFieldValue |PhiMaxPos|
Zone = |Loop|          Var = |VarNumPhiMaxPos|          Index = 1

$!VarSet |Station| += |XbyDSkip|
$!ACTIVEFIELDMAPI = [|2DPlaneZoneNR|]
$!VarSet |PhiMaxPosByD| = (|PhiMaxPos|/|NozzleSize|)
$!If |PhiMaxPosByD| < (0.0 -|JetClearance|)
$!VarSet |PhiMaxPosByD| = -1.9999
$!EndIf
$!GLOBALTHREED SLICE{ORIGIN{Z = |PhiMaxPosByD| }}
$!CREATESLICEZONEFROMPLANE

```

```

Advanced_Post_Processing.mcr

SLICESOURCE = SURFACEZONES
FORCEEXTRACTIONTOSINGLE Zone = YES
COPYCELLCENTEREDVALUES = NO

$!RENAMEDATASETZONE
Zone = |PhiHorizontalZoneNR|
NAME = 'Case_|CaseNr|_Phi-HORIZONTAL x/D = |Station|'

$!EndLoop
$!RemoveVar |VerticalZoneNR|
$!RemoveVar |2DPlaneZoneNR|
$!RemoveVar |PhiHorizontalZoneNR|
$!RemoveVar |PhiMaxPosByD|
$!RemoveVar |Station|
#-----#
$!VarSet |StartDelZones| = (2*|NrStations|+1)
$!VarSet |EndDelZones| = (3*|NrStations|)
$!DELETEZONES [|StartDelZones|~|EndDelZones|]
$!RemoveVar |StartDelZones|
$!RemoveVar |EndDelZones|

=====
$!PLOTTYPE = XYLINE
=====

$!Loop |NrStations|
$!VarSet |VerticalZoneNR| = |Loop|
$!VarSet |PhiHorizontalZoneNR| = (|Loop|+|NrStations|+|NrStations|)
$!ACTIVEFIELDMAPS = [1]
$!LINEMAP [1] ASSIGN{ Zone = |PhiHorizontalZoneNR|}
# ( Find PhiHalf Closest Index ) #

$!VarSet |1stMax| = 0.0
$!VarSet |2ndMax| = 0.0
$!VarSet |3rdMax| = 0.0
$!VarSet |IndexAt1stMax| = 0.0
$!VarSet |IndexAt2ndMax| = 0.0
$!VarSet |IndexAt3rdMax| = 0.0

$!GetFieldValue |PhiMaxPos|
Zone = |VerticalZoneNR| Var = |VarNumPhiMaxPos| Index = 1
$!GetFieldValue |PhiMax|
Zone = |VerticalZoneNR| Var = |VarNumPhiMax| Index = 1
$!GetFieldValue |UMax|
Zone = |VerticalZoneNR| Var = |VarNumUMax| Index = 1
$!GetFieldValue |PhiHalf|
Zone = |VerticalZoneNR| Var = |VarNumPhiHalf| Index = 1

$!ALTERDATA [|PhiHorizontalZoneNR|] EQUATION = '{PhiMax} = |PhiMax| '
$!ALTERDATA [|PhiHorizontalZoneNR|] EQUATION = '{PhiMaxPos} = |PhiMaxPos| '
$!ALTERDATA [|PhiHorizontalZoneNR|] EQUATION = '{UMax} = |UMax| '
$!ALTERDATA [|PhiHorizontalZoneNR|] EQUATION = '{PhiHalf} = |PhiHalf| '

$!Loop |MaxI|

$!GetFieldValue |Phi|
Zone = |PhiHorizontalZoneNR| Var = |VarNumPhi| Index = |Loop|
$!GetFieldValue |PhiMaxPos|
Zone = |PhiHorizontalZoneNR| Var = |VarNumPhiMaxPos| Index = |Loop|
$!GetFieldValue |PhiHalf|
Zone = |PhiHorizontalZoneNR| Var = |VarNumPhiHalf| Index = |Loop|
$!GetFieldValue |Y|
Zone = |PhiHorizontalZoneNR| Var = |VarNumY| Index = |Loop|

$!VarSet |Profile| = (1/(abs(|PhiHalf| - |Phi|)))

$!If |Y| >= 0

$!If |Profile| > |1stMax|
$!VarSet |3rdMax| = |2ndMax|
$!VarSet |2ndMax| = |1stMax|
$!VarSet |1stMax| = |Profile|
$!VarSet |IndexAt3rdMax| = |IndexAt2ndMax|
$!VarSet |IndexAt2ndMax| = |IndexAt1stMax|

```

```

Advanced_Post_Processing.mcr

$!VarSet |IndexAt1stMax| = |Loop|
$!EndIf

$!If |Profile| > |2ndMax|
$!If |Profile| != |1stMax|
$!VarSet |3rdMax| = |2ndMax|
$!VarSet |2ndMax| = |Profile|
$!VarSet |IndexAt3rdMax| = |IndexAt2ndMax|
$!VarSet |IndexAt2ndMax| = |Loop|
$!EndIf
$!EndIf

$!If |Profile| > |3rdMax|
$!If |Profile| != |1stMax|
$!If |Profile| != |2ndMax|
$!VarSet |3rdMax| = |Profile|
$!VarSet |IndexAt3rdMax| = |Loop|
$!EndIf
$!EndIf
$!EndIf

$!EndIf
$!EndLoop

# _____ | Perform a Linear Interpolation to _____ #
# _____ | Identify the Real Location of PhiHalf | _____ #

# _____ INTERPOLATED VARIABLE _____ #

$!VarSet |VARIABLE| = |VarNumPhi|
#-----#

$!VarSet |I1| = |IndexAt1stMax|
$!VarSet |I2| = |IndexAt2ndMax|

$!GetFieldValue |Phi1|
Zone = |PhiHorizontalZoneNR| Var = |VARIABLE| Index = |I1|
$!GetFieldValue |PhiMax|
Zone = |PhiHorizontalZoneNR| Var = |VarNumPhiHalf| Index = 1
$!GetFieldValue |Phi2|
Zone = |PhiHorizontalZoneNR| Var = |VARIABLE| Index = |I2|

$!GetFieldValue |X1|
Zone = |PhiHorizontalZoneNR| Var = |VarNumY| Index = |I1|
$!GetFieldValue |X2|
Zone = |PhiHorizontalZoneNR| Var = |VarNumY| Index = |I2|

$!VarSet |Xmax| = ( |X1| + (|X2| - |X1|) / (|Phi2| - |Phi1|) * (|PhiMax| - |Phi1|) )

$!ALTERDATA [|PhiHorizontalZoneNR|]
EQUATION = '{PhiHalfPosHOR} = |Xmax| '
$!ALTERDATA [|PhiHorizontalZoneNR|]
EQUATION = '{PhiScaledJetDimension} = {Y} / {PhiHalfPosHOR} '

$!RemoveVar |VerticalZoneNR|
$!RemoveVar |PhiHorizontalZoneNR|
$!RemoveVar |1stMax|
$!RemoveVar |2ndMax|
$!RemoveVar |3rdMax|
$!RemoveVar |IndexAt1stMax|
$!RemoveVar |IndexAt2ndMax|
$!RemoveVar |IndexAt3rdMax|
$!RemoveVar |I1|
$!RemoveVar |I2|
$!RemoveVar |Phi1|
$!RemoveVar |PhiMax|
$!RemoveVar |Phi2|
$!RemoveVar |X1|
$!RemoveVar |Xmax|
$!RemoveVar |X2|
#-----#
$!EndLoop

$!ALTERDATA EQUATION = '{UMax} = {UMax} + {Uf}'

```


Advanced_Post_Processing.mcr

```

=====
#===== CREATE ONE DIMENSIONAL ZONE (STREAMWISE DATABASE) =====
#=====

$!VarSet |LastX| = (|NrStations|*|XbyDSkip|)
$!CREATERECTANGULARZONE
  IMAX = |NrStations|  JMAX = 1  KMAX = 1
  X1 = |XbyDSkip|      Y1 = 0      Z1 = 0
  X2 = |LastX|          Y2 = 0      Z2 = 0
  XVAR = 4
  YVAR = 5

$!RENAMEDATASETZONE      Zone = |NumZones|  NAME = 'Case_|CaseNr|_StreamWise'

#-----|          PASS DATA IN THE          |-----#
#-----|          NEW STREAMWISE ZONE          |-----#

$!Loop |NrStations|
  $!VarSet |VerticalZoneNR| = |Loop|
  $!VarSet |UHorizontalZoneNR| = (|Loop|+|NrStations|)
  $!VarSet |PhiHorizontalZoneNR| = (|Loop|+|NrStations|+|NrStations|)
  $!ACTIVEFIELDMAPS = [1]
  $!LINEMAP [1] ASSIGN{ Zone = |PhiHorizontalZoneNR|}

  $!GetFieldValue |XByD|
    Zone = |VerticalZoneNR|      Var = |VarNumXByD|      Index = 1
  $!GetFieldValue |X|
    Zone = |VerticalZoneNR|      Var = |VarNumX|      Index = 1
  $!GetFieldValue |UMax|
    Zone = |VerticalZoneNR|      Var = |VarNumUMax|      Index = 1
  $!GetFieldValue |UMaxPos|
    Zone = |VerticalZoneNR|      Var = |VarNumUMaxPos|      Index = 1
  $!GetFieldValue |UHalfPosVER|
    Zone = |VerticalZoneNR|      Var = |VarNumUHalfPosVER|      Index = 1
  $!GetFieldValue |UHalfPosHOR|
    Zone = |UHorizontalZoneNR|    Var = |VarNumUHalfPosHOR|      Index = 1
  $!GetFieldValue |PhiHalfPosVER|
    Zone = |VerticalZoneNR|      Var = |VarNumPhiHalfPosVER|      Index = 1
  $!GetFieldValue |PhiHalfPosHOR|
    Zone = |PhiHorizontalZoneNR|  Var = |VarNumPhiHalfPosHOR|      Index = 1
  $!GetFieldValue |PhiMax|
    Zone = |VerticalZoneNR|      Var = |VarNumPhiMax|      Index = 1
  $!GetFieldValue |PhiMaxPos|
    Zone = |VerticalZoneNR|      Var = |VarNumPhiMaxPos|      Index = 1

  $!SetFieldValue      Zone = |NumZones|      Var = |VarNumXByD|
  Index = |Loop|      FIELDVALUE = |XByD|      AUTOBRANCH = YES
  $!SetFieldValue      Zone = |NumZones|      Var = |VarNumX|
  Index = |Loop|      FIELDVALUE = |X|      AUTOBRANCH = YES
  $!SetFieldValue      Zone = |NumZones|      Var = |VarNumUMax|
  Index = |Loop|      FIELDVALUE = |UMax|      AUTOBRANCH = YES
  $!SetFieldValue      Zone = |NumZones|      Var = |VarNumUMaxPos|
  Index = |Loop|      FIELDVALUE = |UMaxPos|      AUTOBRANCH = YES
  $!SetFieldValue      Zone = |NumZones|      Var = |VarNumUHalfPosVER|
  Index = |Loop|      FIELDVALUE = |UHalfPosVER|      AUTOBRANCH = YES
  $!SetFieldValue      Zone = |NumZones|      Var = |VarNumUHalfPosHOR|
  Index = |Loop|      FIELDVALUE = |UHalfPosHOR|      AUTOBRANCH = YES
  $!SetFieldValue      Zone = |NumZones|      Var = |VarNumPhiMax|
  Index = |Loop|      FIELDVALUE = |PhiMax|      AUTOBRANCH = YES
  $!SetFieldValue      Zone = |NumZones|      Var = |VarNumPhiMaxPos|
  Index = |Loop|      FIELDVALUE = |PhiMaxPos|      AUTOBRANCH = YES
  $!SetFieldValue      Zone = |NumZones|      Var = |VarNumPhiHalfPosVER|
  Index = |Loop|      FIELDVALUE = |PhiHalfPosVER|      AUTOBRANCH = YES
  $!SetFieldValue      Zone = |NumZones|      Var = |VarNumPhiHalfPosHOR|
  Index = |Loop|      FIELDVALUE = |PhiHalfPosHOR|      AUTOBRANCH = YES
$!EndLoop

$!ALTERDATA EQUATION = '{U/UMax} = {U} / {UMax}'
$!ALTERDATA EQUATION = '{URms/UMax} = {URms} / {UMax}'
$!ALTERDATA EQUATION = '{V/UMax} = {V} / {UMax}'
$!ALTERDATA EQUATION = '{VRms/UMax} = {VRms} / {UMax}'
$!ALTERDATA EQUATION = '{W/UMax} = {W} / {UMax}'

```

```

Advanced_Post_Processing.mcr

$!ALTERDATA EQUATION = '{WRms/UMax}={WRms} / {UMax}'
$!ALTERDATA EQUATION = '{Phi/PhiMax}={Phi} / {PhiMax}'
$!ALTERDATA EQUATION = '{PhiRms/PhiMax}={PhiRms} / {PhiMax}'

$!VarSet |MinUHorizontalZone| = (|NrStations|+1)
$!VarSet |MaxUHorizontalZone| = (|NrStations|+|NrStations|)
$!VarSet |MinPhiHorizontalZone| = (|NrStations|+|NrStations|+1)
$!VarSet |MaxPhiHorizontalZone| = (|NrStations|+|NrStations|+|NrStations|)

#####
##### EXPORT DATA FILES #####
#####

$!WRITEDATASET "|MFBD|/3_CURVES_DATA/1_VERTICAL.plt"
INCLUDETEXT = NO
INCLUDEGEOM = NO
INCLUDECUSTOMLABELS = NO
ASSOCIATELAYOUTWITHDATAFILE = NO
ZONELIST = [1-|NrStations|]
BINARY = YES
USEPOINTFORMAT = NO
PRECISION = 9
TECPLOTVERSIONTOWRITE = TECPLOTCURRENT
$!WRITEDATASET "|MFBD|/3_CURVES_DATA/U-HORIZONTAL.plt"
INCLUDETEXT = NO
INCLUDEGEOM = NO
INCLUDECUSTOMLABELS = NO
ASSOCIATELAYOUTWITHDATAFILE = NO
ZONELIST = [|MinUHorizontalZone|-|MaxUHorizontalZone|]
BINARY = YES
USEPOINTFORMAT = NO
PRECISION = 9
TECPLOTVERSIONTOWRITE = TECPLOTCURRENT
$!WRITEDATASET "|MFBD|/3_CURVES_DATA/Phi-HORIZONTAL.plt"
INCLUDETEXT = NO
INCLUDEGEOM = NO
INCLUDECUSTOMLABELS = NO
ASSOCIATELAYOUTWITHDATAFILE = NO
ZONELIST = [|MinPhiHorizontalZone|-|MaxPhiHorizontalZone|]
BINARY = YES
USEPOINTFORMAT = NO
PRECISION = 9
TECPLOTVERSIONTOWRITE = TECPLOTCURRENT
$!WRITEDATASET "|MFBD|/3_CURVES_DATA/STREAMWISE_DATA.plt"
INCLUDETEXT = NO
INCLUDEGEOM = NO
INCLUDECUSTOMLABELS = NO
ASSOCIATELAYOUTWITHDATAFILE = NO
ZONELIST = [|NumZones|]
BINARY = YES
USEPOINTFORMAT = NO
PRECISION = 9
TECPLOTVERSIONTOWRITE = TECPLOTCURRENT

$!RemoveVar |Zr|
$!RemoveVar |VarNumZ|
$!RemoveVar |Z|
$!RemoveVar |VarNumY|
$!RemoveVar |Y|
$!RemoveVar |VarNumX|
$!RemoveVar |X|
$!RemoveVar |VarNumXByD|
$!RemoveVar |XByD|
$!RemoveVar |VarNumU|
$!RemoveVar |VarNumUMaxPos|
$!RemoveVar |UMaxPos|
$!RemoveVar |VarNumPhi|
$!RemoveVar |VarNumUHalf|
$!RemoveVar |UHalf|
$!RemoveVar |MeshStaggeringOffset|
$!RemoveVar |NozzleSize|
$!RemoveVar |DomainSize|
$!RemoveVar |XbyDSkip|
$!RemoveVar |NrStations|

```

Advanced_Post_Processing.mcr

```
$!RemoveVar |MinUHorizontalZone|
$!RemoveVar |MaxUHorizontalZone|
$!RemoveVar |MinPhiHorizontalZone|
$!RemoveVar |MaxPhiHorizontalZone|

#$!EndLoop #multi-case post processing#
```