A numerical investigation into the aerodynamic characteristics of AeroCity Computational Fluid Dynamics study

M. van Sluis



Challenge the future

## A NUMERICAL INVESTIGATION INTO THE AERODYNAMIC CHARACTERISTICS OF AEROCITY

## COMPUTATIONAL FLUID DYNAMICS STUDY

by

## M. van Sluis

in partial fulfillment of the requirements for the degree of

### Master of Science in Aerospace Engineering

at the Delft University of Technology, to be defended publicly on Friday, March 10, 2017 at 11:00 AM.

Student number:1378961Thesis ID number:123#17#MT#FPPSupervisor:Dr. A. G. Rao,TU DelftThesis committee:Prof. dr. G. Eitelberg,TU DelftDr. A. G. Rao,TU DelftDr. S. J. Hulshoff,TU Delftir. E. van Rees,Movares B.V.

An electronic version of this thesis is available at http://repository.tudelft.nl/.



## **ABSTRACT**

Designed as a contender for the Dutch 'Zuiderzeelijn' project, the AeroCity concept has been proposed as an alternative solution to existing high-speed railway solutions. As an Wing-in-Ground (WIG) effect vehicle, the AeroCity makes use of aerodynamic lift to levitate itself from the track, removing the need for a complex infrastructure. Already since the early days of aviation, it is known that, a wing approaching the ground, experiences a 'cushioning of air' below the wing, which enhances the lift-to-drag ratio. WIG vehicles, like the famous Ekranoplane, are designed specifically to exploit this phenomenon to their benefit. However, in very close ground proximity, the viscous interaction of the flow with the ground plane plays a prominent role. To predict the performance and stability of WIG craft, concise simulation and wind-tunnel experiments are required.

The aim of the current work is gain further insight in the aerodynamic characteristics of AeroCity using CFD simulations involving RANS equations. Previous work conducted on this topic include a CFD sensitivity study into main parameters affecting the aerodynamic performance of AeroCity [1] and a wind-tunnel experiment conducted in the LTT of the TU Delft [2]. Although these studies have attributed to the knowledge about the flow field around the AeroCity, no verification or validation of the research findings have been provided. One of the main tasks of project has been to replicate the previous wind-tunnel experiment, in order to clarify aspects of the findings that are not well understood and provide a basis of validation for the CFD model. Subsequent simulations are performed to investigate several aspects of the configuration in more detail.

Comparison of the numerical results with the previous experimental data shows that the lift and drag characteristics are in good agreement at the lower velocity range of AeroCity, but the drag deviates significantly in the higher velocity range. Analysis of the results has shown that the laminar separation bubble, which is known to exist at the leading edge of the side-plate, is under-predicted by the CFD model. Application of a transition turbulence model significantly improved the modelling of the separation bubble, to such an extent that the simulated local flow is in good agreement with fluorescent oil film traces observed during the windtunnel experiment. Furthermore, it is found that the boundary layer profiles are in good agreement up to the location of maximum thickness. Further downstream, the simulated boundary layer appears to be insensitive to the local adverse pressure gradient, in contrast to the measurements in the wind-tunnel. Application of a damping function for the eddy viscosity in the viscous sub-layer, did not improve the numerical results.

Despite of the noticed limitations of the current numerical model, the influence of the ground boundary condition on the aerodynamic characteristics has been investigated, by applying a moving ground boundary condition. Unlike expected, the influence is very limited in this particular case, due to the scale (1:20) of the wind-tunnel model. Further, inclusion of a track wall showed that the aerodynamic performance is reduced considerably, as the horse-shoe vortex remains close to the vehicle and merges with the upper tip-vortex. Although the parameters of the track geometry were not varied, these findings suggest that, aside from further research, other track geometries are to be considered.

# PREFACE

This thesis is part of the graduation project to fulfil the requirements for obtaining the MSc degree in Aerospace Engineering at Delft University of Technology. The past year and a half, during which I worked on this project, can be described as both challenging and rewarding. The road towards the point of completion of my thesis, has not been without several obstacles along the way. During the journey, several people have provided me with guidance and support. Therefore, I would like to direct some words of gratitude.

First, I would like to express my sincere gratitude to Georg Eitelberg, who provided me with crucial knowledge about wind-tunnel experiments and managed to ask the right critical questions, to regain focus on my goals and improve the quality of my work. Further, I would like to thank André Perpignan for his assistance, during the time that I was stuck on the generation of the initial three-dimensional mesh. Also, I would like to thank Nando Timmer for providing me with measurement data of the boundary layer of the LTT wind-tunnel and his overall preparedness to assist. I would also like to mention Richard Dwight, for the open discussion about the discrepancy in the numerical result, caused by software algorithm. And particularly, I would like to express my gratitude to Arvind Rao. During this project, his patience, kindness and encouragement have helped me to persist and keep my project moving forward.

Finally, I would like to thank my friends and family for their continuous support, both during the last couple of months and throughout my academic career. A special mention is reserved for my father, who provided me with the necessary computer hardware, which has proven to be of great value during this project. And certainly not last, I would like to thank Tara ten Hove, for her love and mental support that I received throughout this period.

*M. van Sluis* Delft, March 2017

# **CONTENTS**

No	Nomenclature xii				
Li	List of Figures xiii				
Li	st of ]	Tables xv	ii		
1	Intr	oduction	1		
•	T		-		
2	2 1	Wing in Cround offect	Э Б		
	2.1	211 Chord dominated effects	5		
		2.1.1 Chord dominated effects	0		
	22	Aerodynamic analysis of AeroCity	2		
	2.2	221 Numerical analysis	2		
		222 Wind-tunnel testing	6		
3	Proj	ject definition 2	3		
	3.1	Mission statement	:3		
	3.2	Research question	25		
		3.2.1 Secondary research questions	25		
	3.3	Methodology	:6		
4	Nun	nerical model 2	29		
	4.1	Reynolds Averages Navier-Stokes equations	29		
	4.2	Turbulence modelling	60		
		4.2.1 Spalart-Almaras	31		
		4.2.2 k-epsilon models	51		
		4.2.3 k-omega SST	52		
		4.2.4 Reynolds Stress Model	3		
	4.3	Discretization	4		
		4.3.1 Second-Order Upwind	4		
		4.3.2 <b>QUICK</b>	5		
		4.3.3 MUSCLE	5		
	4.4	Modelling of the boundary layer	6		
		4.4.1 Near-wall meshing guidelines	8		
	4.5	Solution procedure	8		
5	CFD	Case studies 4	1		
	5.1	NACA 0012	1		
	5.2	NACA 4412	13		
	5.3	NACA 4412 IGE	15		
	5.4	Wing with end-plates	8		
	5.5	Conclusions from the test-cases	52		
e	Set	up and Pro-simulation	3		
0	6 1	Ip and r rc-simulation     5       Experimental setup     5	3 (2		
	0.1 6 2	Wall boundary layer	טי גב		
	0.2 6.3	Mesh construction	ы. С		
	0.5	631 Boundary layer mech	0 7		
	64		;1 ;0		
	6.5	Mesh dependency of results	50 50		
	0.0	meen appendency of results	.0		

Sim	nulation Results	63
7.1	Force measurements	63
7.2	Pressure distribution	65
7.3	Flow visualisation.	67
7.4	Turbulence	69
	7.4.1 Vortex visualization	71
7.5	Boundary layer profiles	73
	7.5.1 Upper surface	73
	7.5.2 Wall boundary layer	74
	7.5.3 Separation bubble	76
7.6	Effect of a moving ground.	77
7.7	Inclusion of a track wall	79
Mod	del adaptation	85
8.1	Refinement of the prism layer.	85
8.2	Transition model	86
8.3	Verification of boundary layer modelling	90
8.4	Damping of the eddy viscosity	92
Con	nclusion	97
9.1	Aerodynamic performance	97
9.2	Validation of the results	98
9.3	Recommendations	99
Ann	andix A. Turbulanca modele	01
	Snalart-Allmaras	01
A 2	k-ensilon Standard	02
A.3	k-ensilon BNG	03
A.4	k-epsilon Realizable.	04
A.5	k-omega BSL	05
A.6	k-omega SST	05
A.6 A.7	k-omega SST	05 06
A.6 A.7 A.8	k-omega SST	05 06 08
A.6 A.7 A.8 App	k-omega SST	105 106 108 11
A.6 A.7 A.8 App App	k-omega SST	105 106 108 11
	Sim 7.1 7.2 7.3 7.4 7.5 7.6 7.7 8.1 8.2 8.3 8.4 9.1 9.2 9.3 <b>App</b> A.1 A.2 A.3 A.4 A.5	Simulation Results         7.1       Force measurements         7.2       Pressure distribution         7.3       Flow visualisation         7.4       Turbulence         7.4       Turbulence         7.4       Turbulence         7.4.1       Vortex visualization         7.5       Boundary layer profiles         7.5.1       Upper surface         7.5.2       Wall boundary layer         7.5.3       Separation bubble         7.5.4       Separation bubble         7.5       Federate wall         7.6       Effect of a moving ground         7.7       Inclusion of a track wall         8.1       Refinement of the prism layer         8.2       Transition model         8.3       Verification of boundary layer modelling         8.4       Damping of the eddy viscosity         9.1       Aerodynamic performance         9.2       Validation of the results         9.3       Recommendations         9.4       Spalart-Allmaras         9.5       Verision Realizable         9.1       Aerodynamic performance         9.2       Validation of the results         9.3       K-epsilon

# NOMENCLATURE

## List of Abbreviations

Symbol	Description	Dimensions	Units
AR	Aspect Ratio		-
HS	Height Stability		
LE	Leading Edge		
TE	Trailing Edge		
BSL	Baseline $(k - \omega \text{ model})$		
CAD	Computer Aided Design		
CAD	Computer Aided Design		
CNC	Computer Numerical Control		
ECC	Error-correcting Code		
EWT	Enhanced Wall Treatment		
FUN3D	Fully Unstructured Navier-Stokes 3D		
FVM	Finite Volume Method		
HART	Hyuga Aerodynamic Research by Towing		
IGE	In Ground Effect		
LES	Large Eddy Simulation		
LEVM	Linear Eddy Viscosity Model		
LTT	Low Turbulence Tunnel		
MAGLEV	Magnetic Levitation		
MUSCL	Monotome Upstream-Centred Scheme for Conserva- tion Laws		
NACA	National Advisory Committee for Aeronautics		
NASA	National Aeronautics and Space Administration		
NEW	Non-Equilibrium Wall-function		
NLR	National Aerospace Laboratory		
OGE	Out-of Ground Effect		
PDE	Partial Differential Equation		
PIV	Particle Image Velocimetry		

PRESTO	Pressure Staggering Option
QUICK	Quadratic Upstream Interpolation for Convective Ki- netics
RAID	Redundant Array of Independent Disks
RANS	Reynold's Averaged Navier-Stokes
RNG	Renormalized Group Method
RSM	Reynolds Stress Model
SA	Spalart-Allmaras
SIMPLE	Semi-Implicit Method for Pressure Linked Equations
SIMPLEC	Semi-Implicit Method for Pressure Linked Equations Consistent
SM	Stability Margin
SOU	Second-Order Upwind
SSD	Solid State Disk
SST	Shear Stress Transport ( $k - \omega$ model)
SW	Standard Wall-function
TGV	Train á Grande Vitesse

Total Pressure Loss Technical University

Total Variation Diminishing

Unsteady Reynold's Averaged Navier-Stokes

**User Defined Function** 

Wind-Tunnel

т

## List of Symbols

TPL

TU TVD

UDF

WT

URANS

Symbol	Description	Dimensions	Units
α	Angle of attack	heta	0
$\delta_{ij}$	Kronecker delta		-
l	Turbulence length scale	L	т
$\epsilon$	Turbulence dissipation rate	$L^2 T^{-3}$	$J \mathrm{kg}^{-1} s^{-1}$
κ	von Kármán constant		-
$\mu$	Viscosity	$WL^{-1}T^{-1}$	$\operatorname{kg} m^{-1} s^{-1}$
$\mu_t$	Turbulent or eddy viscosity	$WL^{-1}T^{-1}$	$\operatorname{kg} m^{-1} s^{-1}$
$\nabla$	Gradient operator		
ν	Kinematic fluid viscosity	$WL^{-1}T^{-1}$	$\mathrm{kg}m^{-1}s^{-1}$

ω	Vorticity	$T^{-1}$	$s^{-1}$
$\Omega_{ij}$	Rotation tensor	$T^{-1}$	$s^{-1}$
$\phi$	Transportable quantity		-
$\phi_{ij}$	Pressure-strain coupling	$WL^{-1}T^{-1}$	$\mathrm{kg}m^{-1}s^{-1}$
ρ	Fluid density	$WL^{-3}$	${ m kg}m^{-3}$
$\sigma_k$	Prandtl number		-
τ	Shear stress	$WT^{-2}L^{-1}$	N
$\tilde{\nu}$	Asymptotic eddy viscosity	$WL^{-1}T^{-1}$	$\mathrm{kg}m^{-1}s^{-1}$
b	Wing span	L	т
С	Chord length	L	m
$C_D$	Drag coefficient		-
$C_L$	Lift coefficient		-
$C_M$	Pitching moment coefficient		-
D	Total drag force		Ν
$D_{L,ij}$	Molecular diffusion	$WL^{-1}T^{-1}$	$\mathrm{kg}m^{-1}s^{-1}$
$D_{T,ij}$	Turbulent diffusion	$WL^{-1}T^{-1}$	$\mathrm{kg}m^{-1}s^{-1}$
$E_{ij}$	Strain rate tensor, see also $S_{ij}$	$T^{-1}$	$s^{-1}$
$e'_{ij}$	Fluctuating rate of deformation tensor	$T^{-1}$	$s^{-1}$
F	Blending function		-
$G_{ij}$	Buoyancy production	$WL^{-1}T^{-1}$	$\mathrm{kg}m^{-1}s^{-1}$
h	Elevation height	L	т
Κ	Mean flow kinetic energy	$WL^2T^{-2}$	J
k	Turbulent kinetic energy	$WL^{2}T^{-2}$	J
L	Total lift force		Ν
L/D	Lift-over-drag ratio		-
р	Pressure	$WL^{-2}$	$\mathrm{kg}m^{-2}$
$P_{ij}$	Turbulence production	$WL^{-1}T^{-1}$	$\mathrm{kg}m^{-1}s^{-1}$
Re	Reynold's number		-
$S_{ij}$	Strain rate tensor	$T^{-1}$	$s^{-1}$
Т	Temperature		K
t	Time scalar	Т	S
$T_{\infty}$	Free stream turbulence intensity		-
и	Cartesian velocity component in x-direction	$LT^{-1}$	$ms^{-1}$
$u^+$	Non-dimensional fluid velocity		-

ν	Cartesian velocity component in y-direction	$LT^{-1}$	$ms^{-1}$
w	Cartesian velocity component in z-direction	$LT^{-1}$	$ms^{-1}$
$X_{\alpha}$	Neutral point w.r.t. angle of attack	L	т
$X_h$	Neutral point w.r.t. elevation height	L	т
$y^+$	Non-dimensional distance to the wall		-

# **LIST OF FIGURES**

1.1	Prototype of the AeroTrain [3]	1
1.2	MD-160 (Lun-class) Ekranoplan [4]	1
1.3	Computer generated image of AeroCity concept by Movares B.V.	2
1.4	Artist impression of the co-existence of AeroCity with existing infrastructure by Movares B.V.	3
2.1	Pressure isobars for a NACA 0012 at $Re = 4.0 \cdot 10^6$ in close ground proximity [5]	6
2.2	Streamlines of the flow for a stationary ground at $h/c = 0.05$ [5]	7
2.3	Streamlines of the flow for various ground boundary conditions $h/c = 0.05$ , $\alpha = 6^{\circ}[5]$	7
2.4	Relative change in lift-coefficient of airfoils in ground effect compared to out-of-ground effect $(h/c = 1.5)$ lift coefficient [6]	8
2.5	Overlay of the PIV measurements of the horizontal velocity component at $h/c = 0.3$ compared with the results of $h/c = 1.5$ , for low-lift conditions (top) and high-lift conditions (bottom). [6]	9
2.6	Streamlines in the y-z plane ( $x/c = 0.75$ ) at $h/c = 0.05$ for a finite wing ( $AR = 2$ ) without (left) and with (right) end-plates [7]	11
2.7	Geometry layout and lift and drag polar of the three configurations for different Reynolds num-	12
2.8	Total pressure loss of the AeroCity baseline configuration [1]	12
2.0	Effect of end plate elevation $k$ and body angle of attack $\alpha$ on the aerodynamic performance of	15
2.3	AeroCity [1]	14
2 10	Stability Margin [1]	16
2.11	Wind-tunnel model [2]	16
2.12	Variation in lift coefficient with <i>Re</i> for two settings of gap height at $h/c = 0.05$	17
2.13	Variation in lift coefficient with <i>Re</i> for two settings of gap height at $h/c = 0.10$	17
2.14	Variation in drag coefficient with <i>Re</i> for two settings of gap height at $h/c = 0.10$	17
2.15	Variation in drag coefficient with <i>Re</i> for two settings of elevation height at $\alpha = 5^{\circ}$	17
2.16	Comparison of lift coefficient for two settings of body angle of attack at similar height parame-	10
0.17	Comparison of drag coefficient for two settings of he dy angle of attack at similar height param	18
2.17	eters	18
2.18	Comparison of lift-over-drag ratios for two settings of body angle of attack at similar height parameters	18
2.19	Comparison of moment coefficients for two settings of body angle of attack at similar height parameters	18
2.20	Flow visualization of the AeroCity upper surface at $U = 100 \text{ m/s} [2]$	19
2.21	Comparison of lift coefficient obtained by experiment and CFD	20
2.22	Comparison of drag coefficient obtained by experiment and CFD	20
2.23	Total pressure contours for 30m/s (left) and 50m/s (right)	21
3.1	Breakdown of the various components involved in the validation process of AeroCity	24
4.1	Blending function $F_1$ for various wall velocity profiles [9]	33
4.2	Example of a two-dimensional control volume. Indicated are the cell centers $c_0$ and $c_1$ , face area surface vector $\vec{A}$ and the vectors between the cell and face centers $\vec{r_0}$ and $\vec{r_1}$ [10]	35
4.3	Sweby's diagram , showing the region for second-order TVD (hi-lighted in grey) [11]	36
4.4	Non-dimensional boundary layer velocity profile as a function of wall distance in comparison with experimental data [12]	38
4.5	Block-schematic of the Fluent solver module for coupled and non-coupled solving methods	39

5.1	Close-up of the structured grid around the airfoil body. Note that domain is split in a laminar part (grey) and a turbulent part (orange)	41
5.2	Detail of the mesh at the trailing edge of the NACA 0012	42
5.3	Overview of the velocity rakes at the trailing edge of NACA 4412 at $\alpha = 13.89^{\circ}$ [13]	44
5.4	Comparison of the non-dimensionalized horizontal u-velocity measured along the velocity-	
	rakes with wind-tunnel results [14]	46
5.5	Verification of Spalart-Allmaras results with CFD results obtained by NASA [13]	46
5.6	Close-up of the NACA 4412 IGE mesh. The domain is divided into a laminar zone (grey) and a fully turbulent zone (green)	47
5.7	Comparison of numerical results for pressure distribution $Re = 800,000$ with experimental data [15][16]	48
5.8	Drawing of the test-setup as used in the experiment by Kikuchi <i>et al</i> [16]	48
5.9	Photograph of the wind-tunnel setup featuring the wing with half-airfoil end-plates [17]	49
5.10	Contours of static pressure over the Clark Y wing using the $k - \omega$ SST model	51
5.11	Contours of skin friction coefficient over the Clark Y wing using the $k - \omega$ SST model	51
5.12	Streamlines around the Clark Y wing, coloured with the local velocity of the flow, using the $k-\omega$ SST model	51
6.1	Photograph of the aerodynamic model installed in the wind-tunnel test-section [2]	54
6.2	Schematic drawing of the Low Turbulence Tunnel (LTT) facility of the TU Delft [18]	55
6.3	Velocity profiles of the LTT upper wall measured w.r.t. the start of the test-section	56
6.4	Plot of a regression curve of the displacement thickness's of the LTT upper wall	56
6.5	Overview of the computational domain	57
6.6	Capture of the surface mesh of AeroCity	58
6.7	Close-up of the surface mesh near the leading edge of AeroCity	58
6.8	Close-up of the surface mesh near the trailing edge of AeroCity	58
6.9	Wall Yplus contours of the AeroCity model and the near-ground plane at $V = 40m/s$	59
6.10	Example of convergence history of $C_L$	60
6.11	Example of convergence history of $C_D$	60
6.12	Dependency of force coefficients on total mesh size	60
6.13	Average time per iteration for various mesh sizes	60
7.1	Comparison of lift and drag coefficients with experimental data [2]	64
7.2	Comparison of pitching moment coefficient and relative deviation with experimental data [2]	64
7.3	Relative deviation with experimental data [2] and graphical location of the centre of pressure	64
7.4	Comparison of $C_n$ pressure distribution at $z/c = 0$ with experimental data at 40m/s [2]	64
7.5	Pressure distribution at various span-wise locations, measured at $U = 40$ m/s	65
7.6	Comparison of the pressure distribution at $z/c=0.00$ and $z/c=0.15$ for two flow velocities with	
	experimental data [2].	65
7.7	Pressure contour plot of the upper (left) and lower (right) surface for $U = 40 \text{ m/s}$	66
7.8	Pressure contour plot of the upper (left) and lower (right) surface for $U = 80 \text{ m/s}$	66
7.9	Isometric views of the streamlines around AeroCity from the front (left) and (right) at $U = 80$ m/s	67
7.10	Comparison of upper surface oil flow pattern between the wind-tunnel ( $U = 100 \text{ m/s}$ ) and CFD	
	$model (U = 80 m/s) \dots \dots$	68
7.11	Surface particle tracks obtained by CFD of the the front potion of AeroCity ( $U = 80$ m/s)	69
7.12	Close-up of the fluorescent oil film of the front region of the side-plate.[2]	69
7.13	Contours of the total pressure loss coefficient at various chord-wise locations ( $U = 40 \text{ m/s}$ )	70
7.14	Contours of lateral w velocity-component at various chord-wise locations ( $U = 40 \text{ m/s}$ )	70
7.15	Iso-surface of Q-criterion ( $Q/Q_{max} = 0.001$ ) and contours of turbulence kinetic energy ( $U = 40$	
_	m/s)	72
7.16	Iso-surface of Q-criterion $(Q/Q_{max} = 0.001)$ and contours of turbulence kinetic energy $(U = 40)$	_
_	m/s)	72
7.17	Iso-surface of Q-criterion ( $Q/Q_{max} = 0.001$ ) and contours of turbulence kinetic energy ( $U = 80$	
- 10	m/s)	72
7.18	Measurement of velocity profile perpendicular to the surface at $z = 0$	74
1.19	measurement of velocity prome perpendicular to the surface at $z = 0$	74

7.20 Meas 7.21 Meas	surement of velocity profile perpendicular to the surface at $z = 0$	74 74
of the	e LTT test-section $\dots$ represented by profiles at various stations upstream of the AeroCity model ( $U$ =	75
40 m	parison of ground velocity promes at various stations upstream of the AeroCity moder $(0 = 1/s)$	75
experies	rimental data [2]	76
7.25 Samp expe	ples of total pressure inside the separation bubble at $y/c=0.12$ and $x/c=0.08$ compared to rimental data [2]	76
7.26 Samp exper	ples of total pressure inside the separation bubble at $y/c=0.12$ and $x/c=0.15$ compared to rimental data [2]	76
7.27 Samp exper	ples of total pressure inside the separation bubble at $y/c=0.12$ and $x/c=0.25$ compared to rimental data [2]	76
7.28 Press	sure distribution at $z/c=0.00$ for two different ground boundary conditions ( $U = 40$ m/s)	78
7.29 Press	sure distribution at $z/c = 0.15$ for two different ground boundary conditions ( $U = 40 \text{ m/s}$ )	78
7.30 Veloc 7.31 Non-	city profile of the boundary layer on the upper surface measured at $x/c=0.02$ and $U=40$ m/s dimensionlized velocity profile of flow channel underneath Aerocity at $x/c=0.10$ and $U=$	78
40 m	$V_{\rm S}$	78 00
7.33 Iso-s	urface of Q-criterion $(Q/Q_{max} = 0.001)$ for $U = 40$ m/s and stationary ground b.c.	30 80
7.34 Iso-s wall	urface of Q-criterion $(Q/Q_{max} = 0.002)$ for $U = 40$ m/s and moving ground b.c. and track	80
7.35 Effec	et of track wall inclusion on $C_P$ distribution at $z/c = 0.00$ and $U = 40$ m/s	81
7.36 Effec	et of track wall inclusion on $C_P$ distribution at $z/c = 0.15$ and $U = 40$ m/s	81
7.37 Close	e-up of the vortices near the aft portion of AeroCity on the inside of the track wall (U=40 m/s) $\{$	82
8.1 Boun	ndary layer velocity profile at $x/c=0.21$ and $U=65$ m/s for two mesh configurations 8	86
8.2 Boun 8.3 Iso-s	Indary layer velocity profile at $x/c = 0.21$ and $U = 65$ m/s for two mesh configurations 8 surface of Q-criterion ( $Q/Q_{max} = 0.001$ ) and turbulence kinetic energy contours at $U = 40$	36
m/s 1 8.4 Iso-s	for $r = 1.30$ urface of Q-criterion ( $Q/Q_{max} = 0.001$ ) and turbulence kinetic energy contours at $U = 40$	57
m/s 1 8.5 Iso-s	for $r = 1.15$	37
m/s o	obtained by Transition SST model	87
8.6 Skin	friction coefficient over the upper surface along $z/c = 0.00$ at $U = 40$ m/s	88
8.7 Veloc 8.8 Oil fl	Low particle traces over the side-plate of AeroCity, obtained with Transition SST model at $V = 65 \text{ m/s}$	38
U = 4	40  m/s	89 80
8.10 Com	parison of velocity profile over a zero pressure gradient flat plate at $r = 0.97$ with numerical	59
data	by NASA [13]	90
8.11 Com data	parison of skin friction coefficient over a zero pressure gradient flat plate with numerical by NASA [13]	90
8.12 Com laver	parison of viscosity ratio over a zero pressure gradient flat plate using a refined boundary mesh $(r = 1.10)$	90
8.13 Plot o	of SST blending functions for a zero pressure gradient flat plate at $x = 0.97$ with numerical by NASA [13]	an
8.14 Plot o	of blending function $F_{SST}$ versus $Y^+$ of the boundary layer at $x = 0.97$	93
8.15 Com ing fi	parison of the velocity profile of the boundary layer at $x = 0.97$ with and without the damp- unction $F_{SST}$	93
8.16 Plot o	of the eddy viscosity ratio along the boundary layer at $x/c = 0.97$ with and without the UDF	
activa 8.17 Plot o	ated	93
UDF	f enabled and data by NASA [13]	93
8.18 Plot o	of the calculated wall distance 'CWallDistance' and the actual wall distance $y$	94

8.19	Velocity profiles of the boundary layer at the upper surface of Aerocity at $x/c= 0.98$ and $U = 65$	
	m/s	4
8.20	Velocity profiles of the boundary layer at the upper surface of Aerocity at $x/c=0.46$ and $U=65$	
	m/s	5
8.21	Velocity profiles of the boundary layer at the upper surface of Aerocity at $x/c=0.65$ and $U=65$	
	m/s 99	5
<b>B.</b> 1	Non-dimensiolized u-velocity measured along the velocity rakes	2
<b>B.2</b>	Non-dimensiolized v-velocity measured along the velocity rakes 112	2
B.3	Non-dimensiolized u-velocity measured along the velocity rakes	3
<b>B.4</b>	Non-dimensiolized v-velocity measured along the velocity rakes	3
<b>B.5</b>	Non-dimensiolized u-velocity measured along the velocity rakes	4
<b>B.6</b>	Non-dimensiolized v-velocity measured along the velocity rakes	4
<b>B.7</b>	Non-dimensiolized u-velocity measured along the velocity rakes	5
<b>B.8</b>	Non-dimensiolized v-velocity measured along the velocity rakes	5
B.9	Non-dimensiolized u-velocity measured along the velocity rakes	6
B.10	Non-dimensiolized v-velocity measured along the velocity rakes	6
<b>B.11</b>	Non-dimensiolized u-velocity measured along the velocity rakes	7
B.12	Non-dimensiolized v-velocity measured along the velocity rakes	7
B.13	Pressure distribution obtained with the SST $k - \omega$ compared to experimental data [14] 118	8
<b>B.14</b>	Verification of SST $k - \omega$ results with CFD results obtained by NASA [13]	8

# **LIST OF TABLES**

Overview of workstation specifications	26
Comparison of coefficients at $\alpha = 10^{\circ}$ for a NACA 0012 with experimental data [19] and numerical results obtained by NASA [13]	43
Comparison of force and moment coefficients with experimental data	47
Specifications of the aerodynamic model [17]	49
Wind-tunnel blockage correction factors [17]	49
Mesh dependency study for three different mesh sizes. Data compared with experimental data	50
Comparison of CED predictions with experimental data by Kumar [17] using different turbu-	50
lence models.	50
Reference values for ambient conditions, as recorded during the wind-tunnel experiment [2]	54
Comparison of force coefficients between two ground boundary conditions for $U = 40$ m/s	77
track wall inclusion for $U = 40 \text{ m/s} \dots \dots$	82
Influence of prism growth factor r on computational parameters	85
Influence of prism growth factor <i>r</i> on predicted force coefficients and aerodynamic efficiency	85
Comparison of Transition SST results with force coefficients obtained by SST model and $r = 1.15$ ( $U = 40$ m/s)	89
	Comparison of coefficients at $\alpha = 10^{\circ}$ for a NACA 0012 with experimental data [19] and numer- ical results obtained by NASA [13]

# 1

## **INTRODUCTION**

Driven by the ever increasing need for faster means of transportation, people have always been searching for faster and more efficient ways to travel. One way to achieve both goals, is to reduce the drag force acting on the vehicle. For wheeled or tracked vehicles, the ground friction drag is often a major source of the total vehicle drag. In the case of railway transportation, the drag can primarily be reduced by minimizing the total vehicle weight and optimization of the aerodynamic efficiency [20]. However, with the rise of high speed trains, the wheels give rise to an additional problem. The high forces and wear on both the wheel and tracks require additional effort in design, construction, and maintenance, increasing the total system cost considerably.

One obvious solution to this problem would be to simply get rid of the wheels. By levitation of the train from the ground, one eliminates one of the major drag components. MAGLEV trains use an alternating induced magnetic field to levitate the vehicle from the surface, using a system of electro-magnets incorporated in the track. Although MAGLEV trains have demonstrated the ability to achieve speeds up to 600 km/h and provide a higher energy efficiency at high speeds, when compared to conventional trains, the high cost and complexity of the infrastructure remain challenges that are to be solved. [20] A different approach was developed by Jean Bertin in the late 1960s with the introduction of the Aerotrain [3]. A picture of the original AeroTrain prototype is shown in Figure 1.1. By allowing the air to create a cushion of air between the track and the train, levitation of the train is accomplished. Similar research was conducted in the United Kingdom and the United States of America, where the Rohr Aerotrain was build under licence by Rohr Industries. Although several prototypes were successfully tested, the large energy consumption required to overcome the initial momentum drag and the rise of conventional high-speed trains, such as the TGV, prevented the technology of being adopted. [21]

Simultaneously, in the USSR, the famous Ekranoplane [4] was developed, as a means for faster transportation over water. The Ekranoplane, shown in Figure 1.2, made use of the so-called Wing-In-Ground (WIG) effect to achieve these relative high speeds, without the penalty of high drag. First studied in the early 1920's [22],



Figure 1.1: Prototype of the AeroTrain [3]

Figure 1.2: MD-160 (Lun-class) Ekranoplan [4]



Figure 1.3: Computer generated image of AeroCity concept by Movares B.V.

the WIG effect, or ground effect, is an aerodynamic phenomena that occurs when a lifting surface is in close proximity to the ground. Already in the early days of aviation, pilots observed a 'cushioning of air' beneath the wings during landing, causing the landing distance to increase. The development of WIG craft, such as the Ekranoplane, sparked a renewed interest in the ground effect phenomenon [23]. In general, an increase in lift and a reduction in drag is observed when approaching the ground [23][15][24]. A second development aiding to the research into the ground effect was the dawn of down force generating surfaces for racing cars [25].

Inspired by the potential of Wing-in-Ground effect craft, the AeroCity concept was developed by Movares B.V. in conjunction with the Dutch aerospace laboratory (NLR). Designed as an alternative high-speed transportation system for the Dutch 'Zuiderzeelijn' project [26], the AeroCity utilizes the WIG effect to remove the need for a complex infrastructure. The promise of the AeroCity concept is to deliver a high-speed transportation solution, that is both energy and cost effective. More-over, the AeroCity would be able to operate over relatively short to intermediate distances, since the acceleration time should be considerably lower compared to existing railway solutions. A computer generated image of the initial concept, as developed by Movares, is shown in Figure 1.3. The overall length of the AeroCity is 21m, width is equal to 8m and the vehicle is 4.2m tall. Depending on the seating arrangement, it has the capacity to transport up to 100 passengers at a design cruise speed of 360 km/h. Total vehicle weight is estimated to be 10,500 kg [1].

Unlike most other WIG craft, which utilize low aspect ratio wings to provide the required lift force, in the case of AeroCity, the entire body acts as a lifting surface. The main body of the vehicle resembles the centre section of a wing. Side-plates on both sides of the vehicle help to maintain the 'air cushion' underneath the vehicle. [27]. Due to the limitations imposed by the infrastructure, the aspect ratio of the AeroCity is low. The addition of the side-plates helps to enhance the effective aspect ratio. [7] Although the AeroCity does require a separate infrastructure, the complexity of the track can be kept relatively low. Since the levitation, unlike a MAGLEV system, is provided by an aerodynamic lift force, the main purpose of the track will be to provide lateral guidance. Electromagnets, incorporated in the track, should keep the AeroCity clear from the track wall, while also propelling the vehicle in forward direction. Since the AeroCity is foreseen to have relatively low noise characteristics, the infrastructure could be integrated well with existing infrastructure. [1]. An impression of the latter is provided in Figure 1.4.

The aerodynamic characteristics of the AeroCity concept have been previously studied by Nouwens [1] and Nasrollahi [2]. Nouwens conducted a Computational Fluid Dynamics (CFD) study, to examine the influence of the main design parameters, such as the angle of attack and elevation height of the body, on the aerody-



Figure 1.4: Artist impression of the co-existence of AeroCity with existing infrastructure by Movares B.V.

namic performance. It was concluded that the early aerodynamic performance estimations are promising and do not reveal any mayor deficiencies of the basic aerodynamic design. To validate the numerical investigation, a series of wind-tunnel experiments has been carried out by Nasrollahi. The experimental data partially confirmed the observations made by Nouwens. The experimental results largely confirmed the observations made during the previous numerical analysis by Nouwens. Nevertheless, the experimental data introduced some new questions which could, at the time, not be answered to a full extent. Despite a replication of the wind-tunnel conditions in an additional CFD experiment, no explanation for the recorded data could be provided.[2]

The present work will be a continuation of the research into the aerodynamic characteristics and performance of the AeroCity concept. Using the knowledge that has already been gathered about the AeroCity's aerodynamic configuration, a series of elaborate CFD simulations will be performed to examine the aerodynamic characteristics of AeroCity in more detail. More-over, it will try to answer some of the questions regarding the aerodynamic phenomena involved, that have remained unanswered since the wind-tunnel experiment. By replication of the conditions in the wind-tunnel and comparison of the numerical results with the experimental data, more insight into the aerodynamic characteristics of AeroCity should be gained. In turn, the wind-tunnel experiment should provide a basis for validation of the CFD model. The latter will significantly improve the credibility of the research findings. Using the validated CFD model, more elaborate simulations can be performed on the aerodynamic model of the AeroCity.

# 2

## **LITERATURE SURVEY**

In order to obtain a thorough overview of the past and active research efforts into the Wing-In-Ground (WIG) effect, a comprehensive literature survey has been conducted. The result of this survey will be discussed in this chapter. In Section 2.1, the aerodynamic phenomena involved of the WIG effect are discussed in more detail. The main purpose is to distinguish the various aerodynamic mechanisms that come into play and identify how this effects the experimental techniques that are required for proper simulation of the WIG-effect. Next, in Section 2.2, the previous research that has been performed on the aerodynamic characteristics of the AeroCity is discussed in more detail.

### **2.1.** WING-IN-GROUND EFFECT

Although often referred to as the ground effect, the WIG effect actually entails multiple physical phenomena that alter the behaviour of the flow around a wing in ground proximity. In general, these effects can be categorized into chord dominated effects and span dominated effects. The distinction between 2D and 3D effects is helpful for analysis, as a fundamental understanding of the results obtained by sectional testing will aid with identifying flow characteristics during finite wing testing. As such, chord dominated effects will be discussed first followed by a review of span dominated flow effects.

### **2.1.1.** CHORD DOMINATED EFFECTS

When an airfoil operates in close vicinity to the ground, the flow is modified due to the presence of the ground surface. In general, the dividing streamline and the stagnation point can be seen to move down. As the clearance between the airfoil and the ground surface reduces, the pressure underneath the airfoil increases, as the air tends to stagnate. This is referred to as the ram effect. Ahmed [28] showed with the use of wind tunnel experiments that for the NACA 0015 symmetrical airfoil, flow is redirected over the upper surface as the flow stagnates underneath the airfoil. The increased flow over the upper surface increased the suction, enhancing the lift coefficient further. As the stagnation point moved down with angle of attack and decreasing ground clearance, the velocities near the suction peak were observed to be considerably higher than compared to the case of the airfoil out of ground effect. The increased mean velocity over the upper surface caused separation of the flow to start almost near the trailing edge, for all angle of attack up to  $\alpha = 10^\circ$ . However, at  $\alpha = 12.5^\circ$ , the increased suction peak caused earlier separation on the upper surface , resulting in a thicker wake.

Another study by Ahmed [29] shows, that for thick symmetrical airfoils at lower ground clearances and relatively low angles of attack, a divergent-convergent path exists beneath the airfoil lower surface and the ground. Due to the Venturi [30] effect a local suction zone is created underneath the airfoil, decreasing the lift. Furthermore, the increased flow velocity enlarges the skin friction and consequently increases the total drag. Ahmed [31] showed that the effect of the convergent-divergent duct could be approximated by assuming a 1D non-viscous flow and using Bernoulli's equation. The resulting pressures did not match exactly, as the loss of pressure due to viscous effects were neglected, but the general trend was in good agreement. Furthermore, Ahmed [15] observed flow separation at the lower surface due to strong suction effect. In the divergent part underneath the airfoil, the flow is unable to fully recover the pressure as a result of a very strong pressure gradient. In order to prevent the negative effects of the Venturi effect, Ahmed [31] suggests to use an airfoil



Figure 2.1: Pressure isobars for a NACA 0012 at  $Re = 4.0 \cdot 10^6$  in close ground proximity [5]

with a relatively rearward position of the location of maximum thickness and a flattish lower surface with a small inclination, at the design cruise angle of attack. A similar solution is opted for by Lee [8] and Fink and Lastinger [23], using a modified Glenn Martin 21 airfoil with a flat lower surface from x/c = 0.7 - 1.0.

A numerical analysis by Hsiun [24] based on the RANS equations shows that the flow over the leading edge of a NACA 4412 at low ground clearances and relatively low Reynolds number ( $Re = 2.0 \cdot 10^5$ ) is influenced by the ground boundary layer thickness. Although the lift coefficient increases with Reynold number, both in and out of ground effect, Hsiun concludes that the effect of Reynolds number is more pronounced at low ground clearances, as the boundary layer thickness increases for lower Re. The thicker boundary layer reduces the mass flow of air underneath the airfoil and consequently, less air is redirected over the leading edge as well. A further discovery made by Hsiun, is the existence of a circulation region underneath the leading edge for a ground clearance of h/c = 0.05 at moderate angles of attack. However, no explanation is given for the phenomenon, other than it is 'because of the very small h/c value, the high angle of attack and the viscous effect'.

A better explanation is given by Yang [5]. In a numerical experiment, Yang shows that the ground boundary layer is subject to an adverse pressure gradient near the leading edge, due to the ramming effect underneath the airfoil (Figure 2.1). As a consequence, the boundary layer grows in thickness accordingly. After the leading edge the flow accelerates under the airfoil and the boundary layer is restored to its original state. However, the presence of the boundary layer under the airfoil changes the effective relative flight height. For a geometric ground clearance of h/c = 0.10 the effective ground clearance is reduced to as much as  $h_e/c = 0.076$  in the case of Yang. When the ground clearance is reduced further, the adverse pressure gradient becomes too large for the boundary layer to overcome and causes the boundary layer to separate from the surface. The separated flow reattaches again downstream [32]. Between the point of separation and reattachment a zone of reversed flow or recirculation exists. As the pressure under the airfoil increases with angle of attack, the separation bubble becomes more prominent with angle of attack. As can be seen from Figure 2.2, the separation bubble alters the streamlines of the flow considerably. Due to the presence of the bubble, the flow is pushed upwards. This causes the stagnation point to move forward. As a consequence, the air flow over the leading edge is has a lower velocity, causing the suction peak to decrease. Furthermore, as the bubble redirects more flow over the upper surface, the ramming effect under the airfoil is reduced as well. Hence, the formation of a separation bubble on the ground, reduces the aerodynamic performance considerably. Interestingly, no ground separation bubble was observed in the wind tunnel experiment of Traub [33]. However, as a relatively short splitter plate was used to act as a ground surface, the developed boundary layer was likely thin enough to overcome the local adverse pressure gradient.

However, what most of the studies above have in common is the use of a fixed ground surface [31] [28] [29] [33] [24]. In a series of wind tunnel experiments on idealized ground-vehicle buff bodies, George [34] found a significant difference in lift and drag between a stationary and a moving ground surface. Also, for the same experiment, Yang [5] examined the difference between the use of three different ground boundary conditions, namely: a stationary ground, a moving wall and a mirror image. As can be clearly seen from Figure 2.3, the streamlines around the airfoil differ significantly. As the ground boundary layer does only develop on the stationary ground, no separation bubble is present on either the mirror image or moving wall bound-



Figure 2.2: Streamlines of the flow for a stationary ground at h/c = 0.05 [5]



Figure 2.3: Streamlines of the flow for various ground boundary conditions h/c = 0.05,  $\alpha = 6^{\circ}[5]$ 

ary condition. Yang concludes that the aerodynamic results from wind tunnel experiment using a fixed floor are either too optimistic due to a decreased effective height or to pessimistic in the case of the formation a ground separation bubble. In a separate study, Barber [35] investigates the effect of four different boundary conditions as applied in the research on ground effects, with the additional boundary condition being the 'slip' wall boundary condition. In the case of the 'slip' condition (u = 1, v = 0), the shear stress at the wall is zero. The difference between the 'slip' wall condition and the 'symmetry wall' condition being that for the image model all normal gradients are zero, where as for the slip wall only the vertical velocity component is equal to zero.

In the numerical computation, the flow around a NACA 4412 in extreme ground effect (h/c = 0.025) was tested for all four boundary conditions [35]. Again, it was found that for the ground stationary case a ground separation bubble exists beneath the leading edge. The 'moving wall' and the 'symmetry wall' showed no sign of recirculation. A noticeable difference between the latter two is that the velocity vectors near the wall for the 'slip' wall are slowing down, where as for the 'moving wall' boundary condition the flow is accelerated near the wall. Interestingly, the 'symmetry wall' condition also shows a region of recirculation below the leading edge, much similar to the 'ground stationary' condition. Barbers suggest that, due to the fact that the that the airfoils are stacked very close together, a stagnation point in the flow if formed, much like a vortex pair in potential flow. When comparing the aerodynamic efficiency of the NACA 4412 in ground effect the 'moving wall' boundary conditions. The only exception being for the airfoil in extreme ground effect (h/c < 0.05). The latter is explained by the fact that at these elevation heights, the moving ground acts as a 'pump', pushing flow under the airfoil. In conjunction with the CFD analysis, Barber performed a series of wind tunnel experiments to support the numerical findings.

In a low-speed wind tunnel equipped with a conveyor belt system, the effect of a stationary or moving ground on the free stream flow was tested [35]. A Particle Image Velocimetry (PIV) system was used to visualize the flow. Clearly visible from the PIV velocity measurements is the presence of the boundary at the wind tunnel walls. In the case that the conveyor belt is switched on, the flow remains almost uniform with a zero velocity gradient near the belt. In the case of the stationary ground, a region of higher turbulence kinetic energy near the simulated ground plane was observed. In the case of the moving wall case however, there was little variation in the kinetic energy compared to the central part of the test section. Since turbulence levels of the flow strongly influence the behaviour of the boundary layer, experimental results obtained in a fixed ground wind tunnel in close ground proximity, are in all likelihood not representative for the actual flow case. Therefore, Barber concludes that the moving wall boundary condition, when performing experiments, either CFD or wind tunnel, into the ground effect is the only justifiable choice.

Since the driving factor behind the WIG effect appears to be predominantly the increased pressure under-



Figure 2.4: Relative change in lift-coefficient of airfoils in ground effect compared to out-of-ground effect (h/c = 1.5) lift coefficient [6]

neath the vehicle, research has been conducted to further enhance the RAM effect. Ockfen [36] investigated the use of a flap for lift enhancement. In a numerical study using CFD, the effect of a split flap and a plain flap on the aerodynamic performance of a NACA 4412 section was tested for various flap deflections. Ockfen defined the flap deflection as the vertical distance  $\gamma_f$  between the trailing edge of the airfoil and the flap. Furthermore, the distance between the trailing edge of the flap and the ground is denoted as  $h_f$ . Modelling of the ground was done by applying a moving wall boundary condition. The configurations were tested at  $Re = 1.0 \cdot 10^6$  for a range of  $\alpha = 2 - 6^\circ$  and h/c = 0.05 - 0.15. It was found that the flap deflection increases lift for all cases tested. The deployment of the flap reduces the effective trailing edge gap and therefore increases the local pressure. In the limiting case, when the flap almost touched the ground ( $h_f = 0$ ), the flow underneath the airfoil is almost completely stagnated. However, a region of recirculation exists on the upper side of the flap as the flow separated from the upper airfoil surface. Furthermore, it was noticed that, when keeping  $h_f$  constant while increasing the ground height h, the lift increased. This is explained by Ockfen due to the fact that a larger amount of air is trapped by the flap, in comparison with lower ground heights. Alternatively, reducing h while keeping the flap deflection  $y_f$  constant, the lift also increased. In all cases the drag was significantly increased. This is mostly due to the fact that pressure drag increases, as the flap creates an additional area perpendicular to the flow. Also, the region of flow separation on the upper surface increases pressure drag. No significant differences were found between the plain flap and the split flap. Furthermore, changes in  $\alpha$  or Re had a minor influence on the aerodynamic performance, as the flap effect was dominant. Ockfen found that a small flap deflection  $h_f/c = 0.025$  yielded to significant improvement of L/D values. However, for flap deflection larger than  $h_f/c = 0.05$ , the aerodynamic efficiency was actually lessened compared to a configuration without a flaps due to the increased pressure drag. Therefore, Ockfen concludes that a small flap deflection using a simple split or plain flap could be used to significantly enhance the lift-to-drag ratio for wings in extreme ground effect.

Apart from observation of the aerodynamics of airfoils in ground effect, one could also attempt to predict the behaviour based on certain airfoil characteristics. This approach was followed by Hase *et al* [6]. In a combined analytical and experimental study, the effect of chamber on the aerodynamic behaviour of an airfoil in ground effect was studied. Using thin airfoil theory, the limiting lift for either a front or rear loaded airfoil can be derived to be:

$$L_{\text{front}} = \rho \, u \Gamma_{\infty} \left( 1 + \frac{1}{1 + 2(h/c)^2} \right) \quad , \qquad L_{\text{rear}} = \rho \, u \Gamma_{\infty} \left( 1 - \frac{c - x}{2h} \sin \beta \right) \tag{2.1}$$

,where  $\beta$  is the trailing edge angle. For the above derivation, a lumped vortex was placed at either the leadingor in close proximity to the trailing edge, while maintaining the Kutta condition at the trailing edge. As one can compute, the limiting case for a front loaded airfoil is that the lift is doubled when  $h \rightarrow 0$ . In contrast, the



Figure 2.5: Overlay of the PIV measurements of the horizontal velocity component at h/c = 0.3 compared with the results of h/c = 1.5, for low-lift conditions (top) and high-lift conditions (bottom). [6]

lift can be shown to decrease for a rear loaded airfoil when the ground is approached. In order to test if this simplified theory holds true, a wind-tunnel experiment was set up. For this purpose, a symmetric NACA 0012, a NACA 6212 and a NACA 6712 were tested. Note that both asymmetric airfoils have equal amount of camber (6%), with the only difference being the location of maximum camber. The experiment was conducted in the M-tunnel of the TU Delft. The chord-based Reynolds number was set to approximately Re = 180,000, with the model having a span of b = 400mm and  $AR \approx 4.4$ . In order to prevent any interference, the ceiling of the test-section was removed. Note that a fixed ground plate was used throughout the experiment. The relative elevation height was varied between h/c = 0.3 and h/c = 1.5.

The experimental data showed that in the case of that the aft-loaded NACA 6712 experiences a negative normalized change in lift coefficient, due to the ground effect. This was illustrated by computing the change in lift at the lowest elevation height, compared with the essentially out-of-ground effect lift coefficient at h/c = 1.5. The results are shown in figure 2.4. As can be clearly seen, the ground-effect only has a positive effect in the case of higher lift conditions. In the case of the aft-loaded NACA 6712, the net-effect of the ground presence is actually negative, while front-loaded NACA 6212 only experiences a modest improvement in lift. Comparison the experimental results with the thin airfoil theory predictions (eq. (2.1)), showed that theory does not agree with the experimental data, due to the fact that the thickness effect should not be neglected. [6] In order to better understand the underlying aerodynamic principles, a series of PIV measurements were taken in both the low-lift and high-lift conditions. The results, again compared with OGE conditions, are shown in figure 2.5. Note that only the change in the horizontal velocity component is presented.

As can be observed, in the case of low-lift conditions, the flow between the lower surface and the ground plane was accelerated due to the channel (Venturi) effect. The net result is a decrease in lift, especially for the rear-loaded NACA 6712. For the case of high-lift conditions, or higher angle-of-attack, the flow is noticed to decelerate below the airfoils. In the case of the NACA 0012 and 6212 airfoils, this leads to a net increase of the lift. For the NACA 6712 airfoil the net effect is approximately zero, as the channelling effect between the lower surface and the ground plane is still partially present. What this study perfectly shows, is that in addition to Ahmed [29] who noted that the position of maximum thickness should be positioned relatively far aft, is that location of maximum camber is favoured to be located relatively towards the front.

The effect of camber is also acknowledged by Gross and Traub [37]. Using the both a lumped vortex approach and thin airfoil theory, it was shown that for potential flow analysis, two components influence the lift in ground effect. First, a cambering effect due to a change in up-wash of the vortex is identified. Secondly, a reduction of the free stream velocity is found due to an upstream velocity component. The expression, based

on the lumped vortex located 1/4 chord, is summarized as:

$$\frac{C_{l,ge}U_{eff}}{C_{l,\alpha}\alpha} = \left(1 + \frac{C_{l,\alpha}}{8\pi} \left(\frac{c}{h}\right) \cos\theta \sin\theta\right) \left(1 - \left(\frac{c}{h}\right) \frac{c_{l,\alpha}KK}{8\pi}\right)^2$$
(2.2)

,where  $C_{l,\alpha}\alpha$  is the lift coefficient measured out of ground effect,  $\theta$  is a angle given by  $\tan^{-1}(c/4h)$  and *KK* is an empirical constant. The latter could be viewed as an incidence angle of an image wing. Note that the first term of eq. (2.2) describes the camber effect and the second term the correction for a reduced free stream velocity. Testing was conducted in a open-return wind-tunnel at Re = 180,000 using a splitter plate to replicate the ground. The airfoil under investigation was a S8036 section. When comparing the lift coefficient normalized by the OGE lift coefficient with the simplified expression of eq. (2.2), the results matched well. Note that the lowest elevation height measured was h/c = 0.1. Based on these findings, it can be concluded that theoretical modelling of the ground effect is possible and a good tool for initial prediction of airfoil behaviour.

#### **2.1.2.** Span dominated effects

Already in the 1920's, Wieselberger [22] started to investigate the WIG effect. Using the wing theory by Prandtl [38] and the multi plane theory developed by Betz [39], Wieselberger showed that the lift-drag polar of an airfoil in proximity of a ground surface could analytically be determined once the polar out of ground effect is known. However, since Wieselberger solution is only valid for wings with a elliptical lift distribution, it is not very useful for the analysis of low-aspect ratio wings, as typically found on WIG craft. In a study performed by Philips and Hunsaker [40] a series of improved formulations for various wing planforms in ground effect were developed. Their study showed that the so-called influence factor is not only a function of the ratio between the distance to the ground and the wing-span (h/b), but is also influenced by the lift coefficient, wing planform and aspect ratio of the wing. Traub [33] used the expression for untapered wings by Philips and Hunsaker to develop an analytical approach for the prediction of lift and drag characteristics for WIG craft. In conjunction with the theoretical approach, Traub performed a series of sectional and finite wing wind tunnel experiments. Using the 2D experimental results as an input for the analytical expressions. Traub shows that predictions for the finite wing are in good agreement with the 3D experiment data, especially for the lower aspect ratio wing (AR = 3.46). For the larger aspect ratio wing (AR = 5.18), the predictions start to deviate below h/c = 0.4. In closer ground proximity, the gap between the experimental results and the predictions is increased for both wings. This is explained by Traub, by the fact that the method by Phillips and Hunsaker, translates a loss in leading edge suction into additional pressure drag. When approaching the ground, an increase in leading edge suction is observed for the S8032 profile.

No different than for wing operation out-of-ground effect, the wing's aspect ratio is of significant influence for the aerodynamic performance. Due to practical limitations, the aspect ratio for WIG craft is generally limited to very low aspect ratio wings. How such wings behave in close ground proximity was examined by Fink and Lastinger [23]. In a series of wind tunnel experiments a modified Glenn Martin 21 profile wing of various aspect ratios was tested. To prevent any viscous interaction of the ground boundary layer, the image model technique by was used. Fink and Lastinger reported an increase in the lift-curve slope and a reduction in induced drag for all aspect ratio wings. However, the effect of a decreased aspect ratio resulted in a reduction of the lift-curve slope, accompanied by an in increase of the induced drag coefficient. As expected, the largest aerodynamic efficiency is obtained for the higher aspect ratio wings at low values for h/c. In a comparison of the experimental results with the theoretical predictions by Wieselberger [22], Fink and Lastinger obtained very similar results for values h/b = 0.3 to 1.0. When reducing the ground distance further, theory and experiment started to deviate considerably. This is in accordance with the observations made by Traub [33]. In order to enhance the performance of the lowest aspect ratio wing (AR = 1), the authors experimented with the use of end-plates extending only below the wing. By preventing the high air pressure air, due to the ramming effect, from escaping near the tips, the aerodynamic performance is enhanced. Indeed, it was shown that the installation of end-plates increased both the lift and aerodynamic efficiency considerably. However, as Fink and Lastinger note, increasing the aspect ratio from AR = 1 to AR = 2 would yield almost as much lift increase and a significantly improved L/D value.

In a more condensed study, Chawla [41] confirms the finding of increased aerodynamic performance for a low aspect ratio (AR = 2.33) wing equipped with end-plates. In his research, Chawla also experimented with the use of a center plate in conjunction with the end-plates. However, adding a center plate did not results into any significant changes in the aerodynamic performance. In a comprehensive numerical study Park [27]



Figure 2.6: Streamlines in the y-z plane (x/c = 0.75) at h/c = 0.05 for a finite wing (AR = 2) without (left) and with (right) end-plates [7]

examined the effect on a square (AR = 1) wing with the modified Glenn Martin 21 airfoil. Using the RANS for a computational fluid dynamics analysis, Park validated the findings of Fink and Lastinger [23]. In a follow on computation, including a moving ground boundary condition, it is found that at each side two wing-tip vortices are generated. In the case of the wing with end-plates, the strength of the vorticity is found to be much higher compared to the plain wing. However, the center of the vortices are pushed more outward laterally and, as a consequence, the two vortices do not merge and diminish relatively quickly after leaving the trailing edge. In the case of the plain wing, the vortices are weaker, but merge near the trailing edge. Hence, Park concludes that in the case of a wing equipped with end-plates 'a jet-like flow tends to push the wing-tip vortex at the lower surface. The distance between the two separated wing-tip vortices is too great to merge at the trailing edge'. Although it is suggested that this will influence the drag favourably, no further analysis of the phenomena is performed.

In a similar numerical experiment, Jung [7] shows that the presence of the end-plates significantly reduces the velocities at the underside of the wing, thereby increasing the ram-effect. Referring to figure 2.6, one can clearly identify the stronger vortex development and lateral shift of the vortex for the wing equipped with end-plates, compared to the wing without end-plates installed. To better understand the behaviour of the wing-tip vortex for a wing in ground effect, Han [42] examined the unsteady behaviour of a trailing vortex sheet in ground effect. With the use of a discrete vortex method the evolution of the wing-tip vortex, for an elliptically loaded wing (without end-plates), is modelled. The results are validated with the method developed by Krasny [43]. Compared to a wing out of ground effect, Han observes that the size of the core of the vortex is reduced when approaching the ground. Furthermore, the position of the vortex is shifted laterally outboard. Han further states that the viscous interaction between the vortex and the ground could result in the detachment of the boundary layer due to enhanced cross flow. This results in the formation of a secondary vortex, as can also be observed in the result of Jung (Figure 2.6). When modelling also the extension of a flap, Han observes that the presence of the ground hinders the flap and tip vortices from rolling up or rotating around each other, despite having the same sense of rotation. This results into two weaker separate vortices compared to a stronger single vortex for a wing/flap combination in free flight.

Although the previous work shows that the installation of end-plates have a beneficial influence on the aerodynamic performance of WIG craft, Park [27] shows that the end-plates alter the longitudinal stability characteristics in ground effect negatively, in the case of a square NACA 4412 wing. In order to express the stability of a WIG vehicle, the so-called Height Stability (*HS*) [44] is used, which can be expressed as:

$$HS = \frac{C_{m,\alpha}}{C_{L,\alpha}} - \frac{C_{m,h}}{C_{L,h}} = X_{\alpha} - X_{h} \le 0$$

$$(2.3)$$

,where  $\alpha$  and h represent the derivative with respect to angle of attack and height. Note that Irodov used a reference frame with the origin on the trailing edge. Therefore, all moment coefficient are expressed around the trailing edge. Expression 2.3 states that the there is height stability when the distance between the neutral point of heights  $X_h$  and the neutral point of angle of attack  $X_\alpha$  is negative or the locations of the neutral points coincide. As the positive distance is measured from the trailing edge upstream, this implies that neutral point  $X_h$  should be ahead of the neutral point  $X_\alpha$ . In terms of airfoil coefficients, height stability is met also when:

$$C_{L,h} \le 0 \tag{2.4}$$

The condition of 2.4 is valid in general for any WIG craft, as lift is enhanced when approaching the ground. However, the condition of 2.4 also implies that  $C_{m,h}$  remains constant, which for an untrimmed wing is not the case. Therefore, the neutral point formulation is more convenient. In the case of Park, a range of angles of attack was found for which the height stability criterion was not met. In the case of the NACA 4412 wing (AR = 1) as tested by Park, the region of height instability did not coincide with angle of attack for maximum L/D. Depending on the airfoil geometry and the required lift coefficient, this could limit the design options. As an alternative for the wing end-plate, Lee [8] investigated the use of wing anhedral for enhancing the lift of WIG craft. In a CFD analysis, a wing of aspect ratio AR = 1, using the modified Glenn Martin 21 airfoil, was tested in 3 configurations: as a plain wing, equipped with end-plates or as wing with anhedral. In the case of end-plates, a constant gap between the ground end the side-plates was maintained, as shown in Figure 2.7. Lee found that the plain wing and the wing equipped with end-plates was not complying with the height stability criterion for a range of  $\alpha = 6 - 10^{\circ}$  and  $\alpha = 4 - 10^{\circ}$  respectively. However, the wing with an anhedral angle did comply with the height stability criterion for all tested angles of attack. According to Lee, this is mainly due to shift in  $X_h$ . In general,  $X_{\alpha}$  moves upstream, whereas  $X_h$  moves downstream for low angles of attack and upstream for higher angles of attack. Since the changes of  $X_{\alpha}$  against angle of attack or height variation are only moderate, the shifting of  $X_h$  is dominant. In terms of aerodynamic performance, the wing with anhedral performs only marginally better than the plain wing, as can be seen in Figure 2.7. Therefore, Lee recommends the combined use of an anhedral angle and end plates to enhance both the aerodynamic performance and stability for future WIG craft.

In a similar research, Jamei et al [45] compared the performance of a compound wing with a square main wing. The compound wing consists of square inner wing section and two reverse taper outer wings with an anhedral angle. The airfoil used for this study is the NACA 6409. It was found that the compound wing is an improvement over the plain wing in terms of L/D, primarily at low elevations. Although Jamei et al do not determine the height stability of the compound wing, it is noted that the centre of pressure has moves slightly (in the order of 10 - 15%) in the direction of the leading edge. As this has a destabilizing effect, the use of an out-of-ground effect tail is suggested. However, as Park [27] discovered, the end-plates alter the longitudinal stability of a the wing.



Figure 2.7: Geometry layout and lift and drag polar of the three configurations for different Reynolds numbers [8]

### **2.2.** AERODYNAMIC ANALYSIS OF AEROCITY

Following the above review of studies into the WIG effect, in this part the previous research into the aerodynamic characteristics of AeroCity will be discussed. To gain better insight in the results of these studies, each work is discussed separately.

#### **2.2.1.** NUMERICAL ANALYSIS

A numerical sensitivity analysis into the effect of the main design parameters on the aerodynamic performance of the AeroCity was conducted by Nouwens [1]. Using Computational Fluid Dynamics (CFD) simulations of the three-dimensional Reynolds Averaged Navier-Stokes (RANS) equations Nouwens investigated the effect of varying aspect ratio, body angle of attack and end-plate elevation. The analysis was performed on a preliminary aerodynamic design (Figure 2.8), which utilizes a NACA 68015 airfoil with a reflexed (S-shaped) camber line. Although NACA 5-series airfoils are typically high speed airfoils, with poor stalling characteristics and a low pitching moment, this is not a drawback for a ground effect vehicle. In fact, the low pitching moment is expected to be beneficial for the height stability criterion. The wing is given an incidence angle of  $\alpha_{body} = 3^{\circ}$  to obtain a more or less level floor. All testing has been performed at Reynold number  $Re = 1.38 \cdot 10^8$ , based on a chord length of 20m and a cruise speed of 100m/s. Before a start was made with the sensitivity analysis, considerable effort was taken to refine the aerodynamic and numerical model. For instance, the shape of the end-plates was modified, as a large portion of total pressure loss was observed at the outside of the end-plate. Initially, the end-plates were given a symmetrical profile. However, it was observed that flow trying to flow around the AeroCity, over the end-plate leading edge, separates due to a high local angle of attack. Nouwens modified the side-plate shape into an a-symmetric profile which eliminated this local flow defect.



Figure 2.8: Total pressure loss of the AeroCity baseline configuration [1]

Having fine-tuned the model, a baseline configuration was tested as a reference model to be used in the sensitivity study. For the numerical computations, the ENFLOW software package by NLR was used and a nonuniform grid of 6 million cells created to capture small perturbations in the flow. With the baseline model (AR = 0.4) an aerodynamic efficiency of L/D = 14.22 was obtained. The height of trailing edge was kept at h = 0.48m (h/c = 0.024) and the end-plate elevation was set equal to h = 0.25m. It was found that the main contributor of the drag is the pressure drag, amounting to almost 65% of the total vehicle drag. As can be seen in Figure 2.8, vortices created by higher pressure air, escaping from underneath the vehicle, contribute significantly to the overall loss of total pressure. The first parameter to be investigated in the sensitivity study, is the angle of attack. Nouwens suggests that a higher aerodynamic efficiency could be achieved at values  $\alpha \ge 3^\circ$  due to an increased ramming-effect, therefore the AeroCity is tested for the range  $\alpha = 3-5^\circ$  for various end-plate elevations. In the case of h = 0m, an Euler wall boundary condition has been used instead of the moving wall boundary condition.

The results for  $C_L$  and  $C_D$  are given in Figure 2.9a and 2.9b respectively. Note that Nouwens did not nondimensionalize the end-plate gap height. The height of the body, with respect to ground, remained unaltered compared to the reference model. As expected, the higher angle attack resulted in the highest lift. The effect of end-plate elevation had a strong effect on the aerodynamic properties, as a significant increase in lift can be observed, at reduced values of end-plate gap height. When examining the pressure distributions, it was found that increasing the angle of attack , besides enhancing the suction peak at the leading edge, strongly influences the pressure side. As the ramming effect is increased with angle of attack, a stronger air cushioning effect is created below the vehicle. The suction side however remains mostly unaffected. By reducing the end-plate gap with the ground, less high pressure air is able to leak underneath away the vehicle, preserving lift. In the case of the drag coefficient, a sharp reduction in total vehicle drag is observed, with the end-plates approaching the ground. As observed in the analysis of the reference model, the leakage flow, due to high pressure air escaping underneath the vehicle, causes the formation of wing-tip vortices. In the case of strong vorticity, a significant portion of the drag is made by the total pressure loss caused by the vortices. In the



Figure 2.9: Effect of end-plate elevation h and body angle of attack  $\alpha$  on the aerodynamic performance of AeroCity [1]

case of  $\alpha = 5^{\circ}$ , a relative larger increase in drag is observed compared to the computations of a lower angle of attack. Nouwens suspects that numerical errors in the computation play a role here. At an increased angle of attack, the trailing edge approaches the ground very closely. In this case it caused the grid cells to deform.

Interestingly, a reduction in drag is also observed when increasing the end-plate elevation height from h = 0.25m onwards. This is explained by Nouwens by the fact that for a larger gap, the higher pressure air is able to escape from underneath the vehicle more smoothly. The lower flow speeds of the leakage flow causes the strength of the vortices to be reduced, which is in accordance with the reduction of the lift. When comparing the total aerodynamic efficiency (Figure 2.9c), one can see that the L/D value is higher for a body angle of attack  $\alpha = 4^{\circ}$ . Due to the higher drag for  $\alpha = 5^{\circ}$ , the L/D is approximately equal, although slightly lower, than the case of  $\alpha = 4^{\circ}$ . However, as Nouwens states, this effect could be exploited for cases of lift regulation, for example at speeds below the design cruise speed. When examining the pitching moments (Figure 2.9d) of the vehicle, it was found that the pitching moment decreases with end-plate elevation and angle of attack. The reference point has been positioned at the quarter chord of the body, at the bottom of the end-plate. Since the high pressure air cushion underneath the vehicle is increased, both in strength and size, with decreasing end-plate elevation, the pitching moment is reduced as the center of pressure moves rearwards. Simultaneously, the suction peak near the leading edge is enlarged and transitions forward with increasing angle of attack. However, this effect is unable to offset the effect of increased ram pressure at the lower side.

Next, Nouwens examined the effect of aspect ratio on the aerodynamic performance of the AeroCity. Since the AeroCity concept is by the design inherently limited to small aspects ratios, the sensitivity to changes in

aspect ratio need to be known in order to perform a trade-off between aerodynamic performance and operational aspects, such as internal capacity or infrastructure design. Due to these limitations, the width of the AeroCity was varied between 4 - 8m while keeping a constant chord length. This amounts to a range of aspect ratios between AR = 0.2 - 0.4. As expected, the higher aspect ratio wings has a larger efficiency for all variations in height and angle of attack compared to the lower aspect ratio models. Both an increase in lift and a decrease in drag were observed for the case of the higher aspect ratio wing. Since the area of flow leakage along the end-plates remains constant, irrespective of aspect ratio, the relative amount a high speed leakage flow under the airfoil is larger for the lower aspect ratio wing. Furthermore, the pitching moment is seen to be increasing with decreasing aspect ratio, implying that the center of pressure moves forward.

Finally, Nouwens investigated the use of a trailing edge flap. By extension of the flap at low ground clearances, the flap partially or fully blocks the flow underneath the vehicle. The key factor in enhancing the lift has shown to be reducing the flow passage underneath the vehicle and therefore increasing the ramming effect, the use of a flap could provide. Therefore it is not surprising that is was found that the full extension of the flap significantly enhances the lift for AeroCity. With the trailing edge gap fully closed, it was found that the body became almost completely insensitive to changes in angle of attack. The latter is partially explained by the fact that Nouwens rotated the body around the quarter chord point when varying the angle of attack. As such, the distance between the trailing edge and the ground changed accordingly. Hence, a change of angle of attack resulted in both a change in angle of attack and trailing edge elevation height. This is contrast to literature [27][8][15][24], where the airfoil is rotated around the trailing edge, keeping the trailing edge gap, with respect to the ground, constant. The effect of an increased suction peak, with an increasing angle of attack, was of minor influence on the total lift produced. In terms of drag, the use of a trailing edge is less beneficial, since the aft body of the vehicle now consists of relatively large blunt body. As a result, the vehicle experiences a larger pressure drag, compared to the case of no flap deployed. Next to the increased wake, also a region of flow separation was observed at the front of the side-plates, like experienced earlier. Due to the complete blocking of the flow underneath the vehicle, more air will be forced to flow around the vehicle. A further optimization of the side-plate shape could reduce this contribution to the drag. Unlike the lift, the drag does increase incrementally with variations in angle of attack. Nouwens concludes that the use of a trailing edge flap could be a useful measure to trim or increment the lift.

In the second part of his work, Nouwens investigates the longitudinal stability of AeroCity concept. In an assessment into the position of the center of pressures, it was found that there is a relatively large variation in location with respect to the main parameters. For the body angles of attack and side-plate elevations under consideration, the center of pressure is located between x/c = 0.32 and x/c = 0.40. Nouwens remarks that this allows for a feasible position of the center of gravity, to comply with the criterion for longitudinal stability  $(C_{M,\alpha} < 0)$ . When varying the angle of attack of the incoming flow, it was found that the AeroCity is stable for variations in flow incidence as  $C_{M,\alpha}$  is negative for all configurations tested. Stability is increased, when reducing the end-plate elevation height. Computation of the Stability Margin (S.M.) concluded that the AeroCity has at least a S.M. = 0.1 depending on the configuration. Although a large stability margin for aircraft usually implies that the airframe posses non-favourable handling characteristics, in the case of AeroCity this is not considered to be a drawback. Nouwens does state that a large value of  $C_{M,\alpha}$  could be problematic for the dynamic stability. When assessing the height stability of the AeroCity, it is found that the vehicle is stable for height variations for all configurations. In fact, relatively large negative values for H.S. are reported, implying again a very firm margin of stability. How this will affect the dynamic handling of AeroCity remains to be seen. A short study by Nouwens into the dynamic stability, with very preliminary data, does not show any reasons for concerns about instability. To conclude his work, Nouwens states that the L/D values obtained with low end-plate elevations is promising for the total efficiency of the AeroCity. The end-plate elevation should be minimized as much as possible, to minimize the strength of the tip vortices and enhance lift. The body angle of attack, is best to be increased to  $\alpha = 4^{\circ}$  to enhance the air cushioning effect. The use of the trailing edge flap is recommended for use below optimal cruise speed. In terms of stability, the dynamic stability needs further investigation once more vehicle data is available. Finally, Nouwens suggest several modifications to the vehicle to enhance the efficiency. As most of the lift is generated by the RAM WIG effect (> 70%) and more lift is generated than required, less attention is required for the upper surface. By removing the end-plates from the upper surface, one allows for a smoother development of lift and prevent the onset of secondary vortices. For lateral control, to fins are located near the aft body. A solution that allows for very small end-plate elevation heights with respect to the ground is a topic that would require additional effort.



### **2.2.2.** WIND-TUNNEL TESTING

In an effort to validate the numerical results of Nouwens [1], Nasrollahi performed a series of windtunnel experiments on the basic AeroCity model. Like the previous CFD study, the goal of the study was to identify the AeroCity performance when varying several main parameters. In this case body elevation, body angle of attack and end-plate ground elevation were selected as variables. For each combination, testing was conducted with either natural or fixed transition of the boundary layer. In the case of fixed transition, the flow was disturbed at x/c = 0.05 by the use of two adjacent layers of double-sided tape. A scale model (1/20) was constructed out of foam blocks using a CNC milling machine. Testing was conducted in the Delft University of Technology Low Turbulence Tunnel (LTT). The model was mounted upside down on roof of the octagonal test section (2.60m x 1.80m x 1.25m), directly connected to the balance through the connection of 4 bolts. A schematic drawing of the model is shown in Figure 2.11. In order to reduce the number of measurements, the variations per parameter were limited to two values. Unlike Nouwens, in this study the body angle of attack is defined by a rotation around the trailing edge point rather than the quarter chord point. Hence, the trailing edge elevation remained independent of body angle of attack. Flow speeds varied between 10m/s to 100m/s.

Shown in Figures 2.12 to 2.15 are the force and moment measurements for the configuration with h/c = 0.05 or h/c = 0.10 and  $\alpha_{body} = 5^\circ$ . The gap height was varied between 7mm and 3mm by adjusting the bolts on which the model was suspended. Measurements were taken with and without boundary layer trip. The following observations can be made:

- The effect of the gap height can clearly be observed in Figure 2.12 and 2.13. Similar to the observations made by Nouwens [1], the lift is increased significantly if the gap height is reduced. In terms of drag coefficient, shown for the h/c = 0.10 configuration in Figure 2.14, the positive effect of a reduced gap height is also visible. The gap between the ground and the end-plate causes the vortices to move further outboard, which should reduce the drag.[7]
- At the lowest flow speed, a sharp drop in lift coefficient can be observed, regardless of the configuration. No explanation is provided by Nasrollahi [2] about the possible cause. However, as the model is mounted directly onto the wind-tunnel roof, it will be ,at least partially, subjected to the tunnel boundary layer. At the lowest flow speeds, the wall boundary layer may be sufficiently thick to directly interact with the flow over the model. With a stationary ground boundary condition, it is known that a laminar separation bubble can exist in front of the vehicle, reducing lift and increasing drag. However, examining the drag coefficient (Figure 2.14) the reduction at the lowest flow speed is less significant.
- Comparing the data shown in Figure 2.12 and 2.13, one can observe that a change in ground elevation height from h/c = 0.10 to h/c = 0.05 has a significant positive effect on the lift coefficient, with the improvement in  $C_L$  being in the order of 10%, depending on the Reynolds number. However in terms


gap height at h/c = 0.05

Figure 2.12: Variation in lift coefficient with Re for two settings of Figure 2.13: Variation in lift coefficient with Re for two settings of gap height at h/c = 0.10



Figure 2.15: Variation in drag coefficient with Re for two settings Figure 2.14: Variation in drag coefficient with Re for two settings of gap height at h/c = 0.10of elevation height at  $\alpha = 5^{\circ}$ 

of drag coefficient, it can be observed form Figure 2.15 that the increase is only very modest, in the order of 2-3%. Hence, the lift-to-drag ratio is found to increase due to a reduction in elevation height.

- The difference between a tripped boundary layer (at x/c = 0.05) or natural transition is very small, for the case of the lift coefficient. Also the drag levels appears to more or less indifferent. The exception however appears to be the configuration with  $\alpha = 5^{\circ}$  and a 3mm gap at h/c = 0.10. For this particular case, a clear distinction between the tripped and non-tripped experimental data can be observed. Or at least at the lower Reynolds numbers, as the difference is diminishing towards the higher flow velocities.
- Due to an unknown cause, the lift coefficients for almost all configurations decrease with increasing Reynolds number. This appears to be strange, as one would expect that the lift would increase with Re (or at least remain constant). The exception appears to be the h/c = 0.10 at  $\alpha = 5^{\circ}$  and a 7mm gap, as the measured lift coefficient remains approximately constant. Apparently, the increase in gap height causes the effect, responsible for the diminishing coefficients with Re, not to occur for this particular configuration.
- In the case of the configuration with h/c = 0.10,  $\alpha = 5^{\circ}$  and a gap height of 7mm, a big drop in drag coefficient is observed at the highest flow velocity. As the difference amounts to over 20%, the effect is very significant and cannot be ignored. Especially considering the fact that the AeroCity is foreseen to cruise at high velocities. As both the natural and forced transition measurements are affected, it is not





Figure 2.16: Comparison of lift coefficient for two settings of body angle of attack at similar height parameters





Figure 2.18: Comparison of lift-over-drag ratios for two settings of Figure 2.19: Comparison of moment coefficients for two settings of body angle of attack at similar height parameters of body angle of attack at similar height parameters

clear what is the causing this phenomenon. Possibly, the anomaly is caused by a measurement error. Since the resultant forces at the lowest velocity are small, it may have operated outside the calibrated range for the balance scale.

Hence, from the above observations it becomes clear that the measurements show some unexpected trends that are not well understood. The most noticeable deviations are the reversed trend of lift coefficient with Reynolds number and the drop in drag coefficient at the highest flow speed. To complicate matters more, the unknown phenomenon responsible for these effects does not appear to be present for all configurations. Especially the configuration at the higher elevation height h/c = 0.10 and gap height of 7mm seem to be only partially effected. Nevertheless, the above results do show the beneficial contribution of a reduced gap and elevation height for the overall aerodynamic efficiency, as has been shown by Nouwens and throughout literature [1] [7].

Besides an investigation into the effect of the two height parameters, the effect of a change in body angle of attack was also investigated. A comparison of the lift coefficient for either  $\alpha_{body} = 3^\circ$  or  $\alpha_{body} = 5^\circ$  is shown in Figure 2.16. From this figure, one can observe the following:

• In accordance with expectations, the lift is seen to increase significantly when the body angle of attack is increased. Like the previous observations made for  $\alpha = 5^\circ$ , the lift coefficient also follows a downward trend with increasing Reynolds number. The difference between a tripped boundary layer or natural transition is almost negligible for lift force measurements.

- The lift-to-drag ratio, shown in Figure 2.18, is increased by a similar margin as the lift coefficient. Hence, the drag penalty is modest and related to the increase of lift coefficient, indicating that no large flow separation is occurring on the body. Note that, as the drag coefficient is more susceptible for the location of boundary layer transition, a difference in *L*/*D* between natural and forced transition can be found. The aerodynamic efficiency of the model without boundary layer trip is slightly higher in both cases.
- Shown in Figure 2.19 are the moment coefficients, taken at the quarter chord position. As can be observed, the coefficients are negative which is required for static stability. However, it can also be seen that the moment coefficients reduces in magnitude with increasing flow speeds quite rapidly. Since the drag is approximately constant, the reduction in nose-down pitching moment could be explained by either reduced lift aft of the quarter-chord position or an increase in lift near the leading edge section of the wing. The latter however, is not very likely given the trend in lift coefficient, shown in Figure 2.16

From the above observations, one can conclude that the angle of attack does not appear to be influencing the phenomenon that causes the reduction in lift with Reynolds number. Instead, the elevation height and gap height seem to be influencing the phenomenon. These two parameters influence the amount of under-side pressurization. The root of the unknown flow interaction is likely an interaction with the tunnel boundary layer. Since the boundary layer of the wind-tunnel wall is allowed to develop upstream of the test-section, as the boundary layer is not sucked or otherwise suppressed, the thickness of the boundary layer in the test-section could be considerable. The associated Reynolds number for the wall boundary layer therefore has a different length scale compared to the chord-based Reynolds number of the model. At a given flow speed, the Reynolds number of the wind-tunnel wall boundary layer will therefore be considerably higher. How thick the boundary layer actually is at the start of the test-section is unknown, as no measurements of the boundary layer are taken. Therefore, it remains an educated guess

Taking a step back and examining the lift-over-drag values, shown in Figure 2.18, it can be seen that the best L/D value achieved is approximately  $L/D \approx 11.5$ . This is considerable less than the values predicted by Nouwens (Figure 2.9c), which exceed a lift-to-drag ration of L/D > 40. On the other hand, the results is more on pair with observations made elsewhere in literature [23] [27]. Nevertheless, the difference in aerodynamic efficiency is very large and relatively low in the case of the wind-tunnel experiment. As this one of the primary performance figures for the AeroCity, it is of crucial importance that this value is established with great confidence.

Having obtained the force and moments coefficients, an investigation into the flow characteristics was performed. For this study, the configuration with  $\alpha_{body} = 3^\circ$ , h/c = 0.05 and gap height 7mm was used. Since the side-plates had to be modified in order to increase the gap height to 7mm, the 3mm configuration could not be tested at this stage. Along the chord direction, 6 pressure tabs were distributed along both the upper surface and end-plate. With Pitot tube measurements of the flow close to the surface, the boundary layer



Figure 2.20: Flow visualization of the AeroCity upper surface at U = 100 m/s [2]



Figure 2.21: Comparison of lift coefficient obtained by experiment and CFD



development was examined. Each measurement was performed twice to eliminate errors. From the observations of the boundary layer profiles, it was concluded that no separation occurs along the upper surface. Near the front of the side-plates, a region of reversed flow was observed. Using a tuft wand, the local flow direction was visualized. Nasrollahi found that the natural transition to turbulent occurred around x/c = 0.25, on both the side-plate and the upper surface. Although no separation of flow occurred near the trailing edge, a separation bubble was observed at the side-plates near the front. This is similar to what Nouwens [1] observed in the numerical computation. From the Pitot pressure measurements, it was clearly observed that a region of reverse flow exist in this location. Further downstream the separated flow reattaches again. Also, a measurement of the pressure over the upper surface was conducted. The lower surface was not mapped accordingly. Visualization of the flow using liquid paraffin wax and a UV lamp enabled to make the streamlines of the flow visible. In Figure 2.20, the flow pattern over the upper surface is shown. One can clearly identify the transition at approximately x/c = 0.24. Further downstream, the two vortices can be observed near the side. After dismounting the model from the test section, it was found that at the lower surface transition occurs slightly later at x/c = 0.28. No separation of flow on either surface was observed.

#### COMPLEMENTARY CFD ANALSYIS

In an effort to gain more insight in the flow behaviour, Nasrollahi also performed a limited CFD calculation. Using the ANSYS Fluent software, a RANS analysis was performed on the AeroCity model. Nasrollahi tried to mimic the exact conditions of the wind-tunnel for optimal agreement with the experimental results. Three different turbulence models were used to capture the behaviour of the boundary layer. An unstructured grid was used to mesh the computational domain. A velocity-based boundary condition was applied at the inlet and outlet of the domain. The Enhanced Wall Treatment method for wall boundary layer modelling was applied for the  $k - \epsilon$  models.

However, Nasrollahi did not only find a large difference in results between the various turbulence models, but also observed significant discrepancies between the numerical and experimental values. For example, shown in Figure 2.21, is the lift coefficient for various flow speeds. As can be observed, the numerical experiment over-estimates the total produced lift. Furthermore, it does not show the same trend with Reynolds number, compared to the experimental data. In terms of drag, the values for  $C_D$  (see Figure 2.22) are comparable with the wind-tunnel results for low Reynolds number. However, as the trend is opposite, the difference grows with increasing flow speed. Although the  $\kappa - \omega$  SST model showed the lowest average error in respect to the wind-tunnel data, the difference in results remains large. Hence, Nasrollahi concludes that the CFD data is not representative for the wind-tunnel experiment. In order to find an explanation for the discrepancy between the two experiments, Nasrollahi looked at the CFD pressure distributions of AeroCity. A region of positive static pressure was noticed near the rear upper surface. However, as Nasrollahi notes, the pressure gradient is not large enough to cause separation of the flow. Also a region of low dynamic pressure in front of the AeroCity was observed. Nasrollahi does not give a physical explanation for the occurrence of this



Figure 2.23: Total pressure contours for 30m/s (left) and 50m/s (right)

high pressure region in front of the vehicle. Instead, it is suggested that the pressure region may influence the inflow boundary. As a solution, a pressure inlet condition rather than a velocity boundary condition is proposed. An alternative solution given is to extend the computational domain further upstream. However, Nasrollahi notes, this will influence the development of the boundary layer.

When consulting the boundary layer thickness's, Nasrollahi observed a merging of the ground and vehicle boundary layer [2]. This is shown in Figure 2.23 by means of total pressure contours. At lower velocities, up to 30m/s, merging occurs underneath the vehicle. Hence, it is noted that this will increase lift as the flow stagnated underneath the body. At higher velocities the merging of the boundary layers occurs downstream of the vehicle. This would explain, Nasrollahi states, the reduction in lift coefficient with increasing flow speeds. Visualization of the flow shows a similar flow pattern as was observed in the wind-tunnel experiments. The existence of the laminar separation bubble near the leading edge of the end-plates is captured well at low flow speeds. However, at a flow speed of 70m/s, the separation of flow seems to be completely removed. The vortices induced at upper surface and near the end-plate gap appear to be captured reasonable well. In conclusion, Nasrollahi states that the general flow behaviour seems to be captured successfully by the CFD calculations. On the downside, the existence of a laminar separation located on the end-plates bubble at flow speeds between 70m/s to 100m/s is not detected. In order to improve the results, a higher quality mesh in which grid cells are aligned with the local flow is suggested.

# 3

# **PROJECT DEFINITION**

In this chapter the objective and framework of the project are discussed in more detail. First, the questions that the project hopes to answer are discussed in Section 3.2. The project aims (Section 3.1) will support the formulation of the main research question. Further, Section 3.3, will discuss the methodology that will be employed to achieve the project goals and answer the main research question.

## **3.1.** MISSION STATEMENT

With much of the aerodynamic performance of the AeroCity still being uncertain or unknown, the ultimate goal would be to fully understand the aerodynamic behaviour of the AeroCity and consequently adapt the configuration. However, to comprehend the full aerodynamic behaviour of AeroCity, a significant research effort is required. In order for the current project, to remain realistic and a little less ambitious, only specific aspects of the aerodynamic behaviour of AeroCity will be investigated. Shown in the breakdown diagram (Figure 3.1), to validate the AeroCity concept, one should consider many different aspects. Although this may appear obvious, it is important to keep in mind that the aerodynamics is only a small portion of the AeroCity are also divided into multiple disciplines. In this case the aerodynamics are split into steady and transient or unsteady aerodynamics. For the current project, the analysis will be limited to the steady aerodynamics. Within this branch of the aerodynamics, the focus will be put on the accurate prediction of force and moment coefficients.

This is identical to the previous efforts, conducted to investigate this aspect of the aerodynamic characteristics of AeroCity [2] [1]. However, as discussed in the literature survey (Chapter 2), neither study provided a means of validation of their respective results. The CFD by Nouwens [1] provided good insight in some of aerodynamic phenomena, but without experimental data to compare the results, the significance of the results, in terms of proceeding to validate the concept, is limited. The same more or less applies to the experimental study by Nasrollahi [2]. Being limited to a fixed ground wind-tunnel set-up, the results inherently do not represent the actual AeroCity operating case. This does not subtract anything from the actual quality of the respective studies, but it does imply that certain question marks about the aerodynamic performance of the AeroCity remain.

Given the fact that proper replication and measurement of the extreme ground effect in a wind-tunnel environment is difficult, and not say the least expensive, further experimental investigations are not deemed feasible at this stage of the design. Instead, the goal of this project will be to investigate, if the use of computation methods is suitable to investigate the Wing-in-Ground effect, and specifically the AeroCity's aerodynamic characteristics. This will allow more elaborate analysis under a variety of circumstances to be evaluated in a relatively short amount of time. However, in order for this effort not to be a replication of the work by Nouwens [1], a solid basis for validation of the model needs to be acquired. This will be the most important task for this project, namely to build sufficient confidence about the validity and accuracy of the numerical model such that it can be used for consequent modelling.



Figure 3.1: Breakdown of the various components involved in the validation process of AeroCity

In order to achieve this goal, the predictions by the numerical model have to be compared with at least a single quality experimental data set. Unlike Nouwens at the time, such a data-set is available in the form of the experiment data by Nasrollahi [2]. Although the experiment itself, due to the absence of a moving ground plane, is not representative for the AeroCity's actual performance, it is still suitable to validate the numerical model. Due to the inclusion of additional viscous effects, the aerodynamics with a stationary ground plane are in a sense more difficult to model. As such, the data can still be used for further validation of AeroCity, even though the stand-alone experimental data cannot. The benefit of utilizing the experiment results by Nasrollahi, is that the testing parameters and geometry are known in detail. Using the same CAD-model and operating conditions, a direct comparison could be made with the numerical results without the introduction of additional unknowns.

If the numerical model is able to accurately replicate the experimental results obtained by Nasrollahi [2] the capability of the model to model the close ground effect is assured. Using the same numerical models and techniques, the investigation into the aerodynamic characteristics of AeroCity can be expanded by inclusion additional effects. By changing the scale of the model or making modifications to the geometrical design, one can asses the respective influence of these changes in a relative quick manner. Something that would be difficult to achieve in a wind-tunnel environment, due to physical or cost constraints. Moreover, the numerical model allows the designer to investigate all available measurement data, without affecting the flow or the use of complicated measuring techniques. In short, a verified and validated numerical model allows the design process to gather pace and limited the number of experimental studies during the process. The above can be summarized by the following mission statement:

#### "Evaluating the steady-state aerodynamic performance characteristics of AeroCity by means of CFD analysis"

Note that the method, namely CFD, is explicitly stated in the mission statement. Up till now, the term 'numerical model' was used exclusively. However, as it was found in Section 2 that viscous effects play a mayor role in extreme ground effect aerodynamics, non-viscous methods are no viable option. Therefore, the numerical model that will used throughout this project will be based on a CFD method. A more thorough description of the methodology can be be found in Section 3.3.

## **3.2.** RESEARCH QUESTION

In order to define the exact scope of the research, a main research question is formulated. As mentioned in Section 3.1, the goal of the project is to develop a trustworthy numerical model to further investigate and develop the aerodynamic configuration of AeroCity. Using the already formulated mission statement, the main question this research project aims to answers is stated as follows:

"What are the steady-state aerodynamic characteristics and performance indicators of AeroCity during the cruise phase?"

The research question comprises of three important aspects of the research project. The first aspect is that the investigations into the aerodynamics of AeroCity will be limited to steady-state aerodynamics. Hence, possible unsteady flow phenomena will not be examined specifically. Second, the research will focus both on the analysis of aerodynamic characteristics as well as the performance indicators of AeroCity. This implies that both qualitative and quantitative aspects of the aerodynamic performance will be of interest. Examples of qualitative analysis will include the studying of flow separation and vortex generation, whereas the quantitative analysis will focus more on lift and drag characteristics of the vehicle. The last aspect of the research question discloses that the research will be limited to the AeroCity, specifically in the cruise phase condition.

#### **3.2.1.** SECONDARY RESEARCH QUESTIONS

In order to answer the main research question, secondary research-questions have been formulated to make the research topic more specific. The first sub-question that is raised is concerned with the wind-tunnel experiment by Nasrollahi [2]:

(S.1) "How can the available results from the wind-tunnel experiment, conducted in the LTT of the TU Delft, be explained? "

The wind-tunnel experiment will be an important factor in the validation of the aerodynamics of AeroCity. Being the only available reference data for the CFD model, the results of the experiments should be well understood. As was found in the literature survey (Chapter 2), the wind-tunnel results show a peculiar trend at the lowest Reynolds numbers. Although this may not be directly relevant for the cruise phase of AeroCity, which starts at higher Reynolds number, it needs to be understood nevertheless. Replication of the windtunnel experiment in CFD will hopefully clarify these unknowns and provide a better understanding of the principle aerodynamic aspects of AeroCity.

Following the above, a second question that could be raised is how the stationary ground boundary condition, applied during the wind-tunnel experiment, affects the aerodynamics of AeroCity model compared to a moving ground boundary condition:

(S.2) "What influence does the ground boundary condition have on the aerodynamic characteristics of the AeroCity model?"

Note that question is related to the wind-tunnel model specifically, as the AeroCity will be subjected to a moving ground at all times. However, by comparing the influence of the boundary condition on the wind-tunnel experiment results, one can determine whether or not testing with a stationary ground could still be useful. If the difference in flow characteristics and coefficients is only marginal or with an almost constant offset, the need for a wind-tunnel with a calibrated conveyor belt system to replicate the moving ground could be removed. Moreover, simulation using a moving ground boundary condition provides insight into the actual aerodynamic characteristics of AeroCity.

The AeroCity is foreseen to be flying inside a U-shaped track, which will be used for housing the electromagnetic propulsion system and lateral guidance. [1] Considering that the track-walls are situated in the region where the vortices originate, it is expected that their presence will have a significant affect on the flow field around AeroCity. [1] Nevertheless, this aspect has not yet been investigated, neither for the AeroCity specifically, nor in literature. Hence, a valid question that could raised be raised in order to answer the main research question is:

Model	HP Z600
Operating system	Windows 7 Professional 64-bit
Processor	Intel Xeon X5670
Number of CPU's	2 (12 cores)
Physical Memory	6× DDR3-1333 8GB ECC Reg.
Graphics card	Nvidia Quaddro FX1800
Storage	HP MK0960GECQK 960GB SSD
	Crucial M400 128GB SSD
	500GB RAID 0 volume

Table 3.1: Overview of workstation specifications

#### (S.3) "How does the inclusion of a U-shaped track wall affect the aerodynamic characteristics of AeroCity?"

Note that analysis will focus on a single U-shape track design only, even though their will be multiple parameters involved that will influence the aerodynamics of AeroCity. However, the prime interest of this analysis part will be to investigate how the track walls influence the aerodynamic characteristics of AeroCity in a qualitative way, rather than to study track design itself in more detail with the purpose of optimizing the performance of AeroCity. The latter is proposed as a separate topic for further research.

Since the research is not only concerned with qualitative analysis, but also interested in the quantitative performance indicators of AeroCity, criteria for the acceptable margin of error should be set. If the prediction by CFD cannot meet these criteria, the validity of the obtained data should be questioned. Since the windtunnel data by Nasrollahi [2] is the only available reference data, deviations of the numerical results will be taken with respect to the this data-set. Since the design of AeroCity is still be believed to be in the conceptual phase, the following criteria for the main performance indicators,  $c_L$  and  $c_D$ , are deemed sufficient:

- The predicted lift coefficient by the CFD model should be  $\Delta C_L \leq 5\%$  of the experimental [2] value
- The predicted drag coefficient by the CFD model should be  $\Delta C_D \leq 10\%$  of the experimental [2] value

Note that the margin, especially for drag prediction, is relatively high. However, the accurate prediction of the drag force, specifically by means of CFD, is difficult to achieve without resorting to very high-fidelity models. Traditional conceptual design method for aeroplanes rely on (semi)-empirical relations to estimate the drag force early on in the design stage. The order of accuracy delivered by these methods is roughly in the same order. However, these relations are not available for the WIG-effect vehicles. For later design and research stages, a margin of 10% will not be acceptable, as the error introduced to the power required scales with  $(\Delta C_D)^3$ 

# **3.3.** METHODOLOGY

In this section the methodology and project set-up is discussed in more detail. Information about the hardware and software to be used, the methods and procedures to be followed and a general overview will be discusses consequently.

The numerical analysis will be performed on the basis of the three-dimensional Reynolds Averaged Navier-Stokes (RANS) equations using the ANSYS Fluent 16.1 commercial software package. This Fluent software is widely used in both industry and academics to compute turbulent flows. Calculations using RANS, compared to the more computationally expensive Large Eddy Simulations (LES), will offer sufficient detail for this project, as unsteady effects are not expected to play a mayor role at this stage of the design of AeroCity. As turbulence, in the case of RANS, is modelled, selection of the most suitable turbulence model will be crucial for obtaining the best results. Several turbulence models are incorporated in Fluent, including the popular two-equation eddy-viscosity models (see Section 4.2). Since the aerodynamic CAD model is already constructed by Nasrollahi, this model can be adopted without further modifications. Before simulation of the wind-tunnel experiment, several test studies, involving both In Ground Effect (IGE) and Out-of Ground Effect (OGE) two-dimensional flow cases, will be performed. An additional three-dimensional study for validation with experimental data from literature<sup>[23]</sup> will be performed to select the most suitable turbulence model.

The technical stage of this project is divided into four phases.

- 1) Two-dimensional validation studies of the ground effect
- 2) Validation of a three-dimensional wing with end-plates with wind-tunnel data [23]
- 3) Replication of the wind-tunnel experiment by Nasrollahi [2]
- 4) Additional simulations to improve and validate the aerodynamic configuration of AeroCity

The first phase has a dual purpose. Primarily, the goal will be to investigate how well various turbulenceand mathematical models are able to replicate some of the wing-in-ground effect studies published in literature [15]. By narrowing down the list of candidate models, the model best suited for modelling of the WIG-effect can be elected. Prior to this phase, the flow around airfoils out-of-ground will also be simulated and compared with existing experimental data. This will ensure that the implementation of the models and construction of the mesh is performed correctly. Although the relevance of these out-of-ground effect simulations for the aim of the project is limited, it will ensure that the steepest portion of learning curve for CFD modelling has been passed once the more relevant simulation phase starts.

In the second phase, a three-dimensional simulation of a body equipped with end-plates will be performed. As discussed in the literature review section (Chapter 2), the number of experimental studies of actual WIG vehicles is very limited. However, the wind-tunnel experiment by Fink and Lastinger [23] appears to be a good reference study for this purpose. The three-dimensional body actually resembles the AeroCity shape quite well, despite the different airfoil geometry and larger aspect ratio (AR = 1). Although the ground condition has been modelled by means of an image model rather than a moving ground, it is believed that the ground effect aerodynamics are modelled sufficiently for the current purpose. Having reduced the number of mathematical models in the previous simulation phase significantly, this three-dimensional simulation will only be performed with a limited set of the most promising turbulence models. Since the experimental model resembles the AeroCity well, the turbulence model that most accurately predicts the flow in terms of lift and drag will be selected for the actual AeroCity analysis.

The third phase will be, as discussed previously, of crucial important for the process of the model validation. Using the build-up knowledge and experience, it will need to be verified whether the model predictions are able to replicate the wind tunnel measurement data. The criteria defined in Section 3.2 will help to determine whether or not the model accuracy is sufficient. In the case that the CFD model performs within the set performance goals, it can be assumed that the model is able to replicate the experiment. By carefully examining the obtained data, a conclusive answer should be formulated what flow phenomena have played a role in the wind-tunnel experiment and caused the reversed trend with Reynold's number. However, in the case the CFD model fails to predict the flow with sufficient accuracy, this might not be possible to achieve. Instead, it should be investigated which aspects of the flow are not represented by the simulations in a proper way. The outcome of this investigation will determine if the CFD model is suitable for further investigation of AeroCity. For example, if the turbulence models do not model the severity of the laminar separation bubble on the side plates properly, the CFD model could still be used to adapt the design . By altering the shape of the side-plates, one could minimize the laminar separation bubble size, despite the fact that the magnitude is not properly estimated. Hence, the CFD model could still be used as a qualitative analysis tool.

The outcome of the third simulation phase will therefore greatly influence the simulations that will be performed in the fourth phase. In a worse case scenario, no additional simulations will be conducted as the model turns out to be incapable of prediction the flow with sufficient accuracy. In the case that only qualitative analysis is possible, further investigations could include modifications to the geometry or the vortex interaction with the track wall for example. This will mainly depend on the ability of the CFD model to replicate certain flow phenomena. Ideally however, the type of investigations proposed for the qualitative analysis are also analysed in a quantitative manner. This can only be achieved however, if the third simulation phase was concluded with complete satisfaction. Nevertheless, the additional simulations are only aimed to show what the possibilities for further research using the CFD model include and what topics are to be investigated further.

# 4

# **NUMERICAL MODEL**

In order to answer the main research question, use will be made of Computational Fluid Dynamics (CFD) to compute the flow field around the AeroCity. For this purpose, the ANSYS Fluent software package has been selected, which is a finite volume code widely used in both research and industry to evaluate a wide variety of engineering flow problems. In this section the equations behind the various models available within Fluent are discussed. Note that the aim of this section is to provide some insight in the theoretical background on which the Fluent software is based. For a full understanding of the mathematical and physical models, the reader is referred to open literature.

## **4.1.** REYNOLDS AVERAGES NAVIER-STOKES EQUATIONS

The basis for the computation of fluid flow are the transport equations for velocity, namely the momentum equations. For each velocity component, a separate transport equation exists. Below, the momentum equations for a two-dimensional steady laminar flow are presented: [11]:

$$\frac{\partial}{\partial x}(\rho u u) + \frac{\partial}{\partial y}(\rho v u) = \frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu\frac{\partial u}{\partial y}\right) - \frac{\partial p}{\partial x} + S_u$$

$$\frac{\partial}{\partial x}(\rho u v) + \frac{\partial}{\partial y}(\rho v v) = \frac{\partial}{\partial x}\left(\mu\frac{\partial v}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial y}\right) - \frac{\partial p}{\partial y} + S_v$$
(4.1)

In order to ensure that conservation of mass is guaranteed, the continuity equation is introduced. It states that the net rate of change of mass in a fluid element is zero:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left( \rho \, \vec{u} \right) \tag{4.2}$$

Note that the time derivative of the fluid density is omitted, in the case of incompressible flow. For the latter case, the above equations are sufficient to describe the characteristics of the flow. However, in the case of flow compressibility (M > 0.3), or heat transfer, an additional transport equation is required to capture the change in density of fluid. A thermodynamic equilibrium should be maintained, meaning that the increase of internal energy of the fluid is equal to the product of extracted or added energy to the fluid and the work done by the fluid. The energy equation is formulated as follows:

$$\frac{\partial i}{\partial t} + \nabla \cdot \left(\rho \, i \, \vec{u}\right) = -p \nabla \cdot \, \vec{u} + \nabla \cdot \left(k \nabla T\right) + \Phi + S_i \tag{4.3}$$

Note that, in the above equation,  $S_i$  is an additional source term and  $\Phi$  is a so-called dissipation function. This function captures all the changes of the internal energy, caused by the effects of viscous stresses. [11]

To be able to solve turbulent flow problems, the instantaneous form of the Navier-Stokes equations, as presented by eq. (4.1), are in practice to computationally expensive. Instead of accounting for all the velocity perturbations induced by turbulence, it is more convenient to model the flow as a mean flow and a fluctuating velocity component. By applying a Reynolds [46] decomposition, the instantaneous motion of the fluid is split into a time-averaged and a fluctuating quantity. By doing so, one can, together with additional models that describe the turbulent behaviour of the flow, obtain approximate solutions for the Navier-Stokes equations, in the case of turbulent flows. Although the solution obtained by the RANS equations is a time-averaged solution, it is well suited for cases where a steady-state solution exists. For transient flow cases, for example involving vortex shedding or fluctuating shock-waves, the time-averaged solution of the RANS equations may not be adequate. Instead, one should opt for an unsteady model, such as the Unsteady RANS (URANS) or Large Edddy Simulation approach. As these latter models come at a far greater computational expense and no large unsteady flow phenomena are expected in the case of AeroCity, the steady RANS equations are considered to be sufficient for accurate prediction of the the aerodynamic characteristics of AeroCity.

The RANS equations for an incompressible flow without additional body forces can be written as:

$$\rho \left[ \frac{\partial \overline{u}_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} \right] = \frac{\partial}{\partial x_i} \left[ \mu \frac{\partial \overline{u}_i}{\partial x_j} - \rho \overline{u'_j u'_i} \right] - \frac{\partial \overline{p}}{\partial x_j}$$
(4.4)

Note that due to the addition of velocity fluctuations, additional apparent shear stresses are introduced to the model. These stresses are referred to as the Reynolds Stresses. For closure of the system, an additional model is required to describe the non-linear Reynolds Stress terms. Examples of such models are described in Section 4.2. For the analysis of incompressible flows it is common to decompose the velocity gradient into a symmetric and anti-symmetric part:

$$\frac{\partial u_i}{\partial x_j} = S_{ij} + \Omega_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right)$$
(4.5)

,where  $S_{ij}$  is the symmetric rate of strain deformation tensor and  $\Omega_{ij}$  the anti-symmetric rate of rotation tensor. It is convenient to write the velocity gradient in this form, as the rate of strain deformation is an indication for the amount of shear stress and the rate of rotation for the amount of vorticity in the flow:

$$\omega = \nabla \times u = 2\Omega \tag{4.6}$$

The above expression simply states that the amount of vorticity of the flow is twice the rate of rotation of the fluid. By adopting this formulation, the RANS equation of eq. (4.4) can be written as:

$$\rho \left[ \frac{\partial \overline{u}_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} \right] = \frac{\partial}{\partial x_j} \left[ 2\mu \overline{S}_{i,j} - \rho \overline{u'_j u'_i} \right] - \frac{\partial \overline{p}}{\partial x_j}$$
(4.7)

### **4.2.** TURBULENCE MODELLING

From the derivation of the RANS equations, it follows that the fluid motion can be described by a mean and a fluctuating velocity component. The fluctuations from the mean fluid velocity are due to turbulence. To be able to solve the RANS equations, one must introduce and additional model to determine the Reynolds Stresses:

$$\tau_{ij} = -\rho \overline{u'_{i} u'_{i}} \tag{4.8}$$

In total, six additional unknowns are introduced by Reynolds Stress that need to be solved. Various turbulence models have been developed over time to close the system of equations, required for determining the time-averaged mean flow. The most commonly used method to close to RANS equations for engineering application are so-called Linear Eddy-Viscosity Model's (LEVM). They rely on the Boussinesq [47] assumption, which relates the Reynolds stresses by a linear relationship with the mean flow strain rate:

$$\tau_{ij} = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) = 2\mu_t S_{ij} - \frac{2}{3}\rho k\delta_{ij}$$
(4.9)

where  $\mu_t$  is the turbulence or eddy viscosity. The turbulence viscosity is not constant but variable in space. However, it is assumed to be isotropic in all directions. For relative simple flow problems, the use of LEVM yields good results. However, for more complex flows, for example with strong flow separation or large swirls, the assumption of isotropic Reynolds stress is no longer valid, as the secondary shear stresses and normal stresses are not well predicted. Nevertheless, this class of turbulence models remains attractive for engineering use due its ability to provide reasonable accurate predictions while not requiring excessive computational effort. In the following subsections, a selection of commonly used turbulence models is discussed in more detail. For an overview of all the equations and assumptions involved, the reader is referred to Appendix A.

### 4.2.1. SPALART-ALMARAS

The Spalart-Allmaras [48] is a one-equation turbulence model that has specifically been designed for aerospace applications, involving wall-bounded flows. It utilizes a modelled transport equation of the eddy viscosity. The transport equation is given by:

$$\frac{\partial}{\partial t} \left( \rho \tilde{v} \right) + \frac{\partial}{\partial x_i} = \frac{1}{\sigma_{\tilde{v}}} \left[ \frac{\partial}{\partial x_j} \left( \left( \mu + \rho \tilde{v} \right) \frac{\partial \tilde{v}}{\partial x_j} \right) + C_{b2} \rho \left( \frac{\partial \tilde{v}}{\partial x_j} \right)^2 \right] + C_{b1} \rho \tilde{S} \tilde{v} - C_{w1} \rho f_w \left( \frac{\tilde{v}}{d} \right)^2 \tag{4.10}$$

,where  $\tilde{v}$  is equal to the eddy viscosity  $\mu$ , except at the near wall region. The last two terms of eq. (4.10) are the terms for turbulent production and destruction respectively. Although many different implementations of the Spalart-Allmaras exist, only the formulation proposed by Bradshaw *et al* [49] is discussed here. One of the advantages of the Spalart-Allmaras model is the computational efficiency and it's capability to provide accurate predictions for flow cases with up to moderate separation. For more details about the Spalart-Allmaras model, the reader is referred to section A.1.

#### **4.2.2.** $k - \epsilon$ MODEL

The  $k - \epsilon$  turbulence model focusses on the mechanisms that affect the turbulence kinetic energy. The instantaneous kinetic energy of a turbulent flow k(t) is the sum of the mean kinetic energy K and the turbulent kinetic energy k.

$$k(t) = K + k \tag{4.11}$$

,where the kinetic energy is simply given by:

$$K = \frac{1}{2} \left( U^2 + V^2 + W^2 \right)$$
  

$$k = \frac{1}{2} \left( \overline{u'}^2 + \overline{v'}^2 + \overline{w'}^2 \right)$$
(4.12)

Because of viscous effects, turbulent flows are dissipative. Kinetic energy is converted to heat due to shear stresses acting on the flow. Hence, if no energy is supplied to the flow, the flow disturbances, or eddies, die out quickly. The rate at which the turbulence dissipates is called the dissipation rate  $\epsilon$ . To obtain values for k and  $\epsilon$  two separate equations need to be solved. First, the equation for the kinetic energy K is treated. The governing equation for K can be found [50] to be:

$$\frac{\partial(\rho K)}{\partial t} + \frac{\partial}{\partial x_i}(\rho K U_i) = \frac{\partial}{\partial x_j} \left( -PU + 2\mu U S_{ij} - \rho U \overline{u'_i u'_j} \right) - 2\mu S_{ij} \cdot S_{ij} - \left( -\rho \overline{u'_i u'_j} \cdot S_{ij} \right)$$
(4.13)

,where in the above expression  $S_{ij}$  represents the mean rate of deformation tensor. On the left hand-side of the above expression entails the rate of change of *K* plus the transport of *K* by convection. The right hand-side term is more elaborate. The first term is equal to the transport of *K* by pressure and viscous stresses minus the transport of *K* by Reynolds stresses. The next term is the rate of dissipation of *K*, while the last term describes the production of turbulence. Hence, in the case of turbulence, kinetic energy of the mean flow it transitioned into kinetic turbulent energy. The equation of *k* is very similar:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left( -\overline{p' u'} + 2\mu \overline{u' S_{ij}} - \rho \overline{u_i u'_i u'_j} \right) - 2\mu \overline{s'_{ij} \cdot s'_{ij}} + \left( -\rho \overline{u'_i u'_j} \cdot S_{ij} \right)$$
(4.14)

,where  $s'_{ij}$  is the fluctuating component rate of deformation tensor. Note the plus sign for the last term on the right hand-side. The above expression contains additional fluctuating terms, such as  $\overline{p'}$ , which are unknown. Instead, often a more simplified model equation for k is used:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + 2\mu_t S_{ij} \cdot S_{ij} - \rho \epsilon$$
(4.15)

The left hand-side remained unchanged, however the right hand-side now states that the change increase of k should be equal to the diffusive transport of k plus the rate of production of k minus the rate of destruction. The Prandtl number  $\sigma_k$  connects the diffusivity of k to the turbulent viscosity  $\mu_k$ . Typically, a value of 1.0 is used. [51] The dissipation of k is described by the sixth term of eq. (4.14), as this captures the work done by the smallest eddies. One can now define the dissipation rate per unit mass  $\epsilon$  as:

$$\epsilon = 2\nu \overline{s'_{ij}s'_{ij}} \tag{4.16}$$

The analytical equation is fairly long and contains some unknown higher order terms and cannot be solved. Instead, a simplified model equation is derived by multiplication of the *k* equation with  $\epsilon/k$ , which results to:

$$\frac{\partial(\rho\epsilon)}{\partial t} + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j} \left[ \frac{\mu_t}{\sigma_\epsilon} \frac{\partial\epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} 2\mu_t S_{ij} \cdot S_{ij} - C_{2\epsilon}\rho \frac{\epsilon^2}{k}$$
(4.17)

The Prandtl number  $\sigma_{\epsilon}$  connects the diffusivity of the dissipation  $\epsilon$  to the eddy viscosity  $\mu_t$ . Typically, a value of 1.30 is used. [51].

The advantage of the standard  $k - \epsilon$  model is that is relatively simple to implement and leads to stable, relatively fast converging solutions. On the downside, it is not able to predict solutions for flows with a strong degree of rotation and separation well. [11] This is partially caused by the fact that the equations of the standard model become numerically unstable when integrated to the wall. [51] For such cases, a separate model should be used to capture the turbulent behaviour of the flow near a wall.

An improved version of the standard  $k - \epsilon$  model was introduced by Yakhot et al. [52] The underlying principle of this improved model is that the RANS equations are re-normalized to account for the effect of smaller scales of eddies. In the standard model the eddy viscosity  $\mu_t$  is determined for a single turbulence length scale. This implies that the turbulent diffusion only occurs at the pre-described length scale and ignores the effect of other scales of motion. To re-normalize the RANS equations, a mathematical and statistical technique called Re-normalization Group Method (RNG) is applied. The RNG procedure expresses the effect of the small scales of motion in terms of larger scale motions. [11]

Like the RNG  $k - \epsilon$  model, the Realizable  $k - \epsilon$  focusses on improving the turbulent dissipation  $\epsilon$  equation. It entails a new formulation for the turbulent viscosity. The Realizable turbulence model is developed by Shih et al. [53] The term 'realizable' means that the satisfies the mathematical and physical constraints on the Reynold stresses, which is not the case for the standard or RNG  $k - \epsilon$  model equations. The Standard, RNG and Realizable  $k - \epsilon$  models are discussed in more detail in sections A.2 to A.4.

#### **4.2.3.** $k - \omega SST$

The  $k - \omega$  Shear Stress Transport (SST) model was developed by Menter [54]. The model name stems from the modified turbulent viscosity equation, which has been modified to account for the transport of the principal turbulent shear stress. It was developed as an improvement of the original  $k - \omega$  model by Wilcox [55]. Although the standard  $k - \omega$  model outperforms the  $k - \epsilon$  turbulence models in laminar sub-layers and the logarithmic region of the boundary layer, the model appears to be strongly influenced by the specified free stream value of  $\omega$  outside the boundary layer [51]. As such, the  $k - \omega$  model is not very suited for boundary layer wake flow. The  $k - \epsilon$  model on the other hand is quite accurate for these regions of the flow [51]. By combing the two models on can add the strengths and remove the weaknesses of each individual model.

The first transport equation is similar to  $k - \epsilon$  models and describes the transport of the turbulent kinetic energy. The second model equation of the  $k - \omega$  model describes the transport of  $\omega$ , the specific turbulent dissipation rate, sometimes also referred to as the turbulent frequency. Instead of using the governing equation of fluid motion, the equation for  $\omega$  is constructed based on the known physical process of turbulence. The process of turbulent dissipation takes place at the level of smallest eddies. However, the rate of dissipation  $\epsilon$  is rate of energy transfer between the turbulent kinetic energy to the smallest eddies. Since the larger eddies determine the turbulent kinetic energy, the dissipation rate  $\epsilon$  is set by the properties of the large eddies. The specific turbulent dissipation rate  $\omega$  can be thought of as the ratio of turbulent dissipation rate to the turbulent mixing energy, or the rate of dissipation of turbulence per unit energy.

$$\frac{\partial(\overline{\rho}\omega)}{\partial t} + \frac{\partial}{\partial x_i}(\overline{\rho}\omega U) = \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_{\omega 2}\mu_t) \frac{\partial \omega}{\partial x_j} \right] + \alpha_2 \frac{\omega}{k} \tau_{ij} \nabla U - \beta_2 \overline{\rho}\omega^2 + 2(1 - F_2)\overline{\rho}\sigma_{\omega 2} \frac{1}{\omega} \nabla k \nabla \omega$$
(4.18)

In the combined equation of eq.(4.18)  $F_1$  is the so-called blending function. The  $k-\omega$  equations are multiplied by a blending function  $F_1$  and the equations of  $k - \epsilon$  model by  $(1 - F_1)$ . It is designed such that near the wall



Figure 4.1: Blending function  $F_1$  for various wall velocity profiles [9]

the function  $F_1$  is equal to one, such that the  $k - \omega$  model is used, and equates to zero away from the wall. in the latter case, the  $k - \epsilon$  equations are used. In compact form, the constant used in eq. (4.18) is expressed as:

$$\phi = F_1 \phi_1 + (1 - F_1) \phi_2 \tag{4.19}$$

,where  $\phi_1$  represents the constants associated with the  $k - \omega$  model ( $F_1 = 1$ ) and  $\phi_2$  the constants of the  $k - \epsilon$  model ( $F_1 = 0$ ). The behaviour of the blending function  $F_1$ , for various near-wall velocity profiles, is shown in Figure 4.1. As one can observe, the transition between use of the  $k - \epsilon$  and  $k - \omega$  model occurs in a small portion of the upper part of the boundary layer. Menter [9] compared the performance of the SST model with the  $k - \epsilon$  Standard model for a flat plate under an adverse pressure gradient. Note that the flow did not separate from the plate at any time. It was found that the SST model was able to almost exactly predict the skin friction, where as the  $k - \epsilon$  consistently over-predicted the amount of skin friction. However, when examining the velocity profiles, it was found that the flow retardation predicted by the SST model is stronger than the  $k - \epsilon$  model and the experimental data. In the case of the turbulent shear stress, both models over-predict the amount of shear stress. However, the SST model matched the experimental data closest. When examining the flow over the upper surface of a NACA 4412 airfoil at high angle of attack, the SST model again proved to be able to predict the velocity profiles at various chord-wise stations with good precision, in contrast to the  $k - \epsilon$  model. [9].

#### **4.2.4.** REYNOLDS STRESS MODEL

Instead of modelling the turbulence by means of energy and dissipation, the Reynolds Stress Model (RSM) calculates the individual Reynold's stresses  $\overline{u'_i u'_j}$  using six differential transport. Hence, the concept of Reynolds stress isotropy of the two-equation models is abandoned. Instead, every Reynolds stress component is computed individually and therefore directional effects are incorporated in the model. The RSM was originally developed by Launder and Rodi [56]. The exact form of the Reynold stress transport equations are obtained by multiplying the exact momentum equation with fluctuation terms and subsequent Reynolds averaging of the product. The transport equation for the Reynold stress  $\overline{u'_i u'_i}$  is given by: [10]

$$\frac{\partial}{\partial t}(\rho \overline{u'_i u'_j}) + \frac{\partial}{\partial x_k}(\rho u_k \overline{u'_i u'_j}) = D_{T,ij} + D_{L,ij} + P_{ij} - G_{ij} + \phi_{ij} - \epsilon_{ij} - F_{ij}$$
(4.20)

,where  $D_{\tau_i}$  is the turbulence diffusion term,  $D_L$  the molecular diffusion, P the rate of turbulence production, G a term to account for buoyancy,  $\phi$  a pressure-strain relation,  $\epsilon$  the dissipation rate of turbulence and F the production of turbulence due to rotational effects. An overview of these terms is given in Appendix A.7. Like the  $k - \epsilon$  models, the Reynold Stress Model makes use of the concept of turbulent kinetic energy and dissipation. The turbulent kinetic energy in the case of RSM is obtained by:

$$k = \frac{1}{2}\overline{u_i'u_j'} \tag{4.21}$$

,which is modelled in this case by Fluent [10]:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + \frac{1}{2} \left( P_{ij} + G_{ij} \right) - \rho \epsilon \left( 1 + 2M_t^2 \right)$$
(4.22)

Again, the turbulence production *P* and buoyancy term *G* occur in eq. (4.22). The term  $M_t$  is the turbulent Mach number, meaning that multiplied with turbulence dissipation rate  $\epsilon$ , rapid fluctuations are damped more quickly.

Although one would expect that the RSM would lead to more accurate results, due to the inclusion of anisotropy of the Reynolds stresses, this is certainly not always the case. Only in flow problems were there is a significant flow anisotropy present, for example in highly swirling or rotating flows, the RSM is in some cases able to provide more accurate predictions. Since the RSM still includes modelling assumptions compared to the exact Reynolds Stress transport equations, for certain flow types not involving high levels of anisotropy, the two-equation models could still prove to be more accurate than the RSM. Taking into account that seven additional transport equations are introduced by the RSM, the computational effort is significantly increased (up to a factor 22) [57] Therefore, the RSM is often not the first choice when selecting a turbulence model. Only when the existing two-equation models appear to be unable to provide accurate results or more insight into the flow properties is needed, the RSM is considered to be an attractive alternative.

# **4.3.** DISCRETIZATION

In order to apply the algebraic transport equations, the equations are discretized such that they can be applied on the grid for numerical evaluation. Fluent makes use of the Finite Volume Method (FVM), in which the computational domain is divided into smaller control volumes. By applying the discretized transport equations on these control volumes and solving the system of equations, one can obtain a solution for the entire domain. To illustrate the basic idea behind dicretization, consider the following transport equation for an arbitrary scalar quantity  $\phi$ :

$$\int_{V} \frac{\partial \rho \phi}{\partial t} \, dV + \oint \rho \phi \vec{u} \cdot d\vec{A} = \oint \Gamma_{\phi} \nabla \phi + \int_{V} S_{\phi} dV \tag{4.23}$$

,where  $\Gamma_{\phi}$  is a diffusion coefficient for  $\phi$ ,  $\vec{A}$  a surface area vector and  $S_{\phi}$  an additional source term. If one applies the above formula to the two-dimensional control volume, as shown in Figure 4.2 one would arrive at:

$$\frac{\partial \rho \phi}{\partial t} V + \sum_{f}^{N} \rho_{f} \vec{v}_{f} \phi_{f} \cdot \vec{A}_{f} = \sum_{f}^{N} \Gamma_{\phi} \nabla \phi_{f} \cdot \vec{A}_{f} + S_{\phi} V$$
(4.24)

The above procedure of transforming the algebraic expressions into discrezited equations can be applied for any scalar transport equation and control volume topology. In Fluent, the scalar values, by default, are stored at the cell centres. However, as can be observed from eq. (4.24), the term  $\phi_f$  arising in the convective term implies that the scalar value at the face center needs to be known. In order to achieve this, the face value is approximated using either central difference or upwind scheme. For convection dominant flows (Peclet number  $Pe \ge 2$ ), an upwind differencing scheme should be preferred.

#### 4.3.1. SECOND-ORDER UPWIND

In the case of an upwind scheme, it is assumed that the  $\phi_f$  at the downstream face is determined solely by the upstream cell center. The most simple form of upwind approximation schemes is a first-order upwind



Figure 4.2: Example of a two-dimensional control volume. Indicated are the cell centers  $c_0$  and  $c_1$ , face area surface vector  $\vec{A}$  and the vectors between the cell and face centers  $\vec{r}_0$  and  $\vec{r}_1$  [10]

scheme, where it is assumed that the scalar value at cell center is the cell average and therefore equal throughout the particular control volume. Since this is a crude approximation, a second order upwind scheme is preferred in most cases. By truncating the Taylor series expansion after the second term, one arrives at:

$$\phi_{f,SOU} = \phi + \nabla \phi \cdot \vec{r} + \mathcal{O}^2 \tag{4.25}$$

The term on the right-hand side that remains is the error that is introduced by truncating the Taylor series. The term is proportional to the control volume size squared, hence the name second-order upwind. In Fluent, the gradient  $\nabla \phi$  is limited, to prevent the creation of a new maximum or minimum at the cell face.

#### 4.3.2. QUICK

In order to reduce the error introduced by truncation of the Taylor series, higher-order schemes have been developed. One of such schemes is the Quadratic Upstream Interpolation for Convective Kinetics (QUICK) scheme. [58] With the QUICK scheme,  $\phi_f$  is determined by a quadratic interpolation using the adjacent cell centers and an upstream cell. For a grid with uniform spacing, the interpolation is defined as follows:

$$\phi_e = \theta(\phi_P + \phi_E) - (1 - \theta)(\phi_W + \phi_E - 2\phi_P) \tag{4.26}$$

In the original QUICK scheme, the default value for  $\theta$  is 1/8. However, in the case of Fluent,  $\theta$  is used as a variable to avoid the local creating of new maxima or minima. Note that if  $\theta = 1$ , the QUICK scheme reduces to a central differencing scheme, while for  $\theta = 0$  a second-order upwind scheme remains. Since the QUICK scheme requires the cell faces to be more or less aligned with the flow direction, it is most suitable for structured quadrilateral or hexahedral schemes. In the case of unstructured grids, Fluent by default will adapt to a second-order upwind scheme. Being a quadratic approximation, the truncation error of the QUICK scheme is of the 3rd order.

#### 4.3.3. MUSCLE

Another third-order scheme available in Fluent is the Monotome Upstream-Centered Scheme for Conservation Laws (MUSCL). The term monotome means that the MUSCL scheme is monotonicity preserving, which means that:

- i No local minima or maxima are created
- ii Existing minima or maxima should be non-decreasing or non-increasing respectively.

In short, it implies that the scheme does not create new undershoots or overshoots of the solution and does not exaggerate existing extremes. Mathematically, for monotonicity to be satisfied, it requires that the total variation cannot increase and should diminish with time. Hence, such schemes are often referred to as Total Variation Diminishing (TVD) schemes. As was shown by Sweby [59], the QUICK scheme does not satisfy the conditions for TVD for certain values of r, which is defined as:



Figure 4.3: Sweby's diagram, showing the region for second-order TVD (hi-lighted in grey) [11]

$$r = \left(\frac{\phi_P - \phi_W}{\phi_E - \phi_P}\right) \tag{4.27}$$

Or in words, *r* is the ratio between the upwind and downwind gradient of  $\phi$  and can be viewed as a measure for the smoothness of the data. A generalized form for a higher-order discretization scheme is introduced:

$$\phi_e = \phi_P + \frac{1}{2} \Psi(r) \left( \phi_E - \phi_P \right) \tag{4.28}$$

The above can be viewed as an first-order upwind discretization scheme with an additional convective flux added. By changing the function  $\Psi(r)$ , called the flux limiter function, one can describe various schemes. With  $\Psi = 0$  a first order upwind discretization is recovered and for  $\Psi(r) = 1$  a central discretization scheme is obtained. Similarly, one can show that for  $\Psi(r) = (3 + r)/4$  the QUICK scheme is represented. Sweby [59] showed only for certain ranges of  $\Psi(r)$  and r second-order schemes can meet the requirements for TVD. This is reflected by the so-called Sweby-diagram, shown in Figure 4.3. As can be observed, each second-order scheme should pass through the point (1, 1) in the  $r - \Psi$  diagram in order to meet the TVD requirement.

In the case of the MUSCL scheme by Bram van Leer [60], the flux limiter function is given by:

$$\Psi(r) = \frac{r + |r|}{1 + r}$$
(4.29)

,which is a smooth function passing through the point (1,1) in the Sweby diagram. In the case of the QUICK scheme, the flux limiter function is a piecewise linear function, rather than a smooth function. The MUSCL scheme implemented in Fluent [10] does not contain any flux-limiters. Instead, it is formulated as blending function between a central difference scheme and a second-order upwind scheme:

$$\phi_e = \theta \,\phi_{e,CD} + (1 - \theta)\phi_{e,SOU} \tag{4.30}$$

where  $\theta$  is again a blending parameter. Because this formulation does not contain a flux limiter, underand overshoots of the solution can still occur. However, the MUSCL scheme can be applied to any arbitrary mesh, including unstructured tetrahedral meshes. Being third-order accurate, the MUSCL scheme reduces numerical diffusion, improving the accuracy compared to second-order upwind schemes.

### **4.4.** MODELLING OF THE BOUNDARY LAYER

The effect of a wall on the turbulent flow is quite significant. Not only due to affected velocity field to satisfy the no-slip condition at the wall, but also by means of enhanced viscous effects. If one defines a Reynolds number  $Re_L$  based on the length scale in the flow direction,  $Re_L$  for a free shear layer is usually very large > 10<sup>5</sup>. [11] Hence, the inertia forces acting on the flow are much more dominant than the contribution of viscous forces. Similarly, if one defines the Reynolds number with respect to vertical distance to the wall *y*, the same holds true if *y* is sufficiently far away from the wall. [11] However, when the wall is approached the viscous forces become more relevant. Very close to the wall, when  $Re_y$  is in the order of 1 and  $y \approx 0$ , the viscous forces become equal or larger than the inertial forces. As a consequence, the flow very near to the wall does not depend on the free stream flow parameters, but is a function of the distance to the wall *y*, the fluid density  $\rho$  and viscosity  $\mu$  and the wall shear stress  $\tau_w$ .

$$u^{+} = \frac{U}{u_t} = f\left(\frac{\rho u_\tau y}{\mu}\right) = f\left(y^{+}\right)$$
(4.31)

Above formula, eq. (4.31), is called the law of the wall [11]. Note that  $u^+$  and  $y^+$  are both dimensionless quantities. Further away from the wall, the flow is not directly affected by viscosity, but rather by the wall shear stress. The appropriate length scale in this case is the boundary layer thickness  $\delta$ . It can be shown [11] that the following relation holds true in this case:

$$\frac{U_{max} - U}{\mu_{\tau}} = g\left(\frac{y}{\delta}\right) \tag{4.32}$$

Eq. (4.32) is referred to as the velocity-defect law [11]. Near the solid surface of the wall the fluid is almost stationary and any turbulent motion is dominated by viscous effects. This layer, called the viscous sub-layer, is often very thin ( $y^+ < 5$ ). Therefore is can be assumed that the shear stress  $\tau(y)$  is approximately constant through this layer and equal to the wall shear stress  $\tau_w$ :

$$\tau(y) = \mu \frac{\partial U}{\partial y} \approx \tau_w \tag{4.33}$$

Using the boundary condition U = 0 if y = 0 and eq. (4.31) one can show that the following simple linear relation holds true:

$$u^{+} = y^{+} \tag{4.34}$$

For obvious reasons, this very thin flow layer directly adjacent to the wall is called the linear sub-layer. Further away from the wall ( $30 < y^+ < 500$ ) a region exists where the viscous and turbulent effects are both important. The shear stress  $\tau$  varies only slowly with the wall distance y and therefore the assumption of eq. (4.33) is maintained. Using an additional assumption for the turbulence length scale, it can be shown [11] that:

$$u^{+} = \frac{1}{\kappa} \ln(y^{+}) + B = \frac{1}{\kappa} \ln(Ey^{+})$$
(4.35)

,where  $\kappa$  is the von Karman constant and E (or B) an additional constant. For smooth walls, their values are respectively  $\kappa \approx 0.4$  and  $E \approx 9.8$  (or  $B \approx 5.5$ ). These constants are valid for any turbulent flow over smooth walls at high Reynolds number. With increasing wall roughness, the value of E decreases. Because of the logarithmic scale of eq. (4.35) often called the log-law and the flow region the log-law region. In Figure 4.4, expressions (4.34) and (4.35) are compared to the experimental results by Schlichting [61]. As one can observe, the approximate functions show close agreement with the experimental results by Schlichting. The region between the linear sub-layer and the log-law region, ( $5 < y^+ < 30$ ) is called the buffer layer. Here the viscous and turbulent stresses are of equal magnitude. Together, these three layers form the inner layer, which is about 10% - 20% of the total thickness of the boundary layer [11]. The most important characteristic of the inner layer is that the shear stress is almost constant and equal to the wall shear stress. The log-law has remains valid up to values of approximately  $y/\delta \approx 0.2$ . Beyond this the velocity-defect law takes effect. This region is called the outer layer. Here, the viscous effects are no longer of direct influence on the flow but rather inertial effects become dominant.

The above expressions form the basis for so-called wall functions. Since the two-equation  $k - \epsilon$  and the  $k - \omega$  BSL models cannot be integrated right up to the wall and require a separate function, that describes the flow at the inner layer of the viscous layer. Note that above formulations for the velocity profiles have been derived based on two-dimensional Couette flow with the assumptions of a small pressure gradient, local equilibrium of turbulence and a constant near-wall stress layer. [51] By turbulence equilibrium is meant that the production of turbulence is equal to the amount of turbulence dissipation. Hence, on should take care that the underlying assumptions of the so-called equilibrium or standard wall functions are applicable to the the flow problem at hand. In the case of aerospace applications, where an adverse pressure gradient is experienced by the boundary layer on the airfoil suction-side, standard wall functions may not produce



Figure 4.4: Non-dimensional boundary layer velocity profile as a function of wall distance in comparison with experimental data [12]

accurate results. Instead, one should opt for a non-equilibrium wall function or select a turbulence model (RSM, SA or  $k - \omega$  *SST*) that do not require wall functions. In the latter case, one should take special care of the near-wall mesh, as will be discussed in 4.4.1.

#### **4.4.1.** NEAR-WALL MESHING GUIDELINES

In general, when using low *Re* adaptations of the RSM and  $k - \omega$ , such that these can be evaluated right up to the wall, the fineness and resolution of the near-wall mesh becomes very important. Because of the strong interaction between the turbulence and the mean flow, the numerical results for turbulent flows are susceptible to even small grid defects. In order to resolve the viscous sub-layer of the turbulent boundary layer, the node of the first layer of cell adjacent to the wall should be with approximately positioned at  $y^+ \approx 1$ . A higher  $y^+$  value for the first cell is tolerable, given that the cell node is still within the viscous sub-layer ( $y^+ \leq 5$ ) [12]. Preferably, there should be five to ten layers of cells between the wall and the location  $y^+ = 20$ , in order to resolve the mean velocity an turbulent quantities in the near-wall viscous region. As mentioned, a high mesh resolution is computationally quite expensive, often the cost of computation is of one order larger [12] compared to the case when wall functions are used.

In the case that wall functions are employed, one can use a lower resolution mesh since the near-wall region is not resolved directly. For standard and non-equilibrium wall functions, the cell node of the first wall adjacent layer should be positioned in the log-law region. A value of  $y^+ \approx 30$  is preferred [10]. In any case, one should try to avoid that the wall adjacent cells are located in the buffer region,  $5 < y^+$ , 30. The upper bound of the log-law region is much dependent on the Reynolds number. However, one should make sure to have at least eight to ten cells within the boundary layer to achieve a reasonable accuracy. A typical moderate Reynolds number flow has a boundary layer that extends up to  $y^+$  values between 300 and 500. In order to check whether the resolution of the mesh is adequate, one should perform a post analysis of the results of the boundary layer thickness and adjust the mesh accordingly. Often a subsequent simulation with a finer mesh is performed to analyse the grid-dependency of the result.

#### **4.5.** SOLUTION PROCEDURE

Referring back to the momentum equations (4.1), one can see that a pressure term is present in the transport equation of each velocity component. Since the equations are coupled, the pressure is an additional unknown that needs to be determined. Without a pressure-gradient available, the velocity field cannot be computed. One way to solve this is to guess a certain pressure field  $p^*$  and to use this to compute the mo-



Figure 4.5: Block-schematic of the Fluent solver module for coupled and non-coupled solving methods

mentum equation. A pressure-correction p' is than later deduced from the convective term to update the pressure field. In short, this is how the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) by Patankar and Spalding deals with the pressure-velocity coupling. [62].

The main approximation of the SIMPLE scheme is that, in the process of updating the intermediate velocity field, only the pressure correction terms are used. Terms involving velocity corrections of neighbouring cells are omitted for simplicity. For the final converged solution, when the guessed and actual pressure field should be identical, this assumption does not affect the solution. However, the process of updating the pressure field by the pressure correction p' can cause instabilities or even divergence of the solution, for example when the guessed pressure field and final pressure field are to far apart. Hence, under-relaxation factors should be applied to stabilize the computation. Under-relaxation factors do not affect the solution, but do increase the computation time considerably. In Fluent, the default under-relaxation factor  $\alpha_{press} = 0.30$ . Note that the residuals in Fluent are directly scaled with the relaxation factors. Hence, one cannot rely simply on a certain threshold for the residuals to asses if the solution has converged or not.

To reduce the need for relaxation factors, and therefore the computational time, a revised version of the SIM-PLE algorithm was introduced. [63] Called the SIMPLE-Consistent (SIMPLEC) algorithm, the so-called dterm, occurring in the process of updating the velocity components with the pressure correction term p', has been modified to reduce the effect of omitting terms. As a result, much larger relaxation factors for the pressure-velocity coupling can be applied. In terms of accuracy, both schemes perform near identical. [10]

The SIMPLE and SIMPLEC algorithm's utilize a segregated pressure-based solver. This implies that the pressure values are stored at the cell centres, where-as the velocity scalars are stored at the face cells. As such, the momentum and pressure correction equations can solved separately. This is in contrast to a coupled approach, where all equations are solved at once. The coupled approach is sometimes experienced to be perform better, but requires up to  $2 \times$  the amount of computational memory [10]. An overview of the solving procedure by the Fluent solver is shown in Figure 4.5. After solving the momentum and continuity equations, the Fluent segregated solver continues to solve the energy equation followed by the turbulence transport equations. Additional transport equations are solved in the last computation step. Having solved the transport equations, the solver will proceed to a next iteration or abort if the convergence criteria have been met.

# 5

# **CFD** CASE STUDIES

In order to gain confidence in the CFD simulations, a series of validation cases have been performed to identify the most suitable turbulence model, learn about proper mesh generation and discover solution techniques to aid convergence of the solution.

# 5.1. NACA 0012

The first validation case that has been examined is a comparison of CFD with a NACA 0012 airfoil in an unbounded flow domain. The results are to be compared with a selection of experimental studies for validation and verified with numerical results obtained by NASA. [13] The case under examination is a two-dimensional NACA 0012 airfoil at incompressible flow conditions (M = 0.15) and Re = 6.0 million. The airfoil geometry is adapted slightly to accommodate a sharp trailing edge. The latter is convenient for the creation of a structured grid. Although this changes the maximum thickness-to-chord ratio, the t/c is actual identical for the aiffoil with a blunt trailing edge and the chord extended. To limit the number of computations, a single condition ( $\alpha = 10^\circ$ ) was selected for comparison. At this angle of attack, the airfoil is still just within the linear part of the lift-curve slope.

The flow domain used for modelling is a conventional structured C-grid, with the upper and upstream boundary extending 12.5*c* in front and the downstream boundary 20*c* behind the airfoil. In order to guarantee that the boundary layer is captured with sufficient accuracy, the cells are clustered around the body using a bias ratio of the edge-meshing parameters. A growth ratio of r = 1.20 was adapted to ensure a smooth transition in cell growth. As found in literature, a mesh size of 120,000 to 200,000 should be sufficient to reduce the error introduced by grid dependency. [64] Although the studies used for validation consist of experiments where



Figure 5.1: Close-up of the structured grid around the airfoil body. Note that domain is split in a laminar part (grey) and a turbulent part (orange)



Figure 5.2: Detail of the mesh at the trailing edge of the NACA 0012

the flow was tripped at approximately x/c = 0.05, natural transition at  $\alpha = 10^{\circ}$  occurs upstream of the trip location. [65] [19] [66] With the use of XFOIL, the transition location at the angle of attack under consideration was determined to be at x/c = 0.015. Although by far the majority of the airfoil is submerged in turbulent flow, it was found that the inclusion of a small laminar flow part had a significant effect on the simulated results.

In order to make sure that the calculation are converge steadily and no additional errors are introduced, special care was taken to improve the quality of the mesh. For example near the trailing edge (see Figure 5.2), one should make sure that the transition of cell volume size between the airfoil and the near-wake is smooth. In order to quantify the mesh quality, several parameters of the mesh can be computed such as the orthogonal quality and skewness of the cells. [10]. Both are a measure for the amount of skewness of the cells. In the ideal case, the skewness is close to 0 and the orthogonal quality approximately equal to 1.0. For the meshes used in this validation case the average orthogonal quality was  $\approx 0.95$  and average skewness 0.11, which is considered to be a mesh of good quality. Since for this comparison two types of turbulence models are being investigated, namely models that resolve the flow up to the wall and models that require a separate wall model, two grids have been developed. The grids are identical in topology, but differ slightly in the number of cells and boundary layer resolution.

In terms of boundary conditions a far-field pressure boundary condition has been applied at all boundaries. This implies that the free stream flow velocity and direction is specified at these locations. The turbulence boundary conditions are specified by means of a turbulence intensity *I* and turbulent viscosity ratio  $\mu/\mu_t$ . Since the experimental data was obtained in low-turbulence wind-tunnels, a turbulence intensity of *I* = 0.1 is assumed. The eddy viscosity ratio was fixed at 3 and the airfoil walls were modelled as no-slip walls. Although the case can be considered to be incompressible, compressible flow was assumed by inclusion of the energy equation and ideal gas law. The SIMPLEC scheme was used for pressure-velocity coupling and a second-order scheme for the pressure term. For spatial discretization, the MUSCLE scheme has been adopted for the energy and momentum equation, while the QUICK scheme is selected for the turbulence quantities. A least-squares cell-based method is adapted for gradient determination. Convergence was accepted when all residuals were below  $1.0e^6$  and no change in coefficient was observed after 500 iterations.

A comparison of the force measurements between CFD and experimental data is shown Table 5.1. As can be observed, all simulation over-predict the lift and the drag by a significant amount. The SST model appears to be able to match the result obtained by Ladson [19] the best. The Spalart-Allmaras follows second, while the  $k - \epsilon$  models perform significantly poorer. The Realizable model was used in this case to represent the  $k - \epsilon$  models. Of the various wall functions used, the Non-Equilibrium Wall-function (NEW) performs noticeably better than the Standard Wall-function (SW) or Enhanced Wall Treatment (EWT). Due to the fact that near the aft part of the airfoil significant flow separation is already present, wall models are not very well suited for the flow case at hand. Nevertheless, the NEW should be wall model of choice due to it's ability to handle adverse pressure gradient better. Also tested was the RSM  $k - \omega$  model, but unfortunately no converged solution could be achieved.

Name	type	$c_L$	$\Delta\%$	$c_D$	$\Delta\%$
Ladson [19]	Wind-tunnel	1.0530	-	0.01146	-
Spalart-Allmaras	CFD	1.0983	+4.3	0.01273	+11.1
SST $k - \omega$	CFD	1.0891	+3.4	0.01267	+ 10.6
Realizable $k - \epsilon$ + EWT	CFD	1.1043	+4.9	0.01391	+21.4
Realizable $k - \epsilon + SW$	CFD	1.1040	+4.8	0.01393	+21.5
Realizable $k - \epsilon$ + NEW	CFD	1.1096	+5.4	0.01337	+16.7
Spalart-Allmaras	NASA FUN3D	1.0983	+ 4.3	0.01242	+ 8.4
SST $k - \omega$	NASA FUN3D	1.0840	+ 3.0	0.01253	+9.4

Table 5.1: Comparison of coefficients at  $\alpha = 10^{\circ}$  for a NACA 0012 with experimental data [19] and numerical results obtained by NASA [13]

Also included in Table 5.1 are numerical results obtained by NASA using the FUN3D finite-volume CFD code [13]. As can be observed, the results are very comparable. Note that the computational domain used in that case are much larger, with the far-field boundary extending up to 500*c* away from the model. Next to FUN3D, multiple CFD codes were assessed to simulate the flow around the NACA 0012. It was found that the differences between the codes were in the order of 1% for lift and 4% for drag. [13] This is explained due to different implementations of the theory in each individual code. Hence, based on these observation the current results have been positively verified and validated, albeit a some significant deviations from experimental data.

# **5.2.** NACA 4412

Having established that the mesh-type and applied methods are adequate to model the flow around an airfoil subjected to flow separation, a second validation study is performed. In this case the purpose is not to establish which model is more accurate to determine the force coefficients, but rather to investigate its ability to capture the flow qualitatively. The flow case that is considered, is a NACA 4412 airfoil at  $\alpha = 13.89$  °. [13] Again, the airfoil has been modified to allow for a sharp trailing edge. Incompressible flow can be assumed (M = 0.15), however compressibility was taken into account. Ambient flow conditions were set to yield to a chord-based Reynolds number Re = 1.52 million.

The most important experimental data for this validation study are the velocity measurements of the separated flow near the trailing edge. Shown in Figure 5.3, are the six velocity rakes placed near the trailing edge of the airfoil. The velocity contours have been acquired from data provided by Coles and Wadcock. [14] Using a computer-controlled hot-wire, thousands of samples were taken in the trailing edge and near-wake region of the airfoil. Note than in Figure 5.3, the velocity has been normalized with a velocity approximately one chord below and downstream of the trailing edge. This is different from traditional normalization with the undisturbed free stream. To account for this, results obtained by CFD should be divided by 0.93 to arrive at similar results. [13]

The grids used for this study are similar to the ones used for the NACA 0012 validation case. Namely, a C-grid topology with upstream and upper/lower boundaries extending 12.5*c* away from the body. The downstream boundary is located 20 chord lengths away. Pressure far-field boundary conditions were applied to all contours of the domain and the mesh size consisted of 192,000 nodes. Sufficient capturing of the boundary layer was ensured by setting  $y^+$  equal to approximately unity over the entire airfoil. The SIMPLEC algorithm was applied, together with the MUSCLE scheme for the momentum and energy equations while the QUICK scheme was adopted for turbulent quantities. Since the actual force measurements are not of prime importance for this study and transition is found to occur right at the leading edge, a fully turbulent domain has been adopted. Instead, two transition turbulence models have been implemented next to the traditional turbulence models as described in Section 4.2. The additional models are the Transition SST  $k - \omega$  and the  $k - kl - \omega$  model. [10].

Shown in Figure 5.4 is a comparison of the result obtain by the SST  $k - \omega$  model compared with the exper-



Figure 5.3: Overview of the velocity rakes at the trailing edge of NACA 4412 at  $\alpha = 13.89^{\circ}$  [13]

imental data by Coles and Wadcock [14]. As can be observed, the results match the wind-tunnel data quite well for the first few rakes. However, approaching the trailing edge further, the results start to deviate slowly. Considering again Figure 5.3, one can see that the zone of recirculation begins approximately at x/c = 0.78, the position of the third rake. Examining the vertical velocity component, shown in Figure B.10, one can see that the v-velocity is not captured very well. This is not specific for the SST turbulence model, since non of the applied turbulence models appears to be able to capture the vertical flow component accurately. An overview of the performance of other turbulence models is presented in Appendix B. Interestingly, the Transition SST model does not provide a better representation of the flow compared to the standard SST model (Figure B.5). Instead, the deviation from the wind-tunnel data is larger. The performance of the other transitional turbulence model, the  $k - kl - \omega$  model, is also not very promising. In contrast, the RSM  $k - \omega$  model performs quite well in this case, as can be observed from Figure B.7 and B.8. The Spalart-Allmaras model also performed reasonably well (Figures B.1 and B.2). The Realizable  $k - \epsilon$  model was also tested, in this case with a Menter-Lechner [10] vplus independent wall model, such that it could be applied on the same mesh. The results are mediocre at best. Having failed to approximate the lift and drag characteristics of the NACA 0012 with mild separation in the previous case, it must be concluded that the  $k-\epsilon$  models are most likely not very suited for further application. Instead, the list of candidate turbulence models is reduced to: SST  $k - \omega$ , Spalart-Allmaras and RSM  $k - \omega$  to narrow down the search and save time as the computations become more computationally expensive.

Despite the fact that the force coefficients were not of prime interest for this study, it is interesting to note that none of the models was able to predict the drag-coefficient by even a considerable margin. In all cases, the drag level was under-predicted by about 55 – 60%. This is explained, if one examines the pressure distribution (see Figure B.13) The CFD results all predict that the flow recovers to approximately stagnation pressure  $(C_p \approx 0)$ , where as in the experiment a zone of considerable lower pressure  $(C_p \approx -1)$  exists at the aft part of the airfoil. Hence, the additional pressure drag caused by the separated flow is not sufficiently taken into account. In terms of lift coefficient, the model predictions are much more accurate ranging from 1.5 - 2%) for the RSM  $k - \omega$  and SST models to about 6 - 7% for the other turbulence models. As a check for possible set-up deficiencies, the flow was also computed for a lower angle of attack ( $\alpha = 8^\circ$ ). In this case, using the Spalart-Allmaras model, the lift coefficient was within < 1% and the drag deviated about -6% of the experimental data. The implication of this observation is, that in the case of severe flow separation, the CFD results may not be capable of useful predictions.

To verify is the implementation of the numerical models have been performed correctly, the current results are compared with the data obtained by NASA [13]. Again, the data acquired by the NASA team with the FUN3D software package has been selected for comparison. As can be observed, the current results is identical, or even slightly better, than the reference data. However, it must be noted that the same comparison with the SST model was less favourable, as the predictions by NASA appear to be slightly more accurate (Figure B.14). Nevertheless, it confirms that the current computations are reliable in terms of their correct implementation. However it does underline the limitations of the CFD model, for example in the case of significant flow separation.

# 5.3. NACA 4412 IN GROUND-EFFECT

Having performed previous validation studies primarily to investigate the suitability of various turbulence models and developed skill and confidence for the primary computations, the following validation case is aimed to investigate the capability of the numerical models itself to model the ground effect. The study that will be used for validation is a wind-tunnel experiment by Ahmed [15], investigating the flow over a NACA 4412 in ground effect. This study was chosen as there are very few extreme ground effects studies published, especially if one considers the implementation of a correct ground boundary condition.

For the experiment, Ahmed used a conveyor belt system to replicate the moving ground. It should be noted that the test-section was an open-test section rather than a closed section. The experiment was conducted at incompressible conditions ( $V_{\infty} = 30.48$ m/s) and a Reynolds number of Re = 0.3 million. A range of elevation heights were examined, ranging from h/c = 0.05 to h/c = 1.00. The incidence angle, equal to the angle of attack in this case, was varied between 0° and 10°. The model, with a chord of 150mm and span of 600mm, was suspended between two plates. The plates extended 125mm in front of the leading edge and were of length 400mm. The dimensions of the square test-section was 800 × 800 × 1000 mm. The conveyor-belt system featured the same dimensions as the test-section floor. Since the angle of attack envisioned for the AeroCity will be low, the case of  $\alpha = 4^\circ$  and h/c = 0.05 is selected for further examination.

The mesh used for this computation is essentially an H-grid with a local C-grid around the airfoil. A close-up of the mesh is shown in Figure 5.6. The domain extends 8*c* upstream, 10*c* in upper direction and 15*c* down-stream. A mesh-size of 395,000 nodes was adopted, influenced by refinement near the ground plane along the entire distance of the domain. In terms of computational models, the SIMPLEC algorithm was applied once again. Since pressure gradients are expected to play an important role, the Pressure Staggered Option (PRESTO!) scheme for pressure-discretization was selected in favour of the default second-order scheme. Discretization of the momentum terms was performed by the MUSCLE scheme, while the turbulent quantities are taken care of with the QUICK scheme.

Unfortunately, initial computations showed a very poor agreement with experimental data. Lift coefficients where off by almost 25%, over-predicting the lift compared to the experiment conducted by Ahmed [15] Examining the pressure distribution, shown in Figure 5.7, it was found that the disagreement is caused by a large degree by the difference in underside pressurization. The suction side matches relatively better, except that the small suction peak near the leading edge is not observed in the experiment. Very similar observations were made by Smith *et al* [67] in a limited numerical study. Although this could point to a possible limitation of the numerical models, the difference could also stem from discrepancies in the experimental set-up. The fact that an open test-section has been used, together with the use of a model that does not span the entire width of the test-section, raise questions about the accuracy of the measured data.

In order to overcome the dilemma of which data to trust, a second experimental study was found that could be used for validation. A group of Japanese researchers has been performing experimental research on airfoil characteristics in extreme ground effect, to investigate their use for the a domestic development of a WIG vehicle, called 'Aerotrain'.[16] As the article is Japanese written, some information regarding the test setup was acquired from complementary studies. [68] [69]. Instead of having a fixed model and moving surroundings, a different method is applied where the model is moving by means of towing. A dedicated facility was constructed to allow for this type of testing, namely the Hyuga Aerodynamic Research by Towing (HART) facility. A schematic drawing of the test-setup is shown in Figure 5.8. The airfoil is suspended in a rig, which itself is



Figure 5.4: Comparison of the non-dimensionalized horizontal u-velocity measured along the velocity-rakes with wind-tunnel results [14]



Figure 5.5: Verification of Spalart-Allmaras results with CFD results obtained by NASA [13]

Name	type	$c_L$	$\Delta\%$	$c_D$	$\Delta\%$	$c_M$	$\Delta\%$
Kikuchi <i>et al</i> [16]	WT	1.1281	-	NA	-	- 0.3739	-
Ahmed [15]	WT	0.8696	- 22.91	0.01124	-	NA	-
Spalart-Allmaras	CFD	1.1283	+ 0.0	0.01097	-2.4	-0.4197	+12.4
SST $k - \omega$	CFD	1.0976	-2.7	0.01137	+ 1.1	-0.4032	+7.8
Transition SST	CFD	1.1379	+0.9	0.00677	- 39.8	-0.4232	+13.2
RSM $k - \omega$	CFD	1.1006	-2.4	0.01150	+ 2.3	-0.4047	+ 8.2

Table 5.2: Comparison of force and moment coefficients with experimental data

mounted on a platform car. Using a set of levers, the angle of attack and elevation height is controlled. The whole rig is than pushed along the 2km track by a passenger car. The test-section itself is fully enclosed and 515m long. The purpose of the towing method is to test at virtually zero free stream turbulence level, which is not possible to achieve in a wind-tunnel and using a moving-belt system due to its unavoidable vibrations, according to Kohama *et al.* [70]

The computational procedure was left intact, with only a change in velocity to comply with the chord-based Re = 800,000 used in the study by Kikuchi *et al.* A mayor difficulty for validation is that unfortunately, the presented data in the article is very difficult to read, leading, in the case of drag measurements, to unworkable data. Hence, validation could only be performed by means of lift- and moment coefficient. To get an indication for the drag levels, the drag is compared with measured drag data by Ahmed [15], even though this data cannot be fully trusted. An overview of the force predictions is shown in Table 5.2. As can be observed, the simulations match the experimental results by Kikuchi *et al* much better. The Spalart-Allmaras model is spot on in terms of lift coefficient. The deviation in terms of moment coefficient is larger, although the margin of error is acceptable given the read-off errors involved. When comparing the drag coefficients with the drag data by Ahmed, one can see that the results are surprisingly close in the case fully turbulent flow is assumed. The Transition SST model actually under-predicts the drag by a significant margin. Although this drag data cannot be used for validation purposes, as it stems from a different data set, is does inspire confidence that the simulations are approximating the reference experiment well. More-over, it provides an indication that transition turbulence models may not be suitable for analysis of WIG-craft due to the large deviation in drag.

Examining the pressure distribution over the airfoil, also shown in Figure 5.7, one can see that the CFD prediction matches the measured data by Kikuchi reasonably well. Interestingly, the experimental data shows a higher suction peak which is not resembled by the simulations. The pressure along the lower side of the airfoil is on the other hand slightly over-predicted by the simulations. This is also noticed by Takashi *et al*, when comparing the results with a potential flow solver combined with a proprietary boundary layer prediction method. [68] The authors suggest that blockage effects of the test-rig could be the cause for this deviation. Despite of all the unquantified uncertainties surrounding this close ground effect case study, is has been shown that the three main candidate turbulence models are at least able to approximate the flow over an



Figure 5.6: Close-up of the NACA 4412 IGE mesh. The domain is divided into a laminar zone (grey) and a fully turbulent zone (green)



Figure 5.7: Comparison of numerical results for pressure distribution Re = 800,000 with experimental data [15][16]



airfoil in extreme ground effect.

## **5.4.** WING WITH END-PLATES

The previous validation studies were all focussed on the two-dimensional behaviour and validation of the numerical models. However, since for the AeroCity, two-dimensional flow will not be of much interest for the purpose of assessing the aerodynamic performance, a three-dimensional validation study should be conducted. Even though the amount of wind-tunnel experiments regarding WIG-craft is very limited, two studies are available for validation. [23] [17]. Since the study by Fink and Lastinger involves a geometry very similar to the AeroCity and has been successfully used for validation purposes in previous CFD studies, their study will be used for further analysis. [27] [8]

The purpose of the experiment by Fink and Lastinger was to evaluate the influence of aspect ratio in the aerodynamics characteristics of a wing in close ground proximity. As such, a series of wings with aspect ratios varying from AR = 1 to AR = 6 were investigated in the wind-tunnel of the Langley Research Center. Part of this investigation was to apply an end-plate to the lowest aspect ratio wing and measure the effectiveness. All testing was conducted at Re = 490,000 and a image-wing was used to apply simulate the ground effect. The airfoil selected for the wings was a Glenn Martin 21 airfoil, which is a moderate to high cambered airfoil of 22% thickness. In order to prevent the Venturi effect from occurring, the lower aft 30% of the airfoil had been flattened. Unfortunately, the geometry of the end-plates is not described in detail. Instead, the geometry of the end-plates was guessed by assuming a wedge-like shape with a rounded leading edge and finite thickness trailing edge. The inner side of the end-plates was kept straight in order not to accelerate or decelerate the flow and the upper-side was blended with the airfoil. The additional area was accounted for, as the end plates are an extension of the plain wing.

For the mesh, an unstructured tetrahedral dominated mesh was selected. Although a multi-block structured mesh was preferred, it was concluded to be too difficult to construct due to regions of curvature in all three dimensions. Since a degree of flow separation and regions of high vorticity are expected, an unstructured mesh may provide similar accuracy as alignment of the mesh with the flow will not be possible in critical areas. The downside is that in terms of memory allocation, an unstructured grid is much more demanding. An initial mesh size of 3.4 million elements was selected for initial analysis. A prism layer consisting of 15 layers was added such that the  $y^+$  of the wall adjacent cells was  $y^+ \approx 1$ . The same numerical models as used for the 2D ground-effect case have been applied here as well. The initial turbulence that was selected is the SST  $k-\omega$  turbulence model, as it consistently performed well in the previous validation cases compared to alternative models. The Spalart-Allmaras remains a strong candidate as well, as it was developed with wall-bounded flow in mind it may perform less in a three-dimensional scenario with considerable vortices.

Even though the fact that the mesh was not optimized, the initial results were disappointing. The deviation in



Figure 5.9: Photograph of the wind-tunnel setup featuring the wing with half-airfoil end-plates [17]

Table 5.3: Specifications of the aerodynamic model [17]

Table 5.4: Wind-tunnel blockage correction factors [17]

Property	Value	Dimension	Contribution	$\Delta u$	
Span	1219	mm	Model and image wing	0.0040	$U_{\infty}$
Chord	610	mm	Wake blockage	0.0010	$U_{\infty}$
Aspect ratio	2.0	-	Struts and fairings	0.0015	$U_{\infty}$
Main airfoil <i>t/c</i>	0.12	-			
Side-plate <i>t</i> / <i>c</i>	0.06	-			
Reference area	772953	mm <sup>2</sup>	Total	0.0065	$U_{\infty}$

lift coefficient alone consisted of about 12% compared to the experimental data. It is believed that the main cause for the deviation is confusion about the actual reference area. Hidden in a footnote, Fink and Lastinger state that the reference area in the case of end-plates installed is adjusted to correct for any additional area. Since the geometry is not accurately described, nor the procedure of the correction of the reference area, it remains guesswork about the exact nature of the correction. Nevertheless, an adjustment of the reference area based on the actual CAD model did not improve the results sufficiently. Carefully examining the validation studies based on the Fink and Lastinger experiment reveals that strangely enough the case without end-plates has been used.[27] [8] Hence, it appears that case with end-plates is not fully reliable for validation. Since a solid basis of trust in the numerical models is required, and the case without end-plates is less relevant, it has been chosen to refer to the second wind-tunnel experiment [17].

The experiment by Kumar [17] was aimed to investigate the general aerodynamic and stability characteristics of a wing in close ground proximity. In terms of setup is was similar to the experiment described previously, except that a different airfoil and aspect ratio (AR = 2.0) were used in this case. Testing was conducted in the low-turbulence wind-tunnel of the Cranfield University. The test-section measures 2.4m by 1.6m and is a closed section. The flow speed was set equal to 100ft/s, which equates to a chord-based Reynold number of  $Re \approx 1.3$  million. A photograph of the test-setup is shown in Figure 5.9. Shown is the wing and the image model, suspended on struts. Note that the gap, which was reported to be 0.5mm is negligible compared to the model itself measuring  $4 \times 2$  ft. Since it was assumed that the amount of leak flow through the gap is negligible, the gap height was ignored for the numerical model. The airfoil section used in the experiment is a Clark Y section, with a thickness-to-chord ratio of t/c = 11.7%. Even though this section is thinner than the AeroCity airfoil (t/c = 15%), the shape and Reynolds number are actually quite similar. The lowest elevation height  $h/c \approx 0.084$  and  $\alpha = 2^{\circ}$  have been selected for further analysis. Note that the elevation height is measured with respect to the quarter-chord point in this case.

Unlike the study by Fink and Lastinger [23], the geometry and the reference area are well described. An

_	# cells	$C_L$	$\Delta\%$	$C_D$	$\Delta\%$	Model	$C_L$	$\Delta\%$	$C_D$	$\Delta\%$
	4.7 <i>m</i>	0.5937	+ 6.59	0.00922	- 16.94	SST	0.5783	+ 3.82	0.00993	-10.54
	13.8 <i>m</i>	0.5783	+3.82	0.00990	- 10.86	SA	0.5808	+4.29	0.01020	-8.09
	18.6 <i>m</i>	0.5767	+3.53	0.00989	-10.80	RSM	0.5658	+1.59	0.00933	- 15.92
_	27.5 <i>m</i>	0.5764	+3.48	0.00993	- 10.54					
1										

Table 5.5: Mesh dependency study for three different mesh sizes. Table 5.6: Comparison of CFD predictions with experimental data Data compared with experimental data by Kumar [17]

by Kumar [17] using different turbulence models.

overview of the model parameters is presented in Table 5.3. For the geometry of the end-plates, two different designs where tested. One where the side-plate cross-section is a full Clark Y profile and a design where the only the outer contour is given by a Clark Y profile, while the inner section is straight. The latter design has been adopted in this case. The side-plates are half the thickness of the main element and blended with the main wing. The reference area for this particular configuration was found to be 8.32 ft<sup>2</sup> or 0.77 m<sup>2</sup>. To account for any blockage effects, the correction factors for the model solid and wake blockage, together with the blockage by the struts, have been determined. However, as can be observed from Table 5.4, the total blockage correction factor is less than  $< 1\% U_{\infty}$ . Therefore the effect of blockage was neglected in the experiment [17].

The cross-section of the computational domain was kept identical to the test-section of the wind-tunnel. Due to the relative computational cost of unstructured grids, the domain was extended only 4c upstream and 8c downstream. Since the wake is expected to be limited and the blockage of the model has been shown to be negligible, this limitation should not influence the results in a significant matter. Instead, it has been opted to study the effect of the mesh size on the force predictions. By subsequent mesh refinement in close proximity of the wing, the number of cells was gradually increased. An overview of the results is shown in Table 5.5. As can be seen, the predictions by the CFD model over-estimate the production of the lift force and under-estimate the drag. Utilizing the most coarse grid, the deviation of the numerical results with the experimental data is too large, since a deviation of the drag coefficient of less than  $\Delta C_D \leq 10\%$  is strived for. Refinement of the mesh yields to much more acceptable levels of the deviation with the wind-tunnel measurements, even though the deviation is still just outside the bounds, set in Section 3.2.1. However, further refinement of the mesh does not appear to significantly improve the results. The gain of doubling the total cell count, is less than  $\Delta C_D < 0.5\%$ . Since the average time per iteration increases exponentially with the total mesh size, a trade-off should be made. Comparison of the data in Table 5.5, shows that a mesh consisting of about 13.8 million cells provides sufficient accuracy, given the required computational load. Although the current analysis involves a single simulation, meaning that the computational time is of less importance, the balance between refinement of the mesh for the increased accuracy at the cost of computational cost, will become more important for the next phase of the project. For the current simulation, the most refined mesh was selected in the end, disregarding the increased computational cost.

In order to investigate if the solution could be improved, the two other candidate turbulence models were used to compute the flow field around the wing. The results are shown in Table 5.6. The Spalart-Allmaras model appears to be more able to accurately predict the drag. However, in terms of lift coefficient the accurate is less compared to the default SST  $k - \omega$  model. In contrast, the RSM  $k - \omega$  model is very close in terms of lift predictions, but fails to compute the drag properly. The latter actually holds true for all three models, as the difference in force coefficient is in all cases too large for quantitative analysis. The cause, for the overprediction of the aerodynamic efficiency by the numerical model, compared to the wind tunnel observations, is unclear. Possibly, the interaction of the flow between the struts and the model caused the some degree of separation in the vicinity of the mounts. Although not reported by Kumar, this could explain the lower drag and higher lift values found by the CFD simulation, as the struts where absent in the numerical model.

Regardless of the small difference in force coefficients, it is interesting to investigate some of the physical quantities. Shown in Figure 5.10 are the static pressure contours over the wing. Note that the pressure levels indicated are not absolute, but relative to the ambient pressure. Looking at the contour levels at the vertical symmetry plane, one can see the typical pressure contours, also found in 2D analysis of airfoils. The stagnation region at the leading edge, accompanied with a zone of higher pressure in front of the wing, is clearly visible. Towards the wing-tips, the regions of lower pressure reduce in size as the side-plates are of lower t/c.



Figure 5.10: Contours of static pressure over the Clark Y wing using the  $k-\omega$  SST model



Figure 5.11: Contours of skin friction coefficient over the Clark Y wing using the  $k - \omega$  SST model



Figure 5.12: Streamlines around the Clark Y wing, coloured with the local velocity of the flow, using the  $k - \omega$  SST model

As a consequence, the suction levels on the side-plates are lower compared to the main element. If one looks closely, a small region of low static pressure can be seen to exist at the upper portion of the side-plate leading edge. This is explained by the fact that the flow approaches the stagnation zone on the wing, but instead deflects around the edge of the wing. With strong local curvature, the flow is accelerated quickly causing a strong, very local, zone of low static pressure. Similarly, a stagnation location at the inner side of the end-plate can be spotted, as flow trying to escape around the wing-tip is trapped by the end-plates.

To investigate if the flow separates from the body, one could plot the streamlines over the body to reveal a departure of the flow from the body. However, one could also examine the skin friction coefficient or wall shear stress contours to identify separation zones. The latter is shown in Figure 5.11. As expected, the wall shear stress is the highest in the regions with a strong curvature, for example at the leading edge just above or below the stagnation line. The location where the the leading edge of the side plate and the main wing join, is marked by a small zone of very high shear stresses. This is again explained by the flow of air deflecting around the wing, rather than following the contour of the main airfoil. Directly below this small zone, the wall shear stress can actually be seen to very low, approaching zero. This is an indication of flow separation. As the side-plates have a contour that consists of an airfoil with a straight lower half, the leading edge of the plates contains a sharp edge. Air flowing around the wing will face the side-plate at a non-zero angle of attack and separate from the body as a consequence. After a short distance, the flow can be seen to re-attach to the surface, indicated by a non-zero wall shear stress. Another zone of flow separation can be identified near the trailing edge, where the wall shear stress is reduced to zero. Although the angle of attack ( $\alpha = 4^{\circ}$ ) is small, the flow faces an additional adverse pressure gradient due to ground effect. Interestingly, the flow does not appear to separate over the entire trailing edge. Near the blending region of the main airfoil and the side-plate, a narrow stretch, extending up to the trailing edge, exists where the wall shear stress is non-zero.

This could be explained examining the streamlines, shown in Figure 5.12, near the blending region between the side-plate and the main wing. Air is flowing over the edge and accelerated, as it follows the strong convex shape of the blending region. As such, the boundary layer locally experiences a less steep adverse pressure gradient and it is able to remain attached to the surface for a longer period of time. Near the aft portion of the wing, the curvature of the blending region becomes to large for the matured boundary layer. As a consequence, the flow separates right at the blending region. Although the amount flow separation is relatively small, it shows that careful design, by means of smaller curvatures near the aft portion of the wing, may attribute to lowering the overall drag. Since the blending region for the wind tunnel model was not described by Kumar [17], the aerodynamic phenomenon described above will have been of different severity in the actual experiment. Next to the inclusion of support struts, this could explain the difference in force coefficients.

# **5.5.** CONCLUSIONS FROM THE TEST-CASES

Reviewing the above simulation cases, it becomes apparent that the precise prediction of the drag force remains very difficult with CFD simulation. This appears to originate from the limitation of turbulence models, to model flow separation in a correct manner. In the case of the NACA 4412 (Section 5.2), very significant flow separation was involved. Although some turbulence models, such as the Spalart-Allmaras and SST models, were able to model the u-velocity of the separated flow with relative good accuracy. However, in terms of the vertical v-velocity component, none of the tested models proved to be proficient. As a result, drag predictions were of by almost 40% or worse. Although this is considered an extreme case, in general, a drag deviation of approximately  $\Delta D \approx 10\%$  should be accounted for. In terms of lift coefficient, the results are better. However, still a deviation of a few percent should be reckoned with.

Although this may appear to be sobering conclusion, one should remember that the tolerances of the actual wind-tunnels experiment could sometimes be of the same order of magnitude. As witnessed with the closeground effect studies, discrepancies in the experimental data or set-up do exist as well. More-over, it can be hard to fully replicate the experimental set-up, as not all data is always provided. For example, the exact model geometry was not described sufficiently in the three-dimensional WIG effect study by Fink and Lastinger [23]. The uncertainty introduced into the simulation by these knowledge gaps, introduce uncertainties in the final results as well. Hence, it is crucial that as much relevant data of the experiment is available in order to distinguish model defects from experimental uncertainties. In the case of this project, this limitation is hopefully minimized, as much of the experimental raw data is available for comparison.
# 6

# **SETUP AND PRE-SIMULATION**

Having completed the various CFD test-cases for selecting the most suitable turbulence model, the actual analysis phase of AeroCity is commenced. This chapter is divided into several parts. First, the set-up of the wind-tunnel experiment is discussed in more detail. Next, the construction of the mesh and pre-simulation steps are discussed in more detail. Finally, some initial results are presented that are used to make adjustments for the final simulations.

### **6.1.** EXPERIMENTAL SETUP

The wind-tunnel experiment that is replicated for the purpose of model validation is the experiment by Nasrollahi [2]. A general description of the experiment is given in Section 2.2.2. Only a single configuration will be used to validate the model, namely the configuration with  $\alpha_{body} = 3^\circ$ , h/c = 0.05 and a gap height  $h_g = 7$ mm. The choice for this configuration is motivated by the fact that the additional boundary layer measurements performed in the experiment are limited to this configuration. The boundary layer data could be very useful to explain any discrepancies between the model and the experiment, in the case these would arise. Note that the gap height is a dimensional quantity, in contrast to the angle of attack and height to chord ratio. For ease of analysis, the gap height from now on will be non-dimensionalized with elevation height. For the current configuration, this yields to  $h_g/h = 0.14$ .

The model was suspended on the roof of the test-section and mounted directly on the balance scale by means of four bolts. A photo of the model installed in the test-section is shown in figure 6.1. Since the bolts are not expected to have a significant impact on the flow around AeroCity, they are not incorporated in the CFD model. The force measurements conducted during the experiment are obtained directly from the balance readings. This implies that no correction factors to account for model or wake blockage have been applied, which is actually confirmed in the report of the experiment [2]. Although disregarding the correction factors reduces the credibility of the experimental findings, it makes a direct comparison between the experimental and numerical data a more straight forward. Flow speeds were varied between 10m/s to 100m/s, which is near the upper limit of the LTT wind-tunnel. These velocities translate to a Reynolds number ranging from Re = 0.7 to Re = 6.8 million. Note that for the highest flow velocities, flow compressibility becomes important. This is however not accounted for, by assuming incompressible flow only. During the CFD simulation, the inlet velocity will be limited to 80 m/s in order to justify the assumption of incompressible flow.

During the wind-tunnel experiment, force and pressure measurements were conducted with both an undisturbed and tripped boundary layer. In the latter case, transition was invoked at x/c = 0.05 by means of a layer of dual-sided tape. Due to presence of the ground boundary layer, it will not be possible to divide the computational domain into a laminar and turbulent zone, hence a full turbulent domain is adopted. Although several transition models are available in Fluent, it was shown (see Section 5.3) that these models appear to be less accurate for the prediction of extreme ground effect flows. Since it was found in Section 2.2.2 that the difference in terms of force measurements is actually very small, this decision should not affect the accuracy of the model significantly. The turbulence model that was found to be most consistent and accurate during the previous investigations of the ground effect (Chapter 5) is the  $k - \omega$  SST model. Turbulence boundary



Figure 6.1: Photograph of the aerodynamic model installed in the wind-tunnel test-section [2]

conditions were set to resemble the turbulent conditions of the LTT, namely with a turbulence intensity of I = 0.10%. It should be noted that the actual LTT turbulence intensity is lower, depending on the flow velocity (I = 0.015% at 20 m/s). However, as the model is not subjected to the undisturbed free stream but rather partially submerged in the wall bounded flow, the slightly higher value for I has been adopted. To complement the turbulence intensity, an eddy viscosity ratio of 3 has been selected. [71]

Since the model is not equipped with any pressure ports, measurements of the local pressure are obtained with a Pitot-static tube, rather than manometer of the LTT wind-tunnel. The Pitot-tube was connected to a digital pressure gauge. By mounting the Pitot-tube on the model upper surface, the pressure distribution over the upper surface was investigated. Since the Pitot tube was elevated above the surface, the height from the surface varied slightly depending on the surface curvature. According to Nasrollahi, the height of the Pitot-tube above the surface varied between 4mm to 10mm. A consequence of this manual approach is, that measurements at the pressure side of the vehicle, are very difficult. Therefore no pressure measurement have been conducted at the pressure side. The same applies for the side plate, of which no static pressure data is available. This is a limitation for the validation process, as the pressure side is of most importance when designing a WIG vehicle.

In terms of ambient conditions, the data is well recorded. An overview of the ambient conditions is presented in Table 6.1. It should be noted that these conditions apply to the measurements conducted with a fixed location for boundary layer transition only. For the natural transition experiment, conducted the same day, the temperature was lower at 13.9°*C*. Whether or not the temperature continued to increase during the experiment is unclear. Nevertheless, it assumed that any temperature variation within the time-frame of experiment can be safely neglected. The reference pressure  $P_{\infty}$  is set as reference pressure during the experiment. No gauge pressure is applied. The density and dynamic viscosity are entered as constant values, instead of obtaining these values from the ideal gas law. Since heat losses are expected to be of minor influence, this simplification should be justifiable, as it removes the need to compute an additional PDE, namely the energy equation. Instead, the change in temperature is computed by means of isentropic relations, as the density is assumed to remain constant.

Table 6.1: Reference values for ambient conditions, as recorded during the wind-tunnel experiment [2]

Property	Symbol	Value	unit
Pressure	$P_{\infty}$	102190	Pa
Temperature	$T_{\infty}$	289.4	Κ
Density	$ ho_\infty$	1.230	kg/m <sup>3</sup>
Kinematic viscosity	ν	$1.46 \cdot 10^{-5}$	$m/s^{-2}$
Dynamic viscosity	$\mu$	$1.79 \cdot 10^{-5}$	$m/s^{-2}$



Figure 6.2: Schematic drawing of the Low Turbulence Tunnel (LTT) facility of the TU Delft [18]

### **6.2.** WALL BOUNDARY LAYER

Since it is suspected that influence of the wind-tunnel wall boundary layer played an important role in the experiment, it can prove useful to investigate the characteristics of this boundary layer in more detail. If one examines the design of the LTT tunnel, shown in Figure 6.2, one can identify the settling chamber and test-sectioned denoted by **A** and **E** respectively. After the bend, the flow passes through a series of flow straighteners, often a series of hexagonal meshes, a fresh boundary layer will start to develop. Since the settling chamber contracts, the boundary layer will be subjected to a favourable pressure gradient. As such, the boundary layer growth will be less substantial than in the case of a flat plate. Measuring the distance between the last flow straightener and beginning of the test-section, the length is approximately 7.40 meters. Since the model is placed in the center of the test-section, the total boundary layer length near the model will approximately be equal to 8.2 meters. A model length of 1.00m has been assumed in this case.

Fortunately, the boundary layer thickness in the test-section of the LTT has been measured in a previous experimental campaign conducted in the LTT (courtesy of W.A. Timmer). Shown in Figure 6.3 are the nondimensionalized velocity profiles of the upper wall boundary layer at various positions in the test-section. Note that the reference position x = 0 is taken at the beginning of the test-section. The free-stream velocity of the flow was kept equal at V = 77.5m/s during the measurements. As can be observed from the figure, the displacement thickness  $\delta_{99}$  is already quite thick right after the entrance of the test-section and continues to grow steadily. Plotted in Figure 6.4 are the actual displacement thickness's, as measured at the various locations. According to the figure, the displacement thickness at the leading edge of the model should be around 26mm. Using a curve-fit, one can approximate the location where the displacement thickness is equal to zero. This location will be referred to as the 'virtual origin' of the boundary layer. According to this regression curve, the virtual origin would be positioned approximately 2.5 m upstream of the test-section.

However, this position is located inside the settling chamber, upstream of the contraction region. Since the boundary layer experiences a favourably pressure gradient, the distance cannot be directly translated into a flat plate length. The latter is important, since the wind-tunnel wall will be modelled by a flat wall. In order to predict the corresponding flat plate boundary layer length, an approximation can be obtained by using a Blasius [72] solution for a flat plate. The formulation for laminar and turbulent flow are given by:

$$\delta_{lam} = \frac{4.91x}{\sqrt{Re_x}}, \qquad \delta_{tur} = \frac{0.37x}{\sqrt[5]{Re_x}}$$
(6.1)

Since the computational domain is fully turbulent, only the turbulent formulation will be of interest in this case. Hence, the relatively small laminar part is neglected. These assumptions only hold true for flat plate, without any pressure gradient and constant velocity. Using these approximations, one would arrive at a plate



length in the order of 1.2m. This is considerably less than the prediction based on the LTT data, which suggests an upstream distance of 2.5m, measured from the start of the test-section.

In order to investigate whether the estimation, that the equivalent turbulent flat plate length is approximately equal to L = 1.20m, is valid, the measured boundary layer profile is compared with an analytical solution. This is indicated by the dotted line in Figure 6.3. The plotted profile computed with the so-called power-law, where the power  $n \approx 7$  [61]. Note that the actual velocity profile does not coincide with the measured velocity profile. Since the power-law method assumes a constant velocity profile, regardless of the local length-based Reynolds number  $Re_x$ , this is inevitable. Although it is possible to derive an analytical velocity profile that accounts for a varying shape factor, such as the entrainment method proposed by Head [73], the implementation is much more cumbersome. Since the equivalent turbulent flat plate is of main interest in this case, a confirmation of the approximate displacement thickness is deemed sufficient. Comparing the locations where the velocity ratio u/U is close to unity, one can observe that the boundary layer thickness's are indeed approximately equal. Hence, it can be concluded that the approximated equivalent turbulent flat plate length of L = 1.20 is a good initial guess.

#### **6.3.** MESH CONSTRUCTION

The computational domain is constructed to resemble the LTT wind-tunnel test-section. The width and height are identical to the wind-tunnel dimensions. However, a compromise is made in terms of the cross-section. The wind-tunnel cross-section, at the test-section, is octagonal, while the computational domain is rectangular. It is assumed that this decision will not affect the flow around AeroCity significantly. Whether this assumption is justified will need to be confirmed at a later stage. In order to reduce the computational cost, the domain is split at the XZ-plane. By means of a symmetry boundary condition, only one half of the model is simulated.

The up- and downstream boundaries of the computational domain are located 5c and 8c downstream of the model respectively. An overview of the computational domain is shown in Figure 6.5. The downstream boundary is modelled by means of a pressure-outlet with zero back-pressure. The boundary should be located far-enough downstream in order not to influence the flow field upstream and allow the wake structure to fully develop. In case of the upstream boundary, it is assumed that a distance of 5c is sufficient from preventing the model pressure field to alter the inlet conditions and vice versa. It should be noted that a larger distance between the model and the inlet plane would be preferred. However, for the sake of computational expense, the domain has been confined to these limits. Nevertheless, it is assumed that the influence the boundaries of the domain are extended sufficiently far away, to neglect their influence on the results. The inlet itself is modelled by a velocity inlet with zero gauge pressure. A uniform velocity profile is specified. The boundary layer length of the ground wall is controlled by splitting the ground surface in a slip and no-slip zone.



Figure 6.5: Overview of the computational domain

In order to minimize the computational load, the fluid volume has been divided into three separate zones. This allows the cell size in the vicinity of the model and the wake to be refined further, while the surrounding flow is captured in less detail. Again, the size of these refined volumes needs to be restricted, as the total number of cells grows very rapidly when the local cell size is reduced. The volume directly surrounding the AeroCity model, referred to as the body of influence, needs the highest level of refinement. The local cell size should be sufficiently small to capture all relevant flow phenomena occurring around the body, such as the formation of vortices and local separation of flow. An image of the generated mesh is shown in Figure 6.6. As an example, one can observe a refinement of the mesh on parts with a strong or complex curvature, such as the joint between the leading edge of the wing and side-plate. The latter can be more clearly seen in Figure 6.7. Similarly, near the trailing edge (Figure 6.8), the local mesh is refined to capture the potential separation of flow accurately. As the wake directly behind the trailing edge is also modelled inside the body of influence, the near to intermediate wake can be captured by a slightly less refined mesh. Since the typical wake structures, such as vortices, grow rapidly in size downstream of the body, the cell size can be increased accordingly. The rest of the flow domain, such as the upstream and wake in the far-downstream, is modelled by a coarser mesh. This is acceptable, as the gradients of the flow properties are relatively small in these areas.

#### **6.3.1.** BOUNDARY LAYER MESH

In order to allow the turbulence models to capture the boundary layers, on both the AeroCity model and the ground plane, with sufficient accuracy, prism layers have been applied. The first important aspect of the prism layers is that the first cell height is small enough such that  $Y^+ \approx 1$ . Although one can usually only check if the mesh is refined enough after the simulation has been completed, one can make a reasonably guess by looking at the wall shear stress of a flat plate [74]. Secondly, the total number of layers within the boundary layer should be sufficient to capture the flow properly. This has been explained in Section 4.4.1 in more detail. Although the boundary layer characteristics are different on every location on AeroCity, a uniform prism layer mesh has been applied. Ideally, one would construct a varying prism layer mesh to capture the boundary layer in the most ideal way, regardless of the location. However, this is very demanding, as a smooth transition of the layers is required at all locations.

The region of the gap between the ground and the end-plate should be given special care. Since the vertical height available (7mm) is limited, there is not enough space for prism layers over the AeroCity model and the ground-plane to co-exist. By locally reducing the number of layers by means of stair-stepping and compression of the layers, intersection of the prism layers can be avoided. Since the velocity gradient of the local flow is expected to be large, due to the significant pressure difference on either side, one needs to make sure that the quality of the cells involved is optimized. This can be achieved by a series of smoothing operations, both locally and globally. To limit the number of prism layers required, a global prism height ratio of r = 1.30 has been selected. Admittedly, this ratio is quite large since a ratio of r = 1.10 to r = 1.05 are even used. However, selection of this relatively large growth ratio enables the use of fewer prism layers, reducing the total number of cells in the mesh considerably. The increased numerical error is the cost that is paid for the gain in computational efficiency.



Figure 6.6: Capture of the surface mesh of AeroCity



Figure 6.7: Close-up of the surface mesh near the leading edge of AeroCity



Figure 6.8: Close-up of the surface mesh near the trailing edge of AeroCity



Figure 6.9: Wall Yplus contours of the AeroCity model and the near-ground plane at V = 40m/s

As mentioned in Section 4.4.1, the yplus value needs to be  $Y^+ \ge 300$  to  $Y^+ \ge 500$  in order to resolve the lower part of the velocity defect region of the boundary layer sufficiently. As such, one should take care that the faces of the last prism layer indeed confirm to  $Y^+ \ge 500$ . In this case, this results into a prism layer thickness of h > 5mm. Note that the  $Y^+$  values have been obtained from an calculation tool and are therefore only estimates of the actual  $Y^+$  value. Post-processing of the simulation data will have to reveal whether the boundary layer is indeed resolved with sufficient accuracy. Unfortunately, the Fluent software does not compute the yplus value for interior cells. Hence, a direct analysis of the yplus value is not possible.

Equal care should be taken to resolve the viscous sub-layer with sufficient accuracy. For a low Reynolds turbulence model, this requires the wall adjacent cells to conform to  $Y^+ \approx 1$ . Shown in Figure 6.9 are the  $Y^+$  contours over the body of the AeroCity model. As can be observed, the actual yplus values are varying quite a bit due to the differences in local velocity. In this case it can be seen that over a large portion of the upper surface  $Y^+ \approx 1$ . However, at the side plate, the contours show  $Y^+ \leq 0.50$  due to the locally reduced velocity of the flow. Although very small values of yplus should also be avoided, it is inevitable in this case. In contrast, higher values of yplus can be observed on the leading edge of the side-plate. Due to the higher than free-stream velocity, the yplus requirement is more stringent in this particular area. Still, with the maximum  $Y^+ < 2.00$  the viscous sub-layer of the boundary layers should be well resolved.

# **6.4.** CONVERGENCE

Another concern one has to deal with when performing CFD calculations, is the convergence criterion. To assure that the solution found is indeed the steady-state solution of the flow problem, a set of objective convergence criteria need to be set. A common approach is to define a minimum threshold for the residuals  $r_i$  of the PDE's, for example  $r_i \leq 1E - 4$ . However, this can lead to pre-mature acceptance of the solution as one should focus on the convergence rate of the force coefficients instead. The residuals of the PDE's, at least in Fluent, are the scaled residuals. This means that they are directly dependent on the relaxation factors applied. In the case that considerable relaxation is applied, this can lead to a false indication of the convergence of the solution. To avoid this, the force coefficients are taken as a convergence indicator instead. If the lift and drag no longer change after a period of time, one can safely assume that the solution has converged. The criteria for convergence that have been adopted for this project are the following:

- Change in lift coefficient after 1000 iterations  $\Delta c_L \leq 1E 4$
- Change in drag coefficient after 500 iterations  $\Delta c_D \le 1E 5$



Figure 6.12: Dependency of force coefficients on total mesh size

Figure 6.13: Average time per iteration for various mesh sizes

The values of the residuals in this case are an indication of the credibility of the solution. If the residuals are still relatively large but no longer decreasing, than the numerical errors involved could have prevented solution from further convergence. On the other hand, if the residuals are sufficiently small and still decreasing after convergence of the force coefficients, then one be can certain that solution has reached a steady-state solution. The threshold for the residuals that has been adopted in this case is  $r_i \leq 5E - 5$ . Typically, the above convergence criteria were met after approximately 5000 iterations, depending on the relaxation factor for the higher-order terms. An example of the convergence of the lift and drag coefficient is shown in Figure 6.10 and 6.11. As can be observed, both coefficients are already within 10% of their respective steady-state values after approximately a 1000 iterations. In general, the lift coefficient converges after about 3000 iterations, where as the drag coefficient takes significantly longer before it reaches a true steady-state value. Hence, the convergence criterion for the drag coefficient is, for these cases, the critical factor to accept convergence of the solution.

#### **6.5.** Mesh dependency of results

Since the amount of computational power is constrained by the hardware available, one should strive to optimize the balance between precision and computation time. In order to investigate which mesh size should be selected and where local refinements are required, a mesh dependency study has been conducted. In terms of mesh refinement, one has multiple options where to refine the mesh. After a few mesh variations, it was found that the solution is most sensitive to changes of the mesh directly surrounding the model, the socalled body of influence. Variations in mesh size of the AeroCity surface mesh did show some deterioration of the results when coarsening the mesh, however the impact on the total mesh size was small. As such, it was decided to have a fairly refined surface mesh on the AeroCity body. With the surface mesh kept constant, a sweep of simulations was performed using different cell volumes for the body of influence.

The results of the mesh dependency study are shown in Figure 6.12. Shown is the deviation of lift and drag coefficient with the experimental data for various mesh sizes. As can be seen, the results are converging to a fixed deviation as the mesh size is increased. Clearly, the smallest mesh size of about 3.5 million cells is limiting the solution due to insufficient refinement of the mesh. However, when the mesh size is raised above approximately 10 million cells, the gain in accuracy with further mesh refinement becomes marginal. The difference in deviation of drag coefficient between approximately 10 and 15 million cells is about 0.50%. The latter mesh size approached the upper limit of mesh size, since the unstructured mesh consumes a lot of computational memory. As such, extending the trend-line to larger mesh sizes was not practically possible.

Aside from memory allocation, there is another practical consideration that one must make when picking a suitable mesh size. As the mesh increases in size, the computation time increases rapidly. this is shown in Figure 6.13, where the average time per iterations is plotted versus the number of cells. As can be observed, the average time per iteration grows exponentially with the number of cells. Considering that the solution is seen to converge after approximately 6,000 iterations, the difference in computation time becomes significant. The mesh of 10 million cells will take about 23 hours to converge, while the mesh containing 15 million cells will be seen to converge after almost 42 hours. Considering that the aim of the project is to use the CFD simulations as analysis and design tool, meaning that any changes to the design or operational conditions should be assessed at a regular basis, the mesh of 10 million cells is preferred. The trade-off between accuracy and computational cost is balanced best in this case.

7

# **SIMULATION RESULTS**

In this chapter the results of the CFD simulations performed on the aerodynamic model of AeroCity will be presented. In the first and most elaborate part, the results from the replicated wind-tunnel experiment [2] will be presented. After a thorough analysis of the aerodynamics of the AeroCity, the ground boundary condition is altered to a moving ground boundary condition to asses the influence of the boundary condition on the aerodynamic performance of the AeroCity model. Finally, a track wall is added to the model to determine what influence the addition of a side wall has on the characteristics of the flow around the model.

# **7.1.** FORCE MEASUREMENTS

Before assessing the flow characteristics of the AeroCity model, it is first examined how the performance indicators of the simulated model compare with the experimental data. Both the lift and drag coefficients are of prime interest here, as together these determine the lift-over-drag of the AeroCity. A high lift to drag ratio is required, in order accomplish the ambition of an energy efficient transportation system. Although the energy efficiency of the system as a whole will also depend on other factors, such as the propulsion system, the L/Dratio will play a crucial role.

The results for the lift and drag coefficient are shown in the Figure 7.1. Starting with the lift coefficient, one can observe that the lift predictions agree well with the experimental data at the lower flow velocity range. However, when the flow velocity is increased, the two data-sets start to deviate. The numerical data in this case over-predicts the lift. The growing deviation of experiment and simulation data is explained by the opposite trend with flow velocity. During the wind-tunnel experiment, it was found that the lift coefficient slowly decreases with increasing flow velocities. The simulations however shows an opposite trend, namely a small positive trend with the free stream flow speed. The latter is more in line with expectations, namely a modest positive trend with Reynolds number *Re*.

Turning the attention to the drag coefficient, a similar observation can be made. That is, a reasonably good agreement of the drag prediction at lower flow velocities, but which increasingly deviates from the experimental values, when the velocity of the free-stream is increased. Where-as the lift coefficient was approximately invariant of the free-stream velocity of the flow, the simulations actually predict a decrease of the drag coefficient. Although it is promising that the CFD simulations agree well with the experimental force measurements in the lower part of the flow velocity domain, it is the aerodynamic performance at higher flow speeds which is most relevant for the AeroCity. With cruise-speeds varying between approximately 40 m/s to 80 m/s, the accuracy of the drag prediction by the CFD model weighs more heavily. Especially, considering that the required power scales with the cubed power of the velocity.

In order to examine the relative deviation, the deviation of the numerical data with the obtained wind-tunnel measurements are show in the upper part of Figure 7.3. The deviations are expressed in terms of percentages. Again, it can be observed that the agreement at the lowest part of the velocity range is quite good. However, after increasing the flow velocity beyond 20 m/s, the curves of the relative deviation depart from each-other. Approaching 80 m/s, the deviation amounts to +6% and -16% for the lift and drag coefficient



Figure 7.1: Comparison of lift and drag coefficients with experimental data [2]





Figure 7.3: Relative deviation with experimental data [2] and graphical location of the centre of pressure

Figure 7.4: Comparison of  $C_p$  pressure distribution at z/c = 0 with experimental data at 40m/s [2]

respectively. Translated to lift-over-drag ratio, the deviation of the CFD prediction with the experiment is off by  $\Delta L/D = 26\%$ . The magnitude of the aerodynamic efficiency, shown in Figure 7.2 varies between  $L/D \approx 11$  to  $L/D \approx 14$ . This is on pair with the observations made by Nouwens [1].

Also shown in Figure 7.2 is the development of the moment coefficient. Note that moment coefficient is taken at the the quarter-chord point, at the lowest point of the side-plate. The moment coefficient shows only a small variation with flow velocity, increasing in absolute magnitude. Again, this is opposite to the observations made during the wind-tunnel experiment. The difference between the moment coefficients increases up to  $\Delta C_{M,0.25} \approx 10\%$ .

In order to investigate how the pitching moment  $C_{M,0.25}$  is affected by the lift and the drag, the centre of pressure is computed. By convention, the centre of pressure is assumed to be on the chord line of the mean aerodynamic chord. For the AeroCity, at  $\alpha = 3^{\circ}$  and h/c = 0.05, the chord line is suspended about  $y/c \approx 0.10$  of the ground. The chord-wise position of the centre of pressure is found to be  $x_{cop}/c \approx 0.40$ . To illustrate the position of the centre of pressure shifts downstream when increasing the velocity of the flow, the difference in chord-wise location only amounts to a few millimetres. Since the aerodynamic moment induced by the lift force is dominant, and both the lift force and moment arm increase with the flow speed, the magnitude of  $C_{M,0.25}$  increases. The decrease of the drag force cannot offset the lift induced aerodynamic moment.





Figure 7.5: Pressure distribution at various span-wise locations, measured at U = 40m/s

Figure 7.6: Comparison of the pressure distribution at z/c=0.00 and z/c=0.15 for two flow velocities with experimental data [2].

However, during the wind-tunnel experiment, where the lift decreased in magnitude and the drag remained almost constant, the magnitude of  $C_{M,0.25}$  was decreased by the same mechanism.

# **7.2.** PRESSURE DISTRIBUTION

To examine the flow around the AeroCity in more detail, the distribution of the pressure is plotted in Figure 7.4. The obtained pressure data, given by  $C_P$ , is compared with the pressure measurement conducted during the wind-tunnel experiment [2]. As can be observed, the two data-sets show good mutual agreement. At approximately x/c=0.30, the experimental pressure measurement shows a small bump in the pressure distribution. Although this could suggest the presence of a small separation bubble, this was actually not observed by Nasrollahi during the experiment. More likely, the data point is slightly off due to a measurement error. As explained in more detail in Section 6.1, the height variation of the pitot-tube was in the order of several millimetres and dependent on the local curvature of the body.

Taking a closer look at the aft portion of the AeroCity, it can be observed that the  $C_P$  levels during the windtunnel experiment were slightly more negative. Whether this is once again due to measurement errors or due to a differences in boundary layer development will need to investigated later. Since the pressure distribution on the upper surface matches well with the experimental data, the difference in terms of lift coefficient should originate from a difference in pressure at the lower surface. Unfortunately, there is no experimental data available for this surface, to be used for comparison.

Note that the pressure distribution of Figure 7.4 was obtained at the mean aerodynamic chord (z/c = 0.00). Since this is the centre section of the wing, the influence of the tip vortices on the local pressure distribution is limited. To investigate how the pressure distribution is affected by the finite span, the  $C_P$  distribution at two outboard span-wise positions are plotted. This is shown in Figure 7.5, with the pressure distribution of the mean chord as reference. As can be seen, the pressure distribution at half-span (z/c = 0.10) is affected the slightly. Most noticeable is the reduced suction on the front part of the upper surface. Also, the  $C_P$  values near the trailing edge can be seen to be different. The adverse pressure gradient along the chord-line is lower up till  $x/c \approx 0.90$ , after which the pressure increases at a higher rate. Like the pressure distribution at z/c=0.00, the pressure converges to a value of  $C_P \approx 0.30$  at the trailing edge.

However, looking at the pressure distribution at 3/4 semi-span (z/c=0.15), it can be observed that the  $C_P$  distribution is altered significantly. The pressure on the upper surface is less negative, up to  $x/c\approx 0.60$  of the chord. After this point, the  $C_P$  values remain lower compared to inboard wing sections. Again, at around x/c=0.90, the pressure increases suddenly and recovers to  $C_P \approx 0.20$ . The flattening of the upper side portion of the  $C_P$  curve, followed by a sudden increase in pressure could indicate a separation bubble. However, considering that the location is far aft and the adverse pressure gradient is low, this is not very likely. Instead, it is suspected that a tip vortex is responsible for enhanced negative  $C_P$  values at the aft portion of the AeroCity.



Figure 7.7: Pressure contour plot of the upper (left) and lower (right) surface for U = 40 m/s



Figure 7.8: Pressure contour plot of the upper (left) and lower (right) surface for U = 80 m/s

This will be investigated in more detail in Section 4.2.

All of the above pressure distribution comparisons were conducted at a single velocity, namely 40 m/s. However, was as found in the previous section (Section 7.1), the lift and drag coefficients varied with the free stream velocity. Hence, it is interesting to see how the pressure distribution is affected by the free stream velocity. This is shown in Figure 7.6, for z/c= 0.00 and z/c= 0.15. If one examines the figure closely, it can be seen that the the  $C_P$  values on the upper surface are slightly more negative in the case of 80 m/s. Since the velocity of the free-stream is doubled, the boundary layer on the upper surface will be reduced in thickness. As a results, the air will experience a slightly stronger curvature flowing over the airfoil. Note that the pressure distribution on the lower surface is almost identical. The reduced thickness of the vehicle boundary layer has an almost negligible effect, although in theory the boundary layer reduces the flow channel between the ground plane and the vehicle.

Although the slow increase of the lift coefficient with the free-stream velocity can be explained on the basis of the above figures, the steady decrease of the drag coefficient cannot. In order to obtain a better overview of the pressure distribution, a contour plot of  $C_P$  is shown in Figure 7.7 and 7.8 for U = 40 m/s and U = 80 m/s respectively. Examination of these contour plots reveals that an area of very low pressure exists at the leading edge of the side-plate. Due to the large adverse pressure gradient at the lower side of the vehicle, a part of flow will flow around the vehicle. Since the curvature of the side plate' leading edge is very strong, the local velocity of the flow will be significantly higher than the free-stream. Judging from the zone of low pressure directly after the side plate' leading edge, the flow locally separates before it attaches again further downstream. Comparing the two contour plots, it appears that the zone of locally separated flow is slightly larger at U = 40 m/s compared to U = 80 m/s. This could be a clue why the drag coefficient is reducing at higher free stream velocities.

Similarly, it can be seen that around the wing-tip edge, a narrow band of relative low pressure progresses downstream. Approximately at  $x/c \approx 0.50$ , the zone extends partially over the upper surface. As a result, the



Figure 7.9: Isometric views of the streamlines around AeroCity from the front (left) and (right) at U = 80 m/s

pressure levels in the vicinity of the wing-tip edge remain more negative for a prolonged chord-wise distance. This matches the observations made earlier in this section, when it was found that the  $C_P$  distribution at z/c= 0.15 shows a region of lower  $C_P$  values starting from about  $x/c\approx 0.60$ . This appears to be caused by a small vortex that transitions inboard downstream. The local velocity due to the strong swirling motion is larger than in the direct vicinity, explaining the reduced values for  $C_P$ . Further analysis on vorticity will be performed in Section 7.3 and 4.2

# **7.3.** FLOW VISUALISATION

To gain more insight into the actual flow around the AeroCity model, visualization of the flow is a powerful tool. Although several experimental techniques are in existence to visualize the flow, such as PIV and the fluorescent-oil film method, the range of options to visualize a certain flow aspect without introducing any disturbance to flow is limited. Although the flow predictions by CFD are not always as accurate when compared to a wind-tunnel experiment, it can be very useful tool to visualize the flow. To investigate how the mean flow pattern around the AeroCity looks like, the streamlines can be computed. This is shown in Figure 7.9. Starting from the front section of AeroCity, three distinct observations can be made.

First, is can be seen that the local velocity of the flow around the leading edge of the side-plate is indeed significantly higher compared to the global velocity range. As mentioned earlier, this is explained by the shape of the side-plate' leading edge. The shape resembles a semi-cylinder. Using potential flow theory, it can be shown that the local velocity at  $\phi = 90^{\circ}$  is equal to 2*U*. Note that the stagnation point is at  $\phi = 0^{\circ}$  and *U* the free-stream velocity. Unlike a potential flow, the flow behind a cylinder in a high Reynold number viscous flow will separate and form a turbulent wake. Since the stagnation point on the cylinder is located on the inner side of the end-plate, the flow is likely to separate on the outer side of the end-plate. Looking at the streamlines more carefully, it can indeed be observed that the these do not follow the contour of the body.

A second aspect that becomes apparent from Figure 7.9, are the streamlines over the upper surface, closest to the wing edge. Shortly after the leading edge, the flow curves towards the wing tip. This can be explained by zone of low pressure on the side-plate, as shown in Figure 7.7. When the air flows over the wing edge, it is accelerated quickly over a short distance. After the thickest point of the wing has been past, the streamlines curve inboard again and the flow decelerates over the aft portion of the wing. The inboard movement of the flow past the thickest point is clearly visualized in the right of Figure 7.9. Notice how the band of streamlines is narrowed in the aft section of the upper surface.

A third distinct feature of the flow that can be observed from Figure 7.9 is the creation of a relatively large vortex. Due to the large adverse pressure gradient underneath the vehicle, the vortex does not have the typ-



#### Skin Friction Coefficient

Figure 7.10: Comparison of upper surface oil flow pattern between the wind-tunnel (U = 100 m/s) and CFD model (U = 80 m/s)

ical horseshoe shape. Instead it is a single vortex, originating from underneath the side-plate (referred to as 'ground vortex'). Looking at the streamlines just in front of the AeroCity, one can see the strong bending of the streamlines close to the ground plane, towards the wing edge. Initially the flow has a low momentum, as it has been slowed down by the high pressure region underneath the wing, but is quickly accelerated once it 'leaks' to the lower pressure side of the end-plate. The high curling of the flow increases the local vorticity and as a consequence a vortex is created. The vortex can be seen to grow in size as it travels downstream. The vortex roll-up can be clearly seen in the right picture of Figure 7.9. A more elaborate analysis of the vortex structure is presented in Section 4.2.

Aside from the above observations, the streamlines of the flow around AeroCity do not reveal any significant flow defects. The flow appears to follow the contours of the upper surface well, with no signs of significant flow separation to be seen. Also in front of the AeroCity model, there are no signs of a separation bubble on the ground plane. To compare the flow close to the AeroCity surface in more detail, a surface particle track is made and compared with fluorescent oil film photographs, made during the wind-tunnel experiment [2]. The results is shown in Figure 7.10. Do note that the oil film patterns have been obtained at a non-identical free-stream velocity. However, for the upper surface this is not considered to be a problem, as the  $C_P$  distribution (Section 7.2) appeared to be invariant of the free-stream velocity. Although the particle tracks in the numerical result are not continuous, which makes them less clear to identify, it can be seen that the surface flow pattern is almost identical. In both case, flow is curling over from the wing centreline and straighten out near the trailing edge. Outside of this band of streamlines, a zone is distinguished were the streamlines show a clear curl. As will be show in Section 4.2, this is caused by a region of negative lateral velocity.

Next to the striking similarity between the experimental and numerical result, one can spot a small zone of disturbances at the trailing edge in the wind-tunnel model. No explanation was given by Nasrollahi [2] for this specific flow phenomenon. Possibly, it is caused by local flow separation. However, referring back to Figure 7.4, this does not immediately become apparent from the measured pressure distribution. Moreover, it should be noted that the fluorescent oil film photograph is an instantaneous capture of the flow. In contrast to the CFD results, which is a time-averaged solution of the flow. Nevertheless, examination of the boundary layer profiles will have to conclude about the nature of the disturbance (Section 7.5).

To enhance the comprehension of the flow characteristics, the particle tracks in Figure 7.10 are color coded



Figure 7.11: Surface particle tracks obtained by CFD of the the front potion of Figure 7.12: Close-up of the fluorescent oil film of the AeroCity (*U* = 80m/s) front region of the side-plate.[2]

with the skin friction coefficient. This reveals that the skin friction coefficient decreases over the concave part of the upper surface. As the flow decelerates, the pressure gradient becomes adverse and as a consequence, the shape factor of the boundary layer increases. The velocity gradient near the surface is therefore reduced, which in turn decreases the local wall shear stress. The lowest skin friction coefficient is observed near the trailing edge, where  $C_f \approx 5.0 \cdot 10^{-5}$ . Note that the aft region near the wing edge features significant higher values for the skin friction coefficient. This is again explained by the larger local velocity induced by the rotational motion of the tip-vortex. The highest wall shear stress, or skin friction coefficient, can be found at the leading edge of side plate being  $C_f \approx 0.0221$ .

Besides a close comparison of the top surface, the flow past the side of the Aerocity is investigated in more detail. Shown in Figure 7.11 are the surface particle tracks of the front region of Aerocity. Observing the particle traces, one can immediately spot the zones of flow circulation near the leading edge of the wing's side-plate. If one compares the particle traces with the fluorescent oil film picture, shown in Figure 7.12, one can see that core of the first circular zone is approximately equal in size and location, namely slightly above the main wing leading edge joint with the end-plate. However, the size of the surrounding recirculation zone appears to extend further downwards compared to the numerical result. Although it is difficult to observe, it can be seen that the second recirculation zone is located further down in the case of the wind-tunnel model. Furthermore, there appears to be a strong upward flow component originating from the bottom of the end-plate' leading edge. Unfortunately, the resolution of the CFD particle traces in this particular region is poor, making it hard to determine whether this is also found in the numerical result. However, close examination reveals that the particle traces are curved upwards only in the middle portion of the side-plate leading edge, feeding into the second recirculation zone. Hence, the size of the two recirculation zones appears to have been larger in the wind-tunnel experiment compared to the CFD result.

# **7.4.** TURBULENCE

Up to this point the flow characteristics of the numerical and the wind-tunnel model differ mainly on the size of the recirculation zone after the side-plate leading edge. However, in order to estimate how much this may



Figure 7.13: Contours of the total pressure loss coefficient at various chord-wise locations (U = 40 m/s)



Figure 7.14: Contours of lateral w velocity-component at various chord-wise locations (U = 40 m/s)

have affected the total drag prediction of AeroCity, the relatively contribution of the drag coefficient needs to be determined. This can be approximated by computing the total pressure loss coefficient, shown in Figure 7.13. The total pressure loss is an indication for the amount of friction losses inside the fluid. Immediately, from Figure 7.13, one can observe a region of significant total pressure loss at the location of the separation bubbles on the AeroCity side-plate. The highest level of the total pressure loss coefficient is as high as  $C_{TPL} \approx 0.029$ . The effect of the separation bubble on the total pressure loss can be seen to remain present downstream up to even x/c $\approx 0.60$ .

A second source of total pressure loss that can be identified is the ground vortex (or horse-shoe vortex). Especially within the vortex core a significant amount of total pressure loss can be seen to occur. The intensity of the loss reduces downstream, as the vortex is growing and the swirling of the flow becomes less. Although the tip vortex can also be seen to contribute to the total pressure loss, the level of frictional losses are much lower compared to the ground vortex. Interestingly, the frictional losses in the boundary layer of the upper surface can also be observed in Figure 7.13. The magnitude of the total pressure loss is largest near the surface and reduces away from the surface. Zooming in on the trailing edge, it can be seen that the zone of total pressure loss grows relatively thick. This is an indication of the boundary layer development, which , based on the observations from Figure 7.13, is quite significant. A more detailed analysis of the boundary layer profiles will be presented in Section 7.5.

In addition to the above observations, one can also observe from Figure 7.13, that the lower tip vortex travels downstream in lateral direction, away from the vehicle. This is unusual, as a typical trailing vortex of a wing follows a path at a much smaller angle to the free-stream flow direction. To explain this, it is interesting to examine the lateral velocity component of the flow around AeroCity. This is shown in Figure 7.14. Apart from the obvious lateral flow around the leading edge of the side-plate, one can identify spurs of high velocity flow stemming from underneath the side-plate. This is 'leakage flow', that is accelerated from underneath the vehicle due to the large pressure difference on both sides of the side-plate. As was found by Nouwens [1], the magnitude of the local velocity is not only influenced by the magnitude of the pressure difference, but also strongly affected by the gap height between the side-plate and the ground plane. The jets of high velocity air 'push' the vortex out in lateral direction, which reduces the influence of the vortex on the flow over the upper surface. More-over, it prevents the lower and upper tip-vortex from merging downstream, even though the strength of the vortex itself is increased.[8]

To balance the mass-flow in lateral direction, a zone of re-circulation in opposite direction exist directly above the area of leakage flow. Another zone of negative lateral velocity can be seen to exist on the aft upper surface. Caused by a potion of the flow bending inboard over the wing's edge, a significant amount of cross-flow exists over the aft upper surface. The cross-flow over the wing' edge induces the tip vortex, which originates at approximately  $x/c \approx 0.40$ . Further downstream, a uniform zone of negative lateral velocity can be seen to exist. The zone extends almost up to the wing centreline, and therefore is expected to have a significant effect on the boundary layer at the aft section of AeroCity.

#### 7.4.1. VORTEX VISUALIZATION

From the previous observations, it becomes obvious that at least two vortices per side exist in the case of AeroCity. To gain further understanding of the vortices, it is helpful to visualize them by means of an iso-surface. In order to do this, the so-called Q-criterion is selected to define the vortex [75]. The Q-criterion states:

$$Q = \frac{1}{2} \left( u_{i,j}^2 - u_{i,j} u_{j,i} \right) - \frac{1}{2} u_{i,j} u_{j,i} - \frac{1}{2} \left( |\Omega|^2 - |S|^2 \right) > 0$$
(7.1)

,where *Q* is the second invariant of  $\nabla u$ . Explained by words, the *Q*-criterion defines the vortex as the regions where the vorticity magnitude is larger than the strain-rate magnitude. As a secondary requirement, it requires the pressure to be lower than the ambient conditions [76]. The results is shown in Figure 7.15. For better comprehension of the vortex structure, contours of the turbulence kinetic energy have been plotted on the iso-surface.

The lower tip vortex can be clearly noticed, originating from the side-plate' leading edge. Aside from the lateral displacement of the vortex core, it can be observed that the vortex starts to dissipate further downstream, still ahead of the trailing edge. The reason for this is two-fold. Primarily, the vortex core transitions away from the vehicle in lateral direction. As a result, the influence of the 'jet-like' flow from underneath the side-plate is reduced. This is enhanced by the fact that the momentum of the leakage flow is reduced downstream of  $x/c \approx 0.60$ , due to a reduction of the pressure difference between both sides of the end-plate. This can be seen more clearly in Figure 7.16, where the turbulence kinetic energy of the leakage flow ahead of this location are higher. Note that the flow from underneath the side-plate is not a uniform flow, but appears to consist of a series of smaller vortices.

Further, the zone of flow separation near the side-plate leading edge can be clearly seen. As already noticed in Figure 7.11, this region consists of two separate recirculation zones, stacked closely together. The upper recirculation zone can be seen to be more prominent in terms of size. The turbulence kinetic energy is high inside and towards the end of the bubble. However, if the free-stream velocity is increased, referring the reader to Figure 7.17, the size of the separation bubbles can be seen to decrease. Note that the turbulence kinetic energy has been scaled accordingly. The lower situated separation bubble appears to have diminished at U = 80



Figure 7.15: Iso-surface of Q-criterion  $(Q/Q_{max} = 0.001)$  and contours of turbulence kinetic energy (U = 40 m/s)



Figure 7.16: Iso-surface of Q-criterion  $(Q/Q_{max} = 0.001)$  and contours of turbulence kinetic energy (U = 40 m/s)



Figure 7.17: Iso-surface of Q-criterion  $(Q/Q_{max} = 0.001)$  and contours of turbulence kinetic energy (U = 80 m/s)

m/s. Apart from the reduced separation bubbles, the flow characteristics and the scaled contours of turbulence kinetic energy appear to be identical. This could explain the decreasing trend of the drag coefficient with the free-stream velocity, while the pressure distribution over the wing remains approximately constant.

Aside from the tip vortices and the separation bubbles situated at the side-plate, a few other interesting flow aspects can be identified from Figure 7.15 to 7.17. First, a small horse-shoe vortex can be seen to exist just in front of the side-plate. It is very small in size and diminishes quickly downstream. More importantly, a wake structure can be identified at the rear of the side-plate. The side-plate has a blunt trailing edge, hence the Kutta condition is not met, until a virtual point downstream of the trailing edge. As a consequence, the flow separates from the body. A local zone of reversed flow exists behind the vehicle, causing additional pressure drag. To reduce this drag component, the trailing edge of the side-plate should be shaped such that the flow leaves the trailing edge in a smooth manner.

# **7.5.** BOUNDARY LAYER PROFILES

Up till now, the numerical results have been compared with the wind-tunnel data on a vehicle-based scale. In order to investigate how the flow over the AeroCity compares on a local level, the boundary layers are compared with the measurements conducted by Nasrollahi [2]. The measurements were conducted by means of a Pitot tube, parallel to the surface and aligned with the direction of the flow. According to the report, testing was conducted at U = 75 m/s [2].

#### 7.5.1. UPPER SURFACE

In order to survey the upper surface of AeroCity, five measurement locations have been selected over the centre-line of the wing. [2]. Although the number of measurement locations are too few to fully capture the boundary layer development of the wing, it is sufficient to capture the general trend. The measured boundary layer profiles are originally presented in terms of total pressure, although this was not specified by Nasrollahi [2]. The comparison between the CFD results and the wind-tunnel measurements are shown in Figure 7.18 to 7.21. Before examining these figures more closely, it should be noted that the CFD results were obtained with a free-stream velocity U = 65 m/s. Since the total pressure levels obtained at U = 75 m/s were found to be significantly higher, it is suspected that the wind-tunnel velocity noted in the report is erroneous [2]. Without any other reference or indication about the actual wind-tunnel conditions, it was found that for U = 65 m/s the total pressure levels match approximately. Since the ambient conditions were not stated either, an exact match of the pressure curves was not obtained. Conversion of the pressure profiles to a velocity profile, using the static pressure measurement obtained during measurement of the pressure distribution, shows that assumption of U = 65 m/s is indeed correct.

With the above in consideration, a closer look is taken at the boundary layer profiles of Figure 7.18 to 7.21. Starting with the boundary layer profile measured at x/c=0.21, it can be seen that the experimental and numerical results are in very good agreement. The displacement thickness is approximately equal and measured  $\delta_{99} = 3.7$  mm in the wind-tunnel. Proceeding to x/c= 0.46, shown in Figure 7.19, is can be seen that a discrepancy between the numerical and experimental data exists. Most obvious is non the non-smooth pressure profile, obtained by CFD simulation. The result is a non-smooth, non-physical boundary layer profile. Although the lower part of the boundary layer appears to be modelled well, the outer region of the boundary region is significantly off. As such, it is not possible to determine the displacement thickness of the simulated boundary layer. The displacement thickness of the boundary layer measured in the wind-tunnel is equal to  $\delta_{99} = 8.8$  mm. The same observations hold true for the boundary layer profile at x/c= 0.65. However, in this case it can be seen that the velocity gradient of the boundary layer measured in the wind-tunnel, is lower. This is an indication that the CFD model does not capture the adverse pressure gradient sufficiently. Examination of the boundary layer profile near the trailing edge (x/c=0.98) shows that the boundary layer of the wind-tunnel model is on the verge of separation. Note that for the computation of the velocity profile in Figure 7.21, the local static pressure was unknown. Instead, the static pressure measured at x/c=0.95 has been used. Hence, the actual velocity profile might deviate slightly.

From the above observations, it can be concluded that the boundary layer development in the CFD model is approximately equal to wind-tunnel experiment, at least in a qualitative sense. The displacement thickness and edge velocity match reasonable well, as expected on the basis of the pressure distribution. However, in a



Figure 7.18: Measurement of velocity profile perpendicular to the Figure 7.19: Measurement of velocity profile perpendicular to the surface at z = 0 surface at z = 0



Figure 7.20: Measurement of velocity profile perpendicular to the Surface at z = 0 Figure 7.21: Measurement of velocity profile perpendicular to the surface at z = 0

qualitative sense there are significant differences between the measured and modelled boundary layers. Most significant is the difference in boundary layer shape factor, over the aft section of AeroCity. It appears that the modelled boundary layer is not very sensitive to the adverse pressure gradient. As this affects the velocity gradient near the surface, a difference in local skin friction drag is to be expected. Another distinct observation are the non-smooth velocity profiles in the outer region of the simulated boundary layer. Since the behaviour is not physically correct, an explanation should be sought in the numerical models. Whether the typical behaviour is a consequence of an improper mesh or a bug elsewhere inside the software is examined in Chapter 8.

#### 7.5.2. WALL BOUNDARY LAYER

In Section 6.2, it was estimated that the virtual origin of the boundary layer on the wind-tunnel, in CFD, should be located approximately 1.9 m upstream of the AeroCity model. Since the wall boundary layer characteristics directly influences the flow field upstream of the model, it should be verified whether the assumption is indeed correct. To test the assumption, the AeroCity model was removed from the computational domain and the boundary conditions adapted to meet the conditions of the LTT boundary layer measurements. The result is shown in Figure 7.22. As can be seen, the non-dimensionalized velocity profiles are not in good agreement. Most noticeable is the discontinuity in the velocity gradient, in the upper portion of the simulated boundary layer. This is similar to the observations made with Figure 7.19 and 7.20. As a result, the displacement thickness of the boundary layer is difficult to determine accurately. Extrapolation of the velocity profile before the 'kink' yields to a displacement thickness approximately equal to the wind-tunnel



Figure 7.22: Comparison of the boundary layer velocity profile at x = 115 measured with respect to the start of the LTT test-section

Figure 7.23: Comparison of ground velocity profiles at various stations upstream of the AeroCity model (U = 40 m/s)

measurement. Nevertheless, the velocity profile in the outer region of the boundary layer significantly deviates from the measured velocity profile in the LTT.

Aside from the deviation in the outer region, the velocity of the entire boundary layer profile is slightly higher compared with the LTT measurement. Hence, besides the non-physical behaviour in the outer region of the simulated boundary layer, some additional fine-tuning could be performed to obtain a better match of the results. However, considering the fact the measurement locations with respect to the model are not exactly known, but approximated instead, the global result is considered to be good enough. Still, the non-physical behaviour in CFD of the outer region of the boundary layer should be investigated to increase the validity of the CFD result.

Having established that the virtual origin of the wall boundary layer is approximately the correct distance upstream of the AeroCity model, the boundary layer development close to the model can be examined. Shown in Figure 7.23 are the velocity profiles of the ground boundary layer at various stations upstream of the AeroCity model. As expected, the boundary layer develops gradually, in terms of thickness, when it progresses downstream. Note how the shape of the velocity profile alters when the boundary layer thickness increases. The discontinuity in the velocity gradient, existing between approximately y = 10 to y = 20 mm, is not found in the velocity profile of x/c = -0.025. Instead, the latter velocity profile shows a smooth variation of the velocity up to the boundary layer edge, located at  $\gamma \approx 36$  mm. Since the boundary layer mesh of the ground plane has a thickness of  $h \approx 16$  mm, it appears that the transition between the prism and tetrahedral mesh influences the outer region of the boundary layer. As mentioned, this will be further examined in Chapter 8. Apparently, when the transition of mesh type occurs before the blending region with the edge conditions, the interference does not occur. Since the velocity profile of the ground boundary layer that the AeroCity model encounters is smooth and the development of the boundary layer does not appear to be affected, it is assumed that the numerical error has no significant impact on the aerodynamics of the AeroCity model in this particular case. This is due to the fact that the leading edge of the wing is situated approximately y = 120 mm above the ground plane, as such the direct influence of the ground plane boundary layer is mainly limited to the lower portion of the end-plates

Another distinctive observation that can be made from Figure 7.23 is that the flow is decelerated significantly when approaching the AeroCity model. At half a chord-length upstream, the edge velocity of the boundary layer is still approximately equal to the undisturbed free-stream  $U_{\infty}$ . However, just upstream of the model at x/c = -0.025, the flow has decelerated to about  $U/U_{\infty} \approx 0.65$ . Since the flow is decelerated strongly in the channel underneath the AeroCity model, the flow near the ground plane is affected already relatively far upstream. Although an adverse pressure gradient exists, the ground boundary layer does not show a tendency towards separation. Hence, a separation bubble at the ground plane near the leading edge, as observed in numerous occasions in literature [5] [15], is not found for this particular configuration.

1.2



Figure 7.24: Samples of total pressure inside the separation bubble Figure 7.25: Samples of total pressure inside the separation bubble at y/c=0.12 and x/c=0.04 compared to experimental data [2] at y/c=0.12 and x/c=0.08 compared to experimental data [2]



Figure 7.26: Samples of total pressure inside the separation bubble Figure 7.27: Samples of total pressure inside the separation bubble at y/c=0.12 and x/c=0.15 compared to experimental data [2] at y/c=0.12 and x/c=0.25 compared to experimental data [2]

### 7.5.3. SEPARATION BUBBLE

Next to the boundary layer measurements on the upper surface of AeroCity, measurement were also taken during the wind-tunnel experiment at the end-plate [2]. The goal of these measurements was to provide validation data for a future CFD study. Unfortunately, uncertainty exists about the exact location of the measurements. Although the chord-wise location of the measurements is documented well, confusion exists about the vertical position on the end-plate. Stated in the report by Nasrollahi is that the measurements have been conducted at a distance of d = 150 mm with respect to the end-plate lower surface. However, a CAD drawing and photographs of the experimental model suggest a different vertical location. However, sampling at either of these locations reveals that there is no resemblance between the numerical and experimental measurements of any kind. Most likely, this is due caused by an offset of the location of the separation bubble in the experiment and simulation. Instead, it is opted to sample along a line through the middle of the separation bubble, in order to make a qualitative comparison between the two data sets.

The vertical position of the separation bubble centre in the CFD result was found to be y/c=0.12 with respect to lower side of the end-plate. Sampling of the total pressure along the centre line of the bubble reveals valuable information about the flow characteristics inside the bubble. Shown in Figure 7.24 and Figure 7.25 are the comparisons of the total pressure profiles inside the separation bubble between the CFD result and the wind-tunnel measurements [2]. In the case of the first measurement location x/c=0.04, it can be seen that the CFD result predicts a much lower, negative, total pressure. Note that the ambient pressure, or operating pressure, has been used as a reference pressure. Presumably, this is identical to the reference pressure used

B.C.	$C_L$	$C_D$	$C_M$	L/D
Stationary ground	0.6106	0.04751	-0.08510	12.85
Moving ground	0.6122	0.04707	-0.08338	13.01
	+0.2%	-0.93%	-2.02%	+1.81%

Table 7.1: Comparison of force coefficients between two ground boundary conditions for U = 40 m/s

during the wind-tunnel measurements, however this is not documented. Although the pressure profile feature a similar shape, the quantitative differences are significant.

Most noticeable, is the much higher pressure close to the surface, measured during the wind-tunnel experiment. Unfortunately, only measurements of the total pressure have been conducted. Since the static pressure is not available for these locations, no distinction can be made between the static and dynamic pressure. It appears that the flow velocity close to the wall is significantly higher than the free stream velocity and shows a distinct reduction near z = 20 mm. Further away from the side-plate, the velocity increases again up to free-stream levels. The CFD result shows a similar profile shape, however the location of lowest velocity is closer to the surface, in the region of z = 10 mm. This is an indication that the size of the separation bubble was significantly higher during the experiment than found in the CFD analysis. This supports the hypothesis that the under-prediction of the separation bubble on the side-plate is a mayor contributor to the under-prediction of the drag coefficient by the current CFD model.

Further downstream, at x/c= 0.08, the experimental data shows that the size of the bubble continues to increase with the total pressure becoming negative at  $z \approx 30$  mm. Despite the non-smooth profile of the CFD result, caused by an increasing mesh cell size away from the body, the outer region of the separation bubble appears to be captured well by the CFD model when compared to the wind-tunnel measurements. Note that the location of minimum total pressure, in the case of CFD, has not been altered significantly. Further downstream, at x/c= 0.15, the wind-tunnel measurements show that the separation bubble still exists. The simulated flow field however indicate that the flow has reattached, as can be observed in Figure 7.26. At the next measuring location (x/c= 0.25) the separation bubble has been diminished in experimental results as well. However, as shown in Figure 7.27, the total pressure levels are observed to be higher in the case of the CFD prediction. Hence, the momentum of the boundary layer after the separation bubble appears to be lower for the wind-tunnel model compared to the numerical model. However it should be stressed that, as the vertical location of the measurements is different, the magnitude of the total pressure cannot be compared one to one.

Nevertheless, despite the uncertainties and different measurement location, which prohibits a validation of the current results, a few conclusion can be drawn from the comparison. First, that the separation bubble during the wind-tunnel experiment was significantly larger in size. Both in terms of thickness as in diameter. This is in agreement with the observations from Figure 7.11 and 7.12, which showed that the zone of recirculation close to the surface was larger in the experimental set-up. A second conclusion that can be draw is that the cell size further away from the body prohibits a smooth transition to the free-stream flow parameters. Although it is not expected that this is of mayor influence on the results, local refinement of the mesh could improve the capture of the separation bubble.

#### **7.6.** EFFECT OF A MOVING GROUND

Although the CFD model does not replicate results of the wind-tunnel experiment to a full extent, it is still of interest to examine the influence of the boundary condition applied to the ground plane. Since the deviation of the CFD prediction with the experimental results are believed to be predominantly caused by an under-prediction of the separation bubble and the effect of the adverse pressure gradient on the upper surface boundary layer, a relative comparison between a stationary and moving ground boundary condition can be made. In order to simulate the moving ground, the ground plane was given an absolute velocity equal to the free-stream velocity. The mesh, operating conditions and other boundary conditions were kept constant.

The influence of the ground boundary condition on the force coefficients is presented in Table 7.1. Examination of the results yields that the differences are small, with a minor increase of the lift coefficient and a small





Figure 7.28: Pressure distribution at z/c=0.00 for two different ground boundary conditions (U = 40 m/s)



Figure 7.29: Pressure distribution at z/c=0.15 for two different ground boundary conditions (U = 40 m/s)



Figure 7.30: Velocity profile of the boundary layer on the upper surface measured at x/c=0.02 and U=40 m/s

Figure 7.31: Non-dimensionlized velocity profile of flow channel underneath Aerocity at x/c= 0.10 and U= 40 m/s

reduction of the drag coefficient. Nevertheless, in terms of aerodynamic efficiency, it translates to an increase of  $\Delta L/D = 1.8\%$ . Given the order of magnitude of the absolute deviation of the CFD results with the experimental data, the difference between the two boundary conditions is almost negligible. Still, the comparison shows that the boundary condition imposed on the ground plane has a direct influence on the performance characteristics of AeroCity. This effect is expected to be enhanced when the height-to-chord ratio is reduced further, as the boundary layer of the ground plane occupies a larger portion of the duct underneath the AeroCity. In terms of moment coefficient, it is found that the magnitude of the pitching moment is reduced.

To investigate this further, the pressure distributions for bot cases are compared, as shown in Figure 7.28 and 7.29. Starting with the pressure distribution along the centreline of AeroCity, it can be seen that the Cp values in general are slightly lower for the case of a moving ground boundary condition. The pressure profiles are very much comparable, except in the front part of the lower surface. The difference in  $C_p$  can be explained by the overall higher momentum flow encountered by the AeroCity model, in the case of the moving ground boundary condition. With a displacement thickness of over  $\delta^* = 35$  mm, the momentum loss introduced by the ground boundary layer is significant at this scale. More outboard, at z/c= 0.15, similar observations can be made. However, a small deviation of the  $C_p$  distribution profile can be observed between  $x/c\approx 0.65$  and  $x/c\approx 0.80$ . The inflection point of the  $C_p$ -curve is shifted upstream, compared to the results obtained with a stationary ground. However, comparison of the boundary layer profiles has shown that these are virtually identical. Only the boundary layer profile at x/c= 0.02, shown in Figure 7.30, is affected in a significant manner. In the case of a moving ground boundary condition, the velocity over the leading edge is noticeably

higher. This is an indication for a shift of the stagnation point downstream, in the case when the moving ground boundary is condition applied. As such, the effective angle of attack is marginally larger and the suction peak more pronounced, in the case the moving ground boundary condition.

Aside from the increased suction over the upper surface, the pressure on the lower surface is increased as well. As mentioned, the momentum loss in the ground boundary layer is an important factor for explaining the difference in force coefficients between the two ground boundary conditions. To illustrate this, the nondimensionalized velocity profiles underneath the AeroCity are plotted in Figure 7.31. Immediately, it can be observed that the ground boundary layer occupies over 25% of the height between the lower surface of AeroCity and the ground plane. In contrast to the stationary ground scenario, the velocity remains more or less constant over the entire height of the flow channel, except very close to the walls. Very close to the ground plane, a large velocity gradient can be observed, as the velocity at the wall is equal to the velocity of the ground plane. Note that the velocity of the flow, over the main portion of the duct underneath the AeroCity, is lower for the moving ground boundary condition. This can be explained by the fact that the ground boundary layer reduces the effective eight of the channel. Hence, to accommodate a comparable mass-flow underneath the vehicle, the flow outside the boundary layer needs to have a higher velocity. In turn, this yields to a slightly lower pressure underneath the vehicle, as was already observed in the  $C_P$ -distributions shown in Figure 7.28 and 7.29.

Having explained the source for the marginally added lift, a closer look of the vortex structures around the AeroCity is taken to examine if any significant differences can be identified that attribute to the lower overall drag of the case with a moving ground boundary condition. The iso-surface plots of the Q-criterion for both boundary conditions are shown in Figure 7.32 and 7.33. At a first glance, the two figures appear identical. However, a few observations can be made. Most noticeable are the apparently larger tip vortices, in the case of the moving ground boundary condition. However, the turbulence kinetic energy contours are comparable. The same holds true for the laminar separation bubbles on the side-plate, which both appear to be larger in size. This is confirmed when the samples of the velocity inside the bubble are compared with the results involving a stationary ground plane. The difference in bubble radius is approximately  $\Delta r \approx 20\%$ . Hence, the drag contribution of the separation bubble is expected to increase. However, this is offset by a lower overall turbulence kinetic energy over the upper and lower surface of AeroCity, combined with a small drag reduction due to the enhanced suction peak.

Nevertheless, the differences are very small given the numerical accuracy. Despite this, it can be concluded that the applying a moving ground boundary condition has a marginal beneficial effect on both the lift and the drag, yielding to a minor improvement in the lift-to-drag ratio. However, this conclusion only applies to the current aerodynamic model. Due to the scale of the model, the boundary layer of the ground plane has a limited impact on the actual aerodynamics around the wing. If the absolute elevation height would be reduced, either by decreasing the scale of the model or by reduction of the height-to-chord ratio, the effect of the ground boundary layer will become more evident. Although fairly accurate results can still be obtained from wind-tunnel experiments involving a stationary ground plane, application of a moving ground boundary ary condition should always be preferred.

# **7.7.** INCLUSION OF A TRACK WALL

Up to this point, it has been established that the CFD results and the data from the wind-tunnel experiment [2] show a reasonable commonality. Furthermore, it was found in the previous section, that the use of a stationary ground boundary condition did not introduce a large discrepancy with the moving ground boundary condition case, according to the present CFD results. However, in terms of the exploration of the aerodynamic characteristics of AeroCity, these results have not introduced any radical new aspects about the aerodynamic configuration of AeroCity. Both Nouwens [1] and Nasrollahi [2] already conducted numerical and experimental research into the aerodynamic model of AeroCity with either a stationary or moving ground boundary condition. Although both studies were conducted independently and used a different aerodynamic model, the general characteristics and sensitivity with respect to height and angle of attack are well documented.

However, one important aspect of the AeroCity has not been investigated yet, or been overlooked. In order to provide lateral guidance, the AeroCity is envisioned to fly inside a U-shaped track. [1] Electro magnets incor-



Figure 7.32: Iso-surface of Q-criterion ( $Q/Q_{max} = 0.001$ ) for U = 40 m/s and stationary ground b.c.



Turbulence Kinetic Energy

Figure 7.33: Iso-surface of Q-criterion  $(Q/Q_{max} = 0.001)$  for U = 40 m/s and moving ground b.c.



Figure 7.34: Iso-surface of Q-criterion ( $Q/Q_{max} = 0.002$ ) for U = 40 m/s and moving ground b.c. and track wall





Figure 7.35: Effect of track wall inclusion on  $C_P$  distribution at z/c = 0.00 and U = 40 m/s

Figure 7.36: Effect of track wall inclusion on  $C_P$  distribution at z/c = 0.15 and U = 40 m/s

porated inside the track wall are to be used for both propulsion and lateral guidance. However, since neither the propulsion system or the lateral guidance control system is known yet, no track design currently exists. As such, a simplified track geometry is designed, to be able to investigate the effect of a track wall inclusion to the aerodynamic characteristics of the AeroCity. In theory, the track wall could attribute to the aerodynamic performance of AeroCity, by preventing flow leakage from underneath the vehicle. This would reduce the tip vortices and increase the pressure on the lower surface. Therefore, ideally, the track should be very close to the end-plate, with a minimum gap distance to allow for a margin of clearance. With no reference study or design as a starting point, a track was designed, such that for the full-scale model (c = 20 m) the lateral gap between the end-plate and the track wall would amount to  $z_w = 0.30$  m. A wall height of  $h_w = 1.00$  m was assumed. Or in non-dimensionalized terms:  $z_w/c = 0.015$  and  $h_w = 0.05$ . Although a further reduction of the lateral gap may be achievable, the current geometry of the track was considered to be a realistic estimation of a possible track wall design.

To investigate the impact on the aerodynamic characteristics of the AeroCity model, the track wall was modelled in CFD by means of a no-slip wall, with an absolute velocity equal to the free-stream. As it turns out, the impact of the track wall is quite significant. An overview of the force and moment coefficients is given in Table 7.2, were the results are compared with those of the simulation without a track wall present. Note that in both cases a moving ground boundary condition was applied. As can be observed from Table 7.2, the lift coefficient is indeed slightly increased compared to the model without a track wall present. However, the drag coefficient is found to be significantly increased, reducing the lift-to-drag ratio by more than  $\Delta L/D \leq -6.0\%$ . Also, the magnitude of the pitching moment was reduced, although the decrease was more more modest. To investigate how the flow around the AeroCity was affected, the pressure distribution is compared with the results with a without a track wall. A the centreline of model z = 0.00, the  $C_P$  values up to x/c= 0.20, are very much comparable to the configuration without a track wall present. However, further downstream a few differences can be noticed. On the upper surface, the pressure becomes less negative, up to about x/c = 0.80. On the lower surface the opposite can be observed, namely slightly less positive pressure over the lower surface. Near the trailing edge, the flow can be seen to leave the body at a lower pressure. Hence, from the pressure distribution measured along the centreline of the vehicle, it appears that the lift would decrease slightly, rather than to increase. However, if one examines the pressure distribution further outboard, at z/c=0.15, as shown in Figure 7.36, one can observe that there is a significant zone of lower pressure on the aft upper surface. This zone of aft suction at the outboard portion of the wing is responsible for the lift increase of the total vehicle and offsets the minor reduction in lift distribution over the majority of the wing's surface.

To explain the sudden decrease of the pressure of the upper surface, the behaviour of the vortices is examined. An iso-surface plot of the Q-criterion is shown in Figure 7.34. Note that the non-dimensional Q-ratio has been increased to enhance the visibility of the results. Incorporation of the track wall has indeed reduced the strength of the lower tip vortex, judging from the reduced turbulence kinetic energy contours. However,



Figure 7.37: Close-up of the vortices near the aft portion of AeroCity on the inside of the track wall (U=40 m/s)

the vortex is not suppressed by the track wall, but rather deflected vertically. In the case without a track wall, the vortex showed a lateral displacement away from the body in downstream direction. In this case, the vortex remains attached to the vehicle and merges with the upper tip vortex, just after mid-chord. Together, the two tip vortices form one big vortex, disrupting the flow over the upper surface. Although the vortex induces a down-wash over the wing, thereby reducing the effective angle of attack, the added vortex lift causes a positive net effect on the lift coefficient.

The smaller leak flow vortices, which otherwise would be dispersed in lateral direction, are also deflected upwards. Moreover, a 'jet-like' flow of highly turbulent flow can be seen to exist between the wall and the vehicle. Once near the trailing edge, the stream of highly turbulent flow is given the space to curl into a separate vortex. This is visualized in Figure 7.37. Together with the merged vortex over the upper surface, the pressure drag on the AeroCity model is increased considerably. In total, the pressure drag amounts to approximately 80% of the total drag, versus 20% viscous drag. Aside from the tip vortices, it could be also be observed from Figure 7.34, that the separation bubble over the side-plate has increased in size. This will in turn also attribute to the higher drag. Since the flow channel between the side-plate and the track wall is limited, a portion of the flow around the side of AeroCity will be deflected upwards. As such, the separation bubble on the lower portion of the side-plate is translated upwards and seen to merge with the upper separation bubble.

In summary, it appears that the current track design has a negative impact on the performance of the AeroCity. Unlike expectations, the inclusion of the track wall did not yield to an increase of the pressure on the lower side, by preventing flow leakage through the gap underneath the end-plate. Instead, the track wall deflects the lower tip-vortex upwards and prevents it to be dispersed in lateral direction. As a result, the tip vortices merge and enhance the vortex lift over the aft section of the vehicle. Admittedly, only a single track design has been investigated. Reduction of the gap between the track wall and the side-plate of the wing could improve the results, by further preventing the lower tip vortex to develop. However, this imposes significant requirements to the lateral control system, since the margin between the vehicle and the wall is further reduced. Another

Table 7.2: Comparison of force coefficients for a moving ground boundary condition with and without track wall inclusion for U = 40 m/s

Case	$C_L$	$C_D$	$C_M$	L/D
without track	0.6122	0.04707	-0.08338	13.01
with track wall	0.6170	0.05075	-0.08247	12.16
	+0.78%	+7.80%	-1.10%	-6.51%

option would be to select a different track design, such as the V-shape track mentioned by Nouwens [1]. An alternative solution could also be to use a guardrail type design. The advantage of such a system, as-side from potential cost savings, is that it still allows the lower tip vortex to be dispersed in lateral direction.

# 8

# **MODEL ADAPTATION**

As discussed in Chapter 7, the CFD simulations and the experimental data obtained during the wind-tunnel experiment [2], are in reasonable agreement. However, it was also found that for certain aspects, the results differed considerably. Not only by a growing deviation of the force coefficients towards higher flow velocities, but also in terms of separation bubble size and boundary layer profiles. On top of this, it was observed that the numerical results showed some peculiar behaviour in the outer region of the boundary layers. Although the results obtained so far give a reasonable estimation of the aerodynamic characteristics of the AeroCity model, improvement of the CFD predictions is considered to be essential to gain confidence for future CFD analysis. To improve the results, several aspects of the CFD model have been investigated. The most important changes to the CFD model that have tested will be discussed in this chapter.

## **8.1.** REFINEMENT OF THE PRISM LAYER

One of the most crucial aspects that prevents a full validation of the CFD model with the experimental data, is the deviation in boundary layer profiles. Both in terms of shape factor as well as an apparent numerical defect in the outer region of the boundary layer, observed in some cases. As mentioned in Chapter 6, the growth ratio of the prism layer mesh was selected to be r = 1.30, due to reasons of computational efficiency. By doing so, it was accepted that this would introduce a possible numerical error, since the growth ratio is typically well below  $r \le 1.20$ . Hence, it should be investigated whether this decision is justified by comparing the gain of computational efficiency with the possible reduction in accuracy.

To do this, the mesh was adapted by selecting a prism layer growth factor of r = 1.15. Since an exponential growth law was selected, the number of prism layer increased from 22 to 32 layers. As a result, the total cell size increased from approximately 10.1 million to around 15.7 million cells. This has a significant effect on the computational efficiency, as shown in Table 8.1. As a consequence of the increased cell count, the time per iteration is increased by more than 70%. Considering the number of simulations that have been performed, this translates to a significant increase in overall computation time. The effect of the increased boundary layer mesh on the predicted force coefficients is however considerable, as can be observed from Table 8.2. Although the lift coefficient remains almost invariant of the mesh refinement, the drag coefficient is affected significantly. The reduction of the drag by more than  $\Delta C_D = 5\%$  has a significant positive effect on the predicted lift to drag ratio.

Table 8.1: Influence of prism growth factor *r* on computational parameters

Table 8.2: Influence of prism growth factor r on predicted force
coefficients and aerodynamic efficiency

	n	#cells	t/iteration
.30	22	10.1 <i>m</i>	14 s
.15	32	15.7 <i>m</i>	24 s
	+45.5%	+55.4%	+71.4%



Figure 8.1: Boundary layer velocity profile at x/c=0.21 and U=65 Figure 8.2: Boundary layer velocity profile at x/c=0.21 and U=65 m/s for two mesh configurations m/s for two mesh configurations

Since the difference in drag coefficient is larger than anticipated, the source of the drag reduction needs to be investigated further. To do this, the iso-surface plots of the Q-criterion of the two mesh configurations are compared, since these plots present a clear inside into the flow mechanisms around the vehicle. These are shown in Figure 8.3 and 8.4. Immediately, it can be spotted that the separation bubble on the side-plate of the vehicle has been reduced considerably. The lower, smaller separation bubble has even been diminished in the case of r = 1.15. Investigation of the drag build-up, reveals that the viscous drag is identical in both cases. Instead, the drag reduction is found to be driven by an decrease of the pressure drag. In line with the previous observations, it is suspected that the decrease of the pressure drag is a direct consequence of the reduction of the separation bubbles. Why exactly the separation bubble has decreased for the refined mesh is not well understood.

In order to confirm this hypothesis, it needs to be established that other factors contributing to the drag have not been affected by the decreased prism growth ratio factor. To do this, the profiles of the boundary layer over the upper surface are compared. This is presented in Figure 8.1 and 8.2. As can be observed, the inner region of the boundary layer profiles are identical. Since the drag component due to skin friction is identical in both cases, this is in line with expectations. In the outer portion of the boundary layer however, small changes between the two results can be identified. Unfortunately, the refined mesh also features the discontinuous velocity gradient in the region were the boundary layer profile is blended with the free-stream. The severity of the non-smoothness of the profile does appear to be reduced. This suggests that further refinement of the prism layers could improve the smoothness of the boundary profiles to an acceptable level. However, for the current project, this would increase the computational time beyond workable levels. Since the discrepancy of the boundary layer profile appears to be in the vicinity of the location between the transition of the prism layer mesh to the tetrahedral mesh, it is suspected that the error is mesh-dependent.

Since the boundary layer profiles are nearly identical, it should follow that the pressure distribution has remained unchanged. A quick comparison of the two data-sets reveals that this is indeed the case. Without any other indication for reduction of pressure drag, it can be concluded that the reduced size of the separation is in all likelihood responsible for the under-prediction of the drag coefficient, if compared to the original figure measured during the wind-tunnel experiment.[2]

## **8.2.** TRANSITION MODEL

Since the separation bubble on the side-plate appears to provide a significant contribution to the total vehicle drag, additional attention is paid to proper modelling of the flow over the side-plate' leading edge. As discussed briefly in Section 7.3, the leading edge of the side-plate is a half-cylinder. Hence, the local flow can be regarded as flow over a cylinder. Despite the fact that a plethora of experimental and numerical studies have been conducted on the flow past a cylinder, it remains an active research topic due to the complex aero-



Figure 8.3: Iso-surface of Q-criterion ( $Q/Q_{max} = 0.001$ ) and turbulence kinetic energy contours at U = 40 m/s for r = 1.30



Figure 8.4: Iso-surface of Q-criterion ( $Q/Q_{max} = 0.001$ ) and turbulence kinetic energy contours at U = 40 m/s for r = 1.15



Turbulence Kinetic Energy

Figure 8.5: Iso-surface of Q-criterion ( $Q/Q_{max} = 0.001$ ) and turbulence kinetic energy contours at U = 40 m/s obtained by Transition SST model



Figure 8.6: Skin friction coefficient over the upper surface along z/c = 0.00 at U = 40 m/s



dynamics involved. Although an in-depth analysis of the flow past a cylinder at this point should be avoided, it is useful to gather some more insight into the aerodynamic phenomena involved.

As discussed, the flow past a cylinder in a higher Reynolds number flow is marked by a laminar or turbulent wake. Middleton and Southard [77] showed that the location of flow separation on a cylinder shifts further to the back of the cylinder when the Reynolds number is increased. It was found that for  $Re_d = 1.0 \cdot 10^5$ , the separation point is located at approximately  $\phi = 80^\circ$ . Furthermore, it is found that a non-periodic laminar zone exist in the wake around  $Re_d = 5.0 \cdot 10^3$  [77]. The fluid within this laminar wake is 'trapped' within the turbulent wake. Up till  $Re_d \approx 400,000$  the boundary layer is, depending on the surface roughness, usually laminar and separation takes place downstream of the separation point.

The diameter based Reynolds number for the side-plate' leading edge is found to be between  $Re_d = 1.7 \cdot 10^4$ and  $Re_d = 1.4 \cdot 10^5$ . Hence, during the experiment the flow around the side-plate leading edge will have been most likely laminar. This is an important notion, as the separation point for a turbulent cylinder wall boundary layer is further downstream. As a consequence, the wake of a sub-critical flow past a cylinder is wider than for a super-critical flow. Moreover, the drag coefficient of a cylinder in sub-critical flow is about  $C_D = 1.20$ , of which 90% is due to pressure drag, while the  $C_D$  value for a cylinder in super-critical flow is 'only'  $C_D \approx 0.30$ . Although the total drag coefficient of the AeroCity model consist of many other drag contributions, it is clear that simulation of the flow by means of a fully turbulent domain may cause an underestimation of the drag component due to under-prediction of the separation bubble . A transitional turbulence model or splitting the computational domain into a separate laminar and turbulent zone, may improve the drag prediction.

Since splitting of the computational domain into a laminar and fully turbulent zone is very cumbersome for this particular case, it is opted to use a transition turbulence model. Although four different transition models are available in ANSYS Fluent, the Transition SST model has been found in Section 5.2 to provide the best results. Although it should be noted, that for an airfoil in extreme ground effect (Section 5.3), the Transition SST model failed to predict the drag with sufficient accuracy. With this in mind, the simulation involving a stationary ground is re-computed using the Transition SST model. Note that the mesh with the increased density of prism layers is used in this case. The resulting force coefficients are shown in Table 8.3. As can be observed, the lift-to-drag ratio is increased further with respect to the SST model utilizing the same mesh. The lift coefficient is observed to increase further, while the total drag coefficient shows a small decrease in magnitude. In earlier computations involving the transition SST model, a similar trend was observed in terms of force coefficients (Section 5.3). However, if one examines the way the total drag coefficient is build up, it is discovered that the pressure drag is increased by more than 330 drag counts. In contrast, the drag due to skin friction has been reduced by almost 370 drag counts. The latter is due to the fact that the laminar flow yields to lower skin friction coefficients, as the flow over the AeroCity model is no longer fully turbulent. Shown in Figure 8.6 are the skin friction coefficients measured along the centre-line of the model. Around x/c = 0.37


Figure 8.8: Oil flow particle traces over the side-plate of AeroCity, obtained with Transition SST model at U = 40 m/s

Figure 8.9: Close-up of the fluorescent oil film at the front region of the side-plate (U = 100 m/s).[2]

a sudden increase of the skin friction coefficients can be observed. This implies that transition of the upper surface boundary layer does occur around this location. This is significant further downstream than as observed by Nasrollahi during the wind-tunnel experiment, where the flow was found to transition to turbulent flow at  $x/c \approx 0.25$ .

Hence, the Transition SST model is too much biased towards laminar flow in this particular case. As a consequence, the drag is under-predicted further. Due to the reduced thickness of the laminar boundary layer, the effective curvature of the wing is enhanced, explaining the modest increase of the lift coefficient. The difference in displacement thickness of the boundary layer can clearly be observed in Figure 8.7. Aside from the difference in displacement thickness, the velocity profile is very different. As such, the application of a transition turbulence model does not prove to be useful in the validation of the CFD model. Nevertheless, it does show how the size of the separation bubble is affected by the inclusion of laminar flow. Sampling of the velocity profile inside the separation bubble reveals that the centre is located approximately 20 mm from the surface. Although the total pressure profile does not resemble the measurements performed by Nasrollahi, the size of the bubble is more on pair with the experimental findings than as found with the fully turbulent SST model. [2] More interestingly, the oil flow particle track pattern is much more on pair with the oil low pattern obtained during the wind-tunnel experiment. A comparison between the two sets of surface traces can be made from Figure 8.8 and 8.9. Although there remain some dissimilarities, such as the location of the bubble centre, the overall pattern is very similar. Since the oil flow pattern stemming from the wind-tunnel experiment was obtained at a higher flow velocity, perfect commonality is not to be expected.

Table 8.3: Comparison of Transition SST results with force coefficients obtained by SST model and r = 1.15 (U = 40 m/s)

	$C_L$	$C_D$	$C_{D,visc}$	$C_{D, press}$	L/D
SST	0.6131	0.04497	0.03450	0.01046	13.63
SST Transition	0.6196	0.04464	0.03786	0.00677	13.88
	+1.1%	-0.7%	+8.9%	-35.2%	+1.8%



Figure 8.10: Comparison of velocity profile over a zero pressure gradient flat plate at x = 0.97 with numerical data by NASA [13]



Figure 8.12: Comparison of viscosity ratio over a zero pressure gradient flat plate using a refined boundary layer mesh (r = 1.10)



Figure 8.11: Comparison of skin friction coefficient over a zero pressure gradient flat plate with numerical data by NASA [13]



Figure 8.13: Plot of SST blending functions for a zero pressure gradient flat plate at x = 0.97 with numerical data by NASA [13]

#### **8.3.** VERIFICATION OF BOUNDARY LAYER MODELLING

Aside from the apparent under-prediction of the separation bubble, the current CFD model also appears to contain a numerical error in the modelling of the outer region of the boundary layer. At this point, the cause for the discrepancy in the numerical result is not known. However, it is suspected that the transition of the prism layer to the tetrahedral mesh is of influence on the blending of the boundary layer with the outer-flow. Despite various mesh alterations, including incrementing the total prism layer thickness, no cure has been found for the problem. To investigate the problem in more detail, a verification study with numerical results provided by NASA is conducted. [13] The same mesh, only with the AeroCity model removed, is used for this comparison. As such, any mesh defects that may play a role, are incorporated in the analysis.

The case involved is a flat plate subjected to a zero pressure gradient. The analysis is focussed on the boundary layer profile at x = 0.97m with respect to the start of the flat plate (x = 0.00m). The plate was subjected to a subsonic flow (M = 0.2) and the plate length based Reynold number  $Re_L = 5.0$  million. Although the numerical results provided by NASA were obtained with compressible flow solvers, incompressible flow has been assumed for the current case. Since the flow is still low subsonic, the error introduced by neglecting compressibility of the flow should be small. More-over, incompressible flow is also used for the analysis of the AeroCity model. For the boundary conditions, the turbulence viscosity ratio was set equal to  $\mu_t/\mu = 0.009$ and the free-stream velocity equal to U = 69.4 m/s. Further, it should be noted that the NASA results were obtained using a two-dimensional grid, in contrast to the three-dimensional mesh used in the current CFD model. Multiple mesh sizes were used throughout the numerical verification by NASA. For this comparison, the results obtained by a  $545 \times 385$  grid were selected. Furthermore, it should be noted that the prism layer growth factor for the three-dimensional mesh was reduced from r = 1.30 to r = 1.20 to reduce the numerical errors in the boundary layer mesh.

Shown in Figure 8.10, is the comparison between the velocity profile obtained with the current three dimensional mesh and the results by NASA. [13] Note that velocity profiles are almost identical, only a small deviation of the results is observed in the most outer region of the boundary layer. Like observed earlier in the current work, there appears to be a discontinuity in the velocity gradient of the velocity profile in the region where the boundary layer is blended with the free-stream. However, aside from this anomaly, the correspondence between the two data-sets is good. Similarly, the development of the skin friction coefficient over the plate length, shown in Figure 8.11, is very much on pair with the numerical data provided by NASA. Only at the very front of the plate, a discrepancy between the results is observed. Considering the relatively coarse surface mesh of the ground upstream of the AeroCity model, this could be resolved by additional mesh refinement near the start of the no-slip ground. The development of the skin friction coefficient is very useful for the analysis of a flat plate, since the friction coefficient is directly related to the velocity gradient near the surface. Although not directly useful for the investigation of the non-smoothness of the velocity profile in the outer region of the boundary layer, it does provide verification regarding the overall accuracy of the boundary layer mesh.

Referring back to the measurement of the boundary layer profile at x= 0.97, the ratio of the viscosity and turbulence viscosity is plotted in Figure 8.12. The development of the eddy viscosity can be seen to be on pair in the region closest to the wall. Close to the location of maximum eddy viscosity, the eddy viscosity for the three-dimensional mesh is subjected to some non-smooth changes. These are likely to be caused by the increased size of the outer prism layer cells, as the growth rate with respect to each previous cell is exponential. The maximum viscosity ratio achieved in the boundary is not exactly equal, however the two curves can be seen to reunite further away from the wall again. Remarkably, a 'bump' in the eddy viscosity ratio can be observed in the result provided by NASA. According to the comment by NASA: *"This behaviour is due to the SST blending between omega and vorticity in the denominator of the equation for eddy viscosity, and is only noticeable on extremely fine grids for this flat plate case."* The formulation for the eddy viscosity, given by Eq. (A.35), is repeated below:

$$\mu_t = \frac{\rho k}{\omega} \frac{1}{\max\left[\frac{1}{a^*}, \frac{SF_2}{\alpha_1 \omega}\right]}$$
(8.1)

The bump is not noticeable in the current CFD result, although a slight kink in the viscosity ratio profile can be observed at the same distance from the wall. The location of the bump is in the region where the blending functions  $F_1$  and  $F_2$  are transitioning from one to zero. As can be seen in Eq.(8.1), the blending function  $F_2$  appears in the denominator of the eddy viscosity formulation. The blending function is a hyperbolic function, with the purpose to transition smoothly from the  $k - \omega$  model constants and the  $k - \epsilon$  model constants for the exterior flow. To illustrate this, the blending functions are plotted over the height of the boundary layer, shown in Figure 8.13. As can be seen, the blending function  $F_2$  starts to decrease, up till zero at the edge of the boundary layer. Note that the 'bump' as observed in the eddy viscosity ratio, is located at roughly the same distance away from the wall. Hence, the blending function  $F_2$  is believed to cause a small disturbance in the otherwise smooth profile of the eddy viscosity. Note that the blending functions are plotted only for the numerical data provided by NASA. [13]Unfortunately, the behaviour of  $F_1$  and  $F_2$  cannot be extracted from Fluent without writing a custom script to access the solver. As such, no comparison of the blending functions for the current simulation can be presented.

Reviewing the above observations, it appears that the non-smooth velocity profile in the outer region of the boundary layers is caused by a 'non-synchronized activation' of the blending function  $F_2$ . Since the blending function is given by  $\tanh(\phi_2)^2$ , the error should be traced back to  $\phi_2$ . Given by Eq. (A.35), the parameter  $\phi_2$  is determined by:

$$\phi_2 = \max\left[2\frac{\sqrt{k}}{0.09\omega y}, \frac{500\mu}{\rho y^2 \omega}\right]$$
(8.2)

Aside from the scalar values k and  $\omega$  and semi-empirical constants, the dimensional distance to the wall y appears both terms within the brackets of Eq.(8.2). Since it was already noticed that the error occurs in the proximity of the transition between the prism and tetrahedral cells, it is suspected that the error stems from a faulty calculation of y when switching between the last prism cell to the first tetrahedral cell.

#### **8.4.** DAMPING OF THE EDDY VISCOSITY

Next to the numerical error occurring in the outer region of the boundary layer profiles, it was also observed (Section 7.5) that the numerically obtained boundary layer is less sensitive for the adverse pressure gradient. As such, the shape of the boundary layer profile is significantly different over the aft portion of the AeroCity model. Hence, the lift coefficient is over-predicted and the onset of flow separation postponed. According to Chitsomboon and Thamthae, this is caused by an over-prediction of the eddy viscosity by the ST model. [78] When investigating the stall behaviour of wind turbine blades, it was found that the SST model postponed the onset of stall to much higher angles of attack, compared to available experimental data. In a response it is stated by the researchers that: "...these over-predictions were due to the fact that the turbulence levels (hence, turbulent eddy viscosities) in the boundary layers were too high, thus enhancing a momentum transfer to the near wall regions which helped the boundary layer to push through the adverse pressure gradient regions more easily than otherwise". In order to reduce the discrepancy between the CFD calculations and the wind-tunnel measurements, an additional damping function for the eddy viscosity, close to the wall, was proposed. The results that were achieved with the modified model are promising. Not only was it observed that the lift coefficient was on pair with the experimental results up to high angles of attack, but also the drag was in better overall agreement. Comparison with the Transition SST model showed that the regular SST model with the custom damping function performed significantly better as well.

In order to investigate if such an approach would enhance the results for the AeroCity case, especially the boundary layer profiles over the aft upper surface, a User Defined Function (UDF) is written. The UDF, included in Appendix C, allows the formulation of the eddy viscosity by Fluent to be over-ruled by a user-defined eddy viscosity. Aside from an improvement of the upper surface boundary layers, it is hoped that the dampening of the eddy viscosity close to the wall also enhances the simulation of the separation bubble over the side-plate. The damping function was slightly modified with respect to the original damping function [78] and is as follows:

$$F_{SST} = 0.09 + \left(1 - 0.09 * \tanh\left((0.03Y^{+})^{4}\right)\right) * \left(0.91 + 0.09\tanh\left((0.05Y^{+})^{8}\right)\right)$$
(8.3)

Examination of (8.3) shows that the damping function is a product of two hyperbolic tangents. Note that it is based on the  $Y^+$ . Unfortunately, in Fluent  $Y^+$  is only defined at the wall adjacent cells and not for the cells further away from the wall. In order to obtain a value for  $Y^+$  nevertheless, the non-dimensional wall distance is determined by means of an empirical relation of the  $Y^+$  and dimensional wall distance *y*. Although this is not in all cases accurate, it is believed to be sufficient for a proof of concept study. Nevertheless, for future work a more precise formulation for  $Y^+$  will be required. Using the fixed relation for  $Y^+$ , the blending function can be computed, as shown in Figure 8.14. Note how the damping function is activated rapidly in the buffer region and gradually returns to one around  $Y^+ \approx 100$ .

Before examining the effect of the damping function on the actual flow around the AeroCity model, the UDF is applied to the zero-pressure gradient flat plate case of Section 8.3. Since there is no pressure gradient involved, the results should be identical. Note that the mesh with a prism layer growth ratio r = 1.20 has been used to allow direct comparison of the results. A comparison of the velocity profile of the boundary layer is presented in Figure 8.15. The velocity, in the case of the damping function enabled, is slightly higher compared to the previous result, however the difference is considered to be negligible. Although it is still hard to see, it can be observed that the velocity profile of the case with the UDF enabled is more continuous, where as the previous result shows multiple inflection points. The final blending to the edge velocity is still abrupt for both cases however. The numerical result by NASA is not included in Figure 8.15 for reasons of clarity. However, with the velocity profile of the previous result, being slightly recessed compared to the NASA result, the current result is actually in good agreement with the velocity data provided by NASA [13].

Continuing to the distribution of the eddy viscosity ratio through the boundary layer thickness, it can be observed from Figure 8.16 that the eddy viscosity has been reduced, due to the damping of the eddy viscosity





Figure 8.14: Plot of blending function  $F_{SST}$  versus  $Y^+$  of the boundary layer at x = 0.97



Figure 8.15: Comparison of the velocity profile of the boundary layer at x = 0.97 with and without the damping function  $F_{SST}$ 



Figure 8.16: Plot of the eddy viscosity ratio along the boundary layer at x/c = 0.97 with and without the UDF activated.

Figure 8.17: Plot of the blending function  $F_2$  along the boundary layer at x/c = 0.97 of the current result with UDF enabled and data by NASA [13]

close to the wall by  $F_{SST}$ . Note that the eddy viscosity profile, obtained with the UDF activated, shows a smoother transition to the edge flow conditions compared to the result of the UDF disabled. This is in agreement with the observations of the velocity profile in Figure 8.15, where the velocity profile at the boundary layer edge was found to be more continuous and smooth for the present result. As the UDF is designed to calculate a custom eddy viscosity, the blending function  $F_2$  is required. The value for  $F_2$  was obtained by directly invoking the function concerned with the computation of  $F_2$  from the solver code. Although the value of  $F_2$  cannot be accessed from within the UI, the UDF allows insight into the behaviour of  $F_2$  by storage of  $F_2$  as a separate scalar in an allocated memory slot. A comparison of the blending function behaviour in Fluent with the data by NASA is shown in Figure 8.17. Immediately, it can be observed that the two curves do not match. The blending function, for the current result, starts to decrease from unity at a smaller distance to the wall. The reason for this is not know at this point, although the enabling of the Low Re correction and Kato-Launder correction might play a role in this case, since the  $\phi_2$  parameter is determined directly by both k and  $\omega$ . [10]

Aside from the fact that the blending function is activated closer to the wall, their behaviour is fairly similar, with only a slight deviation in the gradient of  $F_2$ . Striking is however the observation that the blending function does not appear to reduce to zero near the edge of the boundary layer. Instead, it remains non-zero well outside the boundary layer. As a consequence, the velocity outside the boundary layer is slightly higher than free-stream, with the non-dimensional velocity ratio outside the boundary layer typically being u/U = 1.01.



Figure 8.18: Plot of the calculated wall distance 'CWallDistance' and the actual wall distance y



Noticing again that prism mesh transitions to a tetrahedral mesh around y = 15 mm, the suspicion of a faulty calculation of the dimensional wall distance y is strengthened by these latest observations. The wall distance is computed by Fluent internally with a function called 'CWallDistance'. Plotting of the calculated and actual wall distance, as shown in Figure 8.18, reveals that the gradient of the curve is discontinuous at the location of the last prism layer. As a consequence, since the calculated wall distance y by Fluent suddenly increases disproportionally with respect to the local values of k and  $\omega$  (see Eq. (8.2)). Therefore, the gradient of the  $F_2$  curve is increased disproportionally as well.

Why the dimensional wall distance *y* is not calculated correctly in Fluent, at least in the present case, is not clear. Although the root of the problem is unknown, the symptoms may be weakened if the relative changes in cell size perpendicular to the wall are reduced. Possible workarounds include:

- · Reduction of the prism growth ratio will reduce the size of the prism cells
- · Limitation of the size of the tetrahedral cell directly adjacent to the prism cell
- · Changing the prism layer growth law from exponential to a linear growth of the cell height

The downside of these measures is that the total cell count is increased in each case, assuming that the other mesh parameters remain unchanged. Since it was found that reduction of the prism growth factor from r = 1.30 to r = 1.15 did not eliminate the problem, further refinement of the mesh is expected to be required. Since the computational time would even become significantly larger, this is kept as a future recommendation. Instead, it opted to investigate if the addition of the damping function  $F_{SST}$  does indeed affect the boundary layers subjected to an adverse pressure gradient.

For this purpose, the refined mesh with r = 1.15 has been used. The results are unfortunately marginal, with the lift coefficient remaining unchanged and a minor increase of the drag coefficient by  $\Delta C_D \approx 1\%$ . Hence, it does not appear that the addition of the damping function for the eddy viscosity has any significant impact on the aerodynamics of the model. Comparison of the boundary layer profiles over the aft upper surface of AeroCity confirms this impression, since the boundary layer profiles are virtually identical. In fact, the latest velocity profile shows a marginal higher velocity across the height of the boundary layer, while the opposite was expected. Hence, damping function does not appear to be effective for the current case. This may be caused by the determination of the  $Y^+$  within the UDF, which is invariant of the actual local flow. However, despite efforts to create a custom formulation for  $Y^+$ , that does take the local flow properties into account, this cannot be achieved at this point in time. Examination of the separation bubble on the side-plate of the AeroCity model, reveals that the UDF had a minor impact on the size of the separation bubble. However, the increment is very modest and not on pair with the separation bubble, as modelled by the Transition SST model.



Figure 8.20: Velocity profiles of the boundary layer at the upper surface of Aerocity at x/c=0.46 and U=65 m/s

Figure 8.21: Velocity profiles of the boundary layer at the upper surface of Aerocity at x/c= 0.65 and U = 65 m/s

Since the wall distance calculated by the Fluent algorithm remains inaccurate in the mesh transition region, the only solution is to bypass the problem. By increasing the total thickness of the prism layer mesh sufficiently, such that the discrepancy in the wall distance y is not in the blending region of the boundary layer, a smooth boundary layer profile should be obtained. This hypothesis has been investigated by creating a new mesh utilizing a prism layer of 12 mm in total thickness. A larger total thickness would be preferred, as the boundary layer grows to a larger displacement thickness. However, this increases the challenge to arrive at a decent quality mesh, given the current geometry of the model. In order to limit the growth in total number of cells, a wb-exponential growth law was adopted. The growth ratio r = 1.15 was maintained, leading to a total number of 40 prism cell layers.

The effect of the increased prism layer thickness on the boundary layer velocity profiles can be observed in Figure 8.20 and 8.21. In the previous case (h = 5mm), the velocity profiles at x/c = 0.46 and x/c = 0.65were affected most significantly by the erroneous wall distance calculation. With the increased total prism layer thickness mesh, the discontinuity appears to be no longer present. Both new velocity profiles show a smooth and gradual transition of the velocity to the boundary layer edge conditions. Hence, although the requirement of  $Y^+ \leq 300-500$  was met, a much larger prism layer thickness is required to model the boundary layer with sufficient accuracy. The adjusted boundary layers did not appear to have influenced the lift and drag coefficients, although this is impossible to verify, since the separation bubble over the side-plate was affected by the adjusted prism layer mesh. Clearly, the formation of the separation bubble is very sensitive to changes to the mesh. As such, there remains a direct dependency of the results to the mesh, which was overlooked during the initial mesh dependency investigation 6.5.

# 9

## **CONCLUSION**

The aim of this thesis project was to investigate the aerodynamic characteristics of the AeroCity model, by means of Computation Fluid Dynamics (CFD) simulation of the flow around the vehicle. The research has been a continuation of previous efforts to analyse the aerodynamic configuration of AeroCity. This included a CFD study into the main design parameters of AeroCity [1] and a separate wind-tunnel experiment, conducted in the LTT wind-tunnel of Delft University of Technology. [2] Despite the previous research, several questions and uncertainties about the validity of the results remained.

An important aspect of the project was to obtain a solid basis of validation for the numerical results. To achieve this, several test-cases were simulated with CFD and compared with experimental and reference numerical data (Chapter 5). These test-cases helped to identify the most suited methods and models for the simulation of the Wing-in-Ground (WIG) effect. More-over, it revealed the limitations of the numerical models for certain flow types, such as in the case of severe flow separation. A three-dimensional model of a WIG vehicle, showing strong resemblance with the AeroCity model, was modelled (Section 5.4) for final validation of the CFD model, before proceeding to the analysis of AeroCity.

For the analysis of the AeroCity model, the wind-tunnel experiment was replicated within the CFD model. A single configuration, for which measurements of the boundary layer and separation bubble was available, was selected for this task. In general, the predictions by the CFD model and the data obtained by the previous wind-tunnel experiment were in good agreement. However, as will be discussed in Section 9.2, the CFD predictions also deviated significantly in several aspects. Further, the influence of the boundary condition for the ground plane on the aerodynamic characteristics was examined. Similarly, a U-shaped track wall was added to the model to study the interaction of the vehicle and the track wall.

#### **9.1.** AERODYNAMIC PERFORMANCE

Comparison of the lift and drag characteristics between the CFD predictions and the wind-tunnel measurements, reveals that the predictions in the lower velocity range are in good agreement with the experimental data.[2] However, at the medium- to high velocity range, the deviation between the predictions and measured data grows significantly. Especially the drag coefficient, which is under-predicted by the CFD model by almost 20% at the highest velocity (U = 80 m/s) tested. Since the cruise speeds, envisioned for the AeroCity, are located in the upper velocity range, the drag prediction is not accurate enough for quantitative analysis.

The under-prediction of the drag coefficient, appears to be caused by an under-prediction of the separation bubble, existing on the side-plate of the AeroCity. The size of the separation bubble appears to diminish when the velocity of the free-stream is increased. Since contours of the total pressure loss indicate that the separation bubble is a mayor contributor to the overall pressure drag, this explains the decrease of the drag coefficient when the free-stream velocity was increased beyond U = 40 m/s. The underlying cause for the insufficient capture of the separation bubble, is the assumption of fully turbulent flow. Implementation of the Transition SST model yielded to a significantly larger separation bubble over the side-plate. Direct comparison of fluorescent oil film photographs, made during the previous wind-tunnel experiment, and particle

surface traces, obtained with CFD, showed a strong mutual resemblance, in both shape and size. Unfortunately, the overall result did not improve by the inclusion of a transition turbulence model, since the flow over the upper surface remained laminar too far downstream.

Despite the formation of a separation bubble, the side-plate is beneficial, as it helps to maintain the 'cushion of air' underneath the vehicle. Nevertheless, a small portion of the air is observed to flow through the gap between the ground plane and the end-plate, due to the difference in pressure on either side of the end-plate. This lateral flow component displaces the horseshoe-vortex outboard in lateral direction. As a consequence, the horseshoe-vortex is prevented to merge with the tip vortex. The latter vortex originates on the upper surface of the vehicle and grows in thickness towards the trailing edge. A reasonable amount of vortex lift is generated, as for a significant portion of span, the aft upper surface is directly submerged by the vortex.

One uncertainty for the results of the previous wind-tunnel experiment [2], was how the use of a stationary ground may have influenced the aerodynamic characteristics of the AeroCity model. Simulation of the AeroCity model, including a moving ground boundary condition, showed that the influence on the aerodynamic characteristics is small. Only a very modest improvement of the lift-to-drag ratio was observed, mainly due to a small shift of the stagnation point in the direction downstream. Although the velocity profile in the channel ,between the lower surface and the ground plane, was significantly affected by the presence of the wind-tunnel wall boundary layer, the main body of the AeroCity was not directly affected. It is believed that the influence of the ground boundary condition will become more pronounced, when the absolute elevation height is reduced. As such, inclusion of system to simulate a moving ground in the case of experimental testing should always be preferred.

Although the AeroCity is envisioned to operated on a U-shaped track, the influence of the track on the aerodynamic characteristics of the vehicle had not yet been investigated, neither for the AeroCity nor in literature. Hence, a single geometric track design was adopted and applied to the CFD model. The CFD simulations revealed that the lift-to-drag ratio had decreased, despite a modest increase of the lift coefficient. Two main causes for the significant increase of the drag could be identified. First, the horseshoe-vortex was restricted by the presence of the track wall to disperse in outboard direction. Instead, the vortex was deflected upwards and observed to merge with the tip-vortex. Second, the flow channel between the end-plate and the track wall gave rise to a significant amount of additional pressure drag. Although only a single track geometry was studied, alternative track designs may be more suitable.

#### **9.2.** VALIDATION OF THE RESULTS

Despite a successful validation of the CFD model during the test-cases (Section 5.4), the results obtained for the AeroCity model, do not meet the criteria for the lift- and drag-coefficient (Section 3.2). Aside from the separation bubble on the side-plate of the AeroCity model, which is not captured to a satisfactory extent, other discrepancies between the CFD predictions and the wind-tunnel data were found. Most noticeably, is the comparison of the boundary layer profiles over the aft portion of the upper surface, with the experimental results. Although the absolute displacement thickness's are very much comparable, the shape factor of the boundary layers are not.

During the wind-tunnel experiment [2], it was measured that the boundary layer near the trailing edge was on the verge of separation, whereas the CFD prediction reveals a boundary layer seemingly indifferent of the adverse pressure gradient. Implementation of a damping function for the eddy viscosity, with the aim to make the boundary layer more susceptible for flow separation, did not improve the result. Although no actual flow separation over the upper surface was observed during the wind-tunnel experiment, the limitation of the CFD model to predict boundary layer separation might give rise to a further under-prediction of the drag, for example when the angle of attack is increased. Adding to the uncertainty was a numerical error, in the region where the boundary layer was blended with the boundary layer edge conditions. In the end, it was discovered that this was caused by a faulty calculation of the absolute wall distance by the Fluent algorithms. Although the numerical artefact did not appear to influence the overall results, a solution was explored. By increasing the total thickness of the prism layer mesh, the problem was successfully overcome.

#### **9.3.** RECOMMENDATIONS

Summarizing the above, it can be concluded that the current results have attributed to the knowledge about the aerodynamic characteristics of AeroCity and that the current CFD model is able to predict many aspects of the flow around the AeroCity sufficiently. However, in order to gain further insight into the aerodynamic characteristics and move forward with the design, both the aerodynamic model of the AeroCity and the numerical model should be improved.

For the the aerodynamic model of AeroCity, the following improvements are suggested:

- The shape of the side-plate should be adapted, to prevent the formation of a laminar separation bubble. Preferably, one should adopt a half-profile of the main wing airfoil. The thickness-to-chord ratio should be kept low, as well as maintaining a straight profile on the pressure side of the side-plate. This approach was taken by Kumar [17]. As was shown in Section 5.4, no separation bubble was predicted for this model.
- The elevation height of the model should be reduced for future research. The current level for the liftto-drag ratio is only moderate, being in the order of L/D= 12 - 14. Operating closer to the ground, for instance  $h/c \le 0.025$  should increase the aerodynamic efficiency. Particularly, if the separation bubble on the side-plate can be diminished by alteration of the side-plate design.
- The height-to-chord ratio of the current AeroCity model was actually equal to h/c= 0.065. Note that this is higher than the h/c= 0.05 as stated by Nasrollahi [2] and throughout this report. However, the reference point used by Nasrollahi is located below the trailing edge and as such under-estimates the height-to-chord ratio. Since the same CAD model was used, this does not subtract from the credibility of the results.
- Careful shaping, of the transition between the upper surface and the side-plate plate, may weaken the strength of the tip vortex. By applying a more gradual and smooth curvature, the air flowing from the side-plate towards the upper surface will be accelerated less. However, this will decrease the interior volume of AeroCity. Alternatively, the side-plates can be extended in height, after the location of maximum thickness, to offset the tip-vortex from the upper surface of the main wing. Nevertheless, this will introduce additional friction drag and may give rise to interference problems.
- The use of a trailing edge flap, as proposed by Nouwens [1], should be investigated in more detail. The trailing edge flap should increase the lift, by artificially reducing the height-to-chord ratio. Ideally, this should be combined with a study of the dynamic stability characteristics of the AeroCity. By utilizing the flap as a means to adjust the amount of generated lift, vertical oscillations could be dampened. More-over, it will allow for vertical trim of vehicle in case of various loading conditions.
- The design of the track should be given more attention. As shown in Section 7.7, the current track design has a negative effect on the aerodynamic efficiency. Hence, both the vehicle and the track design should be optimized in order to minimize the negative effects of the track on the aerodynamic performance of AeroCity. Other track designs, such as a guard-rail, are advised to also be investigated. Incorporation of the lateral control system should ideally be incorporated during this analysis.

Aside from the recommendations regarding the AeroCity concept, there are several recommendations to improve the CFD model. Although, with the current CFD model and methods, the aerodynamic characteristics of the AeroCity have been investigated partially successful, improvement of the predictions are deemed necessary to use the CFD model for further analysis.

- Design a hybrid mesh, consisting of hexahedral and tetrahedral cells to enhance the computational efficiency. Use the hexahedral cells to fill the large volumes in the computational domain and apply the tetrahedral cells only near the body or in regions of strong curvature.
- A more elaborate study with transition turbulence models in combination with surface roughness parameters might result in better results. By controlling the surface roughness of the model, the location of boundary layer transition may be influenced. As such, the transition location on the upper surface and the side-plate may be controlled individually.

- Improve the quality of the mesh. Although the overall quality of the mesh, after multiple smoothing iterations, was reasonably good, the mesh quality could be improved on a local level. Especially the area between the bottom of the end-plate and the ground plane could require additional attention.
- Refinement of the prism layer mesh, by reducing the prism layer growth factor to r≤ 1.05 should increase the resolution of the boundary layer. A minor improvement of the skin friction drag prediction is to be expected. Furthermore, it might improve the sensitivity of the numerical boundary layer to the adverse pressure gradient.
- Increase the mesh density in the region of the separation bubble. By locally refining the volume of the cells, the prediction of the separation bubble may be improved, without increasing the total cell count excessively.
- Use of a different software package, other than Fluent, is recommended. Although the interface is userfriendly, more efficient or precise CFD codes may be available for the computation of aerodynamic flows.

# A

### **TURBULENCE MODELS**

In this Appendix, the theory behind a selection of the turbulence models, used throughout this project, are discussed in more detail.

#### A.1. SPALART-ALLMARAS

The Spalart-Allmaras [48] is a one-equation turbulence model that has specifically been designed for aerospace applications. It utilizes a modelled transport equation of the eddy viscosity. The transport equation is given by:

$$\frac{\partial}{\partial t} \left( \rho \tilde{v} \right) + \frac{\partial}{\partial x_i} = \frac{1}{\sigma_{\tilde{v}}} \left[ \frac{\partial}{\partial x_j} \left( \left( \mu + \rho \tilde{v} \right) \frac{\partial \tilde{v}}{\partial x_j} \right) + C_{b2} \rho \left( \frac{\partial \tilde{v}}{\partial x_j} \right)^2 \right] + C_{b1} \rho \tilde{S} \tilde{v} - C_{w1} \rho f_w \left( \frac{\tilde{v}}{d} \right)^2$$
(A.1)

,where  $\tilde{v}$  is equal to the eddy viscosity  $\mu$ , except at the near wall region. The last two terms of eq. (A.1) are the terms for turbulent production and destruction respectively. The turbulent viscosity is modelled by:

$$\mu_t = \rho \tilde{\nu} f_{\nu 1} \tag{A.2}$$

and the scalar quantity  $\tilde{S}$  for the rate of strain:

$$\tilde{S} \equiv S + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{\nu 2} \tag{A.3}$$

, where  $f_{v1}$  and  $f_{v2}$  are coefficients accounting for viscous damping, equated by:

$$f_{\nu 1} = \frac{\chi^3}{\chi^3 + C_{\nu 1}^3}, \quad f_{\nu 2} = 1 - \frac{\chi}{1 + \chi f_{\nu 1}}, \quad \chi = \frac{\tilde{\nu}}{\nu}$$
(A.4)

Furthermore, *d* is the distance to the wall and *S* is a based on the local vorticity magnitude. In the original model by Spalart and Allmaras [48] it is given by:

$$S = \sqrt{2 \,\Omega_{ij} \Omega_{ij}} \tag{A.5}$$

However, it was discovered that only including the generation of vorticity in the turbulence production term was not accurate, as one should also include the effect of the mean strain rate of the flow. Hence, the improved formulation of *S* by Bradshaw *et al* [49] is given by:

$$S \equiv |\Omega_{ij}| + C_{prod} \min\left(0, |S_{ij}| - |\Omega_{ij}|\right)$$
(A.6)

,where the magnitude of the strain rate and rotation tensor are given by:

$$|\Omega_{ij}| = \sqrt{2\Omega_{ij}\Omega_{ij}}, \quad |S_{ij}| = \sqrt{2S_{ij}S_{ij}}$$
(A.7)

The definition for *S* of eq. (A.6) is adopted by Fluent [10]. The values of the coefficients used are as follows:

$$C_{b1} = 0.1355, \quad C_{w1} = \frac{C_{b1}}{\kappa^2} + \frac{(1+C_{b2})}{\sigma_{\tilde{\nu}}} \quad C_{w3} = 0.3 \quad \sigma_{\tilde{\nu}} = \frac{2}{3}$$

$$C_{b2} = 0.662 \quad C_{w2} = 0.3 \quad C_{prod} = 2.0 \quad C_{v1} = 7.1 \quad \kappa = 0.4187$$
(A.8)

#### **A.2.** $k - \epsilon$ STANDARD

The  $k - \epsilon$  turbulence model focusses on the mechanisms that affect the turbulence kinetic energy. The instantaneous kinetic energy of a turbulent flow k(t) is the sum of the mean kinetic energy K and the turbulent kinetic energy k.

$$k(t) = K + k \tag{A.9}$$

,where the kinetic energy is simply given by:

$$K = \frac{1}{2} \left( U^2 + V^2 + W^2 \right)$$

$$k = \frac{1}{2} \left( \overline{u'}^2 + \overline{v'}^2 + \overline{w'}^2 \right)$$
(A.10)

Because of viscous effects, turbulent flows are dissipative. Kinetic energy is converted to heat due to shear stresses acting on the flow. Hence, if no energy is supplied to the flow, the flow disturbances or eddies die out quickly. The rate at which the turbulence dissipates is called the dissipation rate  $\epsilon$ . To obtain values for k and  $\epsilon$  two separate equations need to be solved. First, the equation for the kinetic energy K is treated. The governing equation for K can be found [50] to be:

$$\frac{\partial(\rho K)}{\partial t} + \frac{\partial}{\partial x_i}(\rho K U) = \frac{\partial}{\partial x_j} \left( -PU + 2\mu U E_{ij} - \rho U \overline{u'_i u'_j} \right) - 2\mu S_{ij} \cdot S_{ij} - \left( -\rho \overline{u'_i u'_j} \cdot S_{ij} \right)$$
(A.11)

,where in the above expression  $S_{ij}$  represents the mean rate of deformation tensor. On the left hand-side of the above expression entails the rate of change of *K* plus the transport of *K* by convection. The right hand-side term is more elaborate. The first term is equal to the transport of *K* by pressure and viscous stresses minus the transport of *K* by Reynolds stresses. The next term is the rate of dissipation of *K*, while the last term describes the production of turbulence. Hence, in the case of turbulence, kinetic energy of the mean flow it transitioned into kinetic turbulent energy. The equation of *k* is very similar:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_i}(\rho kU) = \frac{\partial}{\partial x_j} \left( -\overline{p'u'} + 2\mu \overline{u'S_{ij}} - \rho \overline{u_i u'_i u'_j} \right) - 2\mu \overline{s'_{ij} \cdot s'_{ij}} + \left( -\rho \overline{u'_i u'_j} \cdot E_{ij} \right)$$
(A.12)

,where  $s'_{ij}$  is the fluctuating component rate of deformation tensor. Note the plus sign for the last term on the right hand-side. The above expression contains additional fluctuating terms, such as  $\overline{p'}$ , which are unknown. Instead, often a more simplified model equation for k is used:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + 2\mu_t S_{ij} \cdot S_{ij} - \rho \epsilon$$
(A.13)

The left hand-side remained unchanged, however the right hand-side now states that the change increase of k should be equal to the diffusive transport of k plus the rate of production of k minus the rate of destruction. The Prandtl number  $\sigma_k$  connects the diffusivity of k to the turbulent viscosity  $\mu_k$ . Typically, a value of 1.0 is used. [51]

The dissipation of k is described by the sixth term of eq. (A.12), as this captures the work done by the smallest eddies against the viscous stresses. One can now define the dissipation rate per unit mass  $\epsilon$  as:

$$\epsilon = 2\nu \overline{s'_{ij}s'_{ij}} \tag{A.14}$$

The analytical equation is fairly long and contains some unknown higher order terms and cannot be solved. Instead, a simplified model equation is derived by multiplication of the *k* equation with  $\epsilon/k$ , which results to:

$$\frac{\partial(\rho\epsilon)}{\partial t} + \frac{\partial}{\partial x_i}(\rho\epsilon U) = \frac{\partial}{\partial x_j} \left[ \frac{\mu_t}{\sigma_\epsilon} \frac{\partial\epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} 2\mu_t S_{ij} \cdot S_{ij} - C_{2\epsilon} \rho \frac{\epsilon^2}{k}$$
(A.15)

The Prandtl number  $\sigma_{\epsilon}$  connects the diffusivity of the dissipation  $\epsilon$  to the eddy viscosity  $\mu_t$ . Typically, a value of 1.30 is used. [51]. The model constants  $C_{1\epsilon}$  and  $C_{2\epsilon}$  have typically values of 1.44 and 1.92. Once the local values for  $\epsilon$  and k are known, the turbulent viscosity is calculated by:

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \tag{A.16}$$

,where  $C_{\mu}$  is often set equal to 0.09. [51] In order to solve the model equations for k and  $\epsilon$  one must specify the initial values for at the inlet boundary of the domain. The turbulence kinetic energy is usually prescribed as:

$$k_{\infty} = 1.5 \left( T_i U_{\infty} \right)^2 \tag{A.17}$$

,where  $T_i$  is the free stream turbulence intensity. Depending on the flow application, the free stream turbulence can vary anywhere between  $T_i = 0.0001$  to 10 percent of the mean velocity. Similarly, the turbulent viscosity is specified as a percentage of the free stream viscosity. Usually, the initial turbulent conditions are only specified at the inlet of the domain, as the values at the outlet are obtained by extrapolation. At a solid surface, the values for k,  $\epsilon$  and  $\mu_t$  are set equal to zero.

The advantage of the standard  $k - \epsilon$  model is that is relatively simple to implement and leads to stable, relatively fast converging solutions. On the downside, it is not able to predict solutions for flows with a strong degree of rotation and separation very well. [11] This is partially caused by the fact that the equations of the standard model become numerically unstable when integrated to the wall. [51] For such cases, a separate model should be used to capture the turbulent behaviour of the flow near a wall. This is discussed separately in Section 4.4. Furthermore, the model assumes fully turbulent flow. As such, it is only relevant for high *Re* flows. Various adaptations of the standard  $k - \epsilon$  model have been proposed which add Reynolds number dependency functions  $f_1$  and  $f_2$  to the last two terms of eq. A.15, plus introduce an extra term to the k and  $\epsilon$  equations to account for the fact that the dissipation process may not be isotropic. [51].

#### **A.3.** *k* − *ε* RNG

An improved version of the standard  $k-\epsilon$  model was introduced by Yakhot et al. [52] The underlying principle of this improved model is that the RANS equations are re-normalized to account for the effect of smaller scales of eddies. In the standard model the eddy viscosity  $\mu_t$  is determined for a single turbulence length scale. This implies that the turbulent diffusion only occurs at the pre-described length scale and ignores the effect of other scales of motion. To re-normalize the RANS equations, a mathematical statistical technique called Renormalization Group Method (RNG) is applied. The RNG procedure expresses the effect of the small scales of motion in terms of larger scale motions. [11] The mathematics behind the derivation is very elaborate and will not be discussed here. Instead, the final formulation for high Reynolds number flows derived by Yakhot et al are presented below:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_i}(\rho k U) = \frac{\partial}{\partial x_j} \left[ \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right] + \tau_{ij} \cdot S_{ij} - \rho \epsilon$$

$$\frac{\partial(\rho \epsilon)}{\partial t} + \frac{\partial}{\partial x_i}(\rho \epsilon U) = \frac{\partial}{\partial x_j} \left[ \alpha_\epsilon \mu_{eff} \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon}^* \frac{\epsilon}{k} \tau_{ij} \cdot S_{ij} - C_{2\epsilon} \rho \frac{\epsilon^2}{k}$$
(A.18)

,where  $\tau_{ij}$  is the Reynolds Stress term according to the Boussinesq hypothesis [47]:

τ

$$a_{ij} = \rho \overline{u'_j u'_i} = 2\mu_t S_{ij} - \frac{2}{3}\rho k \delta_{ij}$$
(A.19)

Note that in eq. (A.18) the effective viscosity is defined, given by:

$$\mu_{eff} = \mu + \mu_t \qquad \mu_t = \rho C_\mu \frac{k^2}{\epsilon} \tag{A.20}$$

The following coefficients are to be used [11]:

$$C_{\mu} = 0.0845 \quad \alpha_k = \alpha_{\epsilon} = 1.39 \quad C_{1\epsilon} = 1.42 \quad C_{2\epsilon} = 1.68$$
 (A.21)

Furthermore:

$$C_{1\epsilon}^{*} = C_{1\epsilon} - \frac{\eta(1 - \eta/\eta_{0})}{1 + \beta\eta^{3}} \quad , \eta = \frac{k}{\epsilon}\sqrt{2S_{ij} \cdot S_{ij}} \quad \eta_{0} = 4.377 \quad \beta = 0.012$$
(A.22)

Note that in the above expression, only the value of  $\beta$  is adjustable. The current value is obtained from near wall turbulence data. [11] All other values are to be computed by the RNG process. With the standard  $k - \epsilon$  model, it is believed that the  $\epsilon$  model equation is one of the causes for limited accuracy in the case of flows that experience large strain rates. The strain-dependent correction term for the  $C_{1\epsilon}$  constant tries to capture the interaction between the mean shear and the turbulence dissipation  $\epsilon$ . Indeed, Yakhot et al [52] showed that the improved  $k - \epsilon$  model is able to obtain good predictions for flow over a backward facing step. Other users have also experienced improved performance for flows with a high streamline curvature and strain rate. Also, transition is reported to be capture better. [11]. However, some of the limitations of the standard are maintained for the RNG model as well.

#### **A.4.** $k - \epsilon$ REALIZABLE

Like the RNG  $k - \epsilon$  model, the Realizable  $k - \epsilon$  focusses on improving the turbulent dissipation  $\epsilon$  equation. The governing equation for the turbulent kinetic energy k is however identical to the standard  $k - \epsilon$  model. It also entails a new formulation for the turbulent viscosity. The Realizable turbulence model is developed by Shih et al. [53] The term 'realizable' means that the satisfies the mathematical and physical constraints on the Reynold stresses, which is not the case for the standard or RNG  $k - \epsilon$  model equations. To understand this, one should combine the Boussinesq relation of eq. (A.19) and the expression for the eddy viscosity, given by eq. (A.20). By doing so, one obtains the following relation for the normal Reynolds stress in an incompressible strained mean flow [10].

$$\overline{u^2} = \frac{2}{3}k - 2v_t \frac{\partial U}{\partial x} \tag{A.23}$$

,where the relation  $v_t = \mu_t / \rho$  was used to obtain the above result. It can now be shown [10] that the normal stress  $\overline{u^2}$ , which is by definition a positive quantity, can become negative when:

$$\frac{k}{\epsilon}\frac{\partial U}{\partial x} > \frac{1}{3C_{\mu}} \approx 3.7 \tag{A.24}$$

Hence, the system becomes "non-realizable". The Realizable  $k - \epsilon$  model fixes this by instead of having a fixed coefficient  $C_{\mu}$ , making it into a variable:

$$C_{\mu} = \frac{1}{A_0 + A_s \frac{kU^*}{c}}, \quad U^* = \sqrt{S_{ij} S_{ij} + \overline{\Omega_{ij} \Omega_{ij}}}$$
(A.25)

,where  $\overline{\Omega_{ij}}$  is the mean rotation tensor viewed from a rotating reference frame with an angular velocity  $\omega_k$ . The constants  $A_0$  and  $A_s$  are functions of the velocity gradients. For briefness, the exact formulations are not shown here. It can be shown that the eddy viscosity, described by eq. (A.25), is equal to the standard value of 0.09 in the case of the inertial (log-law) layer in a equilibrium boundary layer. The inertial sub-layer is a part of the turbulent boundary layer where the velocity gradient in is governed solely by turbulent momentum exchange, as the direct effect of viscosity has become negligible. With the improved formulation for the eddy viscosity the new transport equation for the turbulence dissipation rate is :

$$\frac{\partial(\rho\epsilon)}{\partial t} + \frac{\partial}{\partial x_i}(\rho\epsilon U) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial\epsilon}{\partial x_j} \right] + \rho C_1 S\epsilon - \rho C_2 \frac{\epsilon^2}{k + \sqrt{v\epsilon}}$$
(A.26)

In the above equation any additional source terms and the effect of buoyancy are ignored. The second term on the right-hand side is the generation of  $\epsilon$  and the last term describes the destruction of  $\epsilon$ . Note that the destruction term does not have a singularity if k vanishes (e.g. k = 0), in contrast to the previous  $k - \epsilon$  models. Also, the production term does not involve k, which is believed to better representation of the energy transfer [10]. The Realizable has been shown to produce more accurate results [53] for a wide variety of flows, including separated and boundary layer flows [11].

#### A.5. $k - \omega$ BSL

Next to the various  $k - \epsilon$  models, the  $k - \omega$  Baseline (BSL) model developed by Wilcox [55] uses the transport equation for k. The second model equation of the  $k - \omega$  model describes the transport of  $\omega$ , the specific turbulent dissipation rate, sometimes also referred to as the turbulent frequency. Instead of using the governing equation of fluid motion, the equation for  $\omega$  is constructed based on the known physical process of turbulence. The process of turbulent dissipation takes place at the level of smallest eddies. However, the rate of dissipation  $\epsilon$  is rate of energy transfer between the turbulent kinetic energy to the smallest eddies. Since the larger eddies determine the turbulent kinetic energy, the dissipation rate  $\epsilon$  is set by the properties of the large eddies. The specific turbulent dissipation rate  $\omega$  can thought of as the ratio of turbulent dissipation rate to the turbulent mixing energy or the rate of dissipation of turbulence per unit energy. The  $k - \epsilon$  equations are transformed into a  $k - \omega$  formulation by using:

$$\epsilon = \beta^* \omega k \tag{A.27}$$

Using the above transformation, the new transport equations for the  $k - \omega$  model can be derived to be: [55]:

$$\frac{\partial(\rho\omega)}{\partial t} + \frac{\partial}{\partial x_i}(\rho\omega U) = \frac{\partial}{\partial x_j} \left[ (\mu + \sigma\mu_t) \frac{\partial\omega}{\partial x_i} \right] + \alpha \frac{\omega}{k} \tau_{ij} \nabla U - \beta \rho \omega^2$$
(A.28)

and

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho k U) = \frac{\partial}{\partial x_j} \left[ (\mu + \sigma^* \mu_t) \frac{\partial k}{\partial x_i} \right] + \tau_{ij} \nabla U - \beta^* \rho \omega k$$
(A.29)

with

$$\alpha = \frac{5}{9} \quad \beta = \frac{3}{40} \quad \beta^* = \frac{9}{100} \quad \sigma = \frac{1}{2} \quad \sigma^* = \frac{1}{2} \tag{A.30}$$

Note that the term  $\tau_{ij} \nabla U$  is the production of turbulent kinetic energy *k*. The  $k-\omega$  has been reported to show improved accuracy for various complex flows, such as an attached boundary layer flow under an adverse pressure gradient [55]. Moreover, the turbulence model can be integrated through the viscous sub-layer, removing the need for additional wall functions.

#### A.6. $k - \omega$ SST

The previous discussed standard  $k-\omega$  model has shown to outperform the  $k-\epsilon$  turbulence models in laminar sub-layers and the logarithmic region of the boundary layer [55]. However, the model appears to be strongly influenced by the specified free stream value of  $\omega$  outside the boundary layer [51]. As such, the  $k-\omega$  model is not very suited for boundary layer wake flow. The  $k-\epsilon$  model on the other hand is quite accurate for these regions of the flow [51]. By combing the two models on can add the strengths and remove the weaknesses of each individual model. This was done by Menter [54], by developing the  $k-\omega$  Shear Stress Transport (SST) model. The model name stems from the modified turbulent viscosity equation, which has been modified to account for the transport of the principal turbulent shear stress. The  $k-\omega$  equations are multiplied by a blending function  $F_1$  and the equations of  $k-\epsilon$  by  $(1-F_1)$ .

$$\frac{\partial(\overline{\rho}\omega)}{\partial t} + \frac{\partial}{\partial x_i}(\overline{\rho}\omega u_i) = \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_{\omega 2}\mu_t) \frac{\partial \omega}{\partial x_j} \right] + \alpha_2 \frac{\omega}{k} \tau_{ij} \nabla U - \beta_2 \overline{\rho} \omega^2 + 2(1 - F_1) \overline{\rho} \sigma_{\omega 2} \frac{1}{\omega} \nabla k \nabla \omega$$
(A.31)

and

$$\frac{\partial(\overline{\rho}k)}{\partial t} + \frac{\partial}{\partial x_i}(\overline{\rho}ku_i) = \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_{k2}\mu_t) \frac{\partial k}{\partial x_i} \right] + \tau_{ij}\nabla U - \beta^* \overline{\rho}\omega k$$
(A.32)

Note that an additional term has emerged in the  $\omega$  transport equation, which can be viewed as a crossdiffusion term. The following coefficients have been introduced in eq. (A.31) and (A.32):

$$\sigma_{k2} = \frac{1}{\sigma_k} \quad \sigma_{\omega 2} = \frac{1}{\sigma_\epsilon} \quad \beta_2 = \beta^* (C_{\epsilon 2} - 1) \quad \beta^* = C_\mu \quad \alpha_2 = (C_{\epsilon 1} - 1)$$
(A.33)

In the combined equation of eq.(A.31)  $F_1$  is the so-called blending function. It is designed such that near the wall the function  $F_1$  is equal to one, such that the  $k - \omega$  model is used, and equates to zero away from the wall. in the latter case, the  $k - \epsilon$  equations are used. In compact form, the constant used in eq. (A.32) are expressed as:

$$\phi = F_1 \phi_1 + (1 - F_1) \phi_2 \tag{A.34}$$

,where  $\phi_1$  represents the constants associated with the  $k - \omega$  model ( $F_1 = 1$ ) and  $\phi_2$  the constants of the  $k - \epsilon$  model ( $F_1 = 0$ ). The blending function  $F_1$  itself is given by the following relation:

$$F_{1} = \tanh(\phi_{1}^{4})$$

$$\phi_{1} = \min\left[\max\left(\frac{\sqrt{k}}{0.09\omega y} \frac{500\mu}{\rho y^{2}\omega}\right), \frac{4\rho k}{\sigma_{\omega,2}D_{\omega}^{+}y^{2}}\right] , \quad \phi_{2} = \max\left[2\frac{\sqrt{k}}{0.09\omega y}, \frac{500\mu}{\rho y^{2}\omega}\right]$$
(A.35)

, where  $D_{\omega}^+$  is the positive part of the cross-diffusion term given by:

$$D_{\omega}^{+} = \max\left[2\rho \frac{1}{\sigma_{\omega,2}} \frac{1}{\omega} \frac{\partial k}{\partial x_{j}} \frac{\partial \omega}{\partial \omega} \partial x_{j}, 10^{-10}\right]$$
(A.36)

A new eddy viscosity was derived to be able to handle adverse pressure gradient flows better for the new model. However, since the new formulation does not hold for the complete flow field, it is blended with the eddy viscosity term used by the  $k - \epsilon$  model.

$$\mu_t = \frac{\rho k}{\omega} \frac{1}{\max\left[\frac{1}{\alpha^*}, \frac{SF_2}{\alpha_1 \omega}\right]}, \quad F_2 = \tanh\left(\phi_2^2\right)$$
(A.37)

,where  $\alpha^*$  is a coefficient that damps the eddy viscosity in low *Re* flow and *S* the magnitude of the strain rate.

#### A.7. REYNOLD STRESS MODEL

The Reynolds Stress Model (RSM) calculates the individual Reynold's stresses  $u'_i u'_j$  using six differential transport. The exact form of the Reynold stress transport equations are given by:

$$\frac{\partial}{\partial t}(\rho \overline{u'_i u'_j}) + \frac{\partial}{\partial x_k}(\rho u_k \overline{u'_i u'_j}) = D_{T,ij} + D_{L,ij} + P_{ij} - G_{ij} + \phi_{ij} - \epsilon_{ij} - F_{ij}$$
(A.38)

,where the turbulent diffusion  $D_{T,ij}$  and the molecular diffusion  $D_{L,ij}$  are given by:

$$D_{T,ij} = \frac{\partial}{\partial x_k} \left[ \rho \overline{u'_k u'_j u'_i} + \overline{p(\delta_{kj} u'_i + \delta_{ik} u'_j)} \right]$$
  

$$D_{L,ij} = \frac{\partial}{\partial x_k} \left[ \mu \frac{\partial}{\partial x_k} (\overline{u'_j u'_i}) \right]$$
(A.39)

, the stress production  $P_{ij}$  and the buoyancy production  $G_{ij}$  by:

$$P_{ij} = -\rho \left( \overline{u'_i u'_k} \frac{\partial u_j}{\partial x_k} + \overline{u'_j u'_k} \frac{\partial u_i}{\partial x_k} \right)$$

$$G_{ij} = -\rho \beta \left( g_i \overline{u'_j \theta} + g_j \overline{u'_i \theta} \right)$$
(A.40)

Further, the pressure strain  $\phi_{ij}$  and the turbulent dissipation  $e_{ij}$  are given by:

$$\phi_{ij} = \overline{p\left(\frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i}\right)}$$

$$\epsilon_{ij} = 2\mu \frac{\overline{\partial u'_i}{\partial x_k} \frac{\partial u'_j}{\partial x_k}}{\partial x_k}$$
(A.41)

and finally  $F_{ij}$ , the production of turbulence due to system rotation is given by:

$$F_{ij} = 2\rho\Omega_k \left( \overline{u'_j u'_m} \epsilon_{ikm} + \overline{u'_i u'_m} \epsilon_{jkm} \right)$$
(A.42)

Of the various terms stated above,  $D_{L,ij}$ ,  $P_{ij}$  and  $F_{ij}$  are exact formulations and do not require any modelling. However, the expressions for  $D_{T,ij}$ ,  $G_{ij}$ ,  $\phi_{ij}$  and  $\epsilon_{ij}$  require some assumptions and simplifications in order to close the formulations.

To model the transport of the turbulent diffusivity, the following simplified expression [79] is used by Fluent [10]:

$$D_{T,ij} = \frac{\partial}{\partial x_k} \left( \frac{\mu_t}{\sigma_k} \frac{\partial \overline{u'_j u'_i}}{\partial x_k} \right)$$
(A.43)

The eddy viscosity  $\mu_t$  is determined by by eq. (A.16), similar as computed in the  $k - \epsilon$  models. The pressure strain term can be modelled in various ways. In Fluent, one can either opt for the linear [56] or the quadratic [80] pressure strain model. In the case of the linear pressure strain model by Launder, an additional low *Re* formulation needs to be adopted for near-wall flows. Since the quadratic formulation by Speziale [80] has been shown to be more effective, for example in predicting rotating shear flow [80] the latter is explained in more detail. The formulation derived by Speziale is given by:

$$\phi_{ij} = -\rho (C_1 \rho \epsilon + C_1^* P) b_{ij} + C_2 \rho \epsilon \left( b_{ij} b_{kj} - \frac{1}{3} b_{mn} b_{mn} \delta_{ij} \right) + \left( C_3 - C_3^* \sqrt{b_{ij} b_{ij}} \right) \rho k \$_{ij} + C_4 \rho k \left( b_{ik} S_{jk} + b_{jk} S_{ik} - \frac{2}{3} b_{mn} S_{mn} \delta_{ij} \right) + C_5 \rho k \left( b_{ik} \Omega_{jk} + b_{jk} \Omega_{ik} \right)$$
(A.44)

,with the following coefficients:

$$C_1 = 3.4$$
  $C_1^* = 1.8$   $C_2 = 4.2$   $C_3 = 0.8$   $C_3^* = 1.3$   $C_4 = 1.25$   $C_5 = 0.4$  (A.45)

Further, in eq. (A.44), the Reynolds-stress anisotropic tensor  $b_{ij}$  is defined as:

$$b_{ij} = -\left(\frac{-\rho \overline{u'_i u'_j} + \frac{2}{3}\rho k\delta_{ij}}{2\rho k}\right)$$
(A.46)

The formulations for the mean strain rate tensor  $S_{ij}$  and the mean rate-of-rotation tensor  $\Omega_{ij}$  are given by the terms of eq. (4.5). Note that in the above formulation,  $\delta_{ij}$  is the Kronecker delta, being either  $\delta_{ij} = 1$  if i = j or  $\delta_{ij} = 0$  if  $i \neq j$ . Further, the term  $g_i$  is the local component of the gravitational vector in the direction of the flow component. As one can observe from eq. (A.44), the turbulent kinetic energy k and the turbulent dissipation rate  $\epsilon$  occur in the expression. The turbulent kinetic energy is obtained by:

$$k = \frac{1}{2}\overline{u'_i u'_j} \tag{A.47}$$

,which is modelled in this case by Fluent [10]:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + \frac{1}{2} \left( P_{ij} + G_{ij} \right) - \rho \epsilon \left( 1 + 2M_t^2 \right)$$
(A.48)

,where  $\sigma_k = 0.82$  and  $M_t = \sqrt{k/a^2}$  the turbulent Mach number. The  $\epsilon$  equation, a similar expression as used in the  $k - \epsilon$  model is used. Modelling of the buoyancy term is obtained by assuming the use of an ideal gas:

$$G_{ij} = -\frac{\mu_t}{\rho\sigma} \left( g_i \frac{\partial \rho}{\partial x_j} + g_j \frac{\partial \rho}{\partial x_i} \right)$$
(A.49)

,where  $g_i$  is the component of the gravitational vector g in the direction of the local velocity component. In terms of boundary conditions the RSM requires input values for the Reynolds stresses  $\overline{u'_i u'_j}$  and the turbulent dissipation rate  $\epsilon$  at the inlet of the computational domain. In Fluent, these can be specified directly or by means of turbulence intensity. At the walls, the Reynolds stresses and  $\epsilon$  are computed in Fluent by use of wall functions (described in Section 4.4). By assuming equilibrium and using the log-law [81], the Reynolds stresses at the cells near the wall are computed by:

$$\frac{\overline{u_{\tau}^{\prime 2}}}{k} = 1.098, \quad \frac{\overline{u_{\eta}^{\prime 2}}}{k} = 0.247, \quad \frac{\overline{u_{\lambda}^{\prime 2}}}{k} = 0.655, \quad -\frac{\overline{u_{\tau}^{\prime 2} u_{\eta}^{\prime 2}}}{k} = 0.255$$
(A.50)

Note that for the above expression, a local coordinate system has been introduced, where  $\tau$  is the tangential coordinate,  $\eta$  the normal coordinate and  $\lambda$  the bi-normal coordinate. The value for  $\kappa$  is obtained by solving eq. (A.48)

#### **A.8.** WALL FUNCTIONS

Since the standard (equilibrium) wall function are not very suited for modelling a turbulent boundary layer with experiencing an adverse pressure gradient, only the non-equilibrium wall function implemented in Fluent is discussed here in more detail. The implemented model is a two-layer model developed by Kim and Choudhury [82]. Compared to the standard wall function, it has been adopted to be sensitized for pressure-gradient effects. The modified log-law formulation is summarized as follows [10]:

$$\frac{\tilde{U}C_{\mu}^{1/4}k^{1/2}}{\tau_{w}/\rho} = \frac{1}{\kappa}\ln\left(E\,\frac{\rho C_{\mu}^{1/4}k^{1/4}y}{\mu}\right) \tag{A.51}$$

,where the pressure-gradient adjusted mean velocity  $\tilde{U}$  is given by:

$$\tilde{U} = U - \frac{1}{2} \frac{dp}{dx} \left[ \frac{y_{\nu}}{\rho \kappa \sqrt{k}} \ln\left(\frac{y}{y_{\nu}}\right) + \frac{y - y_{\nu}}{\rho \kappa \sqrt{k}} + \frac{y_{\nu}^2}{\mu} \right]$$
(A.52)

,with  $y_v$  being the physical viscous sub-layer thickness given by:

$$y_{\nu} \equiv \frac{\mu y_{\nu}^{*}}{\rho C_{\mu}^{1/4} k_{p}^{1/2}}$$
(A.53)

,where  $y_v^* = 11.225$  is the dimensionless thickness of the viscous sub-layer, or the distance  $y^+$  from the wall at which the log-law region starts in Fluent. Note that this is significantly earlier than the  $y^+$  values between 30 - 60 as used for the original log-law equation.

The non-equilibrium wall function uses the two-layer concept for the determination of the turbulence kinetic energy k at the wall-neighbouring cells. These are assumed to consist of a viscous sub-layer and a fully turbulent region. The following assumptions are made:

$$\tau_{t} = \begin{cases} 0, & y < y_{v}, \\ 0, & y > y_{v}, \end{cases} \quad k = \begin{cases} \left(\frac{y}{y_{v}}\right)^{2} k_{p} & y < y_{v} \\ k_{p} & y > y_{v}, \end{cases} \quad \epsilon = \begin{cases} \frac{2vk}{v^{2}} & y < y_{v} \\ \frac{k^{3/2}}{C_{\ell}y} & y > y_{v} \end{cases}$$
(A.54)

,where  $C_{\ell}$  is a coefficient for the turbulence length scale given by  $C_{\ell} = \kappa C_{\mu}^{-3/4}$ . Note that  $k_p$  is the kinetic energy at a certain point p in the flow. With these assumptions about the profiles of k,  $\epsilon$  and  $\tau_t$  the cell-averaged production of k, denoted by  $\overline{G_k}$  and cell-averaged dissipation rate  $\overline{\epsilon}$  can be determined.

$$\overline{G_k} = \frac{1}{y_n} \int_0^{y_n} \tau_t \frac{\partial U}{\partial y} \, dy \equiv \frac{1}{\kappa y_n} \frac{\tau_w^2}{\rho C_\mu^{1/4} k_p^{1/2}} \, \ln\left(\frac{y_n}{y_v}\right) \tag{A.55}$$

and

$$\overline{\epsilon} = \frac{1}{y_n} \int_0^{y_n} \epsilon \, dy \equiv \left[ \frac{2\nu}{y_\nu} + \frac{k_p^{1/2}}{C_\ell} \ln\left(\frac{y_n}{y_\nu}\right) \right] k_p \tag{A.56}$$

The above integrals are valid for quadrilateral and hexahedral cells, as the cell volume average can be approximated with a depth average [10]. For other cell shapes different approximations of the cell volume have to be used. Note that in (A.56) the height of the cell is indicated by  $y_n$ . Since the non-equilibrium model described above accounts for pressure-strain effects and non-equilibrium conditions, it is capable to predict flows with occurrence separation, recirculation and reattachment of the wall layer. This is especially noticeable in the predicted values for the skin friction coefficient compared to the stand equilibrium wall functions. Note that the non-equilibrium wall function implemented in Fluent, as described above, is only available for the  $k - \epsilon$ and RSM.

#### **ENHANCED NEAR-WALL TREATMENT**

Wall functions are an computationally cost-effective solution that yields to reasonable accurate solutions for most high Reynolds number wall-bounded flows. However, for flow cases that depart too much from the ideal conditions, such as flows involving strong separation or three-dimensionality in the near-wall region, the accuracy starts to significantly decrease. For such flows, one should use additional measures such as enhanced mesh resolution in the near-wall region. In this case the wall adjacent cells should be fine enough ( $y^+ \approx 1$ ) to capture the laminar sub-layer. However, this requires a lot of computational power when this is applied over the entire near-wall region. Instead, Fluent offers the choice of 'Enhanced Wall Treatment', which combines a Two-Layer model with enhanced wall functions. The Enhanced Wall Treatment is an option for the  $k - \epsilon$  model and RSM, for the  $k - \omega$  models only the enhanced wall functions are available.

**Two-Layer model** In the Enhanced Wall Treatment, the whole domain is divided into a viscous-effected and a fully turbulent region. The border of the two regions is determined by the formulation of a Reynolds number  $Re_y$  based on the distance to the wall *y*:

$$Re_y = \frac{\rho y \sqrt{k}}{\mu} \tag{A.57}$$

Note that *y* is the shortest distance between the cell centroid an the nearest wall. When  $Re_y > 200$ , the flow is assumed fully turbulent and the flow field is resolved by either the  $k - \epsilon$  model or RSM. For the case that  $Re_y < 200$ , a one-equation model by Wolfstein [83] is used. The same equations for momentum and *k* are used by the  $k - \epsilon$  model and RSM respectively. However, the eddy viscosity term is modelled in a different way:

$$\mu_{t,2L} = \rho C_{\mu} \ell_{\mu} \sqrt{k} \tag{A.58}$$

The length scale in (A.57) is given by: [84]

$$\ell_{\mu} = y C_{\ell}^{*} \left( 1 - e^{-Re_{y}/A_{\mu}} \right) \tag{A.59}$$

The two-layer turbulent viscosity formulation of eq. (A.58) is used to blend with the high-Reynolds number eddy viscosity  $\mu_t$ , given by eq. (A.16), of the outer region. The blended viscosity term is used for the enhanced wall treatment and given by [85]:

$$\mu_{t,enh} = \lambda_{\epsilon} \mu_t + (1 - \lambda_{\epsilon}) \mu_{t,2L} \tag{A.60}$$

The blending function  $\lambda_c$  is defined such equal to one far away from the wall and zero very near the wall. The blending function utilized by the Enhanced Wall Treatment in Fluent is: [10]

$$\lambda_{\epsilon} = \frac{1}{2} \left[ 1 + \tanh\left(\frac{Re_{y} - Re_{y}^{*}}{A}\right) \right]$$
(A.61)

Note that  $Re_y^*$  is the Reynolds number at the border of the viscous-effects and fully turbulent region, previously determined to be  $Re_y^* = 200$ . The constant *A* is used to determine the width of the blending function. It is determined such that the value of  $\lambda_{\epsilon}$  is within 1% of the far-field value for a certain variation  $\Delta Re_y$ .

$$A = \frac{|\Delta R e_y|}{\tanh(0.98)} \tag{A.62}$$

The main purpose of the blending function  $\lambda_{\epsilon}$  is to prevent convergence problems with the  $k - \epsilon$ , when the solution of the outer region does not match with two-layer formulation. Typically,  $\Delta Re_y$  is set between 5% and 20% of  $Re_y^*$ .

To compute the turbulence dissipation rate  $\epsilon$ , the following simple relation is used:

$$\epsilon = \frac{k^{3/2}}{\ell_{\epsilon}} \tag{A.63}$$

,where the length scale  $\ell_{\epsilon}$  is again determined by Chen and Patel [84] (see eq. (A.59)). The following constant are used:

$$C_{\ell}^* = \kappa C_{\mu}^{-3/4}, \quad A_{\mu} = 70, \quad A_{\epsilon} = 2C_{\ell}^*$$
 (A.64)

**Enhanced wall functions** In order to have a single function that describes the entire viscous sub-layer (laminar sub-layer, buffer region and log-law region), the linear and logarithmic laws-of-the-wall are blended as follows: [86]

$$u^{+} = e^{\Gamma} u^{+}_{lam} + e^{1/\Gamma} u^{+}_{turb}$$
(A.65)

,where  $\Gamma$  is a blending function given by:

$$\Gamma = -\frac{a(y^{+})^{4}}{1+by^{+}}$$
(A.66)

with the coefficients a = 0.01 and b = 5. By taking the derivative of eq. (A.65) one can write:

$$\frac{du^{+}}{dy^{+}} = e^{\Gamma} \frac{du^{+}_{lam}}{dy^{+}} + e^{1/\Gamma} \frac{du^{+}_{turb}}{dy^{+}}$$
(A.67)

The derivative form of eq. (A.65) helps to promote convergence in the case that  $y^+$  falls inside the buffer region (3 <  $y^+$  < 10). In order to smoothly blend the linear region and the fully-turbulent (log-law) region, an enhanced formulation of the turbulent wall law has been derived: [10]

$$\frac{du_{turb}^{+}}{dy^{+}} = \frac{1}{\kappa y^{+}} \left[ S' \left( 1 - \beta u^{+} - \gamma \left( u^{+} \right)^{2} \right)^{1/2}$$
(A.68)

,where:

$$S' = \begin{cases} 1 + \alpha y^+ & \text{for } y^+ < y_s^+ \\ 1 + \alpha y_s^+ & \text{for } y^+ \ge y_s^+ \end{cases}$$
(A.69)

and

$$\alpha = \frac{v_w}{\tau_w u^*} \frac{dp}{dx} = \frac{\mu}{\rho^2 (u^*)^3} \frac{dp}{dx}$$

$$\beta = \frac{\sigma_t q_w u^*}{c_p \tau_w T_w} = \frac{\sigma_t q_w}{\rho c_p u^* T_w}$$

$$\gamma = \frac{\sigma_t (u^*)^2}{2c_p T_w}$$
(A.70)

,where  $y_s^+$  is the location at which the slope of log-law is constant. The default value in Fluent is  $y_s^+ = 60$ . The coefficients  $\alpha,\beta$  and  $\gamma$ , given by eq. (A.70), take into account the effect of pressure gradients and thermal effects. Note that if the coefficients of eq. (A.70) reduce to zero, the analytical solution of eq. (A.68) again yields to the standard log-law expression. The integrated form of the laminar term of eq. (A.67) is given by:

$$u^{+} = y^{+} \left( 1 + \frac{\alpha}{2} y^{+} \right) \tag{A.71}$$

Besides the blended velocity profile definition (eq.(A.65)), a similar approach is implemented for the thermal wall profile. However, for the sake of briefness, this is not discussed here. Instead the reader is referred to the Fluent user-guide [10].

# **B** NACA 4412 RESULTS



Figure B.1: Non-dimensiolized u-velocity measured along the velocity rakes



Figure B.2: Non-dimensiolized v-velocity measured along the velocity rakes



Figure B.3: Non-dimensiolized u-velocity measured along the velocity rakes



Figure B.4: Non-dimensiolized v-velocity measured along the velocity rakes



Figure B.5: Non-dimensiolized u-velocity measured along the velocity rakes



Figure B.6: Non-dimensiolized v-velocity measured along the velocity rakes



Figure B.7: Non-dimensiolized u-velocity measured along the velocity rakes



Figure B.8: Non-dimensiolized v-velocity measured along the velocity rakes



Figure B.9: Non-dimensiolized u-velocity measured along the velocity rakes



Figure B.10: Non-dimensiolized v-velocity measured along the velocity rakes



Figure B.11: Non-dimensiolized u-velocity measured along the velocity rakes



Figure B.12: Non-dimensiolized v-velocity measured along the velocity rakes



Figure B.13: Pressure distribution obtained with the SST  $k - \omega$  compared to experimental data [14]



Figure B.14: Verification of SST  $k - \omega$  results with CFD results obtained by NASA [13]

# C

## **USER DEFINED FUNCTION**

```
#include "turb.h"
#include "tarb.in
#include "sg_ke.h"
#include "dx.h"
real F3(cell_t c,Thread *t)
{
       real y
                 = C_WALL_DIST(c, t);
       real arg1;
       real fsst;
       real yPlus = 6.0*pow(10,6)*pow(y,2)+194068*y+0.2744;
       fsst = 0.09 + (1 - 0.09 + tanh(pow((0.03 * yPlus), 4))) * (0.91 + 0.09 * tanh(pow((0.005 * yPlus), 8)));
       arg1 = MIN(fsst,1);
       C\_UDMI(c,t,0) = y;
       C_UDMI(c,t,1) = yPlus;
       return arg1;
 }
 real ALPHA_STAR(cell_t c, Thread *t)
{
       real arg2;
       real Re_t;
       real R_k
                           = 6;
       real alpha_inf = 1;
       real alpha0_inf = 0.072/3;
                   = C_MU_L(c, t);
       real mu
       real k = C_K(c, t);
real omega = C_O(c, t);
       real rho
                   = C_R(c, t);
       \operatorname{Re}_t = (\operatorname{rho} * k) / (\operatorname{mu} * \operatorname{omega});
       arg2 = alpha_inf*((alpha0_inf+Re_t/R_k)/(1+Re_t/R_k));
       return arg2;
```

}

```
DEFINE_TURBULENT_VISCOSITY(user_mu_t,c,t)
{
       real mu_t;
       real arg3;
       real arg4;
                   = C_STRAIN_RATE_MAG(c, t);
       real Smag
       real mu = C_WO_{LVC}
= C_K(c, t);
                   = C_MU_L(c, t);
       real omega = C_O(c,t);
       real rho
                   = C_R(c, t);
       real alpha_1 = 0.31;
                = C_WALL_DIST(c, t);
       real y
       /*real* k_g = C_K_G(c,t);
       real* o_g = C_O_G(c, t);
       real Cross_Diff = Get_kw_Cross_Deriv(k_g, o_g, omega);
       real F1 = get_sst_F1(rho, k, omega, mu, y, Cross_Diff ); */
       real F2 = get_sst_F2(rho, k, omega, mu, y);
       C_UDMI(c,t,2) = F3(c,t);
       C_UDMI(c,t,3) = F2;
       arg3 = 1/ALPHA\_STAR(c,t);
       arg4 = Smag*F2/(alpha_1*omega);
       mu_t = F3(c, t) * (rho * k/omega) * (1/MAX(arg3, arg4));
       return mu_t;
```

### **BIBLIOGRAPHY**

- [1] A.T. Nouwens. Aerodynamic performance and stability analysis of aerocity. Master's thesis, Delft University of Technology, March 2011.
- [2] S. Nasrollahi. Numerical and experimental aerodynamic performance analysis of aerocity. Master's thesis, Delft University of Technology, Januari 2014.
- [3] V Guigueno. Building a high-speed society: France and the Aérotrain, 1962–1974. *Technology and Culture*, 49(1):21–40, 2008.
- [4] L Yun, A. Bliault, and D. Johnny. *WIG craft and Ekranoplan Ground Effect Craft Technology*. Springer, 2010.
- [5] Z.G. Yang, W. Yang, and Q. Jia. Ground viscous effect on 2D flow of wing in ground proximity. *Engineering Applications of Computational Fluid Mechanics*, 4(4):521–531, 2010.
- [6] S. Hase, G. Eitelberg, and L. Veldhuis. Ground effect investigtation for two dimensional airfoil. In Symposium on Experiments and Simulation of Aircraft in Ground Proximity. German-Dutch Wind Tunnels DNW, April 2013.
- [7] J.H. Jung, M.J. Kim, H.S. Yoon, P.A. Hung, H.H. Chun, and D.W. Park. Endplate effect on aerodynamic characteristics of threedimensional wings in close free surface proximity. *International Journal of Naval Architecture and Ocean Engineering*, 4(4):477–487, 2012.
- [8] J. Lee, C.S. Han, and C.H. Bae. Influence of wing configurations on aerodynamic characteristics of wings in ground effect. *Journal of Aircraft*, 47(3):1030–1040, 2010.
- [9] F. R. Menter. Zonal two equation  $k \omega$  turbulence models for aerodynamic flows. Technical report, NASA, 1992. Technical Memorandum 103975.
- [10] ANSYS. Fluent 6.3 User's Guide, September 2006.
- [11] H.K. Versteeg and W. Malalasekera. *An introduction to Computational Fluid Dynamics The Finite Volume Method.* Pearson Education Limited, 2nd edition, 2007.
- [12] J. Tu, G. Heng Yeoh, and C. Liu. Computational Fluid Dynamics A Practical Approach. Elsevier, 2008.
- [13] NASA. Turbulence modeling resource. http://turbmodels.larc.nasa.gov.
- [14] D. Coles and A.J. Wadcock. Flying-hot-wire study of flow past a NACA 4412 airfoil at maximum lift. AIAA Journal, 17(4):321–329, 1979.
- [15] M.R. Ahmed. Aerodynamics of a NACA 4412 airfoil in ground effect. AIAA, 45(1):37–47, January 2007.
- [16] M. Kikuchi, K. Hirano, T. Yuge, K. Iseri, and Y. Kohama. Measurement of aerofoil characteristics by method of towing model. *Transactions of the Japan Society of Mechnical Engineers*, *Part B*, 68(676):3378– 3385, December 2002.
- [17] P.E. Kumar. An experimental investigation into the aerodynamic characteristics of a wing with and without endplates, in ground effect. CoA Report Aero 201, College of Aeronautics (Cranfield, England), March 1968.
- [18] W.A Timmer. Two-dimensional low-Reynolds number wind tunnel results for airfoil NACA 0018. *Wind Engineering*, 32(6):525–537, 2008.
- [19] C.L. Ladson. Effects of independent variation of Mach and Reynolds numbers on the low-speed aerodynamic characteristics of the NACA 0012 airfoil section. NASA Technical Memorandum 4074, NASA, 1988.

- [20] F. Lacôte. 50 years of progress in railway technology. Japan Railway & Transport Review, 27:25–31, 2001.
- [21] R. Hope. Dropping the tracked hovercraft. In *New Scientist*, volume 57, pages 358–360. New Science Publications, Februari 1973.
- [22] C Wieselsberger. Wing resistance near the ground. *Zeitschrift für Flugtechnik und Motorluftshiffart*, 10:145–147, 1922.
- [23] M.P. Fink and J.L. Lastinger. *Aerodynamic characteristics of low-aspect-ratio wings in close proximity to the ground*. National Aeronautics and Space Administration, 1961.
- [24] C.M. Hsiun and C.K. Chen. Aerodynamic characteristics of a two-dimensional airfoil with ground effect. *Journal of Aircraft*, 33(2):386–392, 1996.
- [25] X. Zhang and J. Zerihan. Off-surface aerodynamic measurements of a wing in ground effect. *Journal of Aircraft*, 40(4):716–725, 2003.
- [26] Structuurvisie Zuiderzeelijn, 2009.
- [27] K. Park and J. Lee. Influence of endplate on aerodynamic characteristics of low-aspect-ratio wing in ground effect. *Journal of Mechanical Science and Technology*, 22(12):2578–2589, 2008. Springer.
- [28] M.R. Ahmed and S.D. Sharma. Experimental investigation of the flow field of a symmetrical airfoil in ground effect. In *21st Applied Aerodynamics Conference, Orlando, Florida*, 2003.
- [29] M.R. Ahmed. Flow over thick airfoils in ground effect an investigation on the influence of camber. In *Proceedings of the 24th Congress of International Council of the Aeronautical Sciences*, pages 1–11, 2004.
- [30] G. Venturi. *Recherches experimentales sur le principe de la communication laterale du mouvement dans les fluides.* Houel et Ducros, 1797. Paris.
- [31] M.R. Ahmed. Aerodynamics of a chambered airfoil in ground effect. *International Journal of Fluid Mechanics*, 32(2):157–183, 2005.
- [32] M. Gaster. *The structure and behaviour of laminar separation bubbles*. Her Majesty's Stationary Office, 1969.
- [33] W.T. Lance. Experimental and analytic investigation of ground effect. *Journal of Aircraft*, 52(1):235–243, 2014.
- [34] A.R. George. Aerodynamic effects of shape, camber, pitch, and ground proximity on idealized groundvehicle bodies. *Journal of Fluids Engineering*, 103(4):631–637, 1981. American Society of Mechanical Engineers.
- [35] T.J. Barber. Aerodynamic ground effect: a case study of the integration of CFD and experiment. *International Journal of Vehicle Design*, 40(4):299–316, January 2006.
- [36] A.E. Ockfen and K. Matveev. Aerodynamic characteristics of NACA 4412 airfoil section with flap. *International Journal of Naval Architecture and Ocean Engineering*, 1(1):1–12, 2009.
- [37] J. Gross and L.W. Traub. Experimental and theorectical investigation of ground effect at low Reynolds numbers. *Journal of Aircraft*, 49:576–586, 2012.
- [38] L. Prandtl. Wing theory. Bulletin of the Gottingen Scientific Society, ..., 1919.
- [39] A. Betz. Lift and drag of a wing near a horizontal surface (the ground). *Zeitschrift fur Flugtechnik und Motorluftschiffahrt*, 3:217, 1912.
- [40] W.F. Phillips and D.F. Hunsaker. Lifting-line predictions for induced drag and lift in ground effect. *Journal of Aircraft*, 50(4):1226–1233, 2013.
- [41] M.D. Chawla, L.C. Edwards, and M.E. Franke. Wind-tunnel investigation of wing-in-ground effects. *Journal of Aircraft*, 27(4):289–293, 1990.

- [42] J. Cho and C. Han. Unsteady trailing vortex evolution behind a wing in ground effect. *Journal of Aircraft*, 42(2):429–434, 2005.
- [43] R. Krasny. Computation of vortex sheet roll-up in the Trefftz plane. *Journal of Fluid Mechanics*, 184:123– 155, 1987. Cambridge University Press.
- [44] R.D. Irodov. Criteria of the longitudinal stability of the Ekranoplan. Technical report, Uch. Zap. Tsentr. Aerogidrodinamicheskii Inst., 1974.
- [45] S. Jamei, A. Maimun, S. Mansor, N. Azwadi, and A. Priyanto. Numerical investigation on aerodynamic characteristics of a compound wing-in-ground effect. *Journal of Aircraft*, 49(5):1297–1305, 2012.
- [46] O. Reynolds. On the dynamical theory of incompressible viscous fluids and the determination of the criterion. *Philosophical Transactions of the Royal Society of London*, 186:123–164, 1895.
- [47] J. Boussinesq. Essai sur la theorie des eaux courantes. *Mèmoires présèntes par divers savants a l'Acádemie des Sciences*, 23:1–680, 1877.
- [48] P.R. Spalart and S.R. Allmaras. A one-equation turbulence model for aerodynamic flows. In *AIAA-92-0439*. AIAA, January 1992.
- [49] J. Dacles-Mariani, G.G. Zilliac, S. Chow, and P. Bradshaw. Numerical/experimental study of a wingtip vortex in the near field. *AIAA Journal*, 33(9):1561–1568, 1995.
- [50] Spalding D. B. Launder, B.E. Lectures in mathematical models of turbulence. Technical report, Academic Press (London), 1972.
- [51] K.A. Hoffman. Computational Fluid Dynamics Volume III. EES, 2000.
- [52] Orszag S.A. Thangam S. Gatski T.B. & Speziale C.G. Yakhot, V. Development of turbulence models for shear flows by a double expansion technique. *Physics of Fluids A*, 4:1510–1520, 1992.
- [53] T.H. Shih, W.M. Liou, A Shabbir, Z. Yang, and J. Zhu. A new *k e* eddy viscosity model for high Reynolds number turbulent flows. *Computers Fluids*, 24:227–238, 1995.
- [54] F. R. Menter. Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal, 32:1598–1605, 1994.
- [55] D.C. Wilcox. Reassessment of the scale-determining equation for advanced turbulence models. *AIAA Journal*, 26(11):1299–1310, 1988.
- [56] Reece G. J. Launder, B. E. and W. Rodi. Progress in the development of a Reynolds-stress turbulent closure. *Journal of Fluid Mechanics*, 68(3):537–566, 1975.
- [57] T.M.P. Kumar and D. Chatterjee. Numerical study of turbulent flow over an S-shaped hydrofoil. In *Proceeding IMechE Part C: Journal of Mechnical Engineering Sciences*, volume 222. IMechE, 2008.
- [58] B.P. Leonard. A stable and accurate convective modelling procedure based on quadratic upstream interpolation. *Computer Methods in Applied Mechanics and Engineering*, 19(1):59–98, 1979.
- [59] P.K. Sweby. High resolution schemes using flux limiters for hyperbolic conservation laws. *Numerical Analysis*, 21(5):995–1011, 1984.
- [60] B. Leer van. Towards the ultimate conservative difference scheme. V. A second order sequal to godunov's method. *Journal of Computational Physics*, 32:101–136, 1979.
- [61] H. Schlichting and K. Gersten. Boundary-Layer Theory. McGraw Hill, 1979.
- [62] S.V. Patankar and D.B. Spalding. A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. *International Journal of Heat and Mass Transfer*, 10:1787–1806, 1972.
- [63] J. P. Van Doormal and G. D. Raithby. Enhancements of the SIMPLE method for predicting incompressible fluid flows. *Numerical Heat Transfer*, 7(2):147–163, 1984.

- [64] B. Diskin, J.L. Thomas, C.L. Rumsey, and A. Schwoppe. Grid convergence for turbulent flows (invited). In *53rd AIAA Aerospace Sciencess Meeting*. AIAA SciTech, January 2015.
- [65] N. Gregory and C.L. O'Reilly. Low-speed aerodynamic characteristics of NACA 0012 aerfoil section, including the effects of upper-surface roughness simulating hoar frost. Reports and Memoranda 3726, Aeronautical Research Council, 1973.
- [66] W.J. McCroskey. A critical assessment of wind tunnel results for the NACA 0012 airfoil. NASA Technical Memorandum 100019, US Army Aviation Systems Command, October 1987.
- [67] J.L Smith, H.Z Graham, and J.E. Smith. The validation of an airfoil in the ground effect regime using 2-D CFD analysis. In 26th AIAA Aerodynamic Measurement Technology and Ground Testing Conference, June 2008.
- [68] Y. Takahashi, M. Kikuchi, and K. Hirano. Analysis of ground effects on aerodynamic characteristics of aerofoils using boundary layer approximation. *JSME International Journal*, 49(2):401–409, 2006. Series B.
- [69] Y Takashi, M. Kikuchi, K. Hirano, T. Yuge, T. Moriya, and Y. Kohama. Experiments at the Sunrise-Beach Research Facility of the aerodynamic characteristics on ground effects of aerofoils with a secondary aerofoil. In *Proceedings of the 40th JAXA Workshop on "Investigation and Control of Boundary-Layer Transition*, 2008.
- [70] S. Yoshioka, T Kato, and Y Kohama. Introduction of towing wind tunnel facility in Sunrise-Beach Research Facility. In Proceedings of the 40th JAXA Workshop on "Investigation and Control of Boundary-Layer Transition, 2008.
- [71] P.R. Spalart and C.I Rumsey. Effective inflow conditions for turbulence models in aerodynamic calculations. *AIAA Journal*, 45:2544–2553, 2007.
- [72] H Blasius. Grenzschichten in Flüssigkeiten mit kleiner Reibung. B.G. Teubner, 1908.
- [73] M.R. Head. Entrainment in the turbulent boundary layer. Aeronautical Research Council Reports and Memoranda, 3152:1–16, 1958.
- [74] CFD Online Y+ Estimation Tool. http://www.cfd-online.com/Tools/yplus.php.
- [75] J.C.R. Hunt, A.A. Wray, and P. Moin. Eddies, streams and convergence zones in turbulent flows. In Proceedings of the Summer Program, number 89. Center for Turbulence Research, 1988.
- [76] V. Kolar. Vortex identification: New requirements and limitations. *International Journal of Heat and Fluid Flow*, 28:638–652, 2007.
- [77] G.V. Middleton and J.B. Southrad. *Mechanics of Sedement Movement*. Sociesty of Economic Paleontologists and Mineralogist, 1984.
- [78] T Chitsomboon and C. Thamthae. Adjustment of  $k-\omega$  SST turbulence model for an improved prediction of stalls on wind turbine blades. In *Wind Energy Applications*, Linkoping, Sweden, May 2011. World Renewable Energ Congress.
- [79] B.J. Daly and F.H. Harlow. Transport equations in turbulence. *Physical Fluids*, 13:2634–2649, 1970.
- [80] C.G. Speziale, S. Sarkar, and T.B. Gatski. Modelling the pressure-strain correlation of turbulence: An invariant dynamic systems approach. *Journal of Fluid Mechanics*, 227:245–272, 1991.
- [81] T. von Karman. Mechanische ähnlichkeit und turbulenz. Nachrichten von der Gesellschaft der Wissenschaften zu Göttingen, Fachgruppe 1 (Mathematik), 5:58–76, 1930.
- [82] S.E. Kim and D. Choudhury. A near-wall treatment using wall functions sensitized to pressure gradient. *ASME Separated and Complex Flows*, 217:273–280, 1995.
- [83] M. Wolfstein. The velocity and temperature distribution of one-dimensional flow with turbulence augmentation and pressure gradient. *International Journal of Heat and Mass Transfer*, 12:301–318, 1969.
- [85] T. Jongen. *Simulation and Modeling of Turbulent Incompressible Flows*. PhD thesis, EPF Lausanne, 1992. Lausanne, Switzerland.
- [86] B. Kader. Temperature and concentration profiles in fully turbulent boundary layers. *International Journal of Heat and Mass Transfer*, 24(9):1541–1544, 1981.