



University of
Nottingham

UK | CHINA | MALAYSIA

Large scale model order reduction for drag reduction of the Ahmed body

Peter Rhodes

Thesis submitted to the University of Nottingham
for the degree of Doctor of Philosophy

March 2022

Abstract

Flow control of ground vehicles is a key area of research with the aim of reducing the drag that cars and trucks experience. There are many approaches being researched to achieve this, but a common approach is to use Synthetic Jet Actuators (SJA) located at the rear of the model. The SJAs are used to delay the flow separation at the rear of the model and reduce the size or control the behaviour of the wake. A closed loop control scheme monitors the external flow for changes and allows the SJAs to adapt, improving their efficiency over a larger range of conditions. The most common closed loop controller is the Extremum Seeking Controller (ESC), however, the ESC scheme can take a long time to converge as it requires no prior information about the model. To provide model information to the ESC scheme, a reduced order model (ROM) needs to be created as a CFD model is too slow to run in real time. There are many model order reduction schemes, but this research uses the Input-Output Dynamic Mode Decomposition (IODMD) scheme to reduce the size of the CFD model. These schemes have been shown to be successful on smaller scale systems, but never for the reduction of a system on the scale used in this research.

This research uses CFD simulations of an Ahmed body, coupled with SJAs located at the rear of the model. Unlike most SJA setups, the SJAs in this research are directed into the top of the wake, in the hope of reducing the size of the wake and increasing the static pressure on the rear face of the Ahmed body. An adaptation to the ESC is proposed with a reduced order model (ROM) created to provide information about the flow around the Ahmed body to the ESC scheme. By providing this information to the scheme, the convergence of the ESC scheme can be sped up. This would improve the potential for the closed loop control scheme to be used in real time control.

The 2D CFD simulations of the Ahmed body provide a good drag reduction, similar to results presented in the literature, of up to 20% for the Ahmed body with a 35° rear slant. The 3D CFD simulations also showed promising signs with the SJA setup providing a average reduction in the height of the wake by 3%, but detrimental effects in the wake cause the drag to increase. A ROM was produced which reduced the 3D CFD simulation from 4,000,000 cells in the mesh, to just a single mode to model the effect of the SJAs on the drag coefficient. The adapted ESC scheme used the 2D CFD simulation and showed a remarkable reduction to an almost instant convergence time by using the ROM to predict the optimal SJA velocity before the scheme took over. The adapted ESC scheme was also able to save 33% more power over a 30 second simulation.

Acknowledgements

I would like to thank my supervisors, Dr. Mark Jabbal, Professor Kwing-so Choi and Dr. Harald Pfifer for their help and support throughout this research. Their guidance and knowledge has been invaluable.

I would also like to thank Dr Neville Rieger DSc LLd and the Rieger High Fliers PhD Scholarship for funding this research.

This research was conducted using the University of Nottingham Augusta HPC service and the HPC Midlands Plus Athena HPC service. Thanks go to the HPC IT department for their support.

Contents

Abstract	i
Acknowledgements	ii
Nomenclature	vi
1 Introduction	1
1.1 Motivation	1
1.2 Aims and Objectives	3
1.3 Methodology	4
1.4 Thesis Outline	5
2 Literature Review	7
2.1 Introduction	7
2.2 The Ahmed Body	7
2.2.1 The Front and Mid Sections	8
2.2.2 The Rear Section	9
2.2.3 The Attached Low Drag Regime	10
2.2.4 The High Drag Regime	12
2.2.5 The Separated Low Drag Regime	14
2.2.6 The Effects of Frontal Aspect Ratio	15
2.3 Numerical Modelling	16
2.3.1 2D Numerical Modelling	16
2.3.2 3D Numerical Modelling	17
2.4 Flow Control	19
2.4.1 Passive Flow Control	19
2.4.2 Active Flow Control	22

2.4.3	Synthetic Jet Actuators	29
2.5	Reduced Order Modelling	35
2.6	Closed Loop Control	40
2.6.1	Extremum Seeking Control	40
2.6.2	Alternative Closed Loop Control	44
2.6.3	Summary	46
3	Methodology	49
3.1	Introduction	49
3.2	Numerical Simulations	53
3.2.1	Mesh Independence	53
3.2.2	Numerical Model Setup	55
3.2.3	Synthetic Jet Actuators	59
3.2.4	Calculating the Net Power Savings	62
3.3	Model Order Reduction	64
3.3.1	Introduction	64
3.3.2	Reduction Scheme	66
3.4	Extremum Seeking Control	70
3.5	Summary	73
4	2D Numerical Modelling	76
4.1	2D Numerical Model for the 35° Rear Slant Ahmed Body	76
4.1.1	Introduction	76
4.1.2	Model Validation	77
4.1.3	Flow Control of the Wake Using Synthetic Jet Actuators	88
4.1.4	Net Power Savings for Control with Synthetic Jet Actu- ators	102
4.1.5	Summary	105
5	Reduced Order Modelling	106
5.1	Introduction	106
5.2	Input-Output Dynamic Mode Decomposition	108

5.3	Adapting the Input to Improve the Model Fit	112
5.4	Producing a Lower Order Approximation of the System	115
5.5	Single Mode Approximations of the Drag Coefficient	128
5.6	Modelling Freestream Effects on the Drag Reduction	130
5.7	Net Power Savings for the Reduced Order Model	138
5.8	Extremum Seeking Control	139
5.8.1	Introduction	139
5.8.2	The Basic Perturbation Based Extremum Seeking Control Scheme	141
5.8.3	Testing the Model Capabilities for a Constantly Fluctuating Freestream Velocity	148
5.9	Summary	151
6	3D Results	153
6.1	3D Numerical Modelling	153
6.1.1	Introduction	153
6.1.2	3D Numerical Model Validation	153
6.1.3	Controlling the Wake of a 25° Rear Slant 3D Ahmed Body Using Synthetic Jet Actuators	165
6.1.4	Controlling the Wake of a 35° 3D Ahmed Body Using Synthetic Jet Actuators	174
6.1.5	Inverting the Aspect Ratio of the 35° Rear Slant Ahmed Body	183
6.1.6	Summary	188
6.2	3D Reduced Order Modelling	188
6.2.1	Summary	193
7	Conclusion	194
7.1	Main Results	194
7.2	Main Contributions	196
7.3	Limitations and Future Studies	198

Appendices	211
A Fluent UDF	211

Nomenclature

Acronyms

BPOD	Balanced Proper Orthogonal Decomposition
C	Courant Number
C_D	Drag Coefficient
CFD	Computational Fluid Dynamics
CFL	Courant-Friedrichs-Lewy
CLC	Closed Loop Control
C_p	Pressure Coefficient
c_μ	Momentum Coefficient
DBD	Dielectric Barrier Discharge
DES	Detached Eddy Simulation
DMD	Dynamic Mode Decomposition
ESC	Extremum Seeking Control
FFT	Fast Fourier Transform
GTS	Ground Transportation System
HGV	Heavy Goods Vehicle
HPC	High Performance Computer
IODMD	input-output Dynamic Mode Decomposition
LES	Large Eddy Simulation
LGP	Linear Genetic Programming
LPV	Linear Parameter Varying
LTI	Linear Time Invariant
MEMs	Micro-Electro Mechanical systems
MOR	Model Order Reduction
ODE	Ordinary Differential Equation
PISO	Pressure-Implicit with Splitting of Operators

POD	Proper Orthogonal Decomposition
RANS	Reynolds Averaged Navier Stokes
Re	Reynolds Number
ROM	Reduced Order Model
ROMESC	Reduced Order Model Extremum Seeking Control
RSM	Reynolds Stress Model
SaOB	Suction and Oscillatory Blowing
SIMPLE	Semi-Implicit Method for Pressure Linked Equations
SJA	Synthetic Jet actuator
SST	Shear Stress Transport
SVD	Singular Value Decomposition
TAT	Tall and Thin
TKE	Turbulent Kinetic Energy
UDF	User Defined Function
URANS	Unsteady Reynolds Averaged Navier Stokes
VdP	Van der Pol
ZNMF	Zero Net Mass Flux

Greek Symbols

α	SJA Velocity and C_D Relationship Constant
ϵ	Turbulent Dissipation Rate (Js^{-1})
ϵ_{ij}	Turbulent Dissipation Rate Tensor (Js^{-1})
θ	SJA Jet Pitch Angle ($^\circ$)
λ	Non-Dimensional Distance Between Mesh Elements
μ_t	Turbulent Viscosity (Pas)
ρ	Density (kgm^{-3})
ρ_j	SJA Jet Density (kgm^{-3})
ρ_∞	Freestream Flow Density (kgm^{-3})
$\phi_i(x)$	Spatial Components of Vector Field
ω_j	SJA Oscillating Frequency (Hz)
ω_h	High Pass Filter Cut-off Frequency (Hz)
ω_l	Low Pass Filter Cut-off Frequency (Hz)

ω_p	Perturbation Frequency (Hz)
ω_∞	Shedding Frequency (Hz)

Latin Symbols

a	Perturbation Amplitude (-)
$a_i(t)$	Temporal Coefficients of Vector Field
A_j	Total Area of the SJA Orifices (m ²)
A_∞	Spanwise Cross-Sectional Area (m ²)
A, B, C, D	Linear Operators
Ar, Br, Cr, Dr	Reduced Linear Operators
b	Input Vector
c	Speed of Sound (ms ⁻¹)
d	Distance From Orifice Centre (m)
E	Error in the Velocity Profiles (% error)
F^+	Non-Dimensional Frequency
h	SJA Orifice Exit Radius (m)
I_j	Current of the SJAs (A)
I_n	N-Dimensional Identity Matrix
k	Turbulent Kinetic Energy (m ² s ⁻²)
K	Number of Timesteps
l	Length (m)
Δl	Distance Between Mesh Elements (m)
L	Characteristic Length (m)
n_j	Number of SJAs in Array
n_p	Number of Points
N	Number of Mesh Elements
P_j	SJA Power Used (w)
$P(t)$	Static Pressure (Pa)
Q	Projection Matrix
R_0, R_1	Data Matrices
t	Simulation Time (s)
t_k	Discrete Simulation Time (s)

Δt	Timestep Size (s)
u	System Input
u_k	Discrete Inputs
U	Instantaneous Velocity (ms^{-1})
U_i^{CFD}	Numerical Model Data
U_i^{EXP}	Experimental Data
U_j	Instantaneous SJA Exit Velocity (ms^{-1})
U_{max}	Peak SJA Jet Velocity (ms^{-1})
U_0	Input Vector
U_∞	Freestream Velocity (ms^{-1})
$\overline{u'_i u'_j}$	Reynolds Stress Tensor (Nm^{-2})
V_j	Voltage of the SJAs (v)
W_c	Controllability Gramian
W_o	Observability Gramian
$x(t)$	State Vector
x_h	Length of the Ahmed Body (m)
X	Snapshot Matrix
X_0	Input Snapshot Matrix
X_1	Output Snapshot Matrix
$y(t)$	System Output
y_k	Discrete Outputs
y_h	Height of the Ahmed Body (m)
Y_0	Output Vector
z_h	Width of the Ahmed Body (m)

List of Tables

3.1	Mesh independence test for the 3D 35° Ahmed body.	53
3.2	Table showing the effect on the drag coefficient of varying the angle of the SJAs.	61
3.3	Table showing the effect on the drag coefficient of varying the Jet actuation frequency.	62
3.4	A selection of peak jet velocities converted to momentum coef- ficients for a freestream velocity of 40m/s.	74
4.1	A table showing the percentage net power saving from the CFD simulation for various freestream velocities for the 2D 35° Ahmed body.	103
5.1	A table showing the percentage net power saving predicted by the ROM for various freestream velocities for the 2D 35° Ahmed body.	139
6.1	The drag coefficient for the 25° and 35° Ahmed bodies, obtained by Ahmed [6] and recorded as part of this CFD study.	154
6.2	The drag coefficient for the 25° and 35° Ahmed bodies, taken from Ahmed [6] and recorded as part of this CFD study.	174

List of Figures

2.1	A Diagram of the Ahmed body [7].	8
2.2	Drag coefficient against the rear slant angle [6].	10
2.3	The wake structure behind the squareback configuration of the Ahmed body [11].	11
2.4	Diagram of the vortices around the Ahmed body with a 25° rear slant [8].	12
2.5	2D streamlines in the centreplane of the 25° Ahmed body [13]. .	13
2.6	Surface streamlines on the rear slant of the Ahmed body [15]. 25° rear slant on the left and 35° rear slant on the right.	14
2.7	Plates located at the rear of the model to promote reattachment [25].	21
2.8	A diagram of the Windsor model used by Littlewood [39]. . . .	23
2.9	A diagram of the Suction and Oscillatory Blowing actuator used by Arwatz [48].	26
2.10	Curved overhangs on the rear of the truck to promote the conada effect [49].	28
2.11	A diagram of the GTS model [53].	29
2.12	Idealised representation of the vortices formed by the SJAs [56].	30
2.13	Slot opposite (left) and slot adjacent (right) configurations for the synthetic jet actuator [57].	31
2.14	The D-shaped body used by Pastoor [38].	32
2.15	The streamwise SJAs located on a truck A-pillar [64].	33

2.16	The locations of the SJA orifices at the top of the rear slant for the study conducted by Park [37].	34
3.1	Dimensions of the Ahmed body.	50
3.2	Flowchart showing the overarching methodology for this research.	52
3.3	A cross-section of the mesh used for the CFD simulations.	55
3.4	The setup of the boundary conditions for the CFD simulations.	58
3.5	The SJAs located at the rear of the Ahmed body with θ denoting the SJA angle.	61
3.6	The setup of the boundary conditions for the CFD simulations.	64
3.7	The basic setup of a closed loop control scheme.	65
3.8	The basic setup of a perturbation based extremum seeking controller.	71
3.9	The setup for the reduced order model-based extremum seeking controller.	72
4.1	A schematic of the Ahmed body, with lines added to show the locations of the key velocity profiles discussed in the next section.	78
4.2	A comparison of the velocity profiles on the rear slant of the 35° Ahmed body between the 2D CFD numerical model and the centreline experimental data from the Ahmed body ERCOF-TAC database [104].	79
4.3	A comparison of the velocity profiles at $x/x_h = -0.17$ on the rear slant of the 35° Ahmed body between the 2D numerical model and the centreline experimental data from the Ahmed body ERCOFTAC database [104].	80
4.4	A comparison of the velocity profiles at $x/x_h = -0.1$ on the rear slant of the 35° Ahmed body between the 2D CFD simulations and the centreline experimental data from the Ahmed body ERCOFTAC database [104].	82

4.5	A comparison of the velocity profiles at $x/x_h = 0$ on the rear slant of the 35° Ahmed body between the 2D CFD simulations and the centreline experimental data from the Ahmed body ERCOFTAC database [104].	83
4.6	A plot of the velocity profile shown in Figure 4.4 with an extended y axis to demonstrate the recovery of the velocity to the freestream.	84
4.7	A plot of the turbulent kinetic energy profile at $x/x_h = -0.1$ on the rear slant of the 35° Ahmed body.	85
4.8	Error between the velocity profiles on the slant for the experimental centre plane data from the Ahmed body ERCOFTAC database [104] and the 2D CFD simulations.	86
4.9	A comparison of the velocity profiles in the wake of the 35° Ahmed body between the 2D CFD simulations and the centreline experimental data from the Ahmed body ERCOFTAC database [104].	87
4.10	A comparison of the velocity profiles at $x/x_h = 0.08$ downstream of the 35° Ahmed body between the 2D CFD simulations and the centreline experimental data from the Ahmed body ERCOFTAC database [104].	88
4.11	A comparison of the velocity profiles at $x/x_h = 0.18$ downstream of the 35° Ahmed body between the 2D CFD simulations and the centreline experimental data from the Ahmed body ERCOFTAC database [104].	89
4.12	A comparison of the velocity profiles at $x/x_h = 0.32$ downstream of the 35° Ahmed body between the 2D CFD simulations and the centreline experimental data from the Ahmed body ERCOFTAC database [104].	90

4.13 Error between the velocity profiles in the wake for the experimental centre plane data from the Ahmed body ERCOFTAC database [104] and the 2D CFD simulations.	91
4.14 Percentage C_D change vs Momentum coefficient for the 2D 35° Ahmed body CFD simulation.	92
4.15 Time averaged velocity contour for the baseline (no actuation) case.	93
4.16 Velocity contours for the timestep where the drag coefficient is a (top) minimum (max suction) and (bottom) maximum (max blowing) for $C_\mu = 0.008$	94
4.17 A velocity gradient comparison on the rear faces for the baseline, minimum drag and maximum drag cases.	95
4.18 A pressure coefficient comparison on the rear faces for the baseline, minimum drag and maximum drag cases.	96
4.19 Time averaged velocity contours of (top) the baseline case and (bottom) the the case with $C_\mu = 0.008$	97
4.20 Contours of the velocity differences between the images in Figure 4.19.	98
4.21 Velocity profiles over the rear slant for $C_\mu = 0$, $C_\mu = 0.002$ and $C_\mu = 0.008$	99
4.22 Velocity profile over the apex of the rear slant and the top face for $C_\mu = 0$, $C_\mu = 0.002$ and $C_\mu = 0.008$	100
4.23 Velocity profiles in the wake for $C_\mu = 0$, $C_\mu = 0.002$ and $C_\mu = 0.008$	101
4.24 A zoomed in view of the top of the velocity profiles in the wake for $C_\mu = 0$, $C_\mu = 0.002$ and $C_\mu = 0.008$	102
4.25 Pressure coefficients on the rear faces of the Ahmed body for $C_\mu = 0$, $C_\mu = 0.002$ and $C_\mu = 0.008$	103
4.26 Plot showing the development of counter rotating vortices from the SJAs.	104

5.1	The C_D from the CFD and for the full data and face data ROMs with 500 modes.	110
5.2	The averaged C_D from the CFD and for the full data and face data ROMs with 500 modes.	111
5.3	Oscillation frequencies present in the CFD simulation data.	113
5.4	SJA velocity vs C_D change for the linear model.	114
5.5	Percentage C_D change vs SJA velocity, demonstrating the quadratic relationship.	115
5.6	Magnitude (in dB) vs frequency for the linear model and the CFD system.	116
5.7	Magnitude (in dB) vs frequency for the squared input model and the CFD system.	117
5.8	C_D obtained from the CFD simulation and computed from the static pressure increase.	118
5.9	The C_D from the CFD and for the full data and face data ROMs with 500 modes.	119
5.10	A zoomed in figure of the C_D from the CFD and for the full data and face data ROMs with 500 modes.	120
5.11	Magnitude (in dB) vs frequency for the 500 mode Full data and Face data models, compared with the CFD system.	121
5.12	The averaged C_D from the CFD and for the full data and face data ROMs with 500 modes.	122
5.13	The C_D from the CFD and for the full data and face data ROMs with 200 modes.	123
5.14	Magnitude (in dB) vs frequency for the 200 mode Full data and Face data models, compared with the CFD system.	124
5.15	The averaged C_D from the CFD and for the full data and face data ROMs with 200 modes.	125
5.16	Magnitude (in dB) vs frequency for the 20 mode Full data and Face data models, compared with the CFD system.	126

5.17 The averaged C_D from the CFD and for the full data and face data ROMs with 20 modes.	127
5.18 Magnitude (in dB) vs frequency for the 1 mode Full data and Face data models, compared with the CFD system.	128
5.19 The averaged C_D from the CFD and for the full data and face data ROMs with 1 mode.	129
5.20 The simulation of the 1 mode model built using the C_D , compared to the C_D obtained from the CFD for the 2D 35° Ahmed body.	130
5.21 C_D change vs freestream velocity with a fixed SJA velocity of 60m/s ($C_\mu = 8 \times 10^{-3}$).	131
5.22 Surface of C_D change for various freestream and SJA velocities.	132
5.23 C_D vs SJA velocity for 17.88m/s freestream velocity.	133
5.24 C_D change for SJA and Freestream changes for a single state space ROM.	134
5.25 C_D change for SJA and Freestream changes with linear interpolated data.	135
5.26 C_D change for SJA changes at 40m/s using the linear interpolated ROM compared with the CFD data and the linear interpolated data.	136
5.27 C_D change for SJA changes at 20m/s using the linear interpolated ROM compared with the CFD data and the linear interpolated data.	137
5.28 C_D change for SJA changes at 28m/s using the linear interpolated ROM compared with the CFD data and the linear interpolated data.	138
5.29 A plot showing the net power saving for the CFD model and the ROM with a freestream velocity of 22.35m/s.	140
5.30 A plot showing the net power saving for the CFD model and the ROM with a freestream velocity of 31.29m/s.	141

5.31	The convergence of the momentum coefficient for the basic ESC scheme using zero c_μ initial condition for the SJAs.	142
5.32	Percentage power saved for the basic ESC scheme using zero c_μ initial condition for the SJAs.	143
5.33	Convergence of the basic and ROM ESC schemes to an optimum SJA momentum coefficient for changing freestream conditions. .	144
5.34	Convergence of the basic and ROM ESC schemes to an optimum SJA voltage for changing freestream conditions.	145
5.35	A plot showing the net power saving for both ESC schemes in changing freestream conditions.	146
5.36	A plot showing the net power saving, for both ESC schemes in changing freestream conditions, after passing through a lowpass filter.	147
5.37	The freestream velocity input used to test the ROMESC scheme in constantly changing freestream conditions.	148
5.38	Convergence of the basic and ROM ESC schemes to an optimum SJA voltage for a sinusoidal freestream signal.	149
5.39	The voltage input for the SJAs as determined by the ROMESC controller for varying freestream conditions.	150
5.40	The percentage net power saved by the ROMESC for constantly changing freestream condition.	151
6.1	Velocity profile comparison over the rear slant of the 35° Ahmed body with velocity profile data from the ERCOFTAC database [104].	155
6.2	Velocity profile comparison at the end of the rear slant at $x/x_h = 0$ for the 35° Ahmed body with velocity profile data from the ERCOFTAC database [104].	156

6.3	Velocity profile comparison over the rear slant of the 25° Ahmed body with velocity profile data from the ERCOFTAC database [104].	157
6.4	Velocity profile comparison over the rear slant of the 35° Ahmed body in the plane $z/z_h = 0.25$ with velocity profile data from the ERCOFTAC database [104].	158
6.5	Velocity profile comparison over the rear slant of the 35° Ahmed body in the plane $z/z_h = 0.04$ with velocity profile data from the ERCOFTAC database [104].	159
6.6	Velocity profile comparison at $x/x_h = -0.1$ on the rear slant of the 25° Ahmed body in the plane $z/z_h = 0$ with velocity profile data from the ERCOFTAC database [104].	160
6.7	Velocity profile comparison in the wake of the 35° Ahmed body with velocity profile data from the ERCOFTAC database [104].	161
6.8	Velocity profile comparison in the wake at $x/x_h = 0.1$ for the 35° Ahmed body with velocity profile data from the ERCOFTAC database [104].	162
6.9	Vorticity iso-surface showing the vortical structures around the 25° Ahmed body obtained from the CFD simulation.	163
6.10	Velocity vector plot with streamlines to demonstrate the recirculating region in the wake for the 25° Ahmed body.	164
6.11	Vorticity iso-surface showing the vortical structures around the 35° Ahmed body obtained from the CFD simulation.	165
6.12	Velocity vector plot with streamlines to demonstrate the recirculating region in the wake for the 35° Ahmed body.	166
6.13	C_p plot on the rear faces of the 25° Ahmed body for the CFD simulations in this research and for the experimental C_p [37]. . .	167
6.14	C_p plot on the rear faces of the 35° Ahmed body for the CFD simulations in this research and for the experimental C_p [37]. . .	168

6.15 C_D change for the 35° Ahmed body with SJAs acting normal to the rear slant for the CFD simulation and the experimental C_D [37].	169
6.16 C_D reduction for the 25° Ahmed body with SJAs acting normal to the rear slant for the CFD simulation and the experimental C_D [37].	170
6.17 Velocity contour in the centreplane of the 25° Ahmed body with no actuation (top) and with synthetic jet actuation with $c_\mu = 2.5 \times 10^{-3}$ (bottom).	172
6.18 A plot of the pressure coefficient on the centre line of the rear face of the 25° Ahmed body.	173
6.19 Velocity contour in the centreplane of the 35° Ahmed body with no actuation, $c_\mu = 0$ (top) and with synthetic jet actuation with $c_\mu = 2.5 \times 10^{-3}$ (bottom).	175
6.20 Contour of the differences in velocity for the cases in Figure 6.19.	176
6.21 Velocity profiles in the centreplane of the rear slant for the 35° Ahmed body, showing the effects of actuation for $c_\mu = 0, 6 \times 10^{-4}$ and 2.5×10^{-3}	178
6.22 Percentage differences between the velocity profiles for the actuated flow for $c_\mu = 0, 6 \times 10^{-4}$ and 2.5×10^{-3}	179
6.23 Velocity contour in the plane, $z/z_h = 0.25$, of the 35° Ahmed body with no actuation, $c_\mu = 0$ (top) and with synthetic jet actuation with $c_\mu = 2.5 \times 10^{-3}$ (bottom).	180
6.24 Velocity contour in the plane, $z/z_h = 0.04$, of the 35° Ahmed body with no actuation.	181
6.25 Velocity profiles in the plane, $z/z_h = 0.04$, of the rear slant for the 35° Ahmed body, showing the effects of actuation for $c_\mu = 0, 6 \times 10^{-4}$ and 2.5×10^{-3}	182

6.26	Velocity contour showing the differences in the velocity between the case with $c_\mu = 0$ and 2.5×10^{-3} in the horizontal plane $y/y_h = 0.6$	183
6.27	A plot of the pressure coefficient on the centre line of the rear face of the Ahmed body.	184
6.28	Velocity contour in the centreplane of the TAT body with no actuation.	185
6.29	A plot of the pressure coefficient on the centre line of the rear face of the TAT body.	186
6.30	Velocity contour in the centreplane of the TAT body with synthetic jet actuation with $c_\mu = 2.5 \times 10^{-3}$	187
6.31	The simulation of the 1 mode model built using the C_D , compared to the C_D obtained from the CFD for the 3D 25° Ahmed body.	189
6.32	The simulation of the 1 mode model built using the C_D , compared to the C_D obtained from the CFD for the 3D 35° Ahmed body.	191
6.33	The simulation of the 1 mode model built using the C_D , compared to the C_D obtained from the CFD for the 3D TAT model.	192
6.34	The simulation of the 1 mode model built using the C_D , compared to the C_D obtained from the experimental data obtained by Kourta [67].	192

Chapter 1

Introduction

1.1 Motivation

In 2018, the UK Government set out targets to reduce greenhouse gas emissions over the next couple of decades [1]. The target, by 2025, was a 15% decrease in the greenhouse gas emissions, produced by heavy goods vehicles, from the 2015 levels. A plan was also outlined for the adoption of fuel efficient motor cars to again reduce emissions. The push for low emission vehicles was driven by the fact that transport accounts for 27% of all greenhouse gas emissions that the UK produces, with 90% of that generated from road transport.

The drag that a body experiences scales with the square of its speed. This means that for small increases to the speed of a vehicle, the drag can increase by a large amount. This effect is amplified for vehicles that experience a large drag, such as trucks, which are optimised for carrying goods rather than aerodynamic performance. The more drag that a body experiences, the more driving force is needed to overcome the drag, leading to a larger fuel consumption. In fact, almost 52% of the fuel burned, by a truck trailer, when travelling at 100km/h is used to overcome drag [2]. Therefore, a drag reduction for the body, results in lower fuel consumption, which reduces the running costs of vehicles. A reduced fuel consumption also leads to a reduction in emissions from vehicles resulting in a lower environmental impact from the vehicles as

well. The main areas of drag that a truck trailer experiences are; the front (25%), the underbody (30%), the gap between the tractor and trailer (20%) and the rear wake (25%) [3].

Methods to reduce drag fall into two main categories, passive flow control and active flow control. Passive flow control uses devices and techniques to improve an aspect of the flow around a body, without the use of auxiliary power. The improvement in the flow could be a drag reduction, a lift increase, a mixing enhancement or a noise reduction. Active flow control techniques attempt to improve aspects of the flow using an external energy source.

Passive control has been shown to be effective at reducing drag by modifying the shapes of vehicles or through the use of mechanical devices such as vortex generators. The main aim is to delay separation and increase the momentum in the boundary layer of the objects. Active flow control devices have been shown to produce drag reductions, but this has always been balanced against the power draw for the devices which, until the last decade [4], was generally greater than the power saved. This meant that in most cases, passive flow control techniques provided greater performance gains than active control devices. Advancements in the efficiency of active devices has begun to produce large net power savings, upto 8% [5], which is better than 2 of the passive control schemes discussed here.

A common active flow control device is the synthetic jet actuator (SJA) which uses a vibrating membrane, located in a cavity, to induce an oscillatory suction and blowing flow. A small orifice is used to connect this cavity to the outer flow around a body, producing a small jet flow that interacts with the surrounding flow. This jet flow can be used to increase the momentum in the boundary layer of the body and delay flow separation over a surface. The advantage of the SJA is that it is a zero net mass flux (ZNMF) device, meaning that an air supply is not needed to produce the jets, which causes the SJAs to be lightweight and easy to install. They can also provide large momentum changes to the flow, leading to a high control authority, making them suitable for the control of

high velocity flows.

An advantage that active schemes have over passive schemes is the ability to adapt to outer flow changes through open or closed loop control schemes. The ability to adapt the flow control leads to greater power savings over a range of different flow conditions when compared to passive control schemes. With the advancements in computational power, these adaptations can be computed in real time, to allow a system to adapt to changes in the flow conditions or even possibly to flow states it has never seen before.

One of the keys to providing meaningful real time control is the use of reduced order models (ROM), low dimensional approximations of a larger system which capture the key flow dynamics. Model order reduction (MOR) schemes allow highly complex, turbulent flow to be reduced to a smaller, easier to compute problem and in theory can be used to scale any size system. Most MOR schemes in the literature are limited to reducing systems smaller than 1000 states due to the computational cost associated with producing these ROMs, but with increased computing power there appears to be no reason that larger models could not be scaled in the same way. The combination of closed loop control schemes with ROMs could allow for greater fidelity in real time control situations which could also lead to greater power savings.

1.2 Aims and Objectives

The aim of this research is to produce a novel control scheme, using a large scale order reduction on numerical flow field information around a road vehicle, to produce a net energy saving for that road vehicle.

- **Objective 1:** Produce a 2D CFD simulation of the Ahmed body with a 35° rear slant, combined with a synthetic jet actuator array to provide a drag reduction through control of the wake, with a view to producing a net power saving.

- **Objective 2:** Produce a low dimensional approximation of the CFD simulation, which can be used to predict the flow changes caused by operating the synthetic jet actuator array.
- **Objective 3:** Develop an extended Extremum Seeking Controller, using a low dimensional approximation to improve the performance of the Extremum Seeking Controller for use in real time control.
- **Objective 4:** Produce 3D CFD simulations of various Ahmed bodies, combined with actuator arrays to provide a drag reduction, with a view to producing a net power saving and produce a low dimensional approximation for each model.

1.3 Methodology

The investigation of bluff body aerodynamics is primarily conducted using CFD analysis or wind tunnel experimentation. Both approaches have their benefits and drawbacks. For example, CFD analysis allows quick prototyping of new model setups as new parts do not have to be built or bought, however, large models require significant computation time and power. Experimental investigations require time and resources to build new models, but subsequent tests can be much quicker. The other major benefit of experimental data is the trust that the results are realistic. CFD models have to be validated against experimental data to ensure that the results are realistic, since the models use approximations of the Navier-Stokes equations. CFD models that are more accurate, require greater computation time.

This research uses CFD simulations with an unsteady Reynolds averaged Navier-Stokes model to ensure that the computation required is not too large, whilst also allowing easy access to large amounts of data about the models. CFD simulations simulate the whole of the flow field, so it is easy to get measurements from every part of the flow, whereas, experimental setups have to be

careful to not interfere with the flow when trying to gather data, meaning that it is much harder to collect information about the flow. For the development of a reduced order model, it is very useful to have a large dataset to gather data from and compare results to.

Real time control of fluid flow is a key target for closed loop control schemes to ensure they are viable for use on road vehicles. A common control scheme used is the Extremum Seeking Controller (ESC), since no prior information about the model is needed for use. This is the starting point for this research, with an extension to be developed using a reduced order model approximation of the CFD simulations to provide key information about the system to improve the performance of the ESC scheme. To interface the control scheme with the CFD, a coupled approach between FLUENT and Simulink will need to be established. A data driven approach is proposed with shared files used by the two programs to pass data between the CFD model and the controller.

1.4 Thesis Outline

- **Chapter 2** contains a review of the current literature that is relevant to this research. It begins with a look at the Ahmed body and the various flow structures in the wake for each of the different flow regimes. This is followed up with a review of current active flow control techniques and results, with a specific focus on steady jet actuators and synthetic jet actuators. Finally, there is a review of model order reduction schemes and results and the application of control schemes with and without reduced order model assistance.
- **Chapter 3** explains the methodology used, with a discussion of the specific CFD and actuators setups. There is also an explanation of the technical details for the reduced order model and control schemes used for this research.

- **Chapter 4** focuses on the validation of the 2D CFD simulation of the Ahmed body with the 35° rear slant along with the flow field behaviour due to actuation. The effect on the drag coefficient and the net power saving that the SJAs provide is also presented.
- **Chapter 5** presents the development of the reduced order model, which is based on the 2D CFD simulation. The accuracy of the reduced order model is discussed and the effect of the number of modes is investigated. This chapter ends with an application of the reduced order model in extending the extremum seeking controller. The extended extremum seeking controller is compared to the basic extremum seeking controller to demonstrate the improved performance allowed by the reduced order model.
- **Chapter 6** contains 3D numerical models, with a comprehensive validation, including spanwise effects, of the Ahmed body with the 25° and 35° rear slants. Results for these models with synthetic jet actuation at various jet pitch angles is presented and compared to experimental results. Results for an Ahmed body with an inverted frontal aspect ratio are presented as the aspect ratio provides a closer approximation to a truck trailer. Finally, reduced order models are presented for each of these datasets to demonstrate the ability of the model order reduction scheme to capture 3D flows. A model is also built using experimental data to demonstrate the viability of the model order reduction to experimental data.

Chapter 2

Literature Review

2.1 Introduction

This research spans multiple disciplines, from computational fluid modelling of various Ahmed body setups to flow control over the rear slant of the Ahmed body setups to low dimensional modelling and control theory. As such, this literature review is split across the disciplines and provides a background of the uses and applications of the various work. Whilst most of the low dimensional modelling and control theory applications are related to vehicle shaped bluff bodies, due to the wide scope of these areas of research, applications of the schemes to areas other than bluff bodies are also considered.

2.2 The Ahmed Body

The Ahmed body was developed in 1984 by Ahmed [6] with the aim of producing a standard car shaped bluff body upon which further research could be conducted. A diagram showing the body can be seen in Figure 2.1. It was split up into 3 sections; the front section with curved faces, designed to suppress flow separation at the front; the mid section, to allow a uniform flow to develop and the rear section with a variable slant, designed to produce a consistent wake structure behind the model.

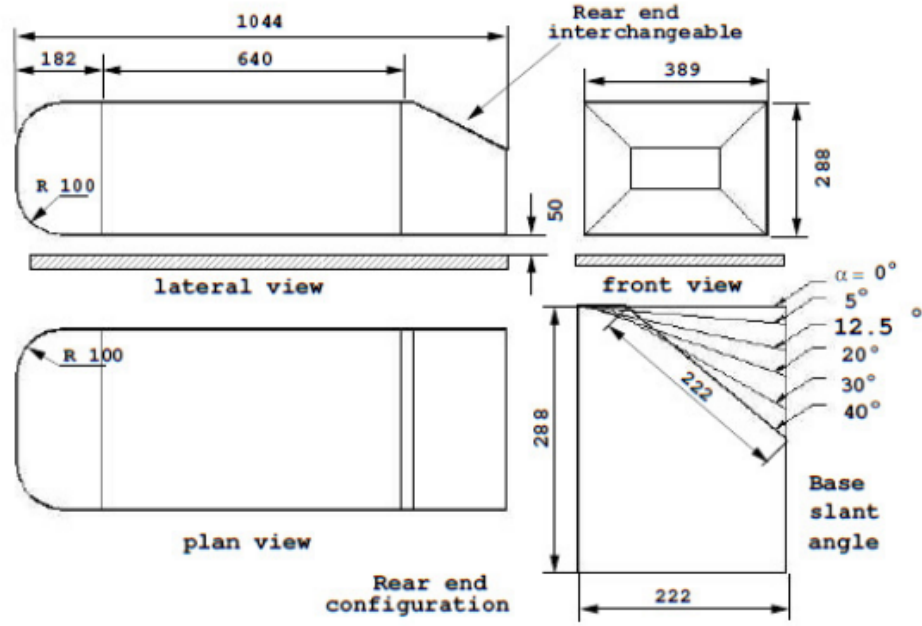


Figure 2.1: A Diagram of the Ahmed body [7].

2.2.1 The Front and Mid Sections

The curved front edges of the Ahmed body were designed to suppress the separation at the front of the body and allow a structured flow to develop over the body. The model was unsuccessful in preventing flow separation, as shown by Zhang *et al.* [8]. Zhang found that small recirculation bubbles formed on the top and side faces of the body, just downstream of the front section. These small recirculation bubbles also cause longitudinal vortices to develop along the top face of the body. It was shown by Khan [9], that the radius of the front curved edges has a major impact on the separation of the flow and therefore the formation of the longitudinal vortices. The radius of the front curves was increased by 20%, resulting in a reduction in the size of the recirculation bubble and a drop in the drag coefficient by 1.7%. Reducing the radius had the opposite effect.

The mid section of the Ahmed body was designed to allow the flow to develop into a uniform structure. The Ahmed body was largely successful at allowing the flow to develop, however, Zhang [8] found that hairpin vortices emanated from the recirculation region and travelled the length of the mid

section. These vortices were formed from the pulsating nature of the front recirculation bubbles and hence caused the flow to be highly unsteady. This caused the structure of the wake behind to model to also be highly unsteady, which resulted in a fluctuating shear layer over the rear slant. The frequency of the fluctuations can be determined by detecting dominant frequencies in the velocity field. Despite the fluctuations in the velocity field, a consistent time averaged wake structure is also present.

2.2.2 The Rear Section

The rear section of the Ahmed body was designed to produce a time averaged wake that demonstrated a consistent large eddy behaviour. The unsteady nature of the instantaneous flow, coupled with the consistent time-averaged wake structure allowed various aspects of the body geometry to be changed and tested. This included a variable rear slant which allowed the effect of the rear slant angle to be investigated. The effect of the rear slant angle is key to the flow structure of the wake and the drag that the body experiences. Ahmed [6] identified a critical angle of approximately 30° which corresponds to the stall angle for the flow. Above this critical angle, the flow separates at the top of the rear slant and below this critical angle the flow remains attached to the rear slant. separation of the flow also has a large bearing on the drag experienced by the Ahmed body which can be seen in Figure 2.2.

Figure 2.2 shows the drag coefficient vs angle of attack determined by Ahmed. There is a clear drop in the drag coefficient for angles greater than the critical angle. This indicates that the separated flow experiences a lower drag force than the attached flow for angles just smaller than the critical angle. This phenomena was explained by Zhang [8] as the flow does not remain attached over the full slant. In fact, the flow separates at the top of the rear slant and reattaches further down the slant. This detachment and reattachment causes a recirculation bubble to be formed which increases in size with the slant angle

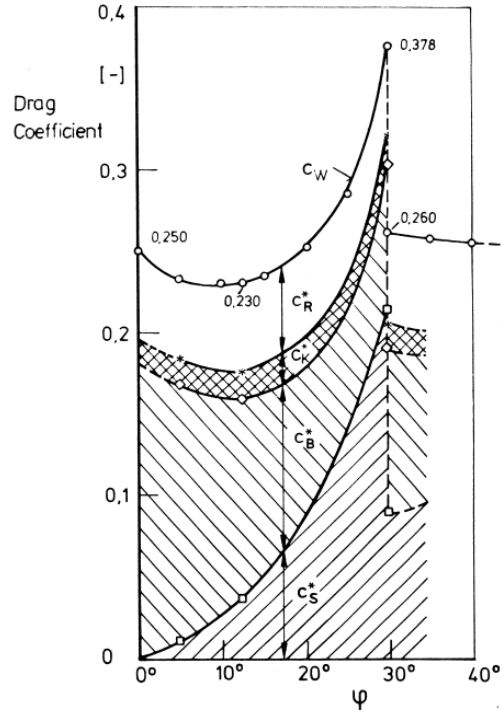


Figure 2.2: Drag coefficient against the rear slant angle [6].

until the critical angle. The recirculation bubble causes a drop in the static pressure on the rear slant, which causes an increase in the drag.

The flow regimes can be split into 3 different regimes [10], the attached low drag regime, for slant angles from 0° upto 20° , the high drag regime, upto 30° and the separated low drag regime for slant angles greater than 30° .

2.2.3 The Attached Low Drag Regime

The attached low drag regime in most cases is studied through squareback bodies. Lahaye [11] studied the squareback configuration of the Ahmed body. It was found in the centreplane of the wake, that 2 recirculation regions were present which can be seen in Figure 2.3. One generated from the flow under the model and one from the flow over the top. It was shown that the lower recirculation region was much larger than the upper recirculation region. This was due to the lower pressure at the base of the model than at the top of the model. The model also produced recirculation regions from the vertical edges of the model acting alternately to the vortex shedding from the top and

bottom of the model. These vortices produced a circular vortex ring behind the model, which is typical of squareback models.

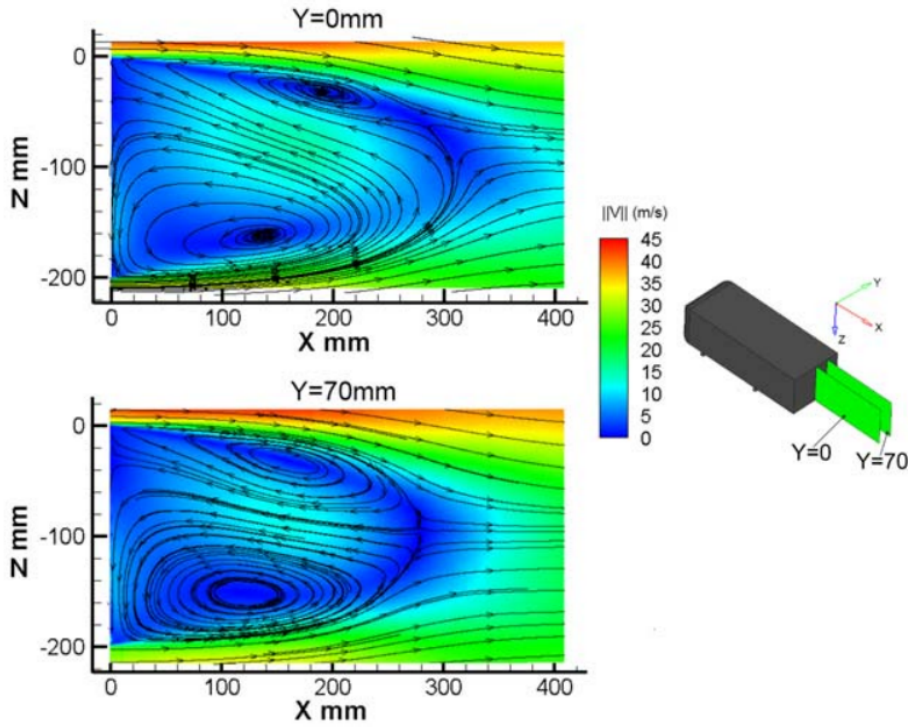


Figure 2.3: *The wake structure behind the squareback configuration of the Ahmed body [11].*

Tientcheu [12] produced a 2D numerical CFD model of the Ahmed body with a 5° rear slant which exhibited the same recirculation bubbles in the wake structure. In fact, upto a slant angle of 12° the structure of the wake remains largely the same, with the shedding vortices from the top and bottom of the model alternating with the vortices shedding from the sides of the model.

For slant angles between 12° and 20° , vortices begin to roll up over the sides of the rear slant. These are known as C-pillar vortices. The effects of these vortices are greatest in the high drag regime and are a key structure in determining the wake behaviour.

For the squareback model, all of the pressure drag experienced by the body is generated by the rear face. As the slant angle is increased, the percentage contribution to the pressure drag by the rear slant also increases. The increase in the pressure drag (5.4% - 66.4%) is greater than the increase in the slant projected area (7% - 38.5%) up to 30° [6]. This shows that the flow structures

generated by the rear slant angle are more significant to the drag, than the base rear face of the body. The body also experiences skin friction drag, with about 20% of the total drag made up of skin friction drag.

2.2.4 The High Drag Regime

The high drag regime has been investigated extensively in the literature as the flow structures in the wake are highly complex. Figure 2.4 shows the vortex breakdown presented by Zhang [8]. For slant angles above 12.5° , the flow separates at the top of the rear slant, producing a recirculation bubble at the top of the rear slant. This recirculation bubble is contained for slant angles under 30° as the flow reattaches to the slant surface.

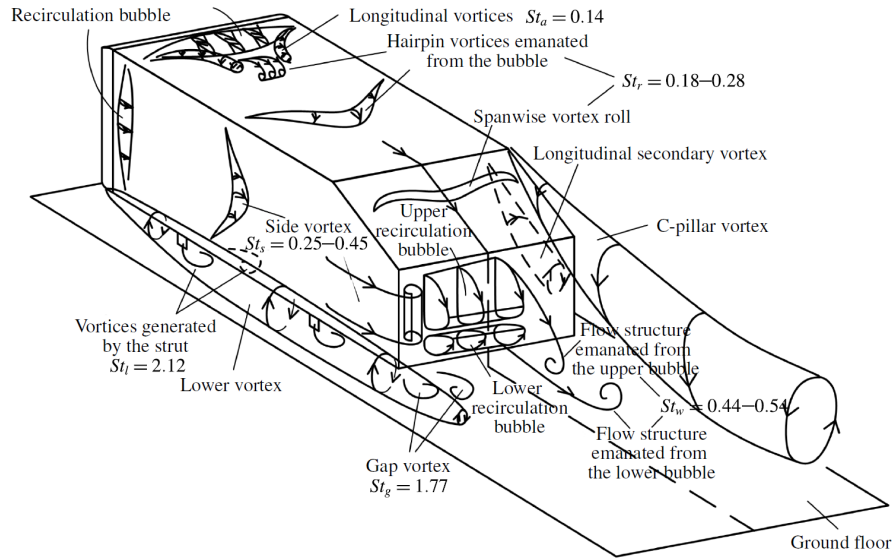


Figure 2.4: Diagram of the vortices around the Ahmed body with a 25° rear slant [8].

The flow separates again at the base of the rear slant, producing recirculation bubbles in the wake behind the lower rear face. These recirculation bubbles are similar to the ones seen for the attached low drag regime, however, the top recirculation bubble is larger than the lower recirculation bubble now. This was also shown by Minguez [13], who used a Large Eddy Simulation (LES) to produce a flow analysis of the 25° Ahmed body. Minguez presented

streamlines, shown in Figure 2.5, in the centreplane of the model. Each labelled point in the Figure represents a flow feature of the 2D Ahmed body. Point A shows the small recirculation region due to the flow separation from the curved edges of the front face and B shows the freestream flow near to the body. C and D indicate the key structures in the wake behind the body, with C demonstrating the recirculation region at the top of the rear slant, followed by the reattachment of the flow and D illustrates the 2 recirculation regions in the base of the wake.

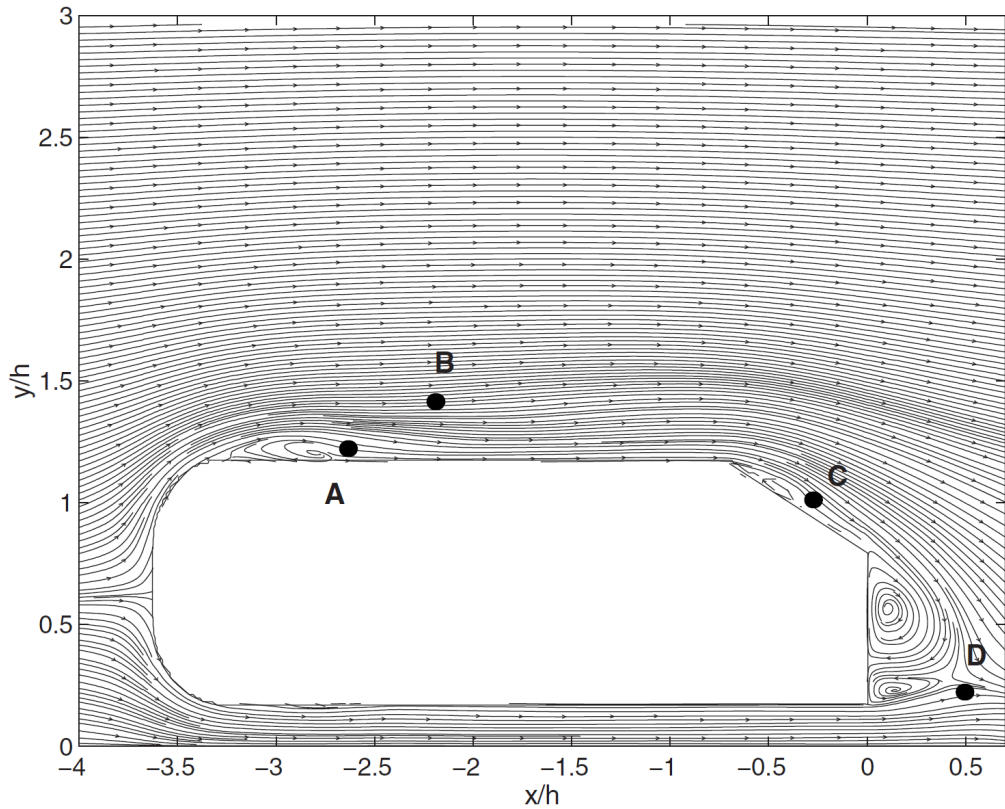


Figure 2.5: 2D streamlines in the centreplane of the 25° Ahmed body [13].

The edges of the slant on the Ahmed body produce 2 counter rotating C-pillar vortices caused by flow rolling up over the edge of the slant. The C-pillar vortices are demonstrated in Figure 2.4. These vortices build up on the rear slant, but the effects remain further downstream of the model. Lienhart [14] produced experimental results for the 25° Ahmed body and demonstrated that the circulation of the flow due to these vortices could still be detected 500mm (half of the length of the Ahmed body) downstream of the body, even after

the effect of recirculating regions in the wake had disappeared.

Although the percentage contribution to the pressure drag, by the rear slant, decreases with slant angle, the overall pressure drag experiences a large increase [6], which causes the increase in the overall drag coefficient. The C-pillar vortices play a substantial role in the pressure drag increase, due to their part in the reattachment of the flow to the slant surface. The pressure drag accounts for almost 85% of the total drag for the high drag regime.

2.2.5 The Separated Low Drag Regime

For slant angles greater than 30° , the flow experiences a separation from the top of the rear slant and does not reattach to the slant surface. Instead, the separated region over the rear slant combines with the separated region behind the lower rear face to produce a single large wake region [6]. Ahmed also showed that the strength of the C-pillar vortices was greatly reduced once the flow had separated and Gillieron [10] proposed that this was because the separated bubble began to dominate the flow post stall. Menter [15] also found that the C-pillar effect on the slant surface was diminished. This can be seen in Figure 2.6, which shows the surface streamlines on the slant surface for the Ahmed body with a 25° and 35° rear slant.

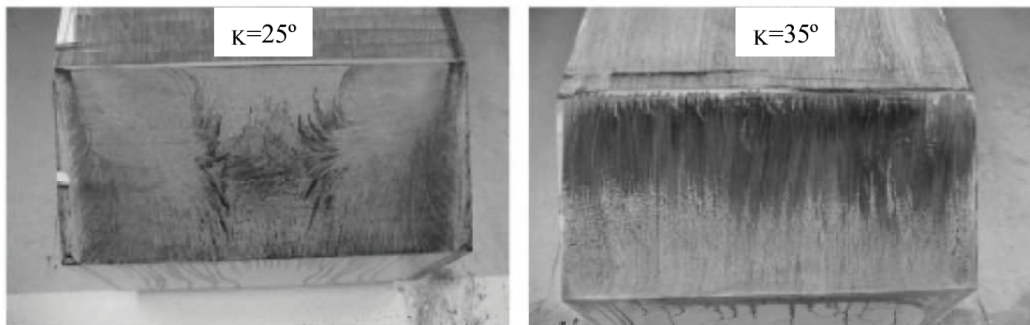


Figure 2.6: Surface streamlines on the rear slant of the Ahmed body [15]. 25° rear slant on the left and 35° rear slant on the right.

The streamlines for the 25° case were impacted by the C-pillar vortices at the edges of the body, demonstrated by the absence of streamlines in this area. The 35° case did not exhibit any effects from the side edges of the rear slant,

demonstrated by the uniform nature of the streamlines across the rear slant. The reduction in strength of the C-pillar vortices is what caused the reduction in the pressure drag compared to the slant angles just smaller than the critical angle. The pressure drag makes up about 80% of the total drag experienced by the Ahmed body in this regime.

2.2.6 The Effects of Frontal Aspect Ratio

It is known that increasing the aspect ratio of a bluff body can cause a reduction in the 3D effects that the body experiences. This is due to the flow phenomena located at the edges of the body being separated from each other. Venning [16] tested various frontal aspect ratios of the 25° Ahmed body to see how the flow around the Ahmed body was affected. The width of the body was varied whilst the height remained constant to produce aspect ratios from 1.05 upto 2.28. It was found that as the aspect ratio was increased, the horseshoe vortex formed behind the model increased in strength. The horseshoe vortex is a direct result of the C-pillar vortices combining with the expansion of the underbody flow, indicating that expansion of the underbody flow has a greater effect than the C-pillar vortices as the aspect ratio is increased.

Venning also showed that as the aspect ratio was increased, the flow over the rear slant did not reattach to the slant surface as was expected for the 25° Ahmed body, but instead exhibited a wake structure more similar to the separated low drag regime, with a single separation at the top of the rear slant and a larger recirculation region made up of the region over the slant and the region behind the lower rear face. This lead to the discussion of a critical aspect ratio, below which, the flow behaves like the high drag regime and above which, the flow behaves more like the separated low drag regime.

2.3 Numerical Modelling

There are 2 main methods that have been used to test new Ahmed body setups, these are CFD simulations and experimental modelling. CFD modelling has been performed for both 2D and 3D variations of the Ahmed body across various rear slant angles. However, the majority of studies focus on the Ahmed body with a 25° or a 35° rear slant as these two slant angles produce the flow structures and effects in the more complex flow regimes identified previously.

2.3.1 2D Numerical Modelling

The most common turbulence models used for 2D simulations of the Ahmed body are the k- ϵ ([9, 10, 17]) and the Reynolds Stress models (RSM) ([18, 19]). These models are Reynolds Averaged Navier Stokes (RANS) models and use a time averaged representation for the motion of a fluid. A turbulence model must then be used to close the system. This reduces the computational cost of approximating the movement of a fluid compared with direct computation of the Navier Stokes equations. The k- ϵ model closes the RANS equations using a two equation model for the turbulence kinetic energy and its rate of dissipation. The RSM uses transport equations for the Reynolds stresses (2.1) and an additional equation for the dissipation rate (2.2) [20].

$$-\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \left(\rho k + \mu_t \frac{\partial u_k}{\partial x_k} \right) \delta_{ij}, \quad (2.1)$$

$$\epsilon_{ij} = \frac{2}{3} \delta_{ij} \rho \epsilon \left(1 + \frac{2k}{a^2} \right), \quad (2.2)$$

where ρ is the density of the fluid, $\overline{u'_i u'_j}$ denotes the Reynolds stress tensor, μ_t is the turbulent viscosity, k is the turbulent kinetic energy, ϵ_{ij} is the dissipation rate tensor and a is the speed of sound.

For 2D simulations, Equation (2.1) provides 3 unique Reynolds stress transport equations, $u'_i u'_i$, $u'_j u'_j$ and $u'_i u'_j$, since $u'_i u'_j = u'_j u'_i$. Ansys Fluent also uses an

additional scale-determining equation, $u'_k u'_k$, for the dissipation rate, leading to a total of 5 extra equations compared to the k- ϵ model. For 3D simulations, Equation (2.1) gives 6 unique Reynolds stress transport equations, since $u'_j u'_i$, $u'_k u'_i$ and $u'_k u'_j$ can be eliminated by symmetry. This means that an additional 7 equations are needed for 3D RSM simulations and a greater computation time when compared with the k- ϵ model.

Khan [9] reports that the k- ϵ model provides a good prediction of the drag coefficient for the 25° Ahmed body, but that the drag coefficient for the 35° Ahmed body is larger than experimental data. Gillieron [10] and Kalyan [17] both report good agreements of their 25° Ahmed body simulations with the experimental data, suggesting that the k- ϵ model provides a good base on which to extend the testing of the Ahmed body. Bambhania [18] reported that the RSM provided a decent agreement with the experimental data for high Reynolds number flow around the 20° and 30° Ahmed bodies. The 30° model here is the low drag version after the flow has separated fully from the rear slant surface. Liu [19] reported that the k- ϵ model had an error of 6.8%, whilst the RSM model had an error of 8.5% when predicting the drag coefficient and its components.

It is not unexpected that the 2D CFD models can predict flow structures similar to the experimental results, as the low drag regimes for the Ahmed body are largely 2D in behaviour. It is a little more surprising that the 2D models can predict the drag and velocity profile shapes of the high drag regime, as the wake structure for the high drag regime is largely 3 dimensional.

2.3.2 3D Numerical Modelling

For 3D CFD models, a common model is the Large Eddy Simulation (LES) model ([7, 21]). Franck [21] tested the Ahmed body with a 12.5° rear slant using the LES model. This study produced a very good approximation to the system, with a lot of the key elements of the flow topology present in

this simulation. The small recirculation regions on the top and side faces just downstream of the front of the model were present along with the subsequent reattachment of the flow shortly after. A small recirculation bubble was located at the top of the rear slant, in the centreline of the model, where the C-pillar vortices were not strong enough to control the separation.

Brondolo [7] used RANS and LES simulations to compare the accuracy of the different turbulence models for the 25° Ahmed body. The two RANS models used were the realizable $k-\epsilon$ model and the $k-\omega$ shear stress transport ($k-\omega$ SST) model. The realizable $k-\epsilon$ model adapts the standard $k-\epsilon$ model 2 equation approach, using an alternative form for the turbulent viscosity and a modified transport equation derived from the exact equation for the transport of velocity fluctuations. The $k-\omega$ model uses a 2 equation model based on the turbulence kinetic energy and the specific dissipation rate. The SST version of the model limits non-physical build up of shear stresses, using a damping function, ensuring that the normal stresses are realizable [15].

Brondolo found that the LES simulation provided a more accurate simulation, as was expected due to the direct calculation of large eddies which transfer most of the momentum, mass and energy in the flow [20]. But it was also found that the RANS models offered a decent alternative for the much reduced computational cost. The realizable $k-\epsilon$ model predicted the attached flow over the rear slant whilst the $k-\omega$ SST model predicted a fully separated flow. The $k-\omega$ SST model produced a better approximation to the forces present on the body, with minimum errors of 3.8% for the SST model and 6.6% for the realizable $k-\epsilon$ model. Neither model was able to fully capture the separation and subsequent reattachment like the LES model.

Meile [22] used the RSM model to simulate both the 25° and 35° Ahmed bodies with Reynolds numbers up to 2.8×10^6 . It was reported that the pressure and velocity fields around the body were in a good agreement with the experimental data and hence produced a good agreement with the forces and drag coefficient. The RANS models are used due to their reduced computational cost, whilst

the LES model is considered to be more accurate due to capturing small scale eddies. Kapadia [23] used a hybrid of the two models called the Detached Eddy Simulation (DES) model. This model was based on the one equation Spalart-Allmaras RANS model with LES computations in regions where the mesh grid is fine enough. It was found that this technique was able to capture the minor flow structures in the flow unlike the RANS model and also produced a good approximation of the drag coefficient (7%) when compared to the experimental values. However, the computational costs are still large in areas of increased accuracy using the LES computations. For example, in the areas close to the SJAs where the mesh density is largest, or in the near wall regions, where the inflation layer is small to provide high fidelity simulations of the boundary layer development.

2.4 Flow Control

Flow control is the manipulation of fluid flowing around an object, changing the normal course of the flow, for aerodynamic gain. These effects are achieved using many different approaches which can all be categorised into active or passive flow control. Passive flow control uses changes to the physical shape of a body, using flaps or other devices. Active flow control uses the addition of an external energy input to modify the flow behaviour, such as supplying energy to drive an actuator.

2.4.1 Passive Flow Control

Passive flow control has many applications in bluff body flow separation control. While the following section is not an exhaustive review of passive flow control techniques, it covers a few key effects of passive flow control which active flow control techniques could reproduce. Heavy goods vehicles are not optimised for aerodynamic performance, hence almost 52% of the fuel burned when travelling at 100km/h is to overcome the drag [2]. While the majority of

the drag is generated by the wake at the rear of the truck, the gaps between the cab and the trailer and the underbody also contribute to the overall drag. Chowdhury [2] used aerodynamic fairing to prevent these gaps from producing large amounts of drag and found that a drag reduction of 26% could be achieved. The shape of the truck with the aerodynamic fairings is also a closer match to the squareback Ahmed body, suggesting that this method could be combined with rear wake control to further improve this drag reduction.

A common approach to control of the rear wake is to use flaps or plates to elongate the wake and reduce the strength of the vortices acting in the wake for the Ahmed body with rear slant angles greater than 12.5° . Tian [24] used flaps located at the top of the rear slant and along the sides of the rear slant for 2 different configurations. The side slants produced optimal results when angled at 80° behind the body, which caused a deflection of the C-pillar vortices and allowed the flow to separate over the rear slant of the 25° body. This caused the flow structure to resemble the 35° Ahmed body with a full separation at the top of the rear slant. This lead to a pressure drag reduction of 21%. There was very little benefit for the 35° Ahmed body utilising this setup.

Plates can be used to reduce the drag for a squareback model as shown by Khalighi [25]. Plates were installed at the rear of the model as seen in Figure 2.7. The mechanism was slightly different here as the plates were used to cause a reattachment of the flow after initial separation from the rear of the body. This causes a stabilization of the near wake and produces the cavity effect behind the model, both of which provide a base pressure increase and a drag reduction.

There have also been studies conducted using diffuser located behind the Ahmed body ([26, 27]). Although most commonly used for producing down-force, the diffuser can also be used to control the wake behind the model. The slant angle of the diffuser acts very similar to the rear slant angle at the top of the Ahmed body, with separation of the flow above a critical angle and the formation of vortices rolling up the side of the diffuser slant. It was noted by

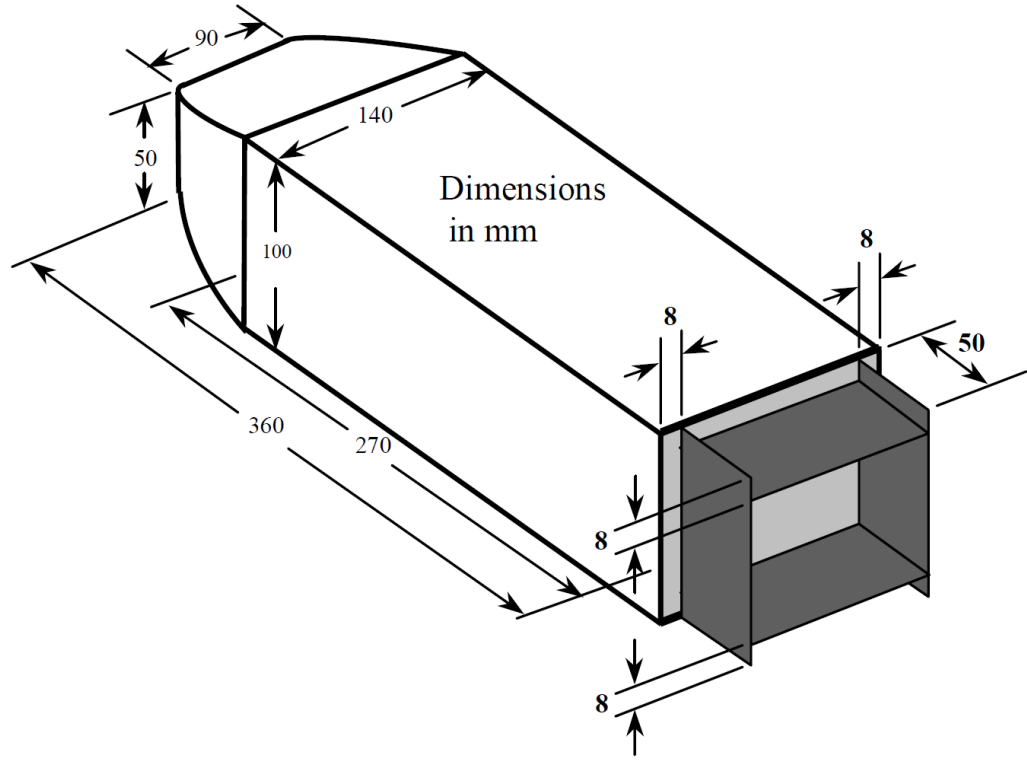


Figure 2.7: Plates located at the rear of the model to promote reattachment [25].

Moghimi [26] that for angles smaller than 8° , a drag reduction of 7.3% could be achieved by increasing the diffuser angle. This reduction was caused by the decrease in the size of the lower recirculating region. Taiming [27] reported similar findings with a critical diffuser angle of 10.46° .

The key mechanisms that are used in passive control to reduce the drag are; to suppress or eliminate recirculating regions and elongate the wake to weaken the longitudinal vortices. Suppressing the recirculating regions in most cases is not possible, but a reduction in the size of these regions leads to an overall reduction in parasitic flow structures and an increase in the base pressure that the body experiences. The elongation of the wake causes the longitudinal vortices that are generated by the front of the model and the side of the rear slants to be weakened and reduce their effects on the wake structure. In the case of C-pillar vortices, weakening the vortices causes the Ahmed body to switch from the high drag regime to the separated low drag regime, resulting in a drag reduction.

2.4.2 Active Flow Control

Active flow control is used to attempt to replicate the behaviour of the passive flow control with the added benefits of being able to adapt to changes in the surrounding flow and not requiring changes to the shape of the body upon which they act. A variety of active flow control devices have been presented in literature mainly focused on various jet control techniques, such as; pulsed jets ([28–30]), plasma jets ([31, 32]), suction and oscillatory blowing (SaOB) jets ([33]), steady jets ([34, 35]) and synthetic jets ([36–38]). The effectiveness of each method is dependent on the setup of the actuator and the Reynolds number of the flow.

A steady jet actuator provides a continuous stream of air flow from an orifice. A common approach with steady jets is to direct the flow in the wake of a bluff body to reduce the size of the wake [34]. A Windsor model was used, which is a squareback model with a slanted front end (Figure 2.8). The jets were located at the top of the rear face and the direction of the jet was varied. The best jet pitch angle, and only angle that produced a drag reduction, was with the jet pointed 45° down into the wake. This caused a deflection in the upper shear layer and reduced the height of the wake. It was reported that the greatest drag reduction of 7.5% was obtained for the highest momentum coefficient tested, however, for the greatest net power saving, a smaller momentum coefficient was preferable.

A similar approach was used by Zhang [35] for the 25° Ahmed body. Steady jet actuators were located at the top of the rear slant, the bottom of the rear slant, the bottom of the lower rear face and down the sides of the slant and rear face. The jet at the top of the rear slant was angled down into the wake and resulted in a delay in separation, increasing the base pressure on the rear slant. The jets on the side of the rear slant were used to weaken the C-pillar vortices and the jet at the base of the rear slant was directed parallel to the flow direction. The jet at the base of the rear slant prompted a reattachment

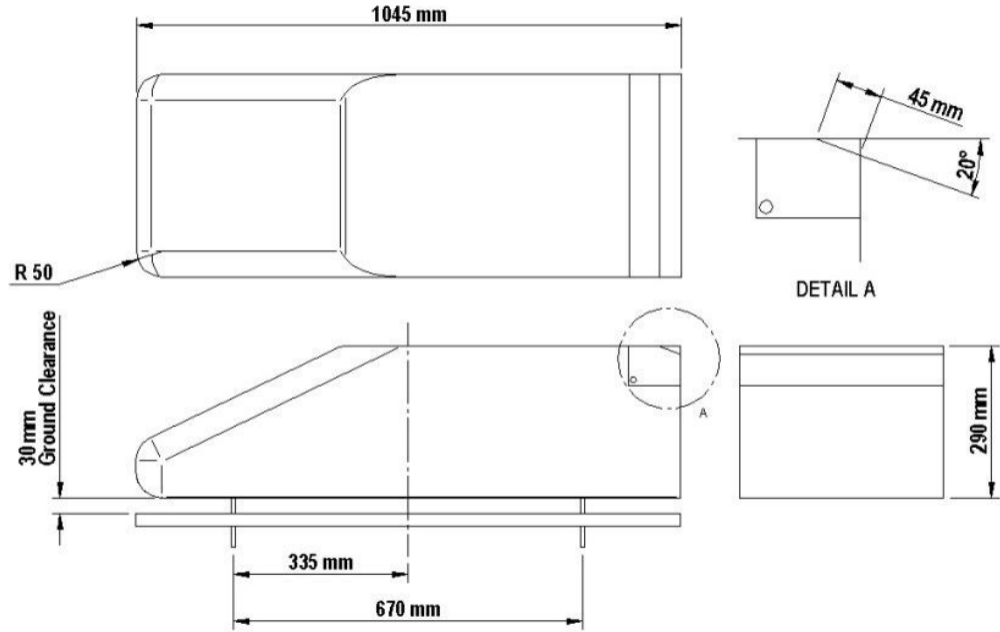


Figure 2.8: A diagram of the Windsor model used by Littlewood [39].

of the flow on the slant surface. The jet at the bottom of the lower rear face was used to deflect the lower shear layer and reduce the wake size. Using a combination of all steady jets, a drag reduction of 29% was achieved, with a 12% contribution to the drag reduction from the top actuator alone.

Geropp [40] also used steady jets directed into the wake to provide a deflection of the upper and lower shear layer at the back of the model and reduce the wake size. The experimental model used was a 2D simplified car model. The model used a suction tube at the front, which carried air through the model to the steady jets located at the back, removing the need for onboard air storage. The jets reduced the wake size and increased the base pressure on the rear face by 50%, which corresponded to a drag reduction of 10%.

Steady jets can also be used in less common setups, with the jets located at the rear corners of an Ahmed body with a jet pitch angle normal to the top surface of the body [41]. A drag reduction of 5% was achieved for the 25° Ahmed body for a Reynolds number of 90,000. Steady jets were also used to initiate an earlier flow separation on the rear window of a simple car shape [42]. The steady jet was located at the base of the rear window and blown

tangentially up the rear window, causing an earlier separation and subsequent reattachment of the flow. This was combined with steady jets blowing normal to the body above and below the rear window to disrupt the boundary layer. Drag reductions of 1% were reported, while a reduction in lift of 5% was also reported.

A large issue to overcome in the use of active flow control devices is the efficiency of the device [43]. For active flow control devices, an external energy input is needed, which requires power to use the device. In a lot of cases, these devices do not produce a large enough power saving to overcome this usage power, which leads to a net power loss. A potential method to overcome this efficiency issue is to use the devices to amplify flow instabilities to provide a larger overall impact. This is a key approach for devices such as the plasma jet actuator, which requires a large voltage and hence power to generate small velocity outputs ([44, 45]).

Plasma or Dielectric Barrier Discharge (DBD) jets are a relatively new actuation device that uses an exposed electrode in the flow and a second electrode encapsulated within the body. The electrodes are asymmetrically aligned and produce a plasma on the surface when operated. This plasma transfers ions to the external flow, injecting a momentum into the boundary layer. Most commonly they are used in flow separation delay [31, 32]. Plasma actuators have a low efficiency as a large portion of the power used is converted to heat and lost. The jets can only provide small velocity injections, Hui [31] was able to generate a peak velocity of 3.22m/s using 18kV. At low wind speeds of 10m/s, a drag reduction of almost 8% was achieved, but that dropped to only 3% for wind speeds of 25m/s. The power usage of the actuators was not presented but likely was far greater than the saving made. Roy [32] also presented drag reductions for a setup with a scaled truck model. Wind speeds of 60 to 70mph were used for this study and drag reductions of up to 15% were reported, however, it was also reported that the power used was 4.5 times that of the power saved.

An alternative to plasma jet actuators that can be used in high speed flows, such as at highway speeds, is the pulsed jet actuator. The pulsed jet actuator uses an onboard compressed air supply with fast switching solenoid valves to produce a pulsed air jet. There have been a few studies that use pulsed jets in conjunction with an Ahmed body to control the wake ([28–30]). These authors all used a squareback Ahmed body with pulsed jets located on the back face of the body. In all cases, the jets were angled down into the wake producing drag reductions of 2.2%, 10% and 10% respectively. These drag reductions did not take into account the power consumed. Barros [28] and Eulalie [29] also report that higher frequency forcing produced better drag reductions than low frequency forcing. It was reported by Barros that the improved drag reductions at higher frequencies was due to the reduced velocity fluctuations and an elongated wake.

Eulalie [29] also presented results for the 25° Ahmed body using pulsed jets and found that again the higher frequency forcing was best, with an increase of the drag by 10% for low frequency forcing but a drag reduction of between 1 and 2% for high frequency forcing. It was also found that several jet slots were better than a single spanwise jet slot, with a drop in the drag from 1 or 2% for a single slot to around 8% when using multiple slots. The drag reduction was caused by a pressure increase on the rear face, by moving the recirculating flow further downstream. This caused a small elongation of the wake.

The opposite of the blown jet is a suction applied to the flow. This has been shown to produce large drag reductions by suppressing flow separation and the effects of longitudinal vortices on the flow. Rouméas [46] studied the effects of continuous suction in the wake of the 25° Ahmed body. The suction slot was located at the top of the rear slant and was able to completely suppress the flow separation over the rear slant. This produced a drag reduction of 17% with the suction jets operated at 60% of the freestream velocity. It was found that the optimum efficiency of the suction jets was at velocities of 38% of the freestream. Suction was also found to be useful in reducing the drag

for a squareback model [47]. The suction slot was located just upstream of the rear face on the top of the model. An evacuation jet was located at the centre of the rear face to expel all the air that was sucked into the model. The combination of these resulted in a drag reduction of up to 75%. The net power saving results were not presented here. It was noted that the blowing jet worked best in lower Reynolds number flows, whilst the suction worked best in higher Reynolds number flows. The Reynolds numbers varied between 10^6 and 10^7 .

Two of the most effective active flow control techniques are continuous suction slots and pulsed blowing from a jet. Arwatz [48] combined the two techniques into a single actuator design, called the suction and oscillatory blowing (SaOB) actuator, to exploit the benefits of each technique. A diagram can be seen in Figure 2.9. The actuator uses an ejector combined with a bistable switching valve. The ejector is used to induce suction and entrain the flow through the actuator to the exit orifice. It was found that the ejector doubled the throughput of the device. The switching valve produced a pulsed jet and could operate at frequencies high enough to effectively delay flow separation.

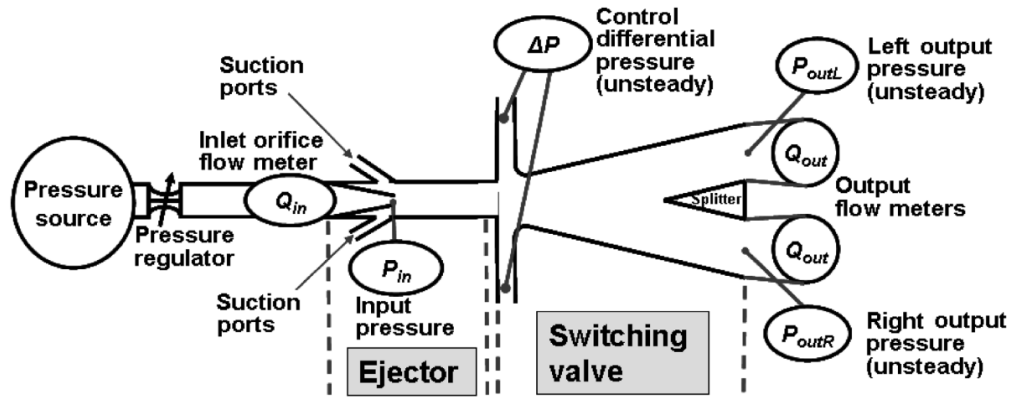


Figure 2.9: A diagram of the Suction and Oscillatory Blowing actuator used by Arwatz [48].

Seifert [33] used SaOB actuators to control the flow around a 2D squareback bluff body and then further on a 3D scaled truck body [49]. For the prior study [33], the SaOBs were located in cylinders which were attached at the

top and bottom edges of the rear face. The suction actuation was able to delay separation around the cylinders by 15-20°, directing the shear layer into the wake, whilst the pulsed jets were located after the separation. The two cylinders produced a drag reduction of 20% which corresponded to a net power saving of 10%. This is largely due to the suction actuation being the most effective flow separation scheme. For [49], a full scale model was used with SaOB actuators located in curved overhangs at the rear of a truck trailer, which can be seen in Figure 2.10. Net power savings of 4-6% were indicated. A similar study was run by ATDynamics [4], with SaOB actuators located in a curved overhang at the rear of a full scale truck trailer. Net power savings of 3% were reported for flow control using the SaOB but passive control techniques fared much better in reducing the fuel consumption of the truck, with fuel savings up to 10%.

Fluid flows act very differently at high Reynolds number compared with low Reynolds number flows, especially for the Ahmed body. For the full scale truck models, the SaOB are operated at large Reynolds numbers and are able to impact the flow and produce drag reductions. The SaOB actuators can also be used in low Reynolds number flow [50]. The SaOB actuators were located at the top of the rear slant of the 25° Ahmed body and operated at Reynolds numbers less than 4000. Drag reductions of upto 14% were reported, with the best actuation frequencies shown to be at the shedding frequency or at half of the shedding frequency.

A collection of other less common actuation techniques have been studied. A combination of steady blowing and pulsed blowing were used to reduce the drag experienced by an airfoil [51]. It was found that the addition of pulsed blowing allowed the steady blowing to be reduced significantly and produce similar effects with a reduced power consumption. Micro-electro mechanical systems (MEMS) emerged as an extension to pulsed jets in 2012 and produced significant reductions in the drag experienced by the 25° Ahmed body [52]. A maximum drag reduction of 8.8% is presented with a net power saving



Figure 2.10: *Curved overhangs on the rear of the truck to promote the conada effect [49].*

reported to be 16 times greater than the power used. The Coanda jet [53] is also reported to have produced useful drag reductions by decreasing the wake size and increasing the base pressure on the rear face of the Ground Transportation System (GTS). The GTS is a simplified geometrical model of a heavy goods vehicle with a square back, shown in Figure 2.11. The Coanda jet is a continuous jet that is blown onto a curved surface which uses the Coanda effect to entrain flow in the boundary layer. For the coanda jet study, [53], the jets were located at the side edges of the back face of the GTS model and cause the flow to remain attached around the sides and into the wake. All of the previous active flow control techniques reviewed use a jet or suction flow to provide control, but active flow control can also refer to controlling

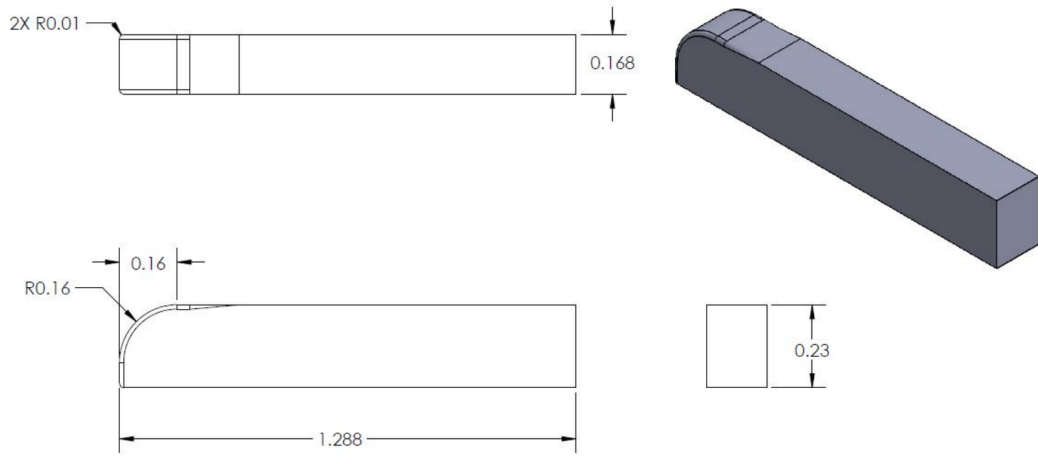


Figure 2.11: A diagram of the GTS model [53].

other devices such as vortex generators [54]. These vortex generators were located on the rear face of the modified Ahmed body (a curved rear face was used rather than a slant and vertical lower face). The electrical input is then used to change the angle of the vortex generators. A small drop in the drag was reported for angles between 0 and 50°.

2.4.3 Synthetic Jet Actuators

Synthetic jet actuators (SJAs) use a vibrating membrane located within a cavity to generate an oscillatory suction and blowing motion. The blowing phase uses a contraction of the space in the cavity to drive the fluid out of the orifice. This causes two counter-rotating vortices to be formed at the orifice exit, as demonstrated in Figure 2.12. The flow is contained mostly in the centreline of the jet. During the suction phase, the outer flow is drawn in through the orifice due to an expansion in the size of the cavity. During this phase, the flow is largely two dimensional and can be thought of as a sink at the orifice [55]. The interaction of the vortices with the suction flow is determined by the strength of the vortex and its distance from the orifice, meaning that if the velocity generated during the blowing phase is large enough, the vortices will propagate to the far-field region. The benefit of these devices is that they are zero net mass flux (ZNMF), meaning that the jets are generated by adding

momentum to the working fluid, hence no extra mass injection is needed. This means that the devices are lightweight and easy to fit to a body for flow control purposes.

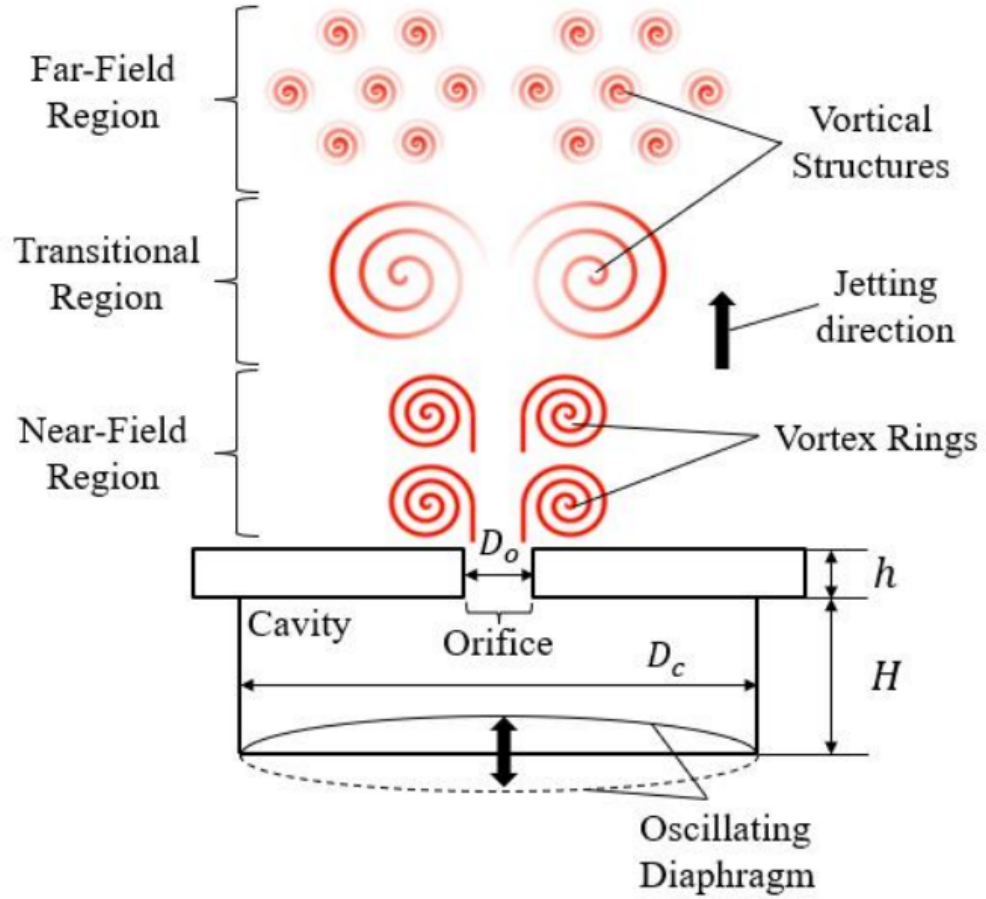


Figure 2.12: Idealised representation of the vortices formed by the SJAs [56].

They can be optimised for their application, by tuning the operating frequency to match a given frequency in the flow, or they can be optimised for the highest peak velocity [57]. Figure 2.13 shows the setup used by Gungordu [57] with the slot adjacent configuration. The most common approach, used in all applications found in the literature review of this thesis, is to use a slot opposite to the vibrating diaphragm. The advantage of the adjacent orifice is that two jets can use the same diaphragm, which reduces the waste cycle of an opposite configuration, improving the efficiency of the devices.

Gungordu [57] produced an SJA that was 39% efficient and could produce jet velocities up to 58m/s. The efficiency of these SJAs is key to reducing the net

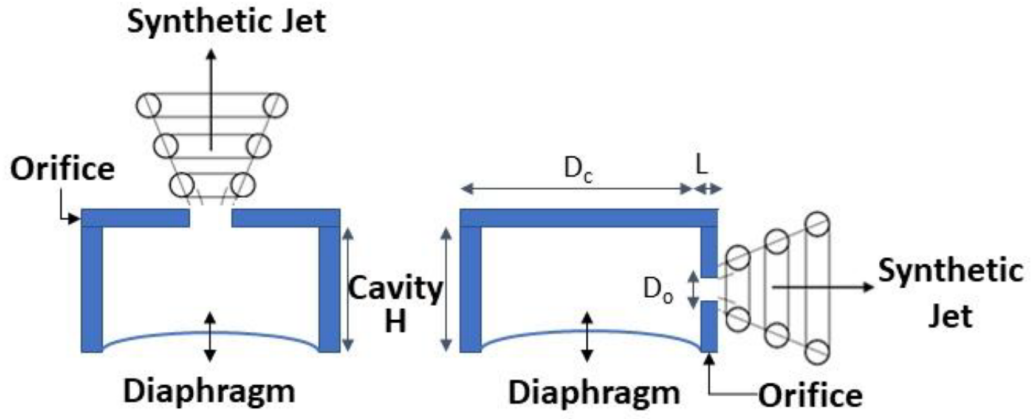


Figure 2.13: Slot opposite (left) and slot adjacent (right) configurations for the synthetic jet actuator [57].

power saving for control purposes. The jet velocity of an SJA is commonly used to mean the peak jet velocity at the centreline of the orifice exit, since the velocity at the orifice is constantly changing. Velocity optimised SJAs [57], use higher frequencies than naturally appear in fluid flows, with a resonance frequency of 2.8kHz obtained by Gungordu.

SJAs are the subject of studies for a wide array of applications, with aerodynamic studies focusing on controlling the lift and drag of bodies. A common use for SJAs is in the control of flow over an airfoil, with particular interest in boosting the lift coefficient, whilst also reducing the drag that the airfoil experiences. Studies by Honoham [58], Amitay [59] and Glezer [60] present large performance gains for the airfoils under actuation. The SJAs were able to delay flow separation from the aerofoil and thereby increase the lift that the aerofoil experienced [58]. The SJAs also increased the lift-to-drag ratio of the airfoil by 27% for high frequency actuation [59] and produced a 15% increase in lift while reducing the drag by 7% [60].

SJAs have also been proven to be effective in the control of flow around a variety of bluff body configurations from cylinders ([58, 61]), to squareback configurations ([36, 38, 62]) and even the Ahmed body ([37]). For the flow around a cylinder, the SJAs were able to delay the flow separation by upto 20° of the cylinder [58]. Naim [61] also showed that by combining the SJAs with

a splitter plate located at the rear of the cylinder, the base pressure increased behind the model and the pressure drag that the cylinder experienced was also reduced.

The studies by Henning [36], Henning [41] and Pastoor [38] are all based on similar setups with a D-shaped bluff body, with a streamwise cross section resembling that of the squareback Ahmed body cross section. The experimental setup can be seen in Figure 2.14. The SJAs were located at the top and bottom of the rear face and angled at 45° to the freestream direction. The SJAs consisted of a loudspeaker in a cavity with a single slot exit of 250mm over the centre portion of the body. The actuation frequency was equal to the shedding frequency of the flow over the body and the momentum coefficient was varied until an optimum was found. Each study produced increasing drag reductions of 10% [36], 15% [41] and 15%, with 4 times the actuation power returned in net power saving [38]. This demonstrates that the SJAs can be used to reduce the net power consumption of a bluff body flow.

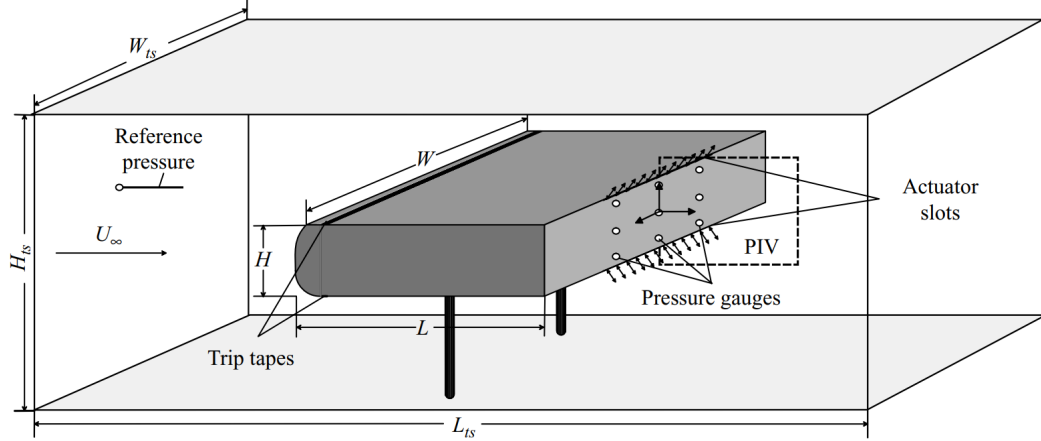


Figure 2.14: The D-shaped body used by Pastoor [38].

SJAs have also been used to delay flow separation around sharp edges as shown by Minelli [62–64]. Each of these studies focuses on the front A-pillar of a truck cabin which is a sharp edge that results in a flow separation. SJAs were located on the A-pillar (Figure 2.15) and used to reattach the flow by deflection of the shear layer towards the side surface of the truck cabin. This resulted in an increase in the base pressure and a drag reduction of upto 34% [62].

This result was improved further by Minelli [64] with LES and experimental analysis. The length of the recirculation bubble was reduced by 80% and the drag was reduced by 50%. These results were obtained using streamwise SJAs blowing tangential to the surface at a Reynolds number of $Re = 5 \times 10^5$.

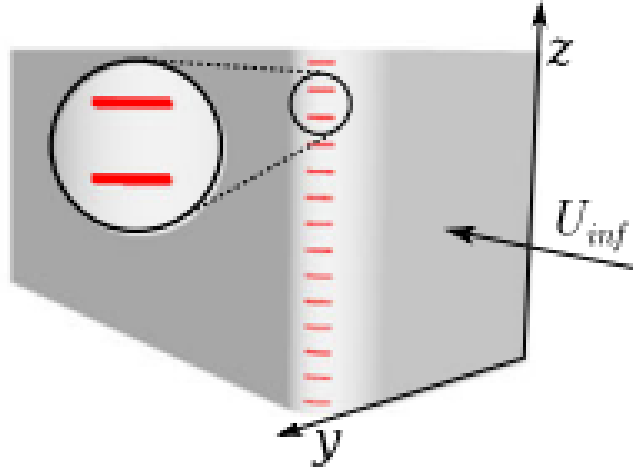


Figure 2.15: *The streamwise SJAs located on a truck A-pillar [64].*

El-Alti [65] used SJAs located at the rear of a truck trailer combined with flaps to reduce the drag. The SJAs delayed the separation of the flow, which remained attached to the flaps at the rear of the trailer and was directed into the wake behind the body. This caused the recirculating regions to move downstream and increased the base pressure on the rear of the truck body leading to a decrease in the pressure drag. Similar results can be seen from the study by Gil [66]. Gil used SJAs located behind the rear of a bullet shaped body. The effects of a single SJA were compared to the effects with multiple SJAs and it was found that the single orifice produced the greatest drag reduction. It was noted that increases in the freestream velocity reduced the effectiveness of the SJAs to control the flow and reduce the drag. A maximum drag reduction of 6% was achieved for Reynolds number between 88,000 and 352,000.

Littlewood [39] used SJAs for control around a cylinder and reported that the small momentum coefficients produced by the SJAs lead to an inability to control the flow at high Reynolds numbers due to the overwhelming cross flow

velocities. Pulsed jets were instead used for future testing as they provided larger momentum coefficients and thereby produced greater drag reductions. The pulsed jets worked well at low actuation frequencies but most of the performance was lost at higher frequencies, with drag increasing by upto 12%. Park [37] used SJAs located at the top of the rear slant of an Ahmed body with a 25° and 35° rear slant angle, which can be seen in Figure 2.16. The Reynolds number was 4.29×10^5 with a freestream velocity of 20m/s. The SJA pitch angle was varied to test for the greatest drag reduction and it was found that the closer to freestream direction that the jets operated in, the greater the drag reduction was. The maximum drag reduction achieved was 5.2% It was also found that the SJAs at the edges of the body had less of an impact on the drag reduction and that by concentrating the jets in the centre of the body, the greatest efficiency could be achieved.

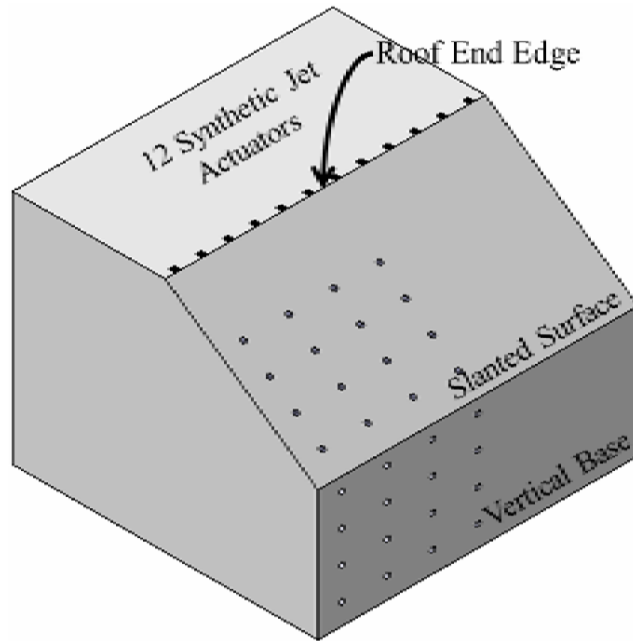


Figure 2.16: The locations of the SJA orifices at the top of the rear slant for the study conducted by Park [37].

Kourta [67] also presents experimental data with SJAs located on the rear slant of the 25° Ahmed body. The location of the jets were varied around the top edge and it was found that the best location was 2mm from the top edge of the rear slant. It was found that changes to the frequency did not have a

major impact on the drag reductions for the body. With the jets operated at 600Hz, the maximum drag reductions were 8.5% at $\text{Re} = 1.2 \times 10^6$ and 6.5% at $\text{Re} = 1.9 \times 10^6$.

2.5 Reduced Order Modelling

Reduced order modelling is the process of taking a high dimensional system and creating a low dimensional approximation that captures the key flow dynamics of the high dimensional system. The most common approach used is the Proper Orthogonal Decomposition (POD) method, which many other model order reduction (MOR) schemes build upon. POD schemes in fluid dynamics are used to decompose physical fields, most commonly velocity fields [68, 69], but can also be applied to pressure fields. The POD approximation is designed to represent a random vector field $\gamma(x, t)$ as a combination of its spatial components, $\phi_i(x)$ and temporal coefficients, $a_i(t)$. These are combined in Equation (2.3) [70].

$$\gamma(x, t) = \sum_{i=1}^{\infty} a_i(t) \phi_i(x) \quad (2.3)$$

The empirical correlation matrix is calculated and the eigenvalue problem is solved to produce the eigenvectors of the field $\phi_i(x)$. These eigenvectors represent an orthonormal set of basis functions. The eigenvalues represent a measurement of the energy that each mode contains and so can be normalised to produce a percentage importance to the system. The number of modes can be varied to balance the computational complexity of the problem and the information loss by removing the low energy modes. A major decision to make when reducing the order of a model is dependent on its intended use. Cohen [70] made the observation that the number of modes required to reconstruct the flow field was different to the number of modes needed to provide a desired control scheme. The control scheme could be operated using a single mode,

but to capture the flow dynamics in reconstructing the flow, 6-8 modes were needed. Most studies using POD have been conducted on small [68] to medium scale problems with upto 8000 mesh elements [69].

An extension of the POD scheme, is the Balanced Proper Orthogonal Decomposition (BPOD) scheme, that uses a balanced truncation of the POD modes. The idea of the balanced truncation is to retain information about the input and output effects on modes when producing the reduced order approximation. This is achieved by computing a controllability and observability Gramians by solving the Lyapunov equations [71]. Consider the linearised input-output system,

$$\dot{x} = Ax + Bu \quad (2.4)$$

$$y = Cx \quad (2.5)$$

where $u(t) \in \mathbb{R}^p$ is a vector of inputs, $y(t) \in \mathbb{R}^q$ is a vector of outputs and $x(t) \in \mathbb{R}^n$ is the state vector. Then the Lyapunov equations are,

$$AW_c + W_c A^* + BB^* = 0 \quad (2.6)$$

$$A^*W_o + W_o A + C^*C = 0 \quad (2.7)$$

where W_c is the controllability Gramian and W_o is the observability Gramian. The controllability Gramian is a measure of how an input excites a given state, while the observability Gramian provides a measure of how effective a state is at exciting the output from the system. By using the best modes from each Gramian, smaller inputs can be used to produce larger outputs as states are more responsive to excitation.

The Gramians have different coordinates, which makes choosing the states to retain hard as the most controllable states are not necessarily the most

observable. A balancing transformation needs to be calculated by transforming the system so that the controllability and observability Gramians are equal and diagonal [72]. This transformation ensures that the most controllable states are also the most observable. This balanced truncation is far too complex to compute for large systems, with more than 1000 states [73]. The balanced truncation can then be applied with the POD modes in the BPOD method. Suppose the Gramians can be rewritten as,

$$W_c = R_0 R_0^* \quad W_o = R_1 R_1^* \quad (2.8)$$

where W_c and W_o are square matrices, then, by the method of snapshots [71], the balancing modes are computed by the SVD of the matrix $R_1^* R_0$, where R_0 and R_1 are data matrices. The BPOD method has a similar computation time to the standard POD scheme whilst providing a better control response due to the retained states having a stronger observability and controllability. The adjoint snapshots do have to be computed for BPOD unlike the POD scheme, which can significantly increase the computational time for very large matrices.

A more recent MOR scheme is the Dynamic Mode Decomposition (DMD) scheme. It differs from the POD scheme as DMD produces a low dimensional subspace and the model from the snapshot data [73]. Snapshot matrices are formed from empirical data, whether that is experimental or numerical, where the columns of the matrices are the vector field snapshots at each timestep. Two matrices are formed, with the last snapshot removed from one and the first snapshot removed from the other. These matrices can then be used to determine a relationship between the timesteps. The typical choice for the low dimensional projection basis is the POD scheme here. The POD modes are used to produce a low dimensional approximation of the model, from which the DMD modes are calculated by computing the eigenmodes of the model approximation. Each DMD mode provides spatial information for a given

temporal frequency in the system [73].

The DMD model alone is not useful for control of a system with inputs and outputs, hence an extension was developed called input output DMD (IODMD). This extension uses a projection of the system into the output subspace, which in turn also accounts for the inputs to the system. The method process is discussed further in Chapter 3. The IODMD scheme produced a similar model to the BPOD scheme for a small scale wind farm approximation [73], with the major benefit of IODMD being that there is no need to compute the adjoint matrix of the BPOD scheme. IODMD has been compared to identification methods, such as the N4SID scheme [74, 75], but IODMD has a benefit over these schemes since it retains information about the inputs and outputs allowing IODMD reduced order models to be used for control purposes [76].

Since IODMD is a time domain model, it is not necessarily true that the reduced system created by the IODMD scheme is stable. A method was proposed by Benner [77] to apply a stabilization to the model derived by the IODMD scheme. The stabilization method used a gradient optimisation based approach. The stabilization approach was successfully applied to a variety of unstable IODMD models whilst the models still retained the properties required to approximate the output response.

A further extension of DMD is the DMDX scheme [77] which separates the system dynamics from the input dynamics. A two step approach is used, first allowing the system to evolve without any inputs to capture the intrinsic flow dynamics and the decay of the system. This step uses a basic DMD approach. The second step is to capture the dynamics of the system under external inputs. Again, this system can be identified using existing methods such as IODMD. The system state is then regarded as a superposition of the 2 states. By separating the actuated system from the intrinsic flow system, the behaviour of the input is decoupled from dependence on the state of the intrinsic system. This could prove useful in cases where changes in the flow state cause a variation in the system response to inputs.

The MOR schemes discussed here can be used to generate Linear Time Invariant (LTI) systems which provide approximations to a nonlinear system at various working points. To fully describe the nonlinear system, the LTI models need to be combined into a Linear Parameter Varying (LPV) system. This usually entails an interpolation between the local LTI models to produce a global LPV model for the system [78, 79] or a linear regression by the least squares method [80, 81]. The issue with interpolating between LTI models is that the local LTI model may not be unique as reduced order models (ROMs) do not retain any information about the structure without careful consideration [79]. This means that the LTI models have to be made coherent before interpolation can be used. To overcome this, structural assumptions can be encoded into the LTI systems. It is also noted by Zhang [79] that when input-output models are used, the coefficients of the LTI systems can be interpolated without issue as these systems are coherent as the input output behaviour retains a partial structure of the system.

An alternative approach is to interpolate the outputs from the local LTI systems [79], as this does not rely on the form of the LTI systems. The downside to this approach is that the output of multiple LTI systems has to be computed. Zhang noted that this approach worked well in cases with slow transitions between working points, however, for systems with fast transitions, an LPV model needed to be determined using interpolation between coherent LTI models.

Bolea [82] proposed a couple of alternative methods to obtain an LPV system and compared the results to the interpolation between LTI schemes. The first alternative was to use physical modelling of the system. That is the laws that govern the flow parameters, the operating conditions and the external factors were accounted for. The other alternative was LPV identification, in which the varying parameter for the LPV system was unknown before simulation. They are measured or estimated upon the start of the simulation. It was found that all 3 methods are consistent and suitable for control purposes.

The LPV models can be effective when obtained from experimental data and used to operate actuator devices [5]. In this study, steady jets were located on a squareback bluff body in cross flow gusts. The LPV controller, in conjunction with the steady jet actuators was able to produce a net power saving of 8%. Motta [83] used a technique based on the Van der Pol (VdP) oscillator to control the wake behind an aerofoil using SJAs. The VDP oscillator was used to construct a low dimensional approximation to the full order system which was able to capture the limit cycle oscillations present in the wake of the wing. It was noted that the SJAs produced effective control as long as the frequency was different from the natural frequencies present in the flow. VDPs are widely used to model turbulent flow, but very few studies use this technique for real time control of a system, however, Motta demonstrated that the model provides a good ability to provide the control for the system.

2.6 Closed Loop Control

Closed loop control (CLC) is the process of measuring changes to the outer flow state and adapting the response of an actuator to compensate for the changes. This can lead to improved performance of the actuator through optimisations for the specific flow conditions, whilst, in some cases, even allowing the model to respond to systems it has not been tested in before. A common CLC scheme is the Extremum Seeking Control (ESC) scheme.

2.6.1 Extremum Seeking Control

The ESC scheme is a gradient, or slope seeking algorithm that attempts to optimise a cost function. The extremum seeking controller can take many forms, but the most common approach is the perturbation based method, which allows for fast adaptations [84]. No model information is needed for the perturbation approach as a small perturbation is provided to the system and the response is measured. This is used to approximate the gradient of

the cost function and the current approximation is moved in the appropriate direction. Krstić [85] showed that the perturbation approach to ESC was stable, hence, providing a strong backing for the ESC scheme. It was also demonstrated that there is a small neighbourhood around the optimum, for which the scheme will always converge, provided that the initial estimate is within the neighbourhood. The full process for ESC is explained in Chapter 3, and this section will focus on the applications of the ESC scheme.

As part of the studies on SJA interactions for the reduction of flow separation behind a squareback Ahmed body, Henning [36, 41] also used ESC to provide a CLC scheme for the model. The ESC scheme was used to control the phase of the SJAs. The ESC scheme was able to determine an optimum and it was found that by using the ESC scheme, only half of the actuation energy was required to achieve the same drag reductions when compared with open loop control. Henning [41] was later able to reduce the time needed to reach an optimal solution by 20% by extending the ESC scheme to use a least-squares algorithm to approximate the slope. The time was reduced further still with the application of an extended Kalman filter. Both of these extensions replace the classic filters and perturbation to the system with a system identification method to approximate the local gradient of the system. Even with the convergence time reductions the ESC scheme is considered slow for real time control.

One of the approaches of flow control is to delay the flow separation, Beaudoin [86] demonstrated this using a rotating cylinder located at the edge of the rear face of a bluff body. An ESC scheme was used to control the rotational frequency of the cylinder to balance the power used and the power saved. The ESC scheme was able to find an optimum, which produced an energy saving. This system is at a steady freestream state and does not provide any information for real time control and adaptations of the controller to changes in the flow state. However, Beaudoin later showed [54] that the ESC scheme could in fact adapt to changes in the outer flow.

This study [54] used an ESC scheme to control the angle of vortex generators on the rear face of an Ahmed body. The ESC scheme set the vortex generator angle to 19° when the freestream flow is 40m/s. The freestream was changed to 20m/s and the ESC scheme was able to react and eventually settled at an angle of 30° . However, it took almost 500 seconds for the ESC scheme to adapt to this change, suggesting that it may not be suitable for real time control.

ESC schemes have also been used to control the flow around cylinders [87]. The ESC scheme was used to control the time delay between the suction and blowing cycle of SaOB actuators. The aim was to control the vortex shedding by matching the frequency with the SaOB actuators. The ESC scheme was able to completely attenuate the shedding but again took about 400 seconds to converge to the optimal, even beginning in the wrong direction and breaking the feedback loop.

ESC schemes can also be used to optimise multiple variables in a system. Kamagata [88] was able to successfully control the lift and drag forces experienced by a bluff body by controlling the rotational velocity of cylinders on the front corners. The cylinders promoted reattachment of the flow to the side surfaces. The convergence of the two outputs to their control targets was much faster than for other studies, requiring just 60 seconds to reach their targets. This suggests that the ESC scheme may be able to provide real time control if the system is optimised far enough. Brackston [89] was also able to successfully implement an ESC controller using multivariate control, which found the optimum forcing within 100 seconds. This ESC scheme attempted to optimise the pressure on the rear of a bullet shaped bluff body and the frequency of pulsed jets. The slow convergence was attributed to the step like manner used to explore the 2D control space.

Advancements in the ESC scheme allowed Papageorgiou [90] to control a deflector on top of a truck trailer cabin with a much faster convergence and response to flow changes. The ESC scheme varied the height of the deflector to reduce the drag that the truck experienced. The controller converged in 25

seconds, which is much faster than any previous studies presented, however, this is still considered too slow to be practical. An observation of note is that the convergence rate of the ESC scheme increases with frequency of the perturbation, meaning that a higher frequency could lead to more practical real time control, however, this could also lead to an interference with the natural frequencies in the flow [84].

An alternative to the perturbation based ESC scheme is the model based ESC scheme [91]. The model based ESC scheme is largely used in the real time optimisation of bioprocesses and chemical reactors. A full or partial knowledge of the system structure is needed for this approach, but the convergence and ability of the system to run in real time is far greater than the perturbation model. Obtaining this knowledge for highly turbulent fluid flows is difficult, which is where the use of ROMs is key.

Extremum seeking control can be extended with other methods to improve the real time performance. By using ESC to optimise the passivation parameters for a car adaptive cruise control system, the performance of the cruise control system can be greatly improved [92]. Passivation theory is not discussed as part of this research, but this clearly demonstrates the wide scope that ESC has and the ability of the ESC scheme to optimise problems.

Alongside the issue of convergence time for the ESC controller, issues with the extrema that the ESC converges to also arise. If the initial estimation of the optimum control variable is too far from the global optimum and too close to a different local optimum, the ESC scheme may converge to the local optimum. Further extensions to the ESC scheme were presented by Li [93, 94] which used various forms of sampling algorithms to sample the whole of the control space, whilst the gradient optimisation controlled the flow. By sampling the whole of the control space, a better minimum could be identified, leading to a better performance of the controller.

2.6.2 Alternative Closed Loop Control

ESC may be the most common form of CLC scheme that is used in the literature, but there are many other approaches that range from feedback control, such as the ESC, to feedforward control designs [95], which measure disturbance variables and take corrective action before they affect the flow and combinations of the two. Kumar [95] presents a review of other control schemes and schemes for tuning control systems which are not covered here.

Feedback controllers use a sensor to measure changes in the flow and react accordingly. There are a few examples of this type of flow in the literature that do not fall into the ESC category [38, 96, 97]. The model proposed in [96] uses a mathematical model of planar Poiseuille flow with MEMs and sensors to detect the formation of instabilities in the flow and suppress them. Pastoor uses a phase control scheme to control the shear layer behind a squareback bluff body. Sensors on the lower face of the body monitor for fluctuations that give rise to the shedding vortices behind the model, while the actuator located on the top of the model attempts to match the shedding, to produce a symmetric shedding from the top and bottom of the model. The controller uses a simple sine function to approximate the fluctuations by matching the phase and frequency of the fluctuations from the base. The feedback model used in [97] consists of a compensator, built from an estimator and controller. The estimator is used to choose a control output based on a given outer flow state and the controller is used to convert that control input to a state change. The feedback controller used by Hall [98], measured the error in the feedback by subtracting the actual flow state from a reference flow state. This allowed the controller to determine if the flow was attached, separated or incipient. The controller then could determine the best actuation velocity to supply to the SJAs to control the aerofoil flow. A method involving multiple time-scales was compared to the single time scale model, which resulted in considerable improvement.

Feedforward controllers use disturbances in the flow to predict and suppress changes before they occur. One such method is the use of statistical learning to develop a model for the flow which can predict changes. This model is known as the Auto Regressive Moving Average Models with exogenous inputs (ARMAX) model [97, 99]. The ARMAX system is very powerful as it requires no numerical modelling of the physical system and the resulting controller is effective for real time control. The model was tested on the flow around a cylinder [99] and over a backward facing step [97] and in both cases was able to provide a good control response.

A feedforward controller can also be used with a numerical model for the system. The Tollmein-Schlichting wave is a streamwise unstable wave that arises when a laminar bounded shear layer transitions to turbulent [100]. Engert used SJAs to delay the formation of the Tollmein-Schlichting instabilities on an aerofoil by producing a counter wave using the principle of wave superposition. An adaptation algorithm is used to calculate this counter wave from the measured disturbances on the aerofoil.

The final alternative discussed in this section is the controller using machine learning, specifically a linear genetic programming (LGP) approach to determine the optimum of a system [101, 102]. Linear genetic programming is an evolutionary algorithm. A randomly sampled set of control laws is established and the ones that perform the best are retained for the next generation. Mutations are applied to all control laws in passing to a new generation and the process is repeated until all control laws converge towards the same performance. In the case of [101], the LGP algorithm is used to maximise the pressure increase on the rear face of a squareback Ahmed body. This method performs well in real time and results in a pressure drag reduction of 22%.

Han [103] wanted to run a controller developed in Simulink on a CFD simulation running in Fluent. The study in [103] presents a data driven approach to achieve this. Simulink and Fluent run in parallel and each print data and retrieve data from the same files. Whilst this is not strictly a CLC scheme, it

provides a framework for testing control schemes in CFD simulations.

2.6.3 Summary

Various turbulence models are used throughout the literature with good results presented with all models. The LES model presents accurate approximations of the eddies when compared to the experimental data, but comes at a heavy computational cost. The DES model also represents a good approximation but the computational cost is still very high. The RANS models are considered much less accurate but represent a more practical turbulence model due to the largely reduced computational cost. Each of the RANS models have strengths and weaknesses when it comes to approximating flows. The realizable $k-\epsilon$ model is best when approximating jet flows and works provides an accurate approximation for the 25° Ahmed body. The $k-\omega$ SST model predicted a separated flow over the 25° rear slant but only has a 7% error in the drag prediction, whereas the realizable $k-\epsilon$ model has an error closer to 18%.

Throughout the literature discussed, various body shapes and sizes are used. For the Ahmed body, the experiments are typically conducted on the same scale as the one developed by Ahmed [6], whilst CFD simulations tend to focus on smaller scale Ahmed bodies, usually 1:2, but even a 1:3 scale model is used. For all the models, the typical inlet flow is 40m/s, but the varying size of the Ahmed bodies means that the Reynolds number varies between 390,000 and 780,000 based on height or 1,400,000 and 2,800,000 based on length. The varied Reynolds number is key to this flow as this can have a drastic effect on the flow structures around the Ahmed body. For lower Reynolds number flows, the separation bubbles on the front and sides of the model may be reduced in size or not form at all. This could lead to the vortices generated from these bubbles also being reduced or not forming leading to changes at the rear of the body.

At the current state of research, passive control techniques tend to produce

a greater drag reduction effect and due to the lack of input power needed, produce the greatest net power saving as well. However, with the increasing efficiency of active control techniques and the potential to produce large changes to the flow structure, active flow control is becoming a viable alternative. The most successful approaches for high speed, highly turbulent flow are suction, pulsed jets and steady blowing. Studies involving SJAs lack the momentum coefficient to make any real changes to the flow structure. There are two methods presented to solve this issue; frequency matching the jets with natural flow frequencies, such as the shedding frequency, to amplify instabilities in the flow structure; or using a velocity optimised SJA to increase the control authority of the actuators. Streamwise SJAs can be used to deflect the shear layer of a flow towards a surface and reattach or even delay flow separation. Steady jets blown into the wake have been shown to also cause deflection of the shear layer and reduce the wake size behind a bluff body, thereby increasing the base pressure and reducing the pressure drag. Jets operated at higher frequencies tend to produce a larger drag reduction in high Reynolds number flows and by operating an SJA at high frequency, the velocity fluctuations could be minimised and the flow react to the jet more like a steady jet with deflection of the shear layer.

Reduced order modelling is common for turbulent flows due to the need to understand the dynamics of these highly complex flows. POD, DMD and their extensions have been shown to produce reliable models for low to medium scale turbulent flows, for systems upto 10,000 states. It has also been shown that the number of modes needed to control the dynamics of the system is lower than the number of modes needed to reconstruct the full flow field from the base elements. LPV models are an extension of the LTI models which allow accurate local models to be developed whilst retaining the ability to produce a global model. The issues with LPV systems focus on the interpolation between non-coherent LTI models, for which alternatives have been developed. LPV models have been used in control schemes to a good effect, resulting in reasonable net

power savings. These models can be combined with closed loop control schemes to improve upon this further.

Extremum seeking control is a commonly used approach for closed loop control schemes, due to its ease of use and its model free nature. However, it is currently unsuitable for real time control due to the slow convergence time upon abrupt changes to the outer flow. A model based approach drastically improves the convergence time but it can prove difficult to obtain the model information. Extensions to the ESC scheme in terms of sampling algorithms show promise in ensuring more accurate approximations to global optimums are reached but do not aid the convergence. Other models have been presented that require numerical models, which is likely unsuitable for highly complex turbulent flows. Models that use statistical data show great promise with real time performance and reasonable accuracy, whilst machine learning provides another approach to real time control. The main issue with machine learning is the large training data sets that are needed and the long training times needed to produce a good model.

Chapter 3

Methodology

3.1 Introduction

The Ahmed body is a well known model used in flow control research, to develop methods to improve the drag reduction of trucks and other land vehicles. It is a bluff body, with curved front edges and a variable slant surface at the rear. Figure 3.1 shows the dimensions of the Ahmed body used as part of this research. An Ahmed body model is used as demonstrated by Ahmed [6], with both a 35° rear slant and a 25° rear slant. These two models represent the low and high drag regimes respectively, with complex flow dynamics at the rear edge of the model, responsible for a large portion of the drag.

For the 35° Ahmed body, with a length based Reynolds number of $Re = 2.9 \times 10^6$, there is an immediate flow separation at the top of the rear slant, leading to a large wake region on the rear slant and in the immediate vicinity of the rear of the body, this causes the Ahmed body to experience a large pressure drag. For the 25° Ahmed body, with the same Reynolds number, there is an initial small separation at the top of the rear slant which is followed by a reattachment of the flow to the rear slant. The flow then separates at the bottom of the rear slant leading to a much smaller wake behind the Ahmed body but an increase in the drag experienced by the body when compared with the 35° Ahmed body. The overall drag experienced by the 25° Ahmed body

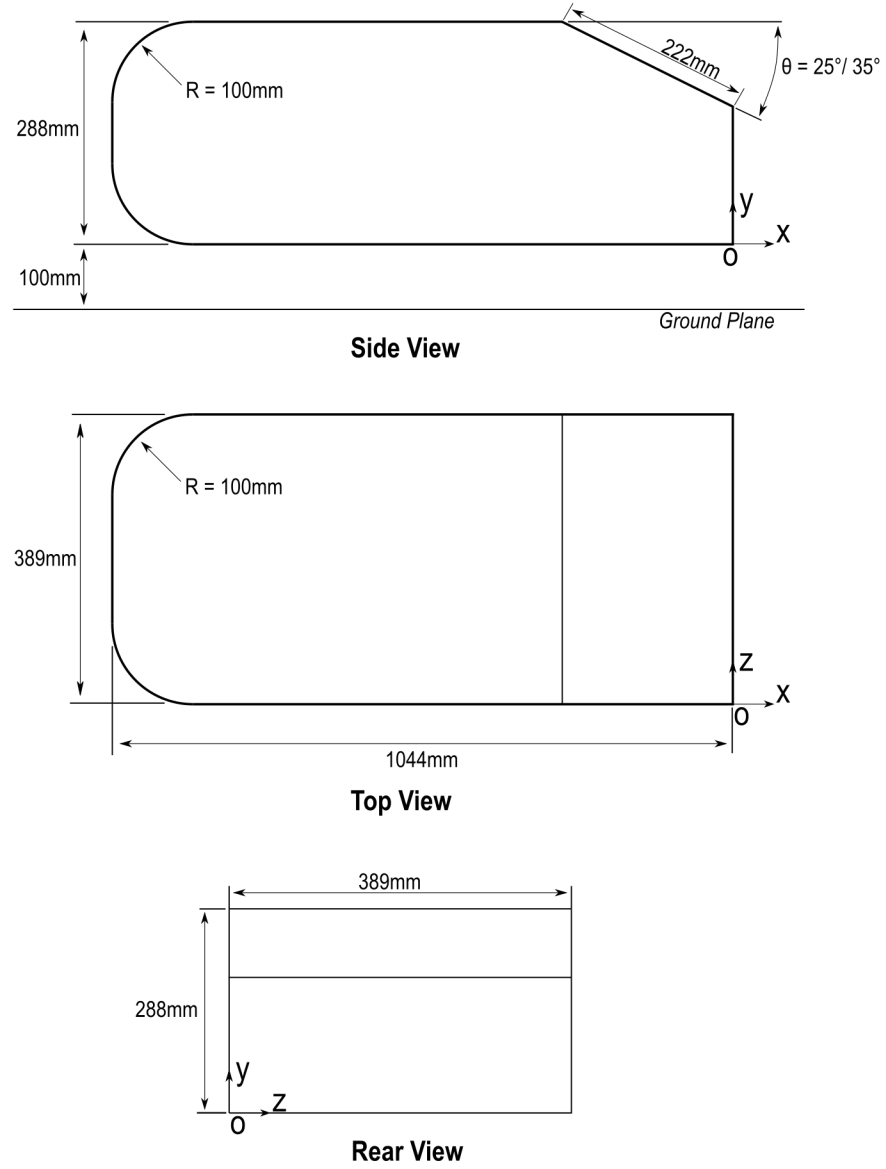


Figure 3.1: Dimensions of the Ahmed body.

is larger due to the increased pressure drag. This difference in the pressure drag can be explained by the effective shape change of the body, caused by the change in the wake shape. The optimal aerodynamic shape is a teardrop with a longer tail producing the best results. The early separation for the 35° Ahmed body causes the silhouette of the body and wake combined to closer resemble this teardrop shape than the 25° Ahmed body, which results in a reduced drag for the 35° Ahmed body. The skin friction drag remains almost constant for every slant angle.

This research aims to control the wake size and it was deemed that the best

way to do this was to target the top of the rear slant. This location was chosen due to the rear slant producing a higher proportion of the pressure drag than the lower rear face [6]. This ruled out SJA locations on the lower rear face of the model, whilst jets located at the side of the rear slant have been shown to be less effective due to the presence of C-pillar vortices [8]. This idea could be extended with extra jets added around the rear faces of the model to complement the SJAs located at the top of the model. A few variations of the jet location at the top of the rear slant were considered, based on results from [37], with small tests conducted to verify the results. It was found that jets located just upstream of the rear slant, on the top face of the model, were less effective at delaying flow separation, whilst jets further down the rear slant, were unable to interact with the shear layer from the top of the model.

The proposed methodology for the research follows the flowchart depicted in Figure 3.2. The initial research focuses on 2D CFD setup and mesh generation, which is followed by a validation study using the ERCOFTAC database [104]. Once the validation has been completed, SJAs will be implemented on the Ahmed body with a variety of parameters tested to determine the best operating location and setup of the SJA array. This stage also consists of testing the SJA behaviour using a velocity inlet and Fluent UDF to model the jet flow. After results have been gathered for the 2D simulation, work can begin on the ROM development. This will entail building and testing the accuracy of the ROM with the data obtained from the 2D simulations. Alongside the ROM development, 3D model building and mesh generation will take place. Once the ROM has been developed, work on the ESC scheme can begin. Again, this will entail building and testing the controller using the 2D simulation data. The ROM and the ESC scheme will then be combined to produce an extended ESC scheme which should improve the performance of the basic scheme. The 3D model development will continue alongside the controller development, with SJA arrays added to the 3D model and tested with various locations and setup parameters to determine the best setup. Once data has been collected

from the 3D CFD simulations, this will also be used to develop a ROM, demonstrating the ability for real time prediction of the effects of the SJAs.

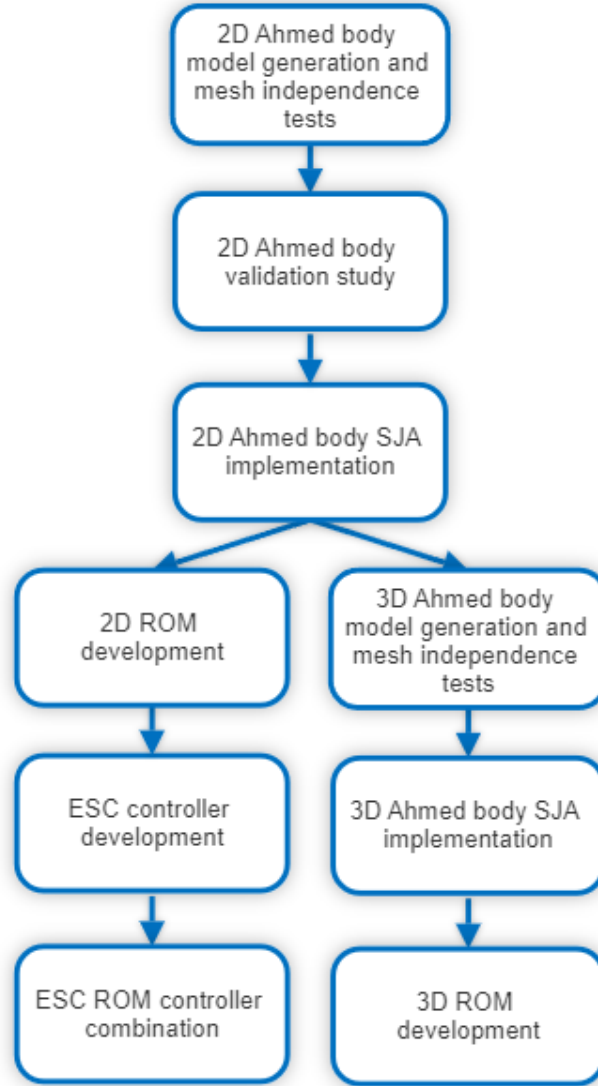


Figure 3.2: Flowchart showing the overarching methodology for this research.

Two common approaches for investigating flow control are, CFD simulations and wind tunnel experimental models. Both setups have their benefits and drawbacks. CFD simulations offer easy manipulation of models and testing parameters at the cost of computing resources, whilst experimental setups require time to gather the materials to build the model, but can be run multiple times with relative ease. For this research, CFD simulations were the focus, as various setups could be tested without needing to rebuild models. It also allowed for the collection of large amounts of data about the model for develop-

ing a control scheme using model information. It would be almost impossible to collect the same amount of data from an experimental model. However, before a CFD model could be built, a mesh had to be created and tested to ensure the baseline results obtained were similar to existing experimental data.

3.2 Numerical Simulations

3.2.1 Mesh Independence

CFD simulations are very sensitive to changes in the mesh, hence, various mesh types and densities were tested to ensure any changes measured were not artefacts of the mesh. The ANSYS workbench meshing tool was used to build a 2D and a 3D mesh. Both the 2D and 3D meshes used the same setup and dimensions for the mesh elements, with the only major difference between the meshes being the number of elements present. An unstructured mesh was used as the base, with large mesh elements in the far field (limited to 0.5m in size) and a more refined mesh close to the Ahmed body. A default expansion ratio was set across the mesh to 1.7. Table 3.1 shows the measured drag coefficient for the 3D 35° Ahmed body mesh independence test. The mesh elements in the far field remained at a similar size, whilst the mesh elements close to the body were decreased in size.

Table 3.1: *Mesh independence test for the 3D 35° Ahmed body.*

Mesh Elements (Millions)	Measured C_D
2	0.267
4	0.265
6	0.266
8	0.265

All meshes produced similar drag coefficient values, which suggests that the smallest mesh should be used to reduce computation power. However, the mesh quality of the 4 million element mesh was much higher than any of the other meshes according to the built in FLUENT mesh analysis tools. This

meant that the simulations converged much quicker, with larger relaxation factors than the other simulations, meaning that the computation time for this mesh was far lower than for the coarsest mesh. Therefore, this mesh was used for the 3D 35° Ahmed body. A similar mesh, using the same body of influences, expansion ratios and surface sizes, was built for the 25° Ahmed body and the Tall and Thin model developed for this research. This model uses similar dimensions to the 35° Ahmed body, however, the height and width measurements have been switched. The rear slant still remains at the rear of the top surface. This model was used as it gives a better representation of the aspect ratio of a truck trailer.

Figure 3.3 shows the mesh schematic for the 3D 35° Ahmed body. The different mesh densities allowed the flow behaviour close to the body to be captured in detail whilst reducing the overall complexity of the mesh, therefore reducing the computation time needed for the simulation. The size of the surface elements was dictated by the size of the elements in the body of influence around the Ahmed body. The elements varied from 2.5mm down to 0.5mm which produced the mesh sizes shown in Table 3.1, with 1.8mm producing the 4 million element mesh.

The mesh close to the body had to be carefully chosen due to the difference in scale of the objects being simulated. The synthetic jet actuators (SJAs) had an orifice diameter of 2mm, whilst the Ahmed body had a width of 389mm. The mesh elements at the orifice exit needed to be smaller than the orifice to capture the flow from the exit, but large enough to ensure the number of elements in the mesh did not increase the computation time too much. To solve this issue, a small region at the top of the rear slant, which can be seen in Figure 3.3, was isolated and mesh density in this region was increased. The maximum mesh element size in this region was 0.5mm, with over 90% of the elements even smaller. This represented a compromise between computation time and performance as shown in Table 3.1.

An inflation layer was used on all faces of the Ahmed body. This ensured that

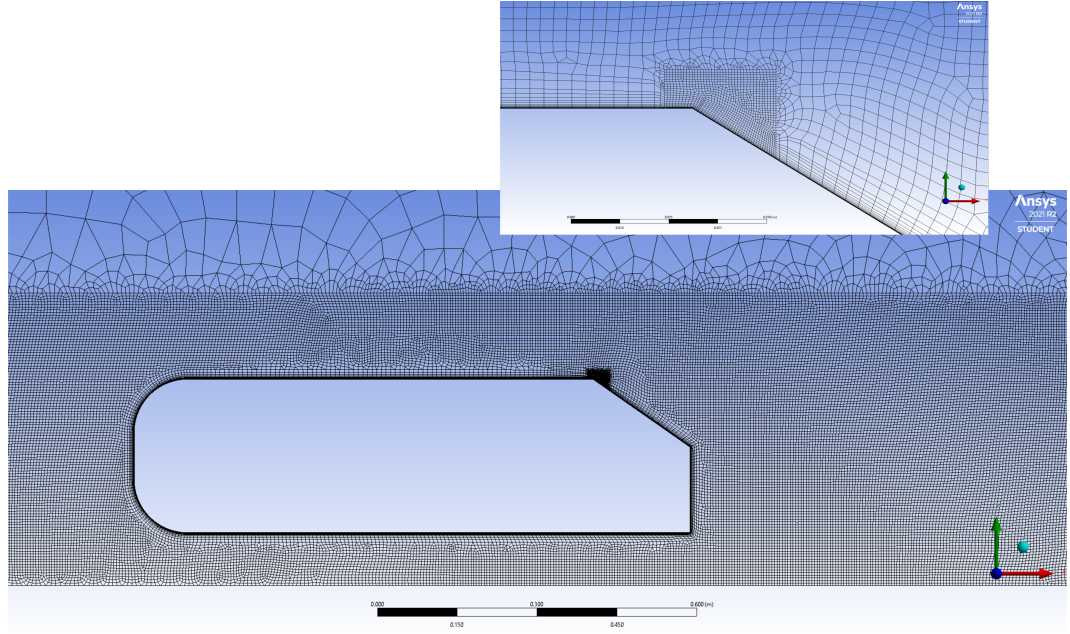


Figure 3.3: A cross-section of the mesh used for the CFD simulations.

the entire boundary layer was captured, whilst also ensuring that the y^+ value was kept small. The simulation used enhanced wall treatment to model the behaviour of the flow close to the body, which requires a y^+ value of between 1 and 10. The inflation layer allowed a y^+ value of 5 to be achieved, whilst again maintaining a small mesh size for the simulation. The inflation layer is hard to see in Figure 3.3 as the layers are so small and blend into the finer mesh around the body. The inflation layer uses an initial layer height of 5×10^{-5} mm with 14 layers across the height of the boundary layer around the model.

The 2D meshes varied from 50,000 to 200,000 elements and demonstrated similar characteristics to the 3D meshes. The drag coefficients for each of the meshes were all within 1% of each other, with the 75,000 element mesh demonstrating the highest quality mesh as measured using the FLUENT mesh analysis tools. The dimensions for this mesh were the same as for the 4 million element 3D mesh and so this mesh was used for the 2D simulations.

3.2.2 Numerical Model Setup

For this research an Unsteady Reynolds Averaged Navier-Stokes (URANS) CFD analysis was used. This approach was used to provide a blend of tran-

sient flow analysis with a reduced computation time compared to a Large-Eddy Simulation (LES). Multiple models were tested for their effectiveness, but ultimately, for this research, the realizable k - ϵ model was used. This is because the realizable k - ϵ model can more accurately predict the spreading rate of planar and round jets than the standard k - ϵ model [20]. The realizable model also provides a more accurate approximation than the standard model when modelling flows with separated regions and recirculating flow, of which the Ahmed body has both. This model was combined with enhanced wall treatment as explained in the previous section. Enhanced wall treatment is a combination of a two-layer model and enhanced wall functions that possess the accuracy of a standard two-layer approach for fine near-wall meshes, but can also be used for meshes where the first near-wall node is located in the turbulent region (i.e. the y^+ value is too large). Due to the low y^+ value in the meshes used for this research, the behaviour achieved from this wall treatment should be close to a two-layer approach.

The two-layer model refers to the domain being split into a viscosity affected region and a fully turbulent region. The viscosity affected region is simulated using the Wolfstein one equation model [20], whilst the turbulent region is simulated using the k - ϵ models. A blending function is then applied across the boundary of the 2 regions to ensure a smooth transition. This wall treatment was necessary for this research due to the complex nature of the near wall flow at the top of the rear slant and the interaction with the SJAs. This represented the best trade-off between accuracy and computation time.

A first order implicit transient scheme was used. The more complex schemes are used to track shocks through the flow and it is recommended to use these schemes when dealing with velocities larger than 100m/s, when the flow is considered to be compressible. The maximum jet velocity considered as part of this research is 100m/s so it was decided that the first order scheme would be sufficient, as the flow is considered to be incompressible. The Pressure-Implicit with Splitting of Operators (PISO) method was used for the pressure-velocity

coupling. This scheme generally has a more aggressive convergence than the Semi-implicit Method for Pressure Linked Equations (SIMPLE) scheme, meaning that fewer iterations are needed to reach a converged solution. To ensure convergence to a high degree of accuracy, residuals of 1×10^{-4} were used for the continuity, turbulent kinetic energy and the dissipation of turbulent kinetic energy, whilst 1×10^{-6} was used for the x, y and z velocity.

To determine the timestep value to use in each CFD simulation, the Courant-Friedrichs-Lewy (CFL) [105] condition was used,

$$C = U_{\infty} \frac{\Delta t}{\Delta l}, \quad (3.1)$$

where C is the Courant number, U_{∞} is the freestream velocity, Δt is the timestep size and Δl is the length between mesh elements. The CFL conditions states that the distance that information travels in a simulation must be less than the distance between mesh elements. The best practice is to keep the Courant value just below 1. In each case, the maximum freestream velocity used was 40m/s and the minimum distance between mesh elements was determined for each case. The characteristic length was decided to be the length of the Ahmed body, 1.044m. This gave a non-dimensional distance between mesh elements of 9.5×10^{-5} using,

$$\lambda = \frac{l}{x_h}, \quad (3.2)$$

where l is the raw length and x_h is the characteristic length for the 3D 35° Ahmed body simulation. The CFL condition gave a timestep size of 2.4×10^{-5} . Figure 3.4 shows the setup of the boundary conditions for the model. A velocity inlet was used upstream of the model, with a uniform inlet velocity set for the simulation. A freestream turbulence intensity of 0.25% was used as this is the value quoted in the ERCOFTAC database [104]. A pressure outlet was used downstream of the model. A wall was used to simulate the ground as this resembled the ground plate used in the ERCOFTAC [104] database results,

that this validation is based upon. A moving wall or symmetry condition could have been used to simulate the rolling road, which would have increased the flow velocity under the body, thereby reducing the velocity over the body. This would have consequences for the proposed control technique, although may improve the power saving as a lower peak velocity would be needed to achieve the same results. The other boundaries were modelled as a symmetry boundary condition to prevent flow across the boundary, whilst limiting the interference in the flow from the boundaries. The boundaries of the Ahmed body were modelled as walls, with no-slip conditions.

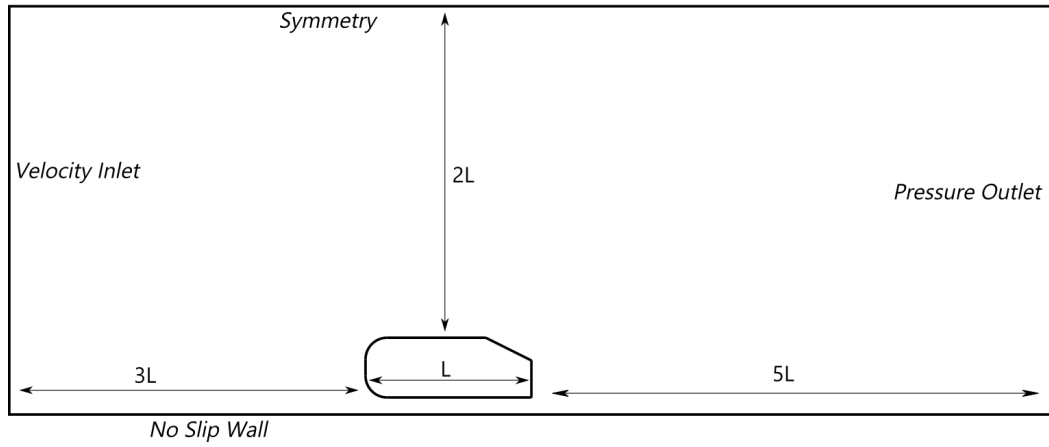


Figure 3.4: The setup of the boundary conditions for the CFD simulations.

The SJAs were modelled as velocity inlets located on the rear slant of the model. Multiple setups were tested, with and without cavities, but it was determined that velocity inlets gave the greatest scope for modifying the model without losing too much information. The inlet velocity profile and angle was handled by a FLUENT user defined function (UDF) to mimic the flow from the jet orifice. The UDF used the laminar pipe flow equation,

$$U = U_{max} \left[1 - \frac{d^2}{h^2} \right], \quad (3.3)$$

where U_{max} is the peak jet velocity at the centre of the orifice exit, d is the distance from the centre point of the orifice and h is the radius of the orifice exit. Equation 3.3 was extended to include a term to model the fluctuating

peak velocity using,

$$U = U_{max} \left[1 - \frac{d^2}{h^2} \right] \sin(2\pi\omega_j t), \quad (3.4)$$

where ω is the SJA oscillating frequency and t is the current time of the flow. The full code example can be seen in Appendix A. This velocity profile was used as it adds a damping function across the profile to match the flow at the orifice edges and avoid any discontinuity. This improves computational stability as well as improving the approximation of the real world SJAs.

3.2.3 Synthetic Jet Actuators

The SJAs in this research are based on a velocity optimised design developed in-house by Gungordu (Personal Communication 2019). It was explained that the SJAs produce a jet of almost 60m/s at an operating frequency of 2.2kHz. The efficiency of these SJAs can be as high as 39%, which is key to maximising the net power saving of the Ahmed body. This research considers jet velocities up to 100m/s, however, in almost every case the jet velocity used is lower than the 60m/s achieved by Gungordu. The SJA velocity was converted to a momentum coefficient using,

$$c_\mu = \frac{\rho_j U_j^2 A_j}{\rho_\infty U_\infty^2 A_\infty}, \quad (3.5)$$

where ρ_j and ρ_∞ are the densities of the jet flow and the freestream flow respectively, U_j and U_∞ are the jet velocity and the freestream velocity respectively, A_∞ is the spanwise cross-sectional area of the Ahmed body and A_j is the total area of the SJA orifices [67]. A non-dimensional frequency [67] was defined using,

$$F^+ = \frac{\omega_j}{\omega_\infty}, \quad (3.6)$$

where ω_j is the frequency of the SJAs and ω_∞ is the shedding frequency of the

body. The SJAs defined above operate at $F^+ = 24$ and produce a maximum momentum coefficient of $c_\mu = 2.5 \times 10^{-3}$. Early CFD simulations were analysed for dominant frequencies to determine the shedding frequency of the Ahmed body. The shedding frequency was found to be 50Hz, which corresponds to a Strouhal number of 0.36.

These non-dimensional values are used throughout the chapters when dealing with CFD simulation results, as it is common practice to allow comparisons of results. However, for the chapter involving controller results (Chapter 5), the raw data is presented with non-dimensional coefficients as this provides more context for the controller. The raw time is used, rather than a non-dimensional time as these results are specific to the setup and cannot really be compared to other setups in the same way that CFD simulations can be.

The literature shows the best location for the SJAs to be at the top of the rear slant, with an angle normal to the rear slant for delaying flow separation. It was also demonstrated that a higher frequency jet actuation provides a greater drag reduction than low frequency actuation. In all cases provided in the literature review for this thesis, the frequency was much lower than the operating frequency used as part of this research, so parameter tests were run to determine the best jet pitch angle and operating frequency. Data was saved at various timesteps so that the development of the flow could be studied as the SJAs interacted with the flow. This data was analysed using CFD-POST in the ANSYS Workbench environment, whilst the more quantitative data was exported to MATLAB to plot figures and analyse patterns in the flow.

For all simulations, a jet actuation of $c_\mu = 2.5 \times 10^{-3}$ was used for the 3D 35° Ahmed body. The baseline drag coefficient is obtained from running the CFD simulation with no SJAs included and is used to compare the various set ups. For the initial tests, the frequency of the jets was set to $F^+ = 24$ and the jet pitch angle was varied. Figure 3.5 shows the location of the jets and how the jet pitch angle was measured. An angle of 0° means that the jet is parallel to the freestream flow, whilst 65° is normal to the rear slant of the 35° Ahmed

body.

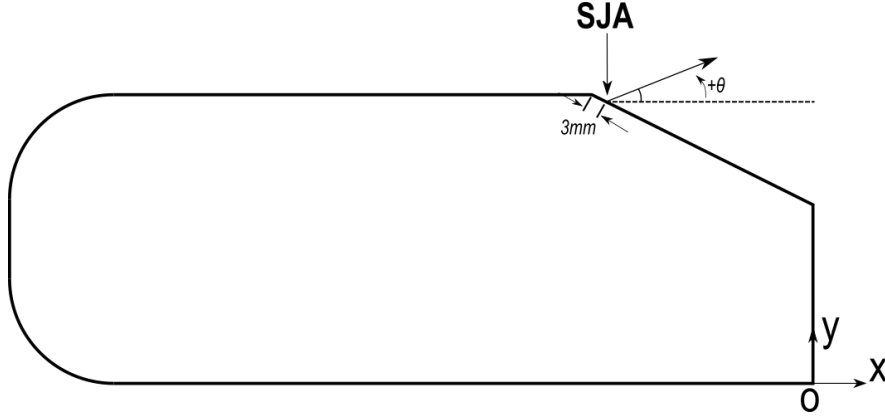


Figure 3.5: The SJAs located at the rear of the Ahmed body with θ denoting the SJA angle.

Table 3.2 shows the results from the jet pitch angle test. It was found that there is an optimum angle for the SJAs, which was similar between the 3D and the 2D simulations, where the jets were directed along the top of the wake. By angling the jets too far into the wake, the jets could not deflect the shear layer, the wake height was unaffected and the drag increased. For an SJA direction parallel to the flow, the shear layer was deflected slightly upwards, the wake height was increased by a small amount, leading to an increase in the drag. Similarly, by increasing the jet angle higher, the wake size was increased more and subsequently, the drag was increased further. For the proposed setup, an angle of -15° was used. This was found to be the optimal of all angles tested as it provided a small shear layer deflection and reduced the height of the wake. The model is very sensitive to this angle change, hence, with more tweaking, the drag could be reduced further.

Table 3.2: Table showing the effect on the drag coefficient of varying the angle of the SJAs.

	Baseline	Jet Pitch Angle ($^\circ$)				
		-30	-15	0	30	65
C_D	0.265	0.290	0.275	0.285	0.3198	0.3206

With the jet pitch angle decided, the SJAs were set to -15° and the frequency of the SJAs was varied. The results are shown in Table 3.3. The larger frequencies produced the smallest drag increases, with the smallest increase generated

by the SJA operating with $F^+ = 24$. At this frequency, the difference in the timescales of the jet flow and the outer flow causes the velocity fluctuations in the top of the wake to be minimal, with the wake height reduction caused by a shear layer deflection, similar to a steady jet interaction. The jet flow acts as a barrier between the fast moving flow over the top of the body and the recirculating region of the wake. The C_D peaks for $F^+ = 1$, where the SJA frequency and the shedding frequency are matched. For frequencies smaller than $F^+ = 1$, the shedding frequency is larger than the SJA operating frequency, meaning that the SJAs have a minimal effect on the flow. A frequency of $F^+ = 24$ was used for this research as it had the lowest overall drag increase in the 3D case, whilst producing the largest reduction of 15% in the 2D case. It was also the optimal operating frequency for the SJAs to produce the largest jet velocity. This results agrees with much of the literature ([28, 29]), suggesting that higher frequency actuation leads to greater performance by the actuators, either pulsed jets or SJAs. With $F^+ = 24$, the SJAs oscillate much faster than the shedding frequency of the flow, producing a barrier between the outer flow and the wake. It is clear from Table 3.3 that higher frequencies experience diminishing gains. If higher frequency SJAs had been used, it is expected that the gain would be minimal.

Table 3.3: Table showing the effect on the drag coefficient of varying the Jet actuation frequency.

	Baseline	Frequency, F^+ (-)			
		0.11	1.1	11	24
C_D	0.265	0.290	0.355	0.280	0.275

3.2.4 Calculating the Net Power Savings

To measure the effectiveness of the SJAs, the net power saving was calculated. This provides a better measure of success than just the drag reduction as it represents an overall increase in the performance of the SJAs. The power saved is calculated using a conversion from the reduction in drag force experienced by

the body, to the energy saved by the actuation. This energy saving is converted to a power saving. This power saving represents the total power saving per metre per second. The power used by the SJAs was computed using real world models developed by Gungordu (personal communication, 2020). The power was obtained using,

$$P_j = n_j I_j V_j, \quad (3.7)$$

where I_j is the current, V_j is the voltage and n_j is the number of SJAs used on the model. This number was halved to account for the slot adjacent design used in this research. The net power saving was then determined by subtracting the power used by the SJAs from the power saved as a result of the drag reduction. To effectively use the SJAs with the CFD simulation, a conversion from voltage to peak jet velocity had to be determined. This was again obtained experimentally, with measurements of the peak jet velocity taken for 10V, 20V, 30V and 40V averaged over multiple tests. Figure 3.6 shows the resultant measurements and the relationship used to approximate this conversion for this research. The SJA velocity is given by,

$$U_j = \frac{71}{40^i} V_j^i, \quad (3.8)$$

where i was determined to be 0.5 and V_j is the voltage supplied to the SJA. This approximate relationship allows the voltage and velocity to be interchanged for any voltage or velocity in the ranges. This would have to be determined for each new SJA and only applies to the current SJAs.

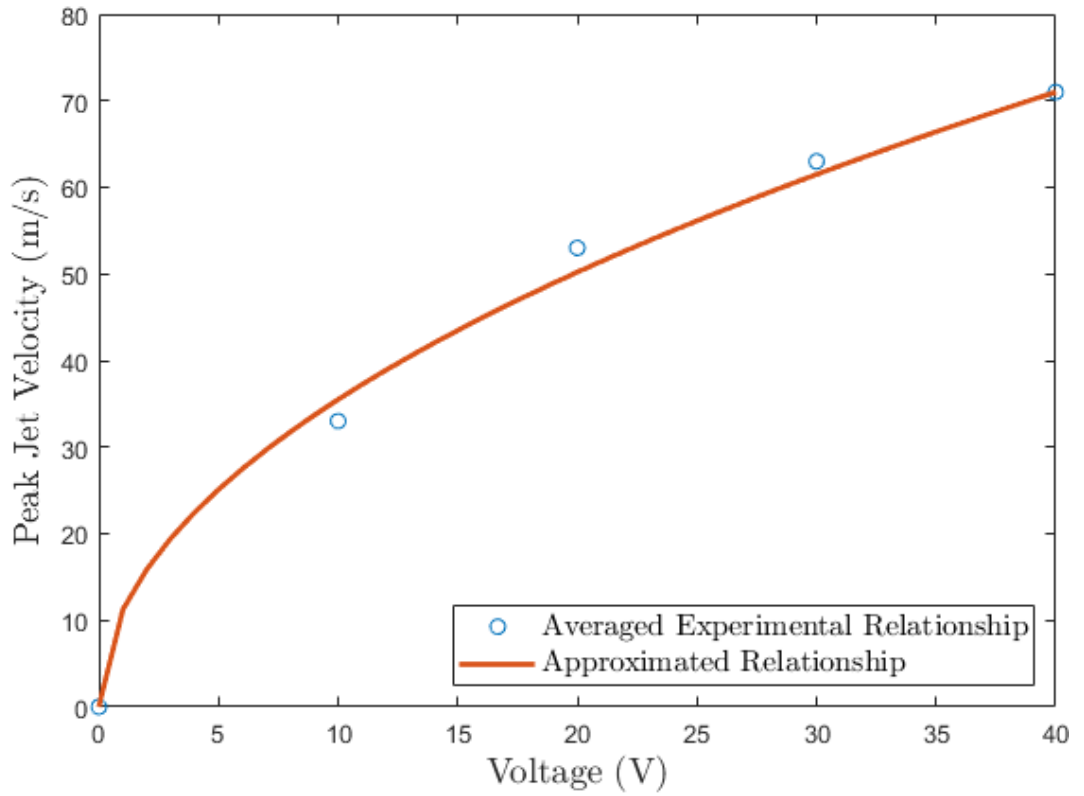


Figure 3.6: *The setup of the boundary conditions for the CFD simulations.*

3.3 Model Order Reduction

3.3.1 Introduction

There are many existing setups that aim to control aerodynamic flow around an object, whether that is to reduce the drag that the object experiences, or to minimise the effects of unwanted flow structures. These setups can be categorised into open loop and closed loop control schemes. The open loop schemes have a set behaviour and are left to operate in the same way for their lifetime. Closed loop schemes measure outside flow and can adapt to exterior stimuli to optimise their control for the current conditions. Open loop control has the drawback that it cannot react to outside stimuli and instead uses a predefined response for the flow state. This can lead to a less optimal performance for the system. Closed loop schemes are much more complex but allow for the control scheme to operate at an optimum for a wider range of conditions. The methods used in closed loop flow control all have slight

variations, however, they all follow the same basic principle.

Figure 3.7 shows the basic setup of a closed loop flow control scheme. A reference input is provided to a controller that produces a signal to send to the actuator to be controlled. The actuator modifies the system and a sensor measures the changes. This error output is fed back into the controller to update the signal and send a new input. The controller here needs to run in real time for the loop to be successful, if it takes too long to compute the optimal value, then the conditions may have changed and the controller will be lagging behind.

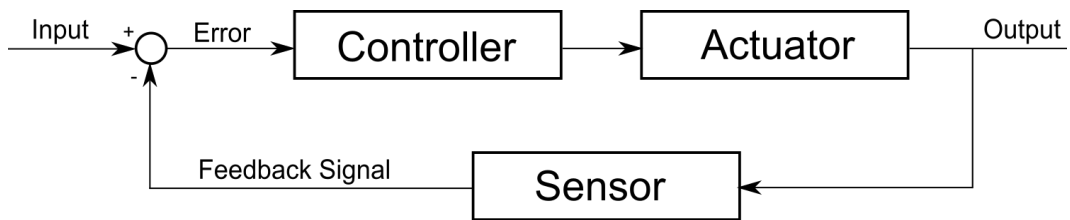


Figure 3.7: *The basic setup of a closed loop control scheme.*

The most common control schemes are variations on the extremum seeking control (ESC) scheme or slope seeking algorithm [41, 93]. This control scheme aims to minimise a parameter based on a cost function. A perturbation to the input signal is used to determine if the change in input leads to an improvement in the output or not. The benefits of this system are that it does not require any model information and so can easily be set up for any model and tested. There are tuning parameters that can be tweaked to improve the speed of convergence of the method or the accuracy of the convergence. The downsides are that the model can take a while to converge if the tuning parameters are not optimal and that the value it converges to could be a local minimum, rather than a global minimum, if the initial condition is not close enough to the optimum value.

By providing the ESC with information about the model, such as a better initial estimate for the optimum value, the performance of the ESC scheme can be improved. If the ESC has a better initial estimate, it is more likely to converge to a more accurate approximation of the global optimum, rather

than get stuck in a local optimum. It will also provide an increase in the convergence speed as the distance between the estimate and the true optimum is smaller.

To provide model information to the controller, a system had to be developed that could predict the optimum SJA velocity for the given freestream velocity. A full CFD simulation would be impossible in this setup, as it was not able to achieve the results in real time, so a reduced order model (ROM) needed to be developed. The reduced order model needed to take an input SJA velocity and convert it to a drag change in the system. This could then be fed into the controller to minimise the overall power used by the system.

3.3.2 Reduction Scheme

The airflow around the Ahmed body is governed by the Navier-Stokes equations, which express the conservation of mass and momentum. The equations can be used to produce a velocity vector at any point in a flow at a given time. It is known that the pressure gradients in the flow around the Ahmed body drive the velocity changes in the flow, suggesting that there is a relationship between the pressure change over time and the velocity change in time. It follows that the pressure change in time could be written as,

$$\dot{P}(t) = \frac{dP}{dt} = g(P(t)), \quad (3.9)$$

where P is the static pressure and g is some function of static pressure which governs the pressure change in time. It then follows that the pressure approximation by the realizable k- ϵ turbulence model could also be written in the form of Equation (3.9).

The POD scheme uses snapshots of a system in order to build a reduced system which contains only the most essential information from the system. A singular value decomposition (SVD) is applied to the snapshots to obtain the reduced order basis on which a system can be projected. Consider the time-invariant

ordinary differential equation (ODE),

$$\dot{x}(t) = f(x(t)), \quad (3.10)$$

where the state variable, $x(t) \in \mathbb{R}^N$, where N is the spatial dimension of $x(t)$ and $f : \mathbb{R}^N \rightarrow \mathbb{R}^N$. Assume also that the snapshot data is sampled at a uniform timestep denoted by $t_0, \dots, t_K$. It is clear that equation (3.9) has the same form, indicating that the POD scheme can be applied to the pressure field [69]. The POD system is used to reduce the system approximation such that,

$$x_{k+1} = Ax_k, \quad (3.11)$$

where $A \in \mathbb{R}^{N \times N}$ is a linear operator, such that if $x_k = x(t_k)$, then $x_{k+1} \approx x(t_{k+1})$ for all $k = 0, \dots, K$. The snapshot data is of the form, $X = [x_0, x_1, \dots, x_K]$, where x_0 is the initial state and the subsequent x_k form the trajectory of the state vector.

The POD modes are computed by calculating the SVD of $X^t X$, from which it is trivial to compute the reduced order basis for the system. An alternative approach that is used is the Dynamic Mode Decomposition (DMD) scheme. The DMD scheme differs from the POD mode scheme as two snapshot matrices are used,

$$X_0 = [x_0, x_1, \dots, x_{K-1}], \quad (3.12)$$

$$X_1 = [x_1, x_2, \dots, x_K]. \quad (3.13)$$

The two snapshot matrices are both similar to the snapshot matrix X , however, X_0 has the last element removed, whilst, X_1 has the first element removed. The problem to solve then becomes,

$$X_1 = AX_0. \quad (3.14)$$

An approximate solution can be found by computing the pseudo-inverse, such that,

$$A \approx X_1 X_0^+. \quad (3.15)$$

The DMD modes are given by the eigenvectors of the matrix A . The pseudo-inverse of the matrix is computed using an inverted SVD. The issue that arises here is that inverting the tiny, nonzero values in the diagonal matrix can lead to large errors, hence, the diagonal matrix is truncated for any values smaller than some tolerance $\xi \in \mathbb{R}^+$. The Input-output dynamic mode decomposition (IODMD) scheme is a variant of the DMD scheme, which adds a dependence on an input function $u : \mathbb{R} \rightarrow \mathbb{R}^M$ and an output function $y : \mathbb{R} \rightarrow \mathbb{R}^Q$, with an output functional $g : \mathbb{R}^N \times \mathbb{R}^M \rightarrow \mathbb{R}^Q$ that also depends on the state x and the input u . For the models in this research, u is the SJA peak velocity at each timestep and y is either a measure of the static pressure change or the change in the drag coefficient from the equilibrium state. The system of ODEs then becomes,

$$\begin{aligned} \dot{x}(t) &= f(x(t), u(t)), \\ y(t) &= g(x(t), u(t)), \end{aligned} \quad (3.16)$$

and the approximated discrete time linear system becomes,

$$\begin{aligned} x_{k+1} &= Ax_k + Bu_k, \\ y_k &= Cx_k + Du_k, \end{aligned} \quad (3.17)$$

where B is the input operator, C is the output operator and D is the feed-

through operator. Similarly to x_k , $u_k = u(t_k)$ is a discrete input function sampled at times t_k . Input and output vectors are created in a similar fashion to the snapshot matrices X_0 and X_1 , with,

$$U_0 = [u_0, u_1, \dots, u_{K-1}], \quad (3.18)$$

$$Y_0 = [y_1, y_2, \dots, y_K]. \quad (3.19)$$

This can then be rearranged into the system to solved,

$$\begin{bmatrix} X_1 \\ Y_0 \end{bmatrix} = \begin{bmatrix} A & B \\ C & D \end{bmatrix} \begin{bmatrix} X_0 \\ U_0 \end{bmatrix}, \quad (3.20)$$

which has the solution,

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix} = \begin{bmatrix} X_1 \\ Y_0 \end{bmatrix} \begin{bmatrix} X_0 \\ U_0 \end{bmatrix}^+. \quad (3.21)$$

The IODMD scheme presented above is used in system identification and does not reduce the size of the system that it is applied to, with the exception of the very small loss of data that arises from the truncation of the SVD in computing the inverse. The final step in the IODMD scheme is to use a truncated projection to reduce the overall order of the system. The solution to the system, then becomes,

$$\begin{bmatrix} Ar & Br \\ Cr & Dr \end{bmatrix} = \begin{bmatrix} Q^* X_1 \\ Y_0 \end{bmatrix} \begin{bmatrix} Q^* X_0 \\ U_0 \end{bmatrix}^+, \quad (3.22)$$

where r is used to denote the reduced version of the matrix and $Q \in \mathbb{R}^{N \times n}$ is the projection matrix, with $n \ll N$ and $Q^* Q = I_n$, where I_n denotes the identity matrix. A popular truncating projection is POD, which causes the snapshot matrix, X_0 , to be reduced to its POD modes. The size of the reduced order model can be chosen based on the number of POD modes used in the

projection.

Initially, the IODMD scheme is used to reduce the large scale CFD simulation to a smaller model, however, in later cases, where the model did not need to be reduced, the IODMD scheme was still used to identify the system and produce a state space model for the CFD model.

The U_0 and Y_0 input and output vectors were built based on the variables that change and are measured respectively. The voltage of the SJAs is modified, which in turn modifies the velocity of the SJAs. For ease of use with the CFD simulations and to allow easy adaptations of the voltage to velocity ratio of the SJAs, the velocity was used in the modelling and hence was used in the u_0 input. In the controller, the voltage is used and is converted using the relationship in equation 3.8. The output from the system is the net power saved from the drag change. This was obtained by using the drag change from the system which was then combined with the power usage in the controller. This meant that the Y_0 vector contained the information about the drag changes in the system. This was taken in the form of pressures on the body which were then converted into an approximated drag coefficient change. This actually allows the model to be reduced to just the C_D output as most of the behaviour required is already encoded in the drag coefficient.

3.4 Extremum Seeking Control

The ESC scheme works to find a point or set of points that maximise or minimise an objective function. This is done using an explorative gradient search approach, which uses a small perturbation to the approximated set point to see which direction the set point should be moved to improve the approximation. This is decided using the change in the output of the system. Figure 3.8 shows the basic setup of an ES control scheme for use on a dynamic system.

Consider the dynamic system in (3.16), where $y = g(x(t), u(t))$ is the objective

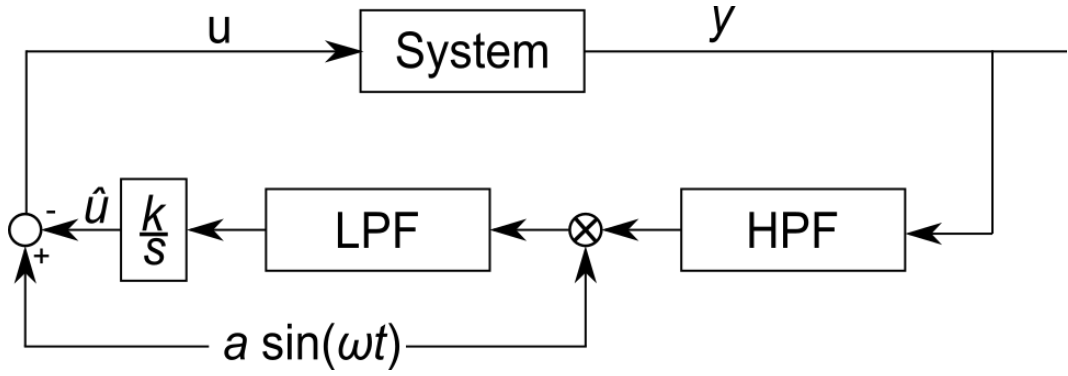


Figure 3.8: The basic setup of a perturbation based extremum seeking controller.

function. The system is combined with an excitation signal in the form of a sine wave perturbation. The output of this system is fed into a high pass filter to attenuate the low frequency oscillations. This removes the average value from the signal, leaving only the high frequency variations. The high pass filter state, shown in Figure 3.8, is represented by η and can be summarised by,

$$\dot{\eta} = -\omega\eta + \omega_h y, \quad (3.23)$$

where ω_h is the cut-off frequency of the high pass filter. This signal is again combined with the sine wave perturbation and fed into a low pass filter with a cut-off frequency of ω_l . The low pass filter is used to approximate the gradient of the signal, denoted by $\xi = (dg/du)$. The gradient can be summarised by,

$$\dot{\xi} = -\omega\xi + \omega_l(y - \eta)a \sin(\omega t), \quad (3.24)$$

where a is the amplitude and ω is the frequency of the perturbation signal. The perturbation signals in this example both use the same amplitude and frequency, but these can use independent values if required. The final step of the scheme is the integral controller, with a gain of k , which attempts to drive the estimated gradient to zero. The integral controller output is denoted by $\hat{u}$, which, when combined with the perturbation signal, makes up the input to the dynamic system for the next loop. This final step can be summarised by,

$$\dot{\hat{u}} = k\xi, \quad u = \hat{u} + a \sin(\omega t). \quad (3.25)$$

It is assumed that the system has optimum performance at $y = y^*$, which occurs when $u = u^*$ and $x = l(u^*)$, where x is a function of the control law u . For control of the SJAs in this research, the ESC scheme is used to maximise the net power saving for the simulation. The scheme used in this research is similar to the base scheme shown above, however, the signal is fed through a saturation block to enforce limits on the voltage. This is due to the physical practicality of the problem as the SJAs have a maximum effective limit. It also enforces a lower limit of zero, since a negative velocity here just indicates the jets operating 180° out of phase, which has the exact same effect as positive jet velocity at the same magnitude. This just reduces the complexity of the input signal and removes the issue with the scheme getting stuck at 0 as improvements are seen in both directions.

Figure 3.9 shows the setup for the model-based ESC scheme that is used as part of this research. The ESC control scheme explained here is combined with the ROM discussed earlier in the chapter, in an attempt to improve the performance of the ESC scheme. The controller model uses a very basic approach to implementing this, by measuring for changes in the freestream flow and resetting the integrator block if a freestream change is detected. Figure 3.9 shows the block diagram for the new ESC scheme.

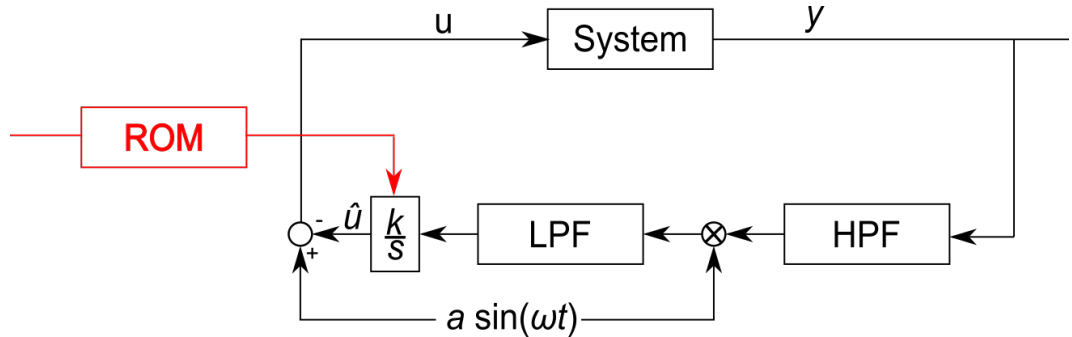


Figure 3.9: The setup for the reduced order model-based extremum seeking controller.

The differences to Figure 3.8 are highlighted in red. The ROM (LPV) model

monitors for changes in the freestream velocity. Once a change is detected, the ROM produces an approximation to the voltage that would produce the maximum net power saving. The optimum voltage is fed into the integrator block, which is reset, and uses the optimum voltage as the initial condition. A small tolerance in the freestream velocity change, which can be tweaked, is used to ensure that the ESC scheme is allowed to evolve for small fluctuations in the flow, but in the event of a large, abrupt change, the scheme converges very quickly to the new state.

This has 2 major benefits over the basic ESC scheme, in that the tuning can be much tighter, as the initial value should be reasonably close to the optimum, whilst not compromising on the speed of convergence. It should also ensure that an equal or more accurate approximation of the global optimum is reached, as the ESC scheme should converge to the global optimum, within the modelled domain. This may not be a global optimum in all cases as this may lay outside of the modelled domain, but the standard ESC scheme also would not reach this optimum.

3.5 Summary

CFD simulations for both 2D and 3D simulations of the Ahmed body with various rear slant angles was set up, using the same approaches for all models. A $k-\epsilon$ model was decided on due to the improved approximations for jet flows and the increased accuracy in separated flows. Pressure coupling equations and wall functions were carefully chosen to enhance the ability of the simulation to describe the interaction of the SJAs and the outer flow over the model. High quality meshes were developed for each simulation, taking into account the difference in mesh element sizes needed around the SJA orifices and the rest of the body, with an attempt to optimise the computational power and time required for each simulation. A time step was determined using the CFL condition. Velocity inlet boundary conditions, combined with a user

Table 3.4: *A selection of peak jet velocities converted to momentum coefficients for a freestream velocity of 40m/s.*

Jet Velocity (m/s)		30	60	90
c_μ	2D	2×10^{-3}	8×10^{-3}	1.8×10^{-2}
	3D	6×10^{-4}	2.5×10^{-3}	5.6×10^{-3}

defined flow profiles, were used to simulate the SJAs as they provided the greatest flexibility in modifying the behaviour of the SJAs whilst maintaining a reasonable fidelity in the simulation.

The SJA setup was investigated, with the optimum jet pitch angle determined to be -15° and the optimum actuation frequency determined to be 2.2kHz for the current setup. The peak velocity of the jets was converted to a momentum coefficient based on the freestream conditions for each simulation, with a selection of momentum coefficients presented in Table 3.4 for a freestream velocity of 40m/s. The 2D and 3D momentum coefficients differ due to the difference in area between the models; the same peak jet velocities are used. The 2D simulation only uses the heights of the model and the SJAs, whereas, the 3D simulations use the frontal cross-section and the total area of the SJA orifices.

The equation for the power consumed by the SJA array was determined, based on the power for each SJA and the number of SJAs present in the array. This number could be halved due to the intended use of the slot adjacent design for the SJAs. A relationship between the velocity and the voltage was computed from experimental data to allow easy conversion between the velocity for the CFD simulation input and the voltage needed to compute the power used.

An approximation to the Navier-Stokes equation was used to represent the evolution of the pressure field as a linear time-invariant ODE, which can be combined with a control law to allow a model order reduction using the IODMD scheme. The input to the control system was the SJA peak jet velocity, whilst the output was chosen to be the change in the static pressure field or the change in the drag coefficient. Both of these outputs are similar and it is dependent

on the information available in each case as to which was used.

The ESC model was described, with an extension proposed to reset the integrator block whenever an external change was measured. The ROM designed using the IODMD model would be used to predict the optimum SJA voltage, before the ESC scheme could take over and fine tune the exact input to the system. It was determined that the SJAs should only ever have a positive input velocity, as a negative jet velocity just represented a phase change by 180° . An upper limit was also added to the SJA input voltage to prevent the model operating outside of the modelled region.

Chapter 4

2D Numerical Modelling

4.1 2D Numerical Model for the 35° Rear Slant Ahmed Body

4.1.1 Introduction

CFD simulations can be used to simulate highly complex fluid flows and have many benefits over experimental setups when it comes to testing various configurations of models or collecting detailed information about the flow. There is a large computational cost for 3D CFD simulations and the results have to be carefully validated against experimental data to ensure that physically realistic results are obtained. Therefore, for this work, 2D numerical models were key to allow a quick turnaround in large data sets and enabled various reduced order model schemes to be built and tested. This chapter shows the development of a 2D model for the 35° Ahmed body and explores how SJAs interact with the wake to cause a drag reduction. The simulation results are validated against the centreplane data from the ERCOFTAC database [104] due to the lack of 3 dimensional effects across the centre of the model; the 3 dimensionality comes from the edges of the flow where the C-pillar vortices act.

The 2D CFD model uses an Ahmed body as was shown in Figure 3.1. This

is coupled with SJAs with a jet pitch angle directed into the top of the wake as seen in Figure 3.5. The aim of this setup was to produce similar drag reductions to studies using steady jets blowing into the wake, such as [34, 79]. Current SJA setups use a small peak jet velocity with a frequency close to that of the natural shedding frequency of the body to achieve a delayed separation over the rear slant. This study uses velocity optimised SJAs with a large jet peak velocity operated at an actuation frequency of 2.2kHz or $F^+ = 24$. This frequency was chosen as it is closely matched with the resonant frequency of the SJAs, which produces the largest peak jet velocity of 60m/s or $c_\mu = 2.5 \times 10^{-3}$ for the 3D simulations and 8×10^{-3} for the 2D simulations as explained in Chapter 3. The frequencies were tested in Chapter 3 and it was shown that the higher F^+ values produced a larger drag reduction for the current setup of up to 15%.

This chapter references time-averaged velocity data when comparing the experimental data and the numerical model. All time averages are taken over 0.5 seconds of simulation time after the flow has settled to an equilibrium state. This is not a steady state, as there is still variation from the vortex shedding behind the model, but rather an oscillatory equilibrium state. This ensures that only the flow dynamics are captured and removes any convergence effects from the data.

4.1.2 Model Validation

Due to CFD turbulence models being approximations of real world systems, the results can be unreliable, hence the need to validate the numerical models against their real world counterparts. It is difficult to validate 2D numerical models against real world data as there is no accounting for the 3D effects of real world flows, such as vortices and the complex recirculating flow that appears in the wake of the body. However, by comparing 3D centreplane experimental data to the 2D numerical model, an indication of the accuracy

can be obtained. These are compared under the assumption that the span of the model is relatively large, such that in the near vicinity of the centreplane flow changes are considered negligible. The 35° Ahmed body is considered to be a mostly 2 dimensional flow around the centreplane as the 3 dimensional effects are located at the edges of the rear slant where the C-pillar vortices act. The 2D 35° Ahmed body CFD numerical model has been compared to 3D centreplane data for the Ahmed body from the experimental data obtained from the ERCOFTAC database [104]. Figure 4.1 presents a schematic showing the locations of the profiles discussed in the next section. Points A, B and C relate to the locations of the velocity profiles over the rear slant, whilst points D, E and F relate to velocity profiles in the wake.

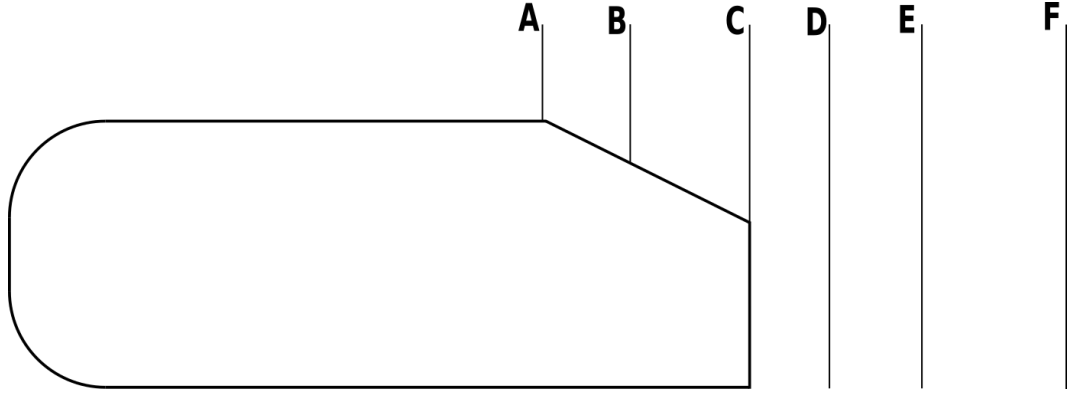


Figure 4.1: A schematic of the Ahmed body, with lines added to show the locations of the key velocity profiles discussed in the next section.

The ERCOFTAC database has a variety of data for the 35° Ahmed body, including velocity profiles on the rear slant and in the wake. The velocity profiles on the slant obtained from the CFD simulations can be seen in Figure 4.8. Both the height and length scales have been normalised by dividing by the height and length of the model respectively.

Figure 4.8 shows the development of the flow over the rear slant of the body, with the profiles matching up in the separated recirculation region, but over predicting the velocity in the freestream flow. The development of the flow in both the numerical model and experimental data demonstrates a similar behaviour, with the flow separating near the top of the rear slant (between

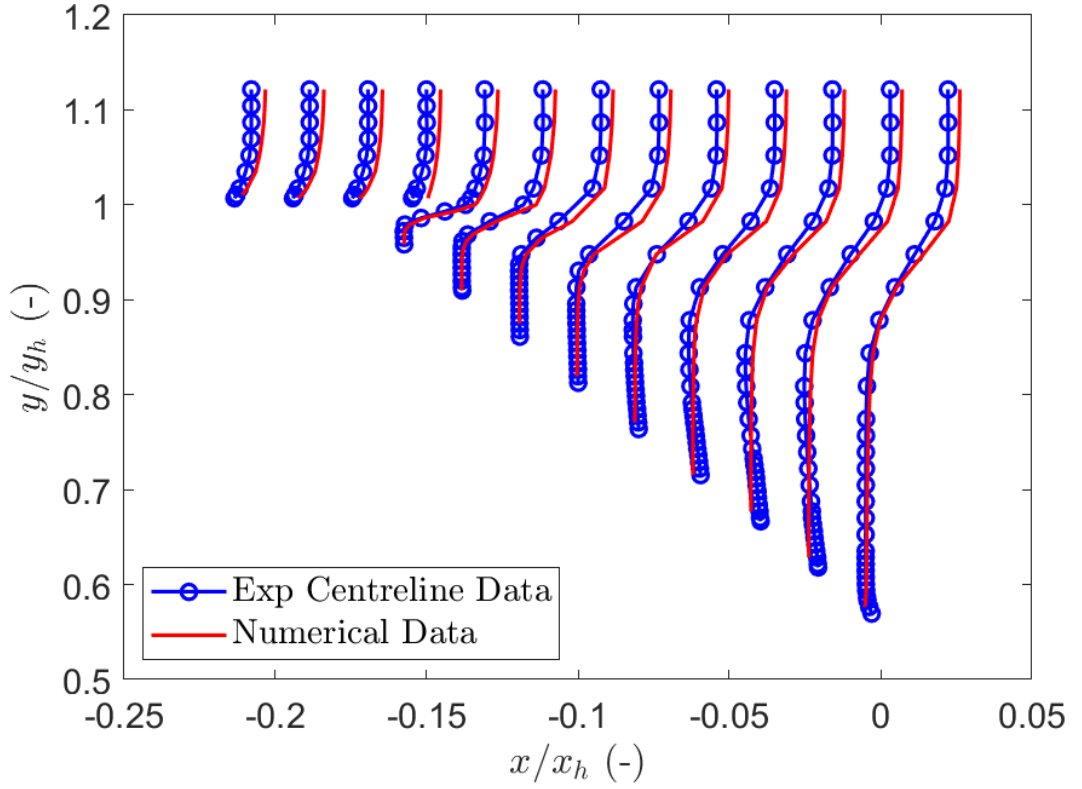


Figure 4.2: A comparison of the velocity profiles on the rear slant of the 35° Ahmed body between the 2D CFD numerical model and the centreline experimental data from the Ahmed body ERCOFTAC database [104].

$x/x_h = -0.15$ and $x/x_h = -0.125$) and a clear recirculating region present below the top of the wake at approximately $y/y_h = 1$.

Figures 4.3, 4.4 and 4.5 present a closer look at 3 of the profiles in Figure 4.8 at $x/x_h = -0.17$, -0.1 and 0 respectively. These locations represent key areas of the flow, at the top of the rear slant just before the separation point (A in Figure 4.1), in the middle of the rear slant (B in Figure 4.1) and at the end of the rear slant (C in Figure 4.1). The profiles in these figures are plotted with the normalised time averaged velocity on the x-axis, to give an indication of the differences in the profiles.

It is clear from Figure 4.3 that there is an over prediction in the flow over the body by $U/U_\infty = 0.2$ or 20%. The over prediction is constant across the full height of the profile and does not seem to cause the flow over the rear slant to deviate too much from the experimental data with an average error of 6.67% across the profiles on the rear slant. It was thought that the increased velocity

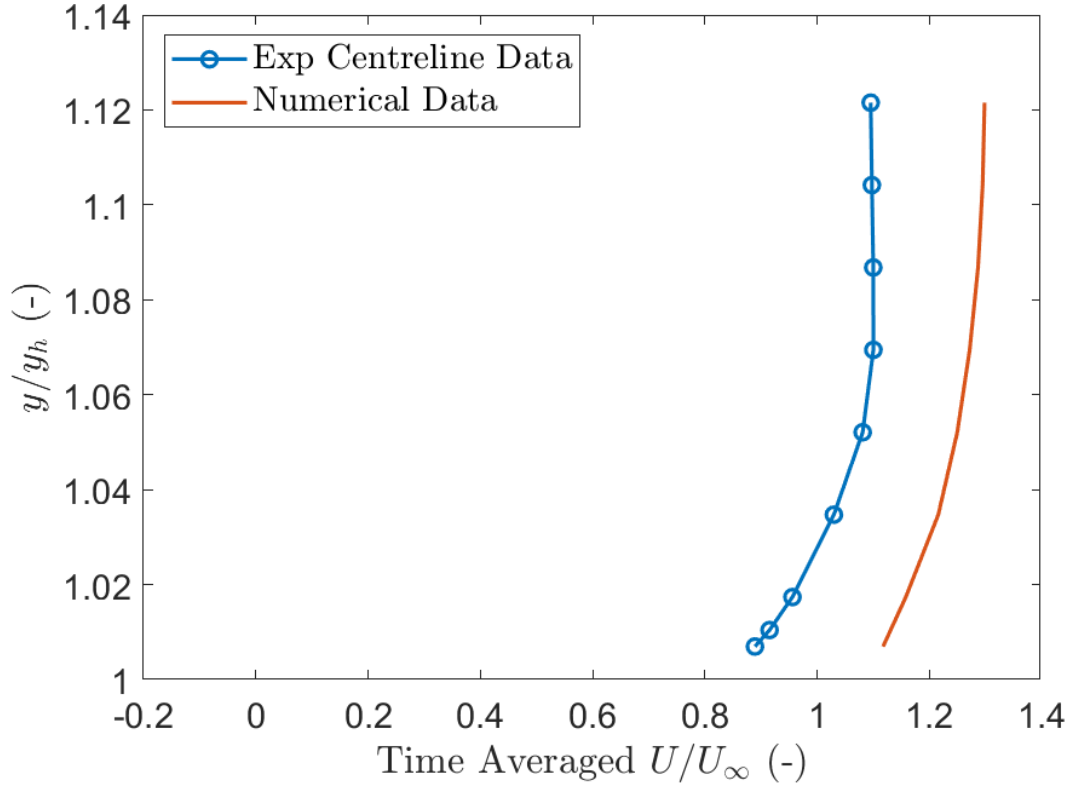


Figure 4.3: A comparison of the velocity profiles at $x/x_h = -0.17$ on the rear slant of the 35° Ahmed body between the 2D numerical model and the centreline experimental data from the Ahmed body ERCOFTAC database [104].

over the model was due to the larger blockage ratio for the 2D simulation, however, increasing the height of the domain did not change the flow velocity at all. Another key design choice that could have led to the over prediction, is the choice in boundary conditions. As stated previously, a moving wall under the body would likely reduce the flow over the top of the model, affecting the over prediction. Also, the use of a uniform velocity inlet could produce the same behaviour. If the level of turbulence in the inlet is not matched to the experimental data, then the development of the flow over the body could be very different. Alternatively, the 2D inlet condition does not account for 3D variations in the inlet velocity which could also cause the over prediction in the velocity. The increased flow velocity is attributed to the lack of 3D effects in the model and around the Ahmed body.

Changes to the flow velocity can lead to errors in the separation point location. Whilst the exact separation point for the experimental model cannot

be determined as the data is not presented in the ERCOFTAC database, the separation point for both the numerical model and the experimental data, is located at the top of the model between $x/x_h = -0.15$ and $x/x_h = -0.125$. This coupled with the small errors over the rear slant, suggest that the numerical model predicts the separation of the flow to a reasonable accuracy. The faster flow over the Ahmed body will likely reduce the effectiveness of the SJAs and the control scheme as a larger momentum coefficient will be needed to interact with the flow.

This can be seen further in Figure 4.4, where the over prediction by 20% in the freestream flow is present, however, in the recirculation region, the profiles match up very well. The CFD simulation is able to predict the negative flow in this area, indicating a recirculating region, or separated flow. Using the definition that the wake height is the point at which the flow velocity is equal to 99% of the freestream velocity and by accounting for the increased freestream velocity in the CFD simulation, both datasets give a similar value for the height of the wake. This can also be seen in the velocity profile at the rear of the body in Figure 4.5. The key information from these profiles is the agreement in the height of the wake and the behaviour of the recirculating region. The apparent agreement in the separation point at the top of the wake and the overall development of the flow over the rear slant are also important results suggesting that this CFD simulation is reliable when predicting the flow over the rear slant.

Further confidence is provided using Figure 4.6, which presents the velocity profile shown in Figure 4.4 with an extended height. This demonstrates that the velocity overshoot, present close to the body, recovers to the freestream velocity further from the Ahmed body.

Similarly, the turbulent kinetic energy (TKE) profiles can be studied to indicate the behaviour of the simulation. Figure 4.7 shows the TKE profile at $x/x_h = -0.1$, located at point B in Figure 4.1. There is a spike in the TKE to around 53J/kg in the wake over the rear slant demonstrating a fully developed

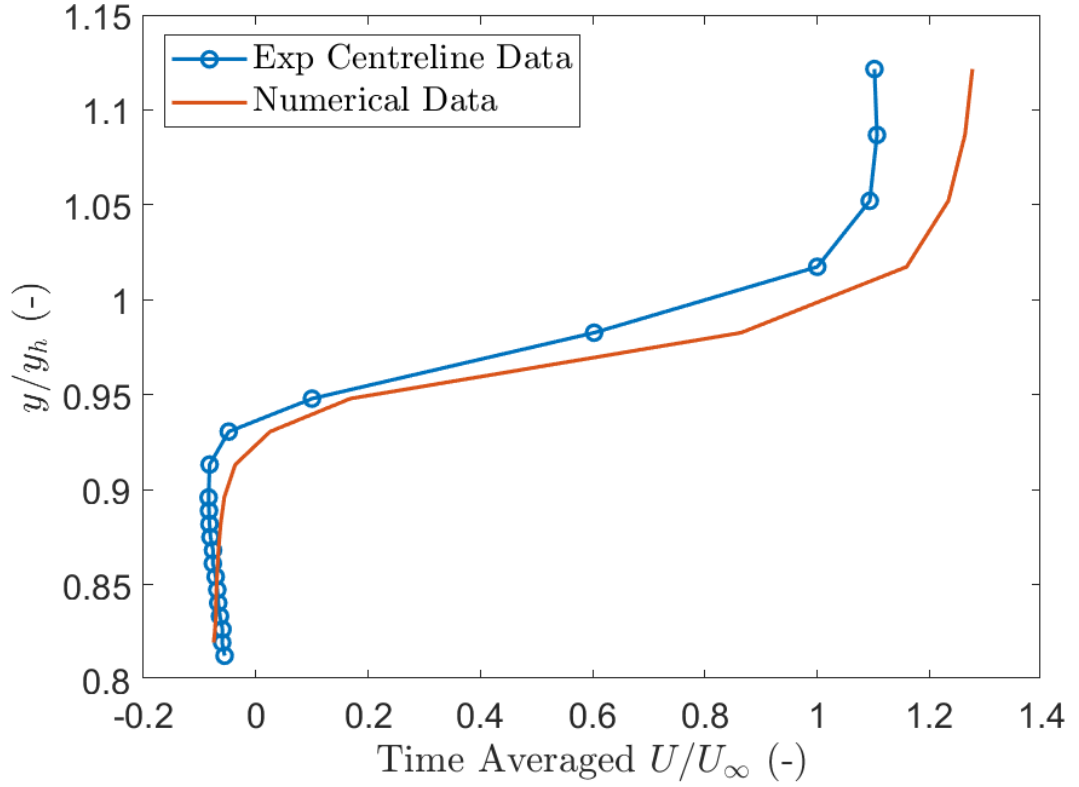


Figure 4.4: A comparison of the velocity profiles at $x/x_h = -0.1$ on the rear slant of the 35° Ahmed body between the 2D CFD simulations and the centreline experimental data from the Ahmed body ERCOFTAC database [104].

turbulent flow, before the TKE decays to 0J/kg further from the body. This behaviour is expected for this setup which provides further evidence that the CFD simulation is reliable.

A measure of the accuracy of the CFD model was obtained by quantifying the error between the two profiles. The error was computed using Equation (4.1),

$$E = \frac{\sum_{i=1}^{n_p} |U_i^{CFD} - U_i^{EXP}|}{n_p U_\infty} \quad (4.1)$$

where subscript i is used to denote the data at a given height, CFD and EXP are used to denote the numerical model data and experimental data from the Ahmed body ERCOFTAC database [104] respectively, U_∞ is used to denote the freestream velocity and n_p is the number of points. This method computes differences in the velocities at a given set of heights and takes the average. Due to the nature of the flow and the large changes in velocity for small changes in height, the errors obtained from this method can be quite large.

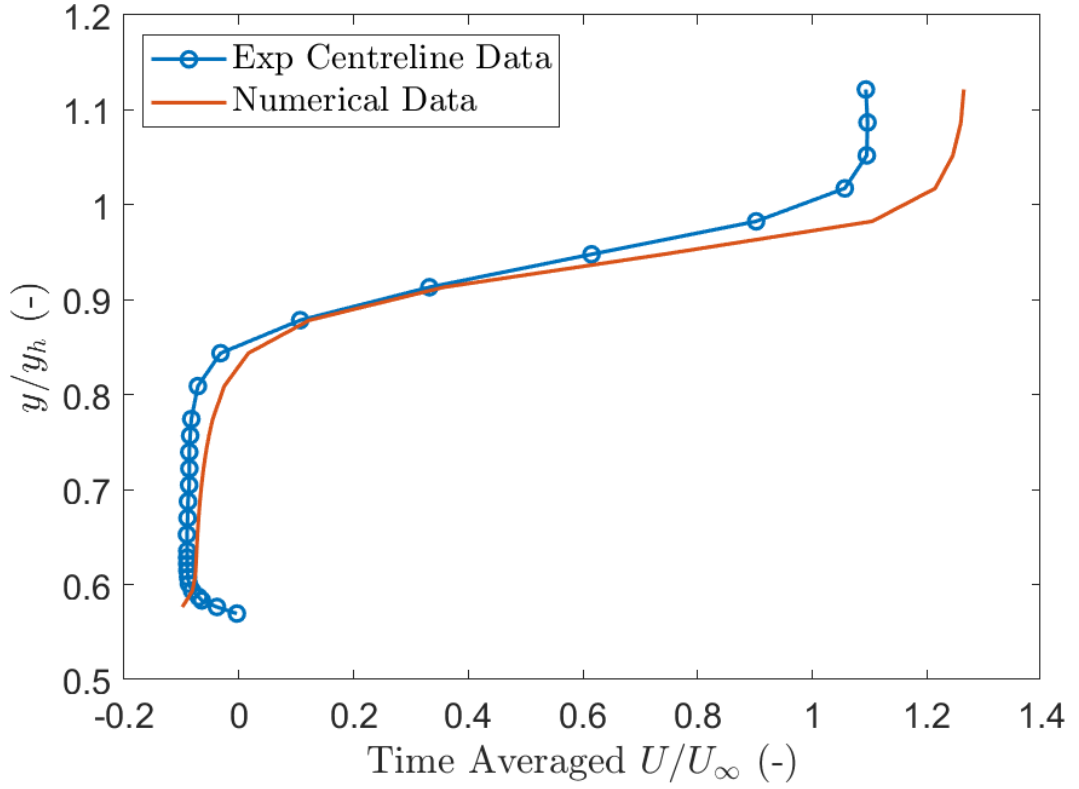


Figure 4.5: A comparison of the velocity profiles at $x/x_h = 0$ on the rear slant of the 35° Ahmed body between the 2D CFD simulations and the centreline experimental data from the Ahmed body ERCOFTAC database [104].

The maximum error in Figure 4.8 is just before the separation point at the top of the rear slant. This corresponds with Figure 4.3 and appears to be due to the over prediction of the velocity. The error drops by up to 15% when the CFD results are scaled to compensate for the over prediction in the freestream flow. The average error in the profiles then is 3.5%. This scaling is only used to indicate the accuracy of the numerical model over the slant and is not used in the reduced order models developed later in this research. The error is also much lower for the profiles where there is a large separated region, which agrees with the observations from the figures. It was decided that the CFD model was reliable when looking at the flow behaviour over the rear slant, which suggested that the flow in the wake should give a reasonable agreement as well. A similar analysis was also carried out for the wake profiles.

Figure 4.13 shows the velocity profiles from the 2D CFD simulation and the centreline data from the ERCOFTAC experiments. The first few profiles show

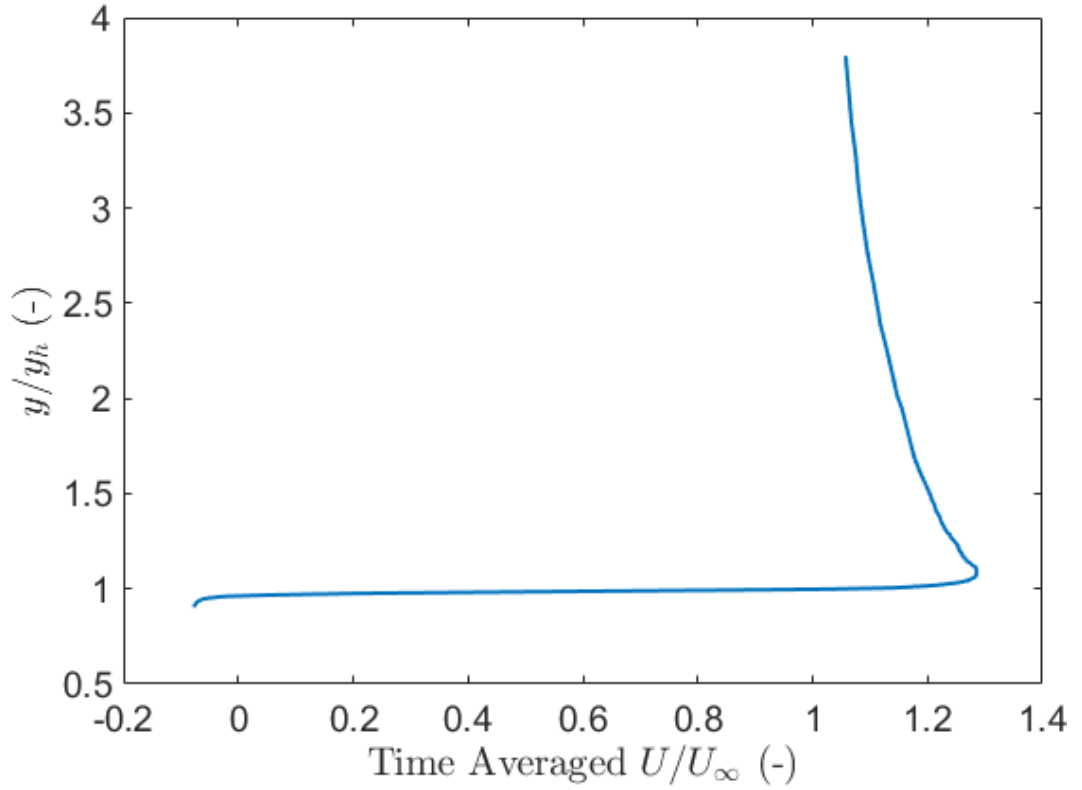


Figure 4.6: A plot of the velocity profile shown in Figure 4.4 with an extended y axis to demonstrate the recovery of the velocity to the freestream.

that the over prediction in the freestream flow from the slant profiles is continued here, however, the profiles that are further downstream of $x/x_h = 0.35$, show that the over prediction has disappeared, with both datasets showing a very good agreement at the top of the profile at $x/x_h = 0.64$. There are more disagreements in the wake profiles, with the CFD simulation predicting a faster flow under the body when compared with the centreline experimental data. This is likely due to the CFD simulation not having any obstructions under the body, whilst the experimental model had struts for the model to stand on. There is also a larger gap under the model for the CFD simulations, resulting in a larger expansion of the flow from under the body.

Figure 4.10 shows the wake profile at $x/x_h = 0.08$ behind the body (D in Figure 4.1) in more detail. The over prediction of the freestream values is clear at the top of the profile as expected and the CFD simulation over predicts the velocity at the base of the wake as pointed out above. In fact, the CFD simulation predicts an almost symmetrical wake around $y/y_h = 0.4$, whilst

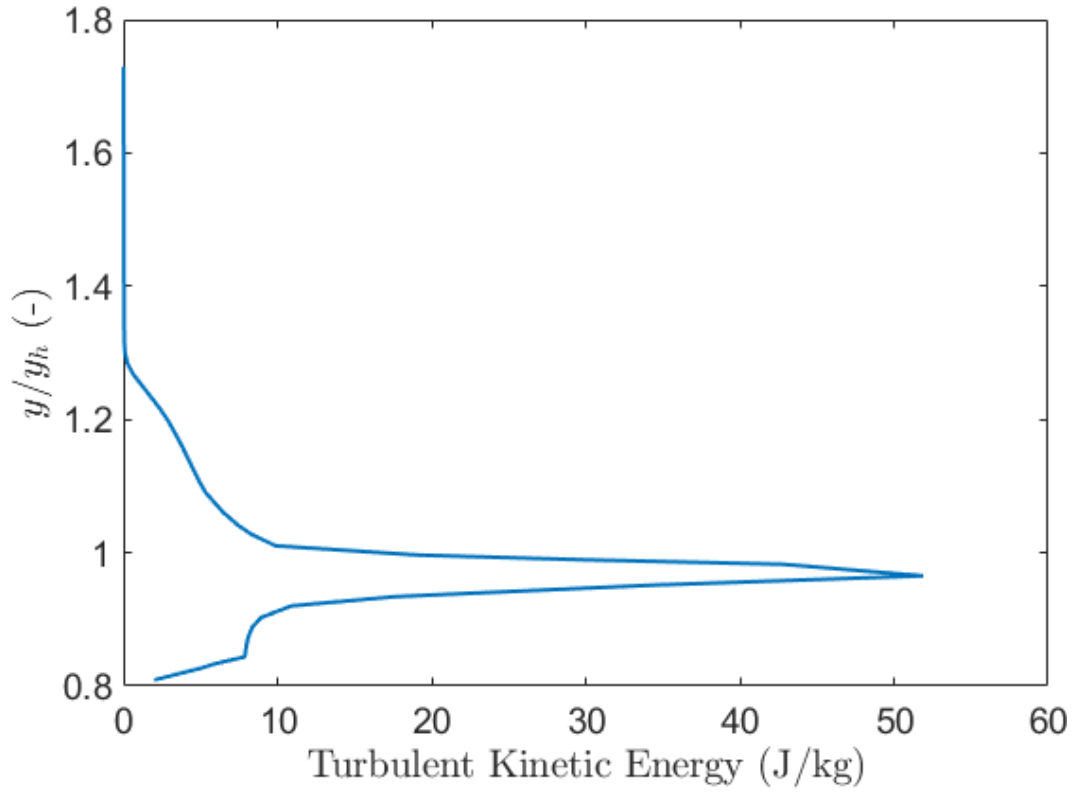


Figure 4.7: A plot of the turbulent kinetic energy profile at $x/x_h = -0.1$ on the rear slant of the 35° Ahmed body.

the experimental data presents an asymmetric wake. This could be due to a variety of differences between the simulations. There is also 3D effects present in the experimental wake, which cannot be captured by a 2D simulation. Alternatively, the flow differences could arise from the differences in floor gaps and the absence of struts holding the model up in the CFD simulation. The CFD simulation does predict the height of the wake and the transition from freestream flow to a recirculating region with reasonable accuracy. This trend continues with the wake profiles further behind the body. Figure 4.11 shows the profile at $x/x_h = 0.18$ behind the body (E in Figure 4.1) in more detail. Similar observations can be made to that of Figure 4.10, suggesting that the issues propagate through the wake and that the model does not accurately model the shape of the wake. Figure 4.12 shows the profile at $x/x_h = 0.32$ (F in Figure 4.1). At this point behind the model, the experimental wake has begun to drop slightly, which is evidenced by the lower wake height than the CFD simulation and the lower base of the wake. This is likely due to the

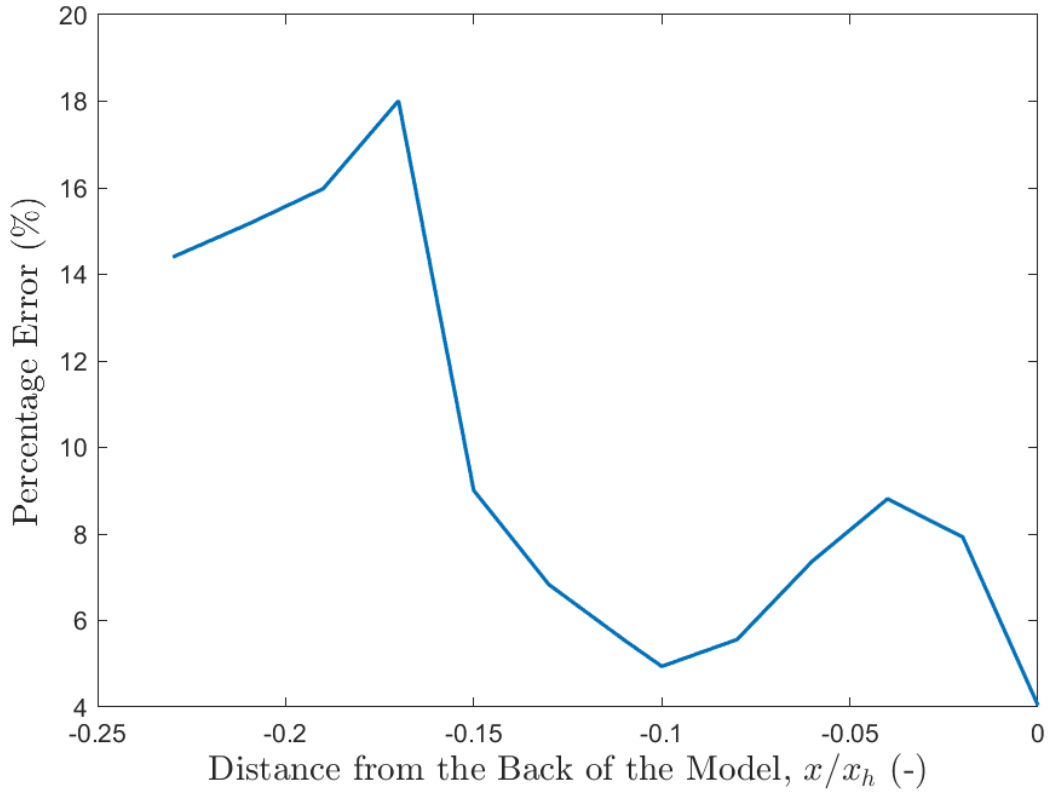


Figure 4.8: Error between the velocity profiles on the slant for the experimental centre plane data from the Ahmed body ERCOFTAC database [104] and the 2D CFD simulations.

increased flow velocity in the CFD simulation producing less of a shear layer deflection, causing the flow to be directed parallel to the flow and elongating the wake.

The CFD simulation is still able to accurately predict the height of the wake. This suggests that the simulation is suitable to test the control scheme which aims to deflect the shear layer into the wake, thereby, reduce the height of the wake and the overall size of the wake to reduce the drag coefficient. Equation (4.1) was again used to quantify the difference between the 2 profiles and the results can be seen in Figure 4.13.

The errors in Figure 4.13 have a larger average than the errors over the rear slant of the body. The wake profiles suffer from the same issues as the slant profiles when it comes to measuring the error, since there are large changes in the velocity for small changes in height. This means that the errors can look quite big. These profiles suffer from the over prediction of the freestream

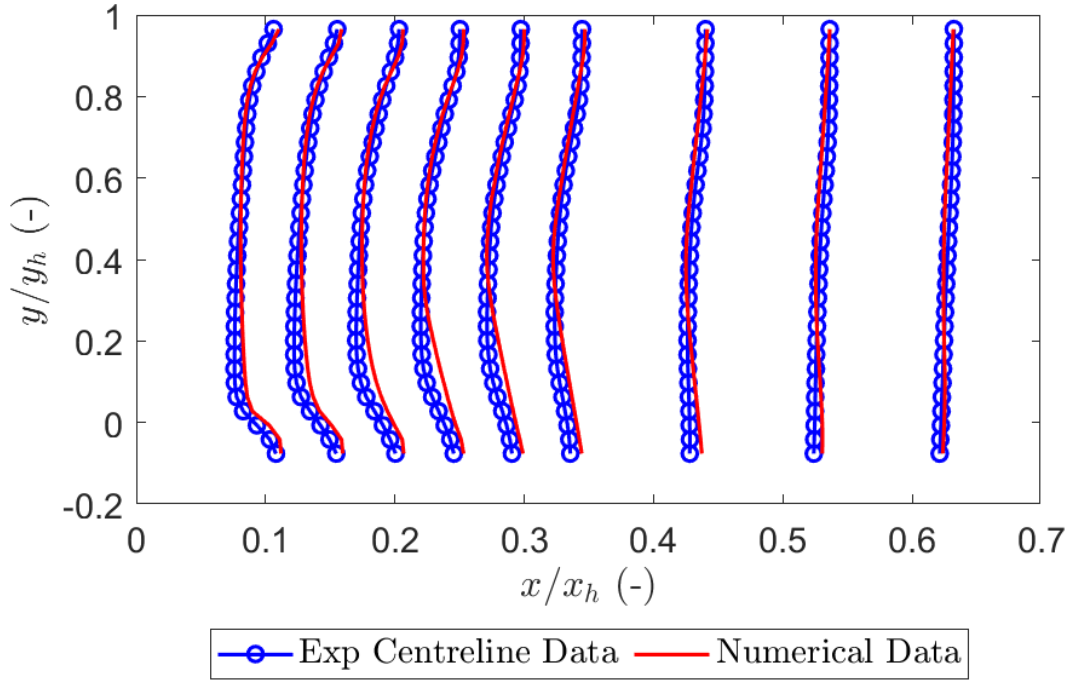


Figure 4.9: A comparison of the velocity profiles in the wake of the 35° Ahmed body between the 2D CFD simulations and the centreline experimental data from the Ahmed body ERCOFTAC database [104].

flow as the slant profiles did, but the errors here are mainly caused by the asymmetry in the lower part of the wake; the CFD simulations are unable to predict the slower flow velocity in the base of the wake.

It was deemed that the 2D model gave a reasonable agreement to the centreline experimental data for the key areas of the flow. That being the flow over the rear slant which is where the control scheme is focussed and the prediction for the height of the wake. The issues with the base of the wake should not play a major role in testing the effects of the SJAs at the top of the slant. The only problematic area is the increased flow velocity over the body, however, this should only cause the SJAs to be less effective, so it was determined that this should not affect the results too much. The overall flow development was similar to the experimental data and so should give a good indication of the effects of the SJAs. Also, the reduced computing resources and time needed to run the 2D simulations compared to a 3D simulation was key as it allowed

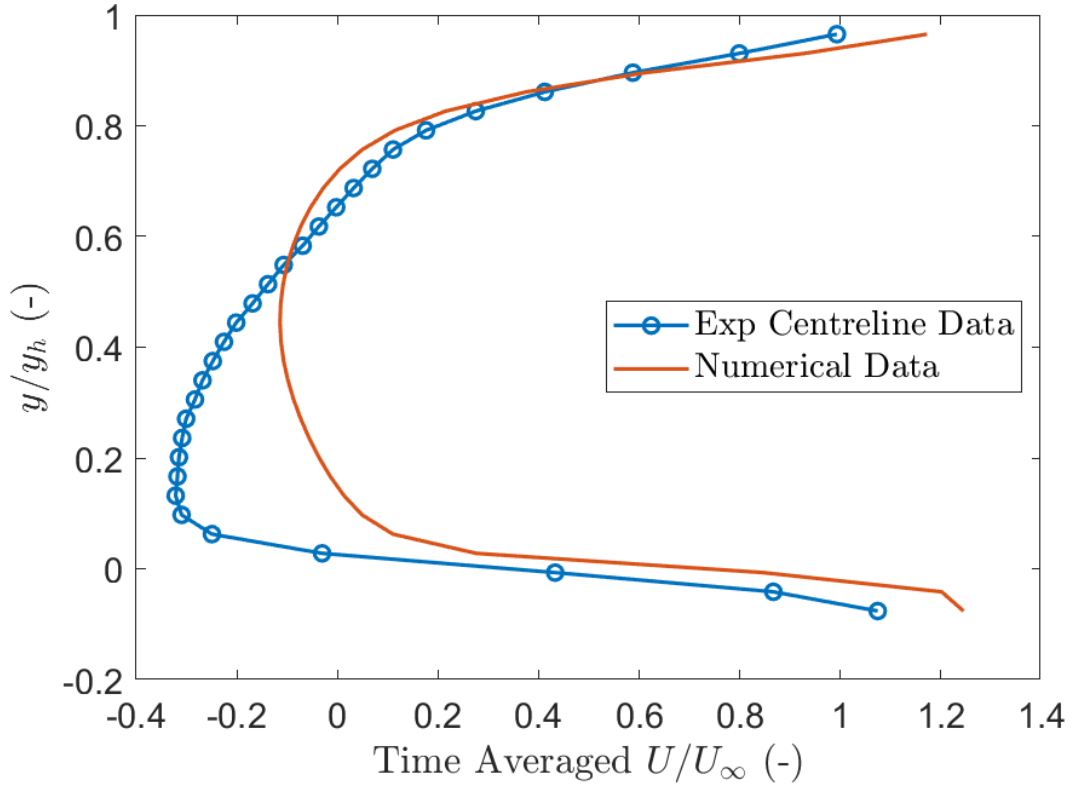


Figure 4.10: A comparison of the velocity profiles at $x/x_h = 0.08$ downstream of the 35° Ahmed body between the 2D CFD simulations and the centreline experimental data from the Ahmed body ERCOFTAC database [104].

data for lots of different setups to be collected. This meant the reduced order model could be built and tested much easier than for a 3D model.

4.1.3 Flow Control of the Wake Using Synthetic Jet Actuators

For the 2D 35° Ahmed body, the SJAs are pointed down the rear slant and into the top of the wake. The aim being to deflect the shear layer into the wake and reduce the separation drag. This has been shown to be reasonably successful with steady jet actuators [34, 79], but has not been tested with SJAs yet. The SJAs were pointed at an angle of -15° to the freestream flow direction as this was found to be the optimum direction of the jets (Table 3.2). At this angle, the jets are pointed along the top of the wake and interact with the top of the wake. With jets pointed further down the rear slant, the drag reduction was not as good and in extreme cases even increased the drag

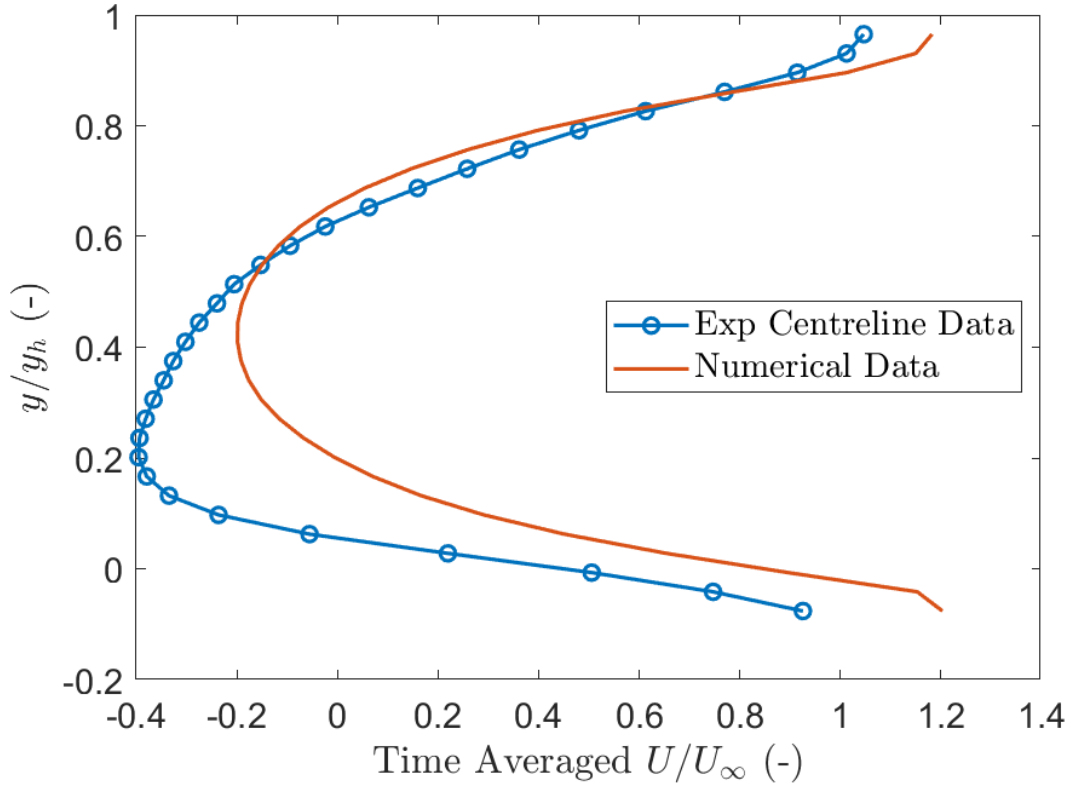


Figure 4.11: A comparison of the velocity profiles at $x/x_h = 0.18$ downstream of the 35° Ahmed body between the 2D CFD simulations and the centreline experimental data from the Ahmed body ERCOFTAC database [104].

coefficient when compared with the baseline flow. If the jets were pointed parallel to the freestream flow, the jets caused a small deflection of the shear layer, increasing the height of the wake and causing a small increase to the drag coefficient from the baseline case.

By operating the SJAs at 2.2kHz, it ensured that there would be no frequency matching, which can amplify the effects of the jets. The drag reduction results were promising, if a little higher than would be expected given the literature. The model was tested with various momentum coefficients ranging from 0 (no jet flow) up to 0.022. The percentage drag coefficient change for each momentum coefficient can be seen in Figure 4.14. The momentum coefficient was calculated using Equation (3.5).

There appears to be a linear relationship between the momentum coefficient of the jet and the percentage change in the drag coefficient. This relationship breaks down if the momentum coefficient gets too large. Whilst 4 of the

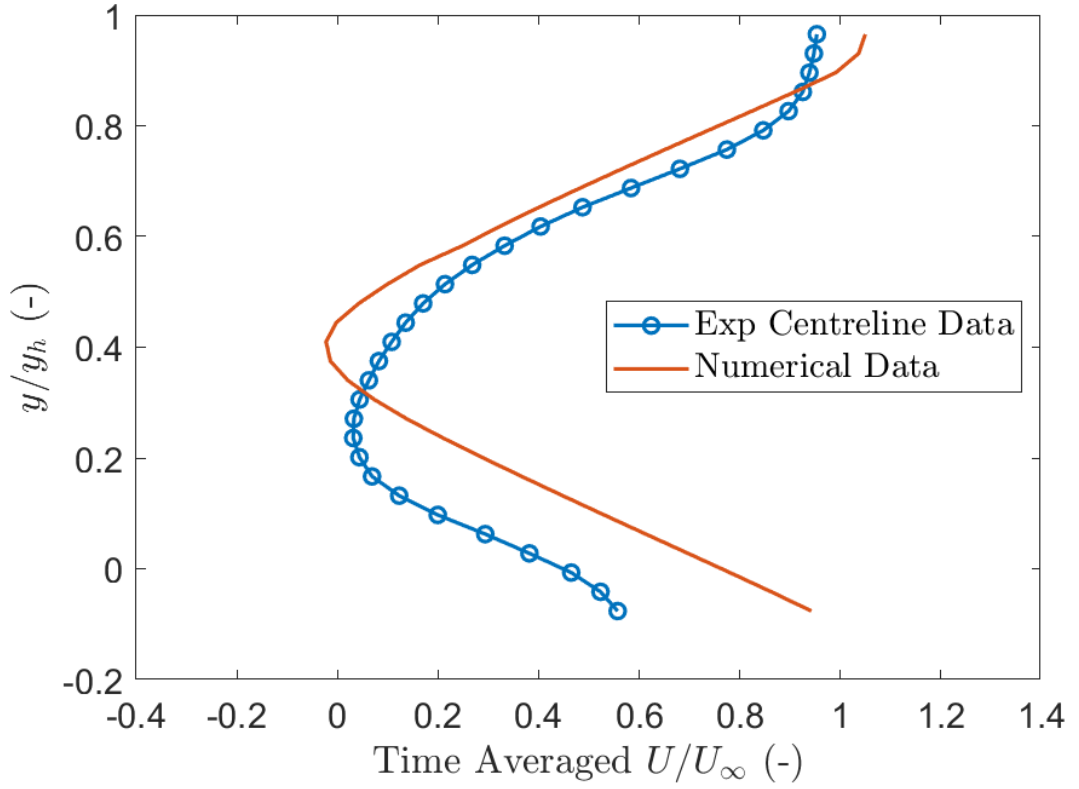


Figure 4.12: A comparison of the velocity profiles at $x/x_h = 0.32$ downstream of the 35° Ahmed body between the 2D CFD simulations and the centreline experimental data from the Ahmed body ERCOFTAC database [104].

momentum coefficients tested are beyond the physical limits of the SJAs that this research is based upon, all values for this freestream velocity are below the critical momentum coefficient, where the relationship breaks down. The maximum drag reduction for the SJAs based on these results would be a 10% reduction in the C_D which corresponds to $C_\mu \approx 1 \times 10^{-2}$. This is in agreement with other studies that used SJAs at the top of the rear slant [37, 67], however, it is a little higher than is expected for steady jets acting at the top of the rear slant [34]. This could be due to the lack of 3D effects in the 2D simulation as the C-pillar vortices present in the 3D case reduce the effectiveness of SJAs at the spanwise edges of the body.

To investigate the interaction between the SJAs and the wake of the Ahmed body, a momentum coefficient of $C_\mu = 8 \times 10^{-3}$ was compared to the flow with no actuation. This corresponded to a C_D reduction of almost 6% so any changes to the flow would be significant. Figures 4.15 and 4.16 show velocity

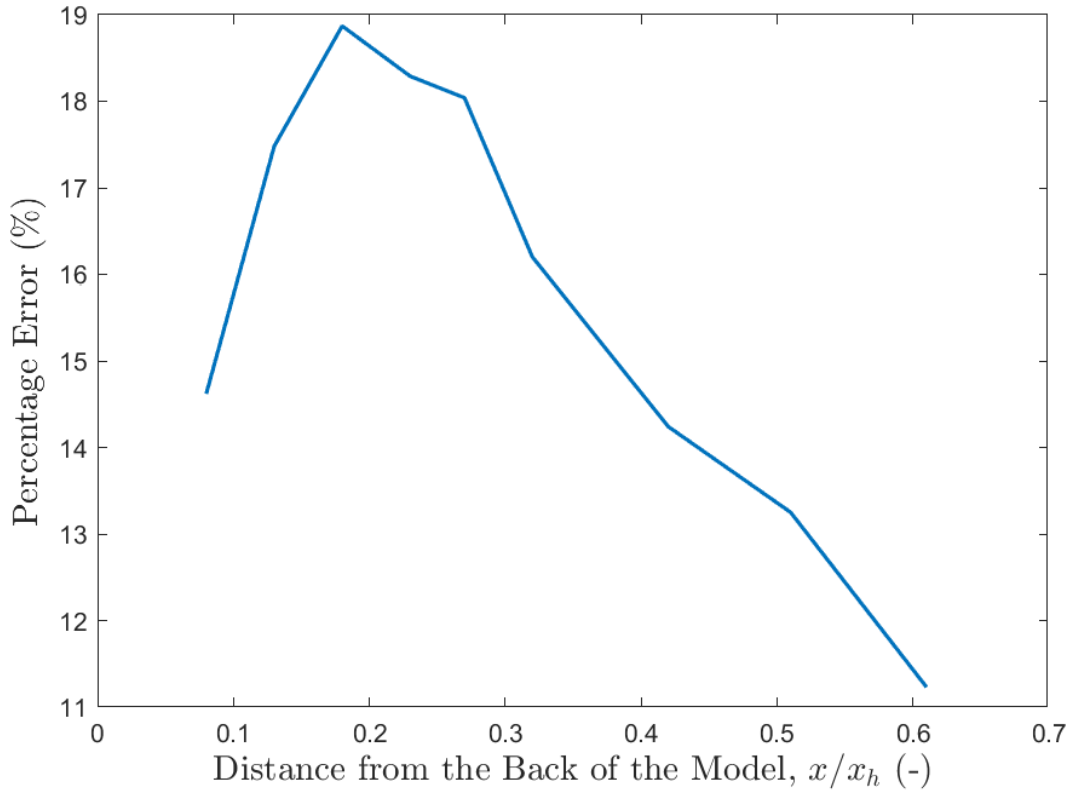


Figure 4.13: Error between the velocity profiles in the wake for the experimental centre plane data from the Ahmed body ERCOFTAC database [104] and the 2D CFD simulations.

contour plots for the CFD model. Figure 4.15 is the time averaged baseline flow with no jet actuation, which shows the behaviour of the flow at the top of the rear slant. The dark blue area indicates a small magnitude flow velocity, but more importantly, is a region of recirculating flow as seen in Figures 4.3, 4.4 and 4.5. The lighter blue contours (20-32 m/s) show the top of the wake. It is interesting to note that the recirculating region does not begin at the apex of the top of the model and rear slant, but slightly below. It can be seen from Figure 4.17 that the separation point in this case does occur at the apex of the slant.

The drag coefficient naturally experiences oscillations in unsteady flow, due to the vortex shedding behind the model, hence why a time averaged contour was used for Figure 4.15. With the SJAs in operation, the C_D oscillates even faster subject to the oscillation frequency of the SJAs. The top image in Figure 4.16 shows the velocity contour at a snapshot in time where the C_D value was at

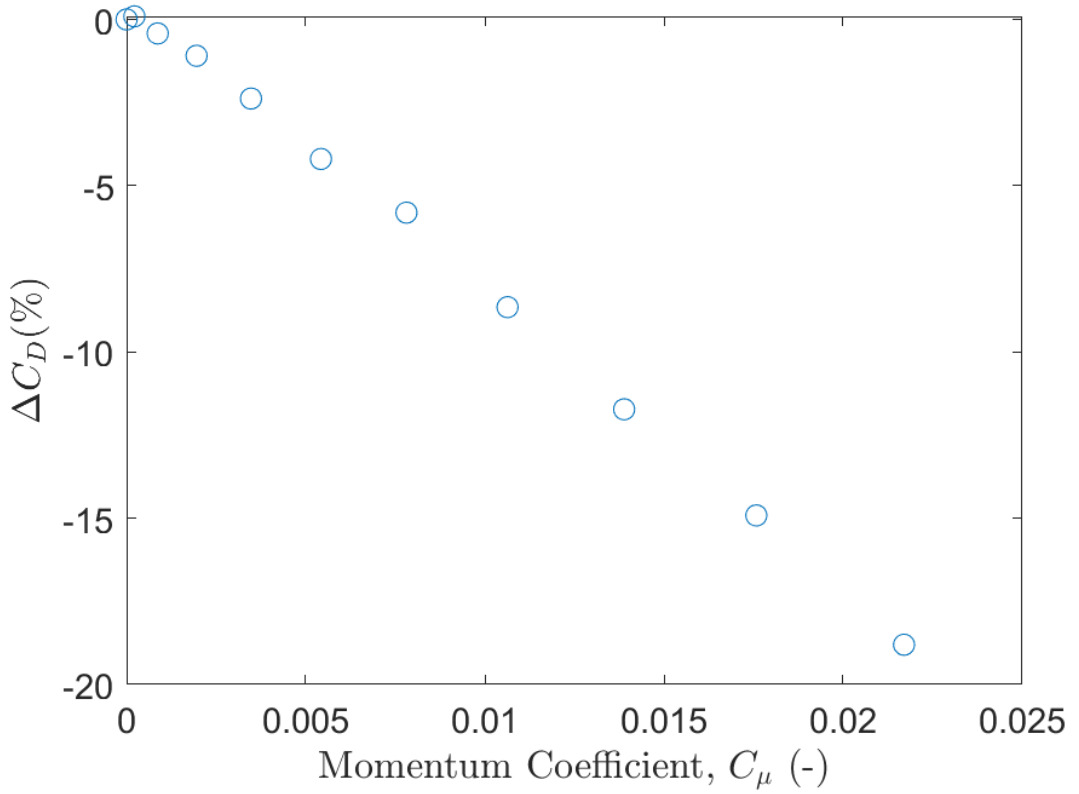


Figure 4.14: Percentage C_D change vs Momentum coefficient for the 2D 35° Ahmed body CFD simulation.

a minimum. This occurs a short time after the SJA has reached maximum suction in its cycle. The black arrow indicates the SJA location and direction of the flow. The key thing to notice here is the delay in separation that the suction cycle is able to generate. The delayed separation increases the static base pressure at the top of the rear slant, resulting in a pressure coefficient increase, which can be seen in Figure 4.18. This leads to a reduction in the pressure drag that the body experiences, which produces the drag reduction. The velocity contours also show a larger velocity magnitude over the apex of the top of the model and the rear slant. This is corroborated by Figure 4.22, which shows a significant increase in the velocity magnitude just upstream of the slant.

The bottom image in Figure 4.16 shows the velocity contour for the time snapshot where the drag coefficient is at a maximum. This occurs a short time after the maximum blowing in the SJA cycle. It is apparent here that the flow separation has again been delayed, due to the deflection of the shear

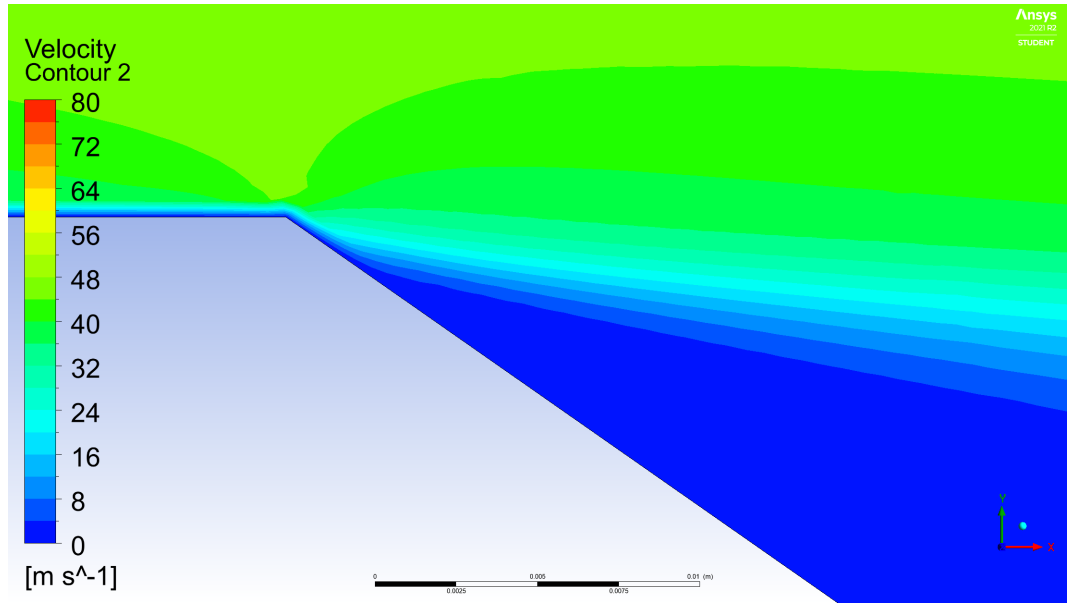


Figure 4.15: Time averaged velocity contour for the baseline (no actuation) case.

layer, resulting in a higher induced velocity over the top edge compared to the baseline case. These changes are less pronounced than for the minimum drag contour, but would still be expected to produce a drag reduction. However, there is a small recirculating region just below the SJA in the bottom image of Figure 4.16. This recirculating region decreases the pressure on the rear face and despite the beneficial changes, the drag coefficient is actually increased compared to the baseline case. The magnitude of this increase is smaller than the magnitude of the decrease for the minimum drag coefficient which produces a net drag reduction over the full SJA cycle.

These results were to be expected as suction actuators are among the most effective means of producing a drag reduction for high Reynolds number bluff body flows. The main issues with suction actuators are the need to store or expel the air that is sucked in. SJAs do not require any onboard storage as they are ZNMF devices and so can be much easier to install as they require much less room.

The separation point for the 3 cases is illustrated in Figure 4.17 which shows the velocity gradient, du/dy at the top of the rear slant. A velocity gradient equal to zero represents a separation point. The black line at the top of Figure

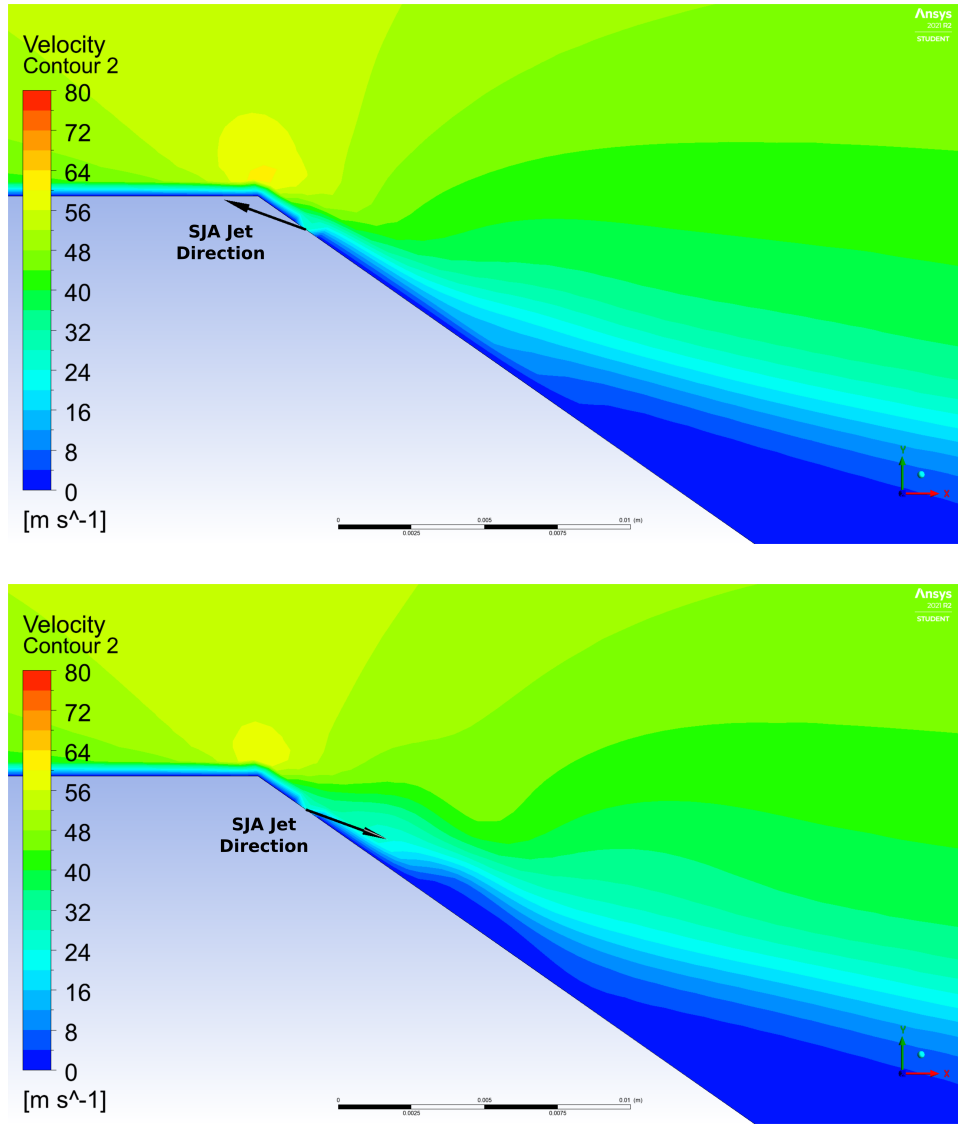


Figure 4.16: Velocity contours for the timestep where the drag coefficient is a (top) minimum (max suction) and (bottom) maximum (max blowing) for $C_\mu = 0.008$.

4.17 shows the SJA location at $y/y_h = 0.996$. The baseline case with no jet actuation crosses $du/dy = 0$ at a height of $y/y_h = 0.99$. The maximum suction of the jet cycle produces the minimum drag case which crosses $du/dy = 0$ at a height of $y/y_h = 0.965$. The blowing portion of the jet cycle produces the maximum drag case which crosses $du/dy = 0$ twice. The first time it crosses is at $y/y_h = 0.98$, but crosses back just below this, suggesting that the recirculating region, just below the SJA, causes the flow to separate and reattach before separating again further down the slant. In fact, the full separation for the maximum drag case is slightly further down the rear slant, at $y/y_h = 0.96$,

than the full separation for the minimum drag case. The key takeaway here is that the operation of the SJAs causes a delay in separation, with the suction cycle producing the greatest drag reduction and the blowing cycle causing an unwanted recirculating region just below. This corresponds with the study by Rouméas [46], which showed that suction at the top of the slant was able to suppress the flow separation over the rear slant.

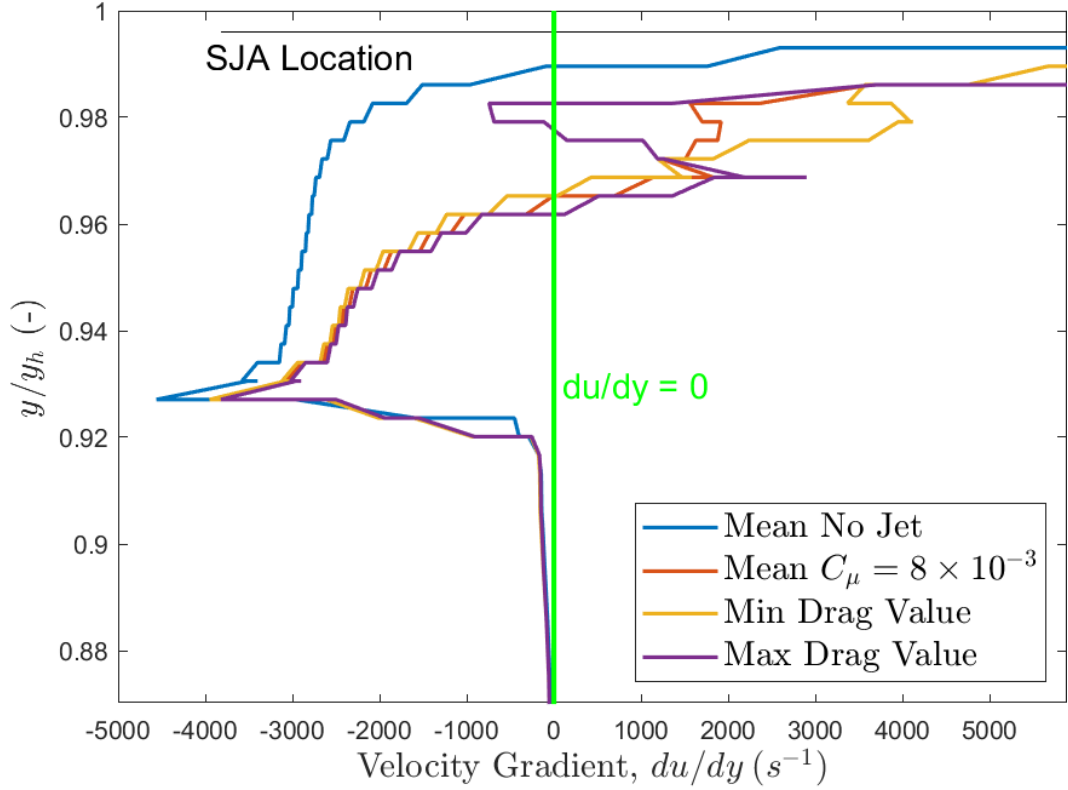


Figure 4.17: A velocity gradient comparison on the rear faces for the baseline, minimum drag and maximum drag cases.

The effects on the rear pressure can be seen in Figure 4.18. There is again a black line right at the top of the plot that indicates the location of the SJA, while the black line in the centre of the figure at $y/y_h = 0.55$ indicates the intersection of the bottom of the slant and the top of the rear face. It is clear that the delayed separation in the minimum drag case leads to a large pressure recovery on the rear slant, which is sustained along the full length of the slant and even onto the rear face below the slant. The recirculating region in the maximum drag case causes an initial drop in the pressure on the rear face, indicated by the switchback at the top of the plot for the maximum drag

case. This leads to a reduced pressure on the rear slant which is propagated onto the rear face. It is clear that the pressure increase on the rear slant for the minimum drag case is larger than the decrease in pressure caused by the baseline, which is what leads to an overall drag reduction.

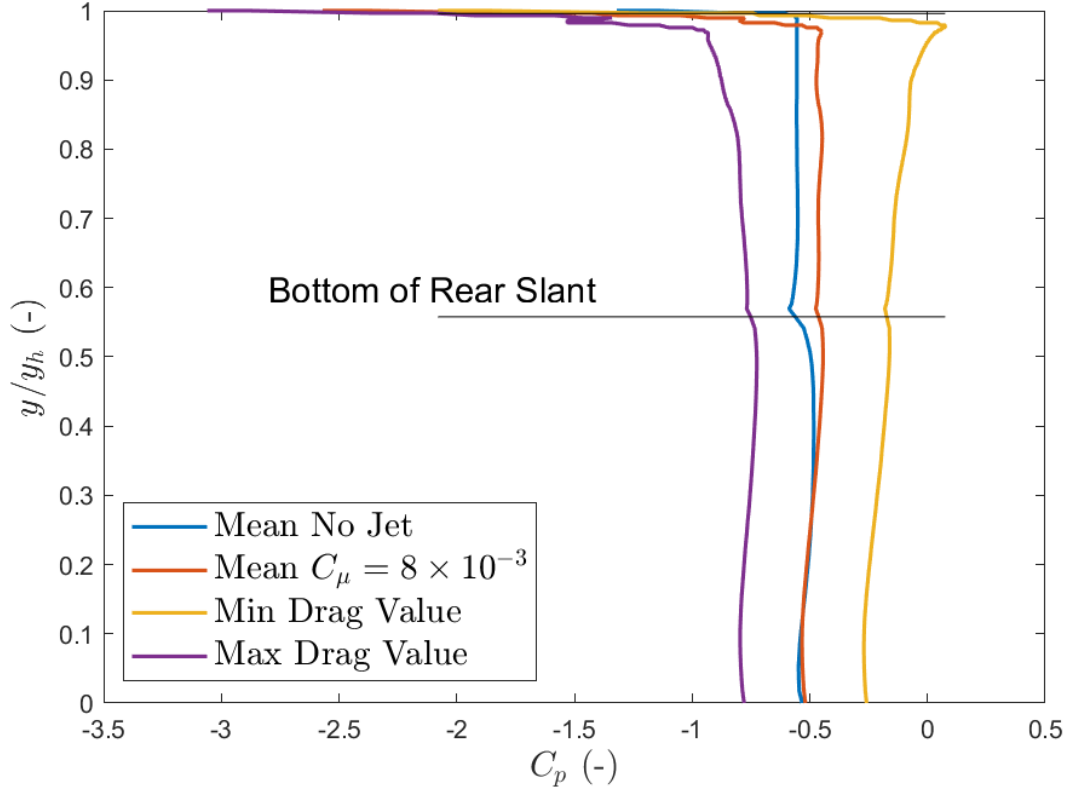


Figure 4.18: A pressure coefficient comparison on the rear faces for the baseline, minimum drag and maximum drag cases.

Figure 4.19 shows the time averaged velocity contour plots of the wake behind the model for the baseline case and the case with $C_\mu = 0.008$. The two contours are very similar with the biggest difference being the increased flow over the top edge of the slant. The bottom image in Figure 4.19 shows the case with $C_\mu = 0.008$ and it can be seen that there is a larger yellow contour here compared with the baseline contour. There is also a small delay in the separation on the rear slant and a small reduction in the height of the wake. However, this is easier to see in Figure 4.20, which shows the differences in the velocity contours in Figure 4.19.

The positive contours in Figure 4.20 indicate a faster velocity in the case with $C_\mu = 0.008$. It is clear now that there is a faster region in the top of the

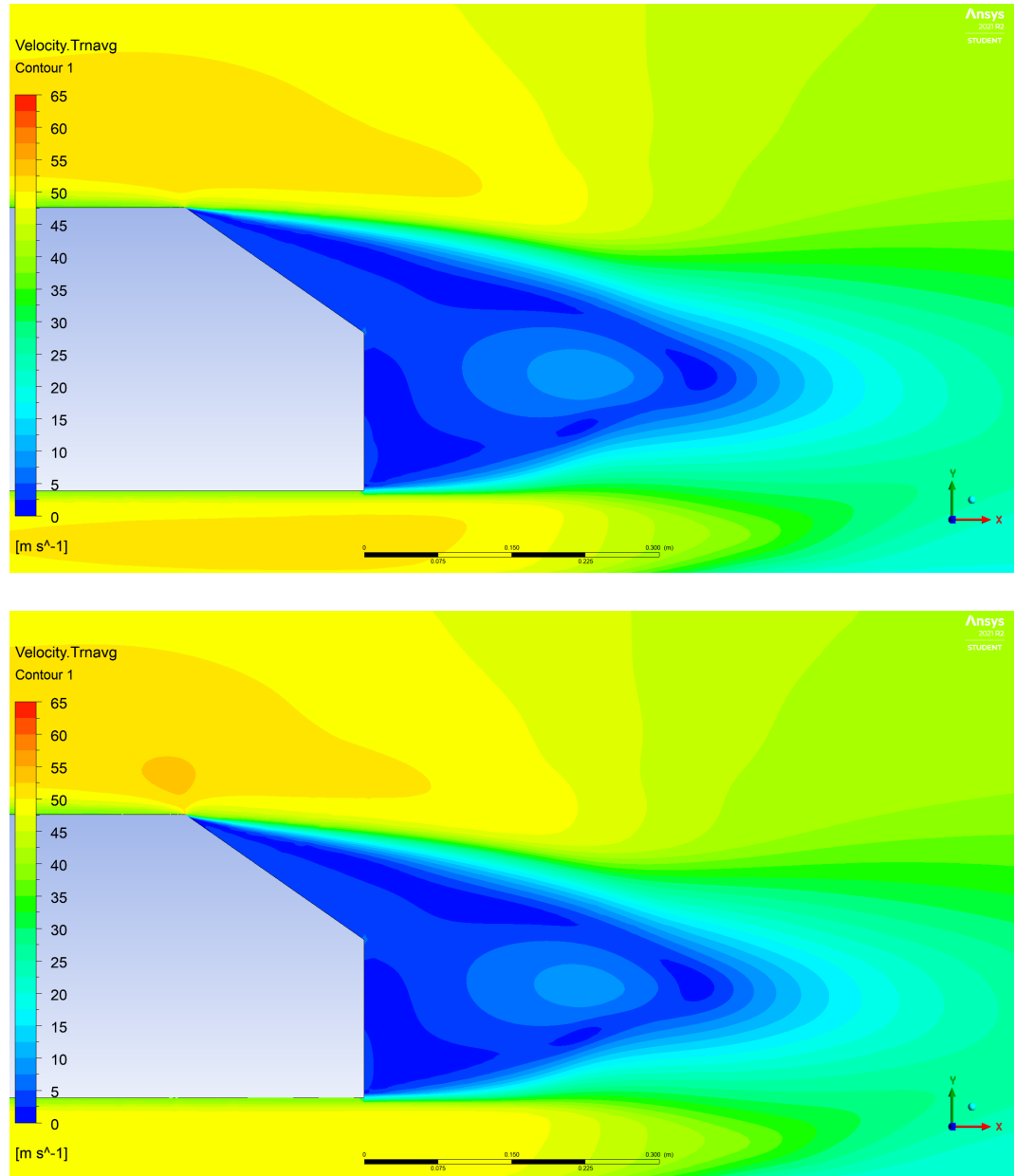


Figure 4.19: Time averaged velocity contours of (top) the baseline case and (bottom) the case with $C_\mu = 0.008$.

wake of the actuated case. The increase in velocity is highest at the exit of the SJA where the flow velocity is increased by 45%. This is to be expected as the SJA is operated at 150% of the freestream velocity in this case. The large green contour line along the top of the wake shows a sustained increase in the velocity for the actuated case. The increase in velocity here is upto 20%, which is a significant increase, suggesting that the top of the wake is lower under actuation. There are a few other minor changes to the wake, but these are inconsequential to the flow, showing that the reduction in the height of the wake leads to a reduction in the size of the wake.

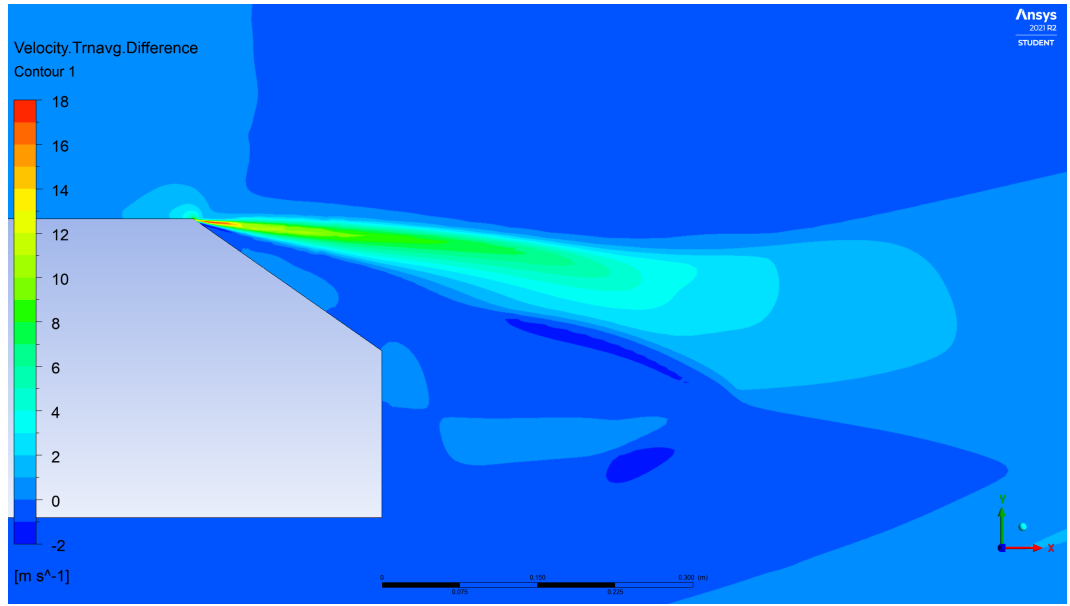


Figure 4.20: Contours of the velocity differences between the images in Figure 4.19.

The drop in the wake height is further confirmed by the velocity profiles over the rear slant presented in Figure 4.21. The baseline velocity profiles over the rear slant are compared to the velocity profiles for the actuated cases with $C_\mu = 0.002$ and $C_\mu = 0.008$. It is clear that the flow velocity is larger and the height of the wake is lower as the momentum coefficient is increased as was hypothesized from Figures 4.19 and 4.20. It is also obvious from these profiles that the jets have very little to no effect on the recirculating region other than reducing the height of the wake. The literature would suggest that the recirculation regions should be moved further downstream resulting in a drag

reduction. The reduction in wake height appears very minor, however, since it extends for the majority of the length of the wake, it has a large effect on the static base pressure, reducing the drag coefficient by upto 6% for $C_\mu = 0.008$.

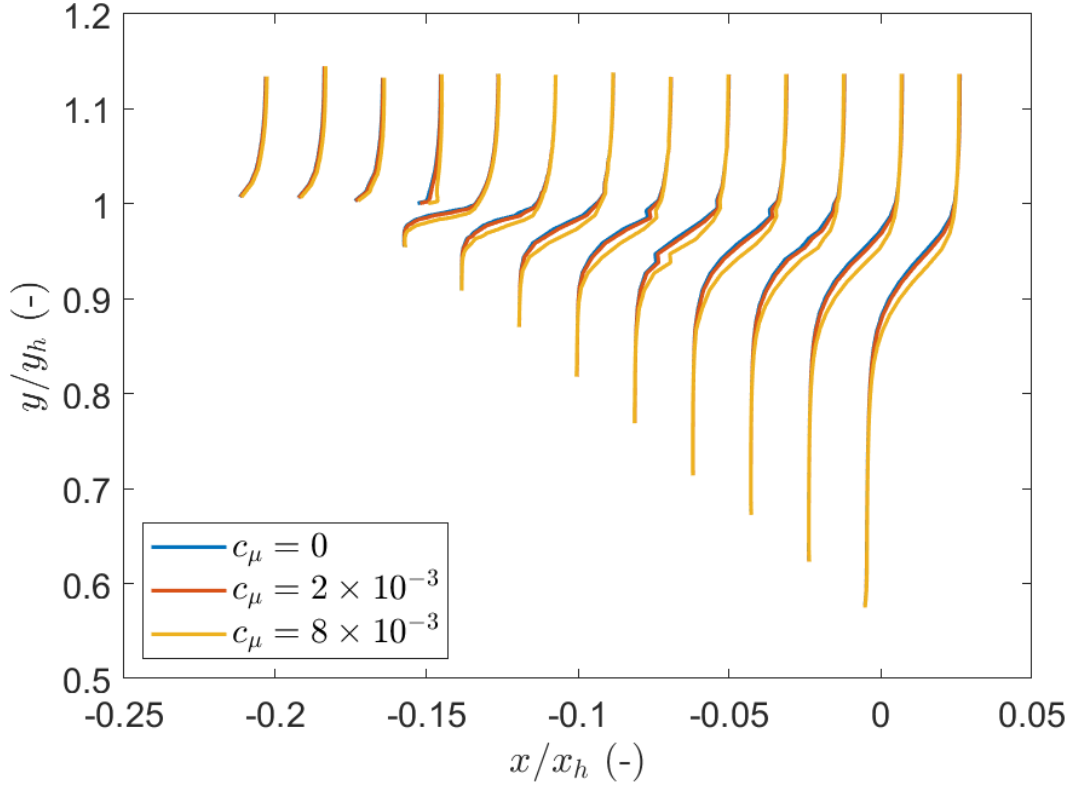


Figure 4.21: Velocity profiles over the rear slant for $C_\mu = 0$, $C_\mu = 0.002$ and $C_\mu = 0.008$.

Perhaps the most interesting part of Figure 4.21 is the velocity profile at $x/x_h = -0.17$ which shows the effects of the SJAs have on the flow just upstream of them. This profile can be seen in more detail in Figure 4.22. The SJAs induce a faster velocity over the top of the rear slant compared to the baseline. The greater SJA momentum coefficients further increase the induced velocity over the top of the rear slant.

The increased velocity likely leads to the reduced height of the wake as the higher pressure flow tries to flow into the lower pressure wake region. The subsequent reduction in the wake height likely causes the increase in rear pressure and the reduction in the drag coefficient. This effect can be seen in all of the profiles over the rear slant and into the wake.

The height reduction in the wake downstream of the body is less significant

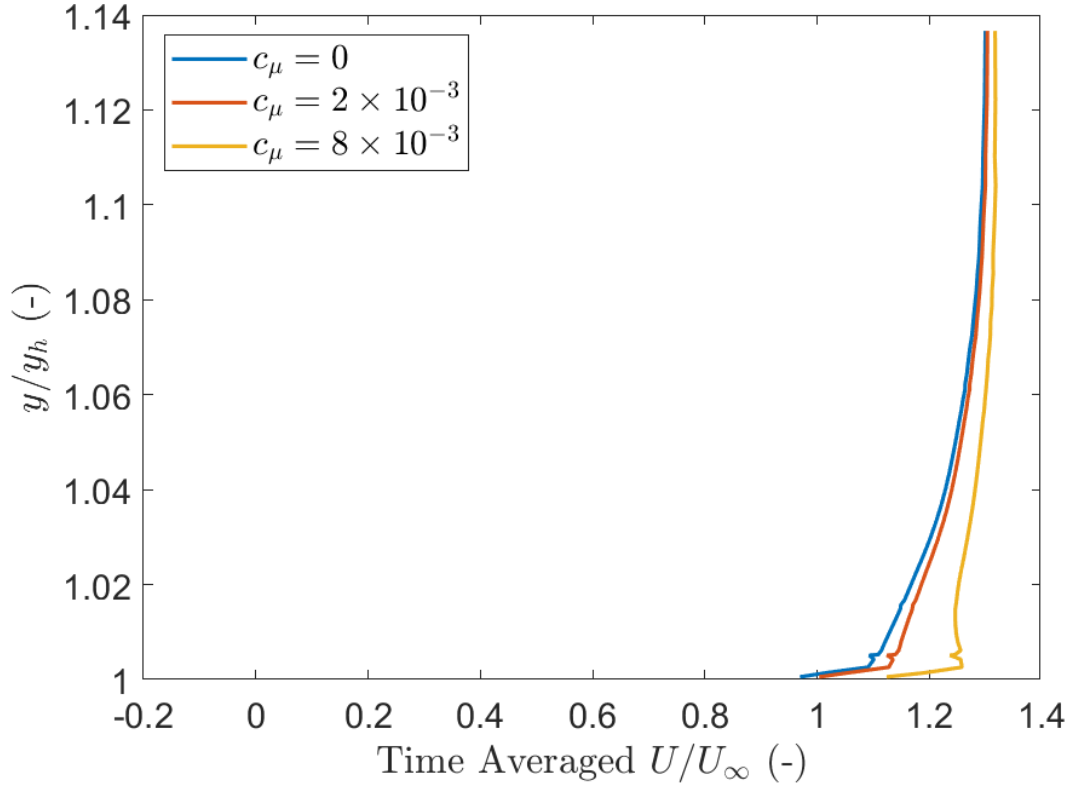


Figure 4.22: Velocity profile over the apex of the rear slant and the top face for $C_\mu = 0$, $C_\mu = 0.002$ and $C_\mu = 0.008$.

than over the slant as the effects of the SJAs are reduced further from the body as seen in Figure 4.20. The velocity profiles can be seen in Figure 4.23, which shows a small change in the wake height for the first few profiles, but a reduced effect in the profiles further downstream of the body. The profiles in Figure 4.23 also show the minimal effect of the SJAs on the velocity in the recirculation regions with little to no change in the profiles with the actuation present. The change is easier to see in Figure 4.24, which shows a zoomed in image of the top of the velocity profiles.

Despite the small reductions in the wake, the static pressure gain on the rear face is significant which leads to a significant drag reduction. The pressure coefficient plot on the rear face of the Ahmed body for the baseline case and the cases with $C_\mu = 0.002$ and $C_\mu = 0.008$ can be seen in Figure 4.25. For the baseline case there is very low pressure at the top of the rear slant when y/y_h is close to 1, with a constant low pressure over the rear slant of approximately -0.56. There is a change in the behaviour of the C_p on the lower rear face,

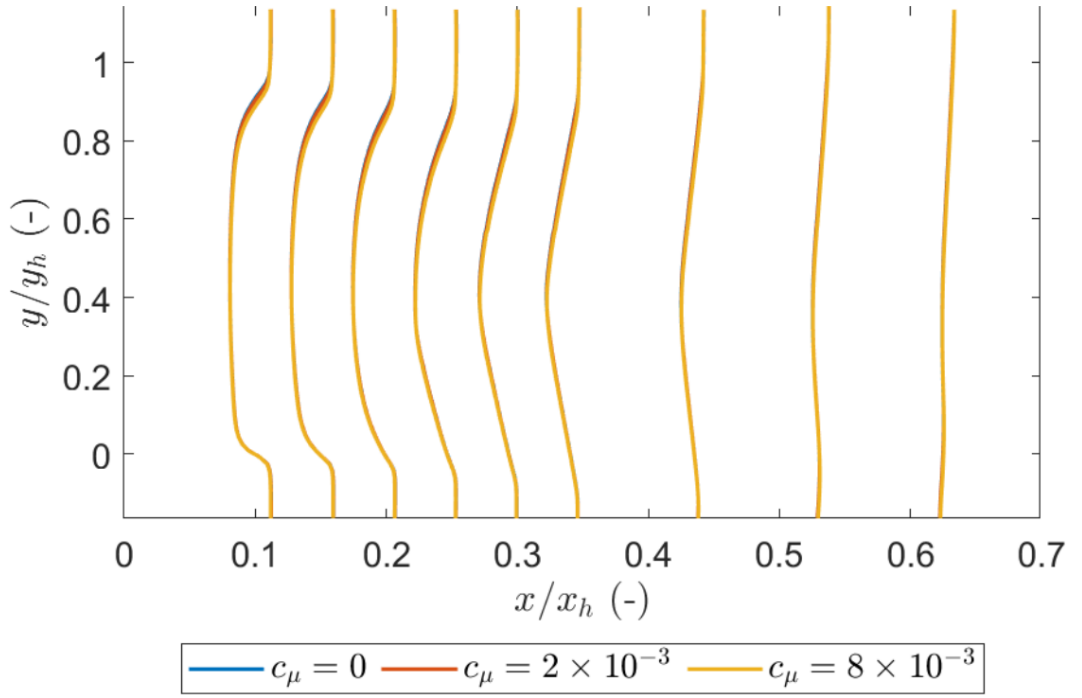


Figure 4.23: Velocity profiles in the wake for $C_\mu = 0$, $C_\mu = 0.002$ and $C_\mu = 0.008$.

below $y/y_h = 0.55$. There is a small spike in the pressure in the upper half of the lower rear face, likely caused by the interaction of the upper and lower recirculation regions in the wake. For each actuated case it is clear that there is a constant pressure increase over all of the rear faces, caused by the delay in flow separation and the wake height reduction. The gain in rear pressure for the case with $C_\mu = 0.008$ is more significant than the gain for $C_\mu = 0.002$ with a gain in C_p of almost 0.03 over all of the rear faces. The static pressure gain is mainly produced by the increased flow velocity over the top of the rear slant, having a significant impact on the wake height over the rear slant. The reduction in wake height further downstream of the model is much less significant and so has less of an effect on the static pressure increase.

Figure 4.26 shows the propagation of 2 counter rotating vortices from the SJA orifice. These vortices travel along the top of the wake. The strength of the vortices quickly diminish after leaving the SJA as the freestream flow weakens them. This would suggest that the main effects from the SJAs are located in the immediate vicinity of the orifice exits. However, it is clear from Figure 4.20 that the effects of the SJAs propagate further downstream. This means

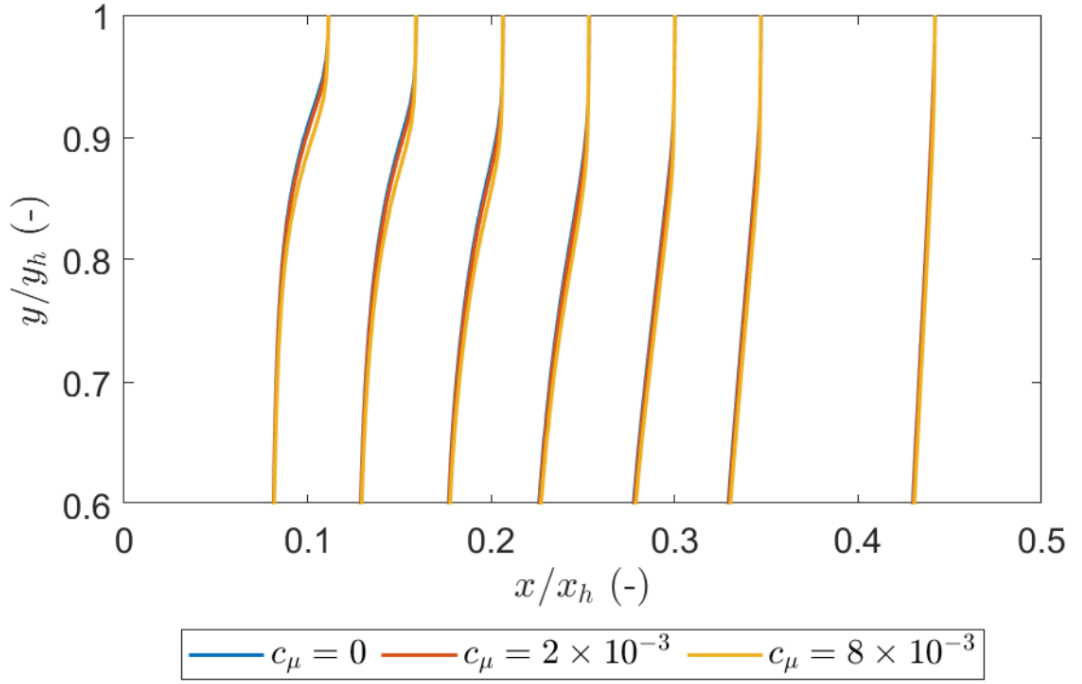


Figure 4.24: A zoomed in view of the top of the velocity profiles in the wake for $C_\mu = 0$, $C_\mu = 0.002$ and $C_\mu = 0.008$.

that the SJAs interact with the flow in close proximity to the orifices, but the interaction causes a prolonged benefit for the length of the wake.

4.1.4 Net Power Savings for Control with Synthetic Jet Actuators

The 2D 35° Ahmed body was simulated with a range of SJA velocities, at various freestream velocities, and the maximum net power saved for each freestream velocity was calculated. The results can be seen in Table 4.1 along with the momentum coefficient used to obtain the maximum net power saving at each different freestream velocity.

Clearly, the jets had little impact at a freestream velocity of 4.47m/s, only achieving a power saving of 0.49%. The SJAs were able to produce the greatest power savings between freestream velocities of 17.88 and 22.35m/s. For the larger freestream velocities, the effectiveness of the jets began to drop again. This is due to the exponential power usage of the SJAs, determined using the model in Equations (3.7) and (3.8), overtaking the drag reductions

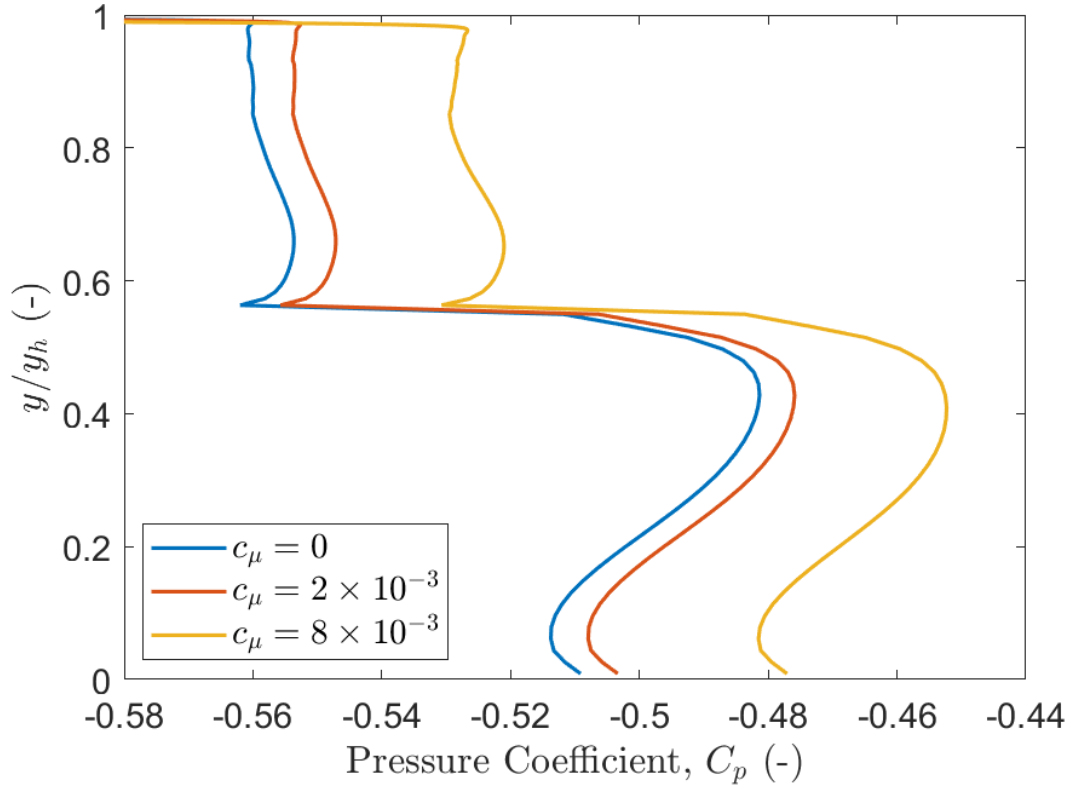


Figure 4.25: Pressure coefficients on the rear faces of the Ahmed body for $C_\mu = 0$, $C_\mu = 0.002$ and $C_\mu = 0.008$.

achieved for the larger jet velocities, which can be seen in Figure 4.14. The optimum momentum coefficient of the SJAs also showed a downward trend as the freestream velocity increased, which contributed to the decline in the power saving achieved. If the actuator power costs are neglected, larger drag reductions can still be achieved for the higher freestream velocities, until the point that the SJAs dominate the flow field and the model breaks down.

Table 4.1: A table showing the percentage net power saving from the CFD simulation for various freestream velocities for the 2D 35° Ahmed body.

Freestream Velocity (m/s)	Momentum Coefficient, $C_\mu \times 10^{-3}$ (-)	Net Power Saving (%)
4.47	4.4	0.49
8.94	4.9	1.98
13.41	4.4	4.01
17.88	4.0	5.70
22.35	3.3	5.86
26.82	2.6	3.84
31.29	2.2	3.12
35.76	1.8	2.47
40.23	1.5	2.13

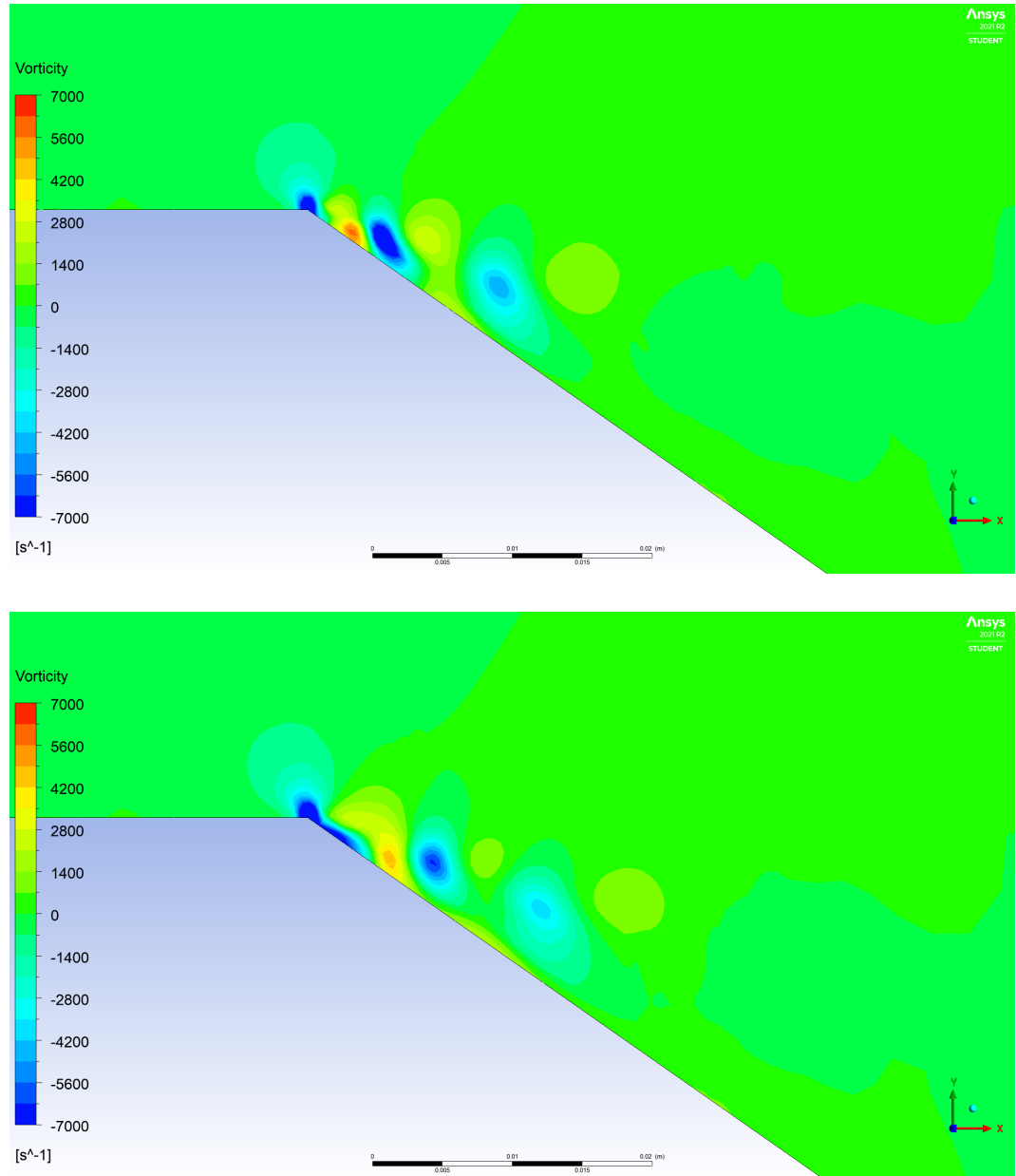


Figure 4.26: Plot showing the development of counter rotating vortices from the SJAs.

4.1.5 Summary

In this chapter, it has been shown that the 2D CFD model is able to approximate the centreplane flow behaviour of the experimental data from the Ahmed body ERCOFTAC database [104] with a reasonable accuracy. The error in the rear slant and wake profiles is upto 20%, however, for the slant profiles, a large portion of this error (upto 15%) propagates from the increased flow velocity over the top of the model. Despite this, the numerical model is able to predict the wake height to within 2% and the separation point to within 3mm, which are the key areas of the flow that this research aims to control. A synthetic jet actuator array was added to the rear slant of the model, which produced drag reductions of upto 19%. The drag reduction observed was similar to other studies in the literature, suggesting that the current approach was reasonable for further study.

The method of drag reduction was explored and it was found that the SJAs induced a larger flow velocity over the top edge of the slant and delayed flow separation on the rear slant. This led to a shear layer deflection into the wake which in turn reduced the size of the wake and increased the pressure recovery on the rear slant and face. The main benefit of the SJAs was caused by the suction cycle of the SJAs, where the initial separation was delayed by 0.017m or 8% of the rear slant and the pressure on the rear slant increased by $C_p = 0.03$. This happens despite the increased vorticity caused by the reversing of the flow direction at the exit of the SJA orifice. The reduction in drag during the suction cycle of the jets was also larger than the detrimental drag increase caused during the blowing cycle of the jets, where the jets caused a small recirculation region to form just below the orifice, which caused a drop in the rear pressure on the slant, and led to a drag increase.

Chapter 5

Reduced Order Modelling

5.1 Introduction

The majority of closed loop control schemes presented in the literature are based on an ESC approach [36, 92]. The ESC uses a gradient search algorithm to find a local extrema for a given cost function. When applied to drag reduction schemes, this usually takes the form of reducing the drag coefficient or increasing the pressure recovery on the rear of the body. The ESC model has multiple approaches that can be used to determine the gradient, but this research focuses on the perturbation based approach. This approach uses a small sine wave perturbation to a system to investigate the nature of the system in a close neighbourhood around the current set point. This determines the ‘direction’ that the set point should move to reach optimal operation, based on a cost function.

The extremum seeking control (ESC) scheme is largely dependent on the tuning parameters in the scheme, such as the amplitude and frequency of the perturbation, the cut-off frequencies of the filters and the gain applied to the system. These tuning parameters are all explained further in Chapter 5.8. This means that although model information is not needed, a lot of optimisation is needed to get the best performance from the scheme. Even for optimal ESC schemes, the convergence times for the model can be large, meaning that the

model is unsuitable for real time control as is needed for a closed loop control scheme. The convergence times of the scheme are limited by the exploratory behaviour of the perturbation and as such, generally, do not react well to large, abrupt changes in the system. One such case is the initial estimation to the optimum for the scheme. By combining this ESC scheme with information about the model to improve the initial estimation for the system, the large convergence times should be reduced.

The CFD model from Chapter 4 is used to provide the information about the model. CFD models are useful to gather model information as it is easy to collect a large dataset to fully understand the flow. This also presents an issue since CFD models are slow to simulate and hence unsuitable for real time control.

A reduced order model (ROM), uses patterns in the data along with a new set of basis functions to capture the key information from the model. The basis functions or modes are the building blocks which are used in a linear combination to describe the full scale model. A ROM also removes the less important information, or less energetic modes, meaning that the size and complexity of the model is reduced. There are various methods for reduced order modelling, but this research focuses on the Input-Output Dynamic Mode Decomposition (IODMD) scheme [73]. This scheme is an adaptation of another scheme, called Dynamic Mode Decomposition (DMD), for use with systems that require an input and an output. The DMD scheme uses a Proper Orthogonal Decomposition (POD) approach to identify a new set of basis functions, before applying a system identification method to project the key information onto the new basis functions. The IODMD scheme builds on this by incorporating input and output matrices that can be used to simulate the temporal behaviour of the system.

The IODMD scheme is used to build a ROM for the CFD simulation in Chapter 4. The key information for this setup is the change in the drag coefficient (C_D) as the SJA velocity is changed, which represent the outputs and inputs

respectively. For a real world model, the C_D cannot be obtained directly and instead has to be computed using pressures on and around the body. Hence, the ROM uses a static pressure change at the rear of the vehicle, which is converted to a C_D change. The ROMs are built using one set of input and output data and then tested using a second set of inputs. The ROM output is compared to the second set of outputs to determine the effectiveness of the model. This use of independent training and testing sets demonstrate the independence between the inputs, for example, setting the SJAs to 60m/s provides the same performance regardless of previous SJA velocities.

5.2 Input-Output Dynamic Mode Decomposition

The IODMD scheme has been used in the literature to successfully reduce the scale of a fluid problem such as a small wind turbine model [73]. This research takes the IODMD approach further and applies it to much larger scale models. The scheme uses a set of ‘snapshots’ obtained from the CFD simulation to produce a relationship between the SJA velocity input and the C_D change output. The snapshots are a series of vectors that contain the static pressure field at a given timestep. By combining the vectors into a snapshot matrix, where the columns represent monotonic increasing timesteps, a complete time history of the pressure field can be obtained. The mechanism for the model order reduction, is the common POD approach, applied to the snapshot matrix, which uses a Singular Value Decomposition (SVD) to factorise the matrix and produce an orthonormal eigenbasis. An appropriate truncation is applied to the eigenbasis, to retain the most important modes in the system. The DMD scheme then uses a system identification method, similar to N4SID [74, 75], to determine the relationship between the SJA velocity and the C_D .

The validity of this approach was tested using a 500 mode ROM built for the

2D 35° Ahmed body CFD simulation with a single SJA velocity of 60m/s and the freestream velocity fixed to 40m/s. This corresponds to $C_\mu = 0.0008$. The CFD simulation was allowed to settle to an unactuated equilibrium condition before the snapshots were collected for the remainder of the simulation. This meant that the initial snapshots in the matrix captured the unactuated behaviour of the system, whilst the remaining snapshots captured the actuated system. Various snapshot timesteps were tested, but it was decided that the best way to capture the most information was to capture a snapshot at each of the simulation steps, meaning that the time between snapshots was equal to the simulation timestep of 1×10^{-5} s. The simulation in total ran for 3.5 seconds, with data captured from 1 second, with the SJA active for the final 1.5 seconds, meaning that 3,300 cycles of the SJA and 125 cycles of the vortex shedding were captured.

Figure 5.1 presents the step response for this model. Three sets of data are presented which show the C_D change obtained directly from the CFD simulation alongside the output from 2 different versions of the ROM. The full domain model was built using static pressure field data from the full simulation domain, including the large areas away from the body that are used to reduce the blockage ratio in a CFD simulation. It was decided that this information was adding unnecessary complexity to the reduced order model and that it was mostly being removed in the POD anyway. Therefore, a ROM was also built using just the static pressure data obtained from the surfaces of the model, which has been dubbed the face domain ROM. The ROMs for both models use 500 modes as the model faces only have 561 data points and so it allowed a direct comparison between the models.

All the lines oscillate with a frequency of 2.2kHz as this is the SJA frequency used in the CFD simulation. The timestep of the simulation was set to 1×10^{-5} s to ensure that each cycle of the SJAs was captured in enough detail. This leads to the data being difficult to distinguish, so Figure 5.2 shows the same lines after passing through a low-pass filter to remove the high frequency SJA oscil-

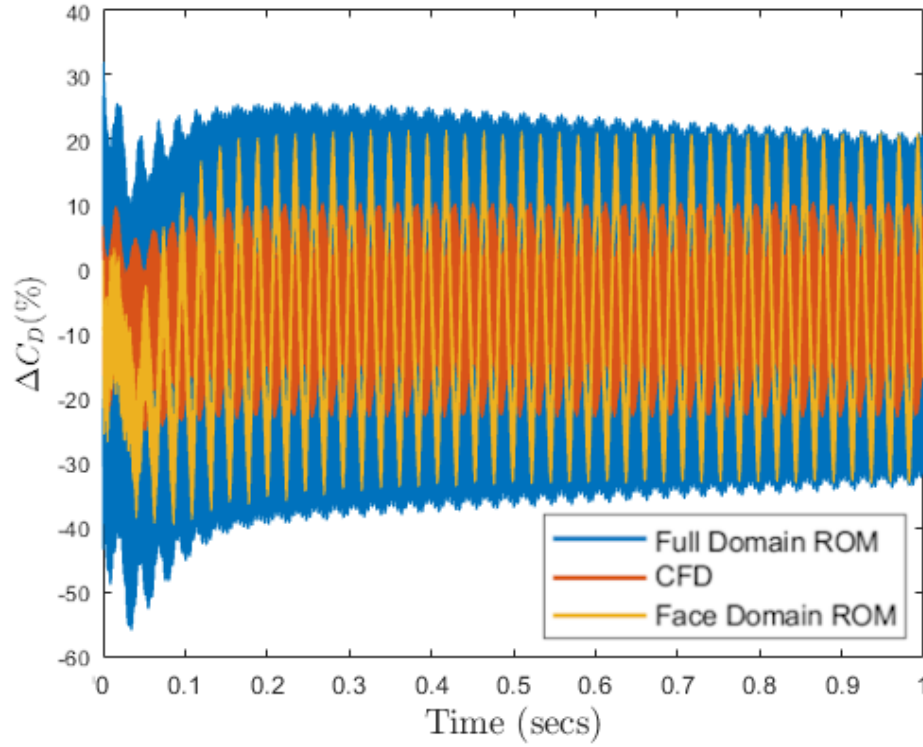


Figure 5.1: The C_D from the CFD and for the full data and face data ROMs with 500 modes.

lations. This is just post processing to make the data easier to visualise and in no way affects the reduced order modelling procedure. The shedding frequency of the CFD simulation is approximately 50Hz, which is what produces the C_D oscillation in Figure 5.2. This oscillation continues indefinitely regardless of the SJAs. It is clear from Figure 5.1 that the full domain ROM over predicts the amplitude of the oscillations. By comparing this to Figure 5.2, it would appear that the ROM over predicts the amplitude of the SJA response, but under predicts the variation due to the vortex shedding in the CFD simulation. This is due to the nature of the scheme which creates a linear map between an input and an output for a single CFD simulation, including the equilibrium state and any SJA velocities that were tested. The shedding frequency causes issues, because it adds noise to the output signal.

The ROM built with the full domain dataset also experiences a decay in the oscillations, likely due to behaviour captured from the pressure field in the wake and behind the model, where the effects of the SJAs decay further downstream

of the body. The ROM built with the face domain data does not experience this decay. The decay for the full data is reduced as the number of modes is reduced to 500 and the remaining modes are almost all obtained from the surface of the body, accounting for 99.9% of the variance in the pressure. This supports the fact that the decay is caused by these downstream effects.

It is clear that the model cannot accurately predict the oscillations of the CFD simulation in the current setup. The full domain model would likely produce a better approximation to the oscillations with more modes, but it was deemed unnecessary for this research as the key information was the mean C_D from the simulation and increasing the number of modes would increase the computational complexity of the model. Both ROMs and the CFD data oscillate around $C_D = -0.036$, suggesting that the face domain ROM is suitable for this use case as it captures all of the relevant data, whilst using less computing resources.

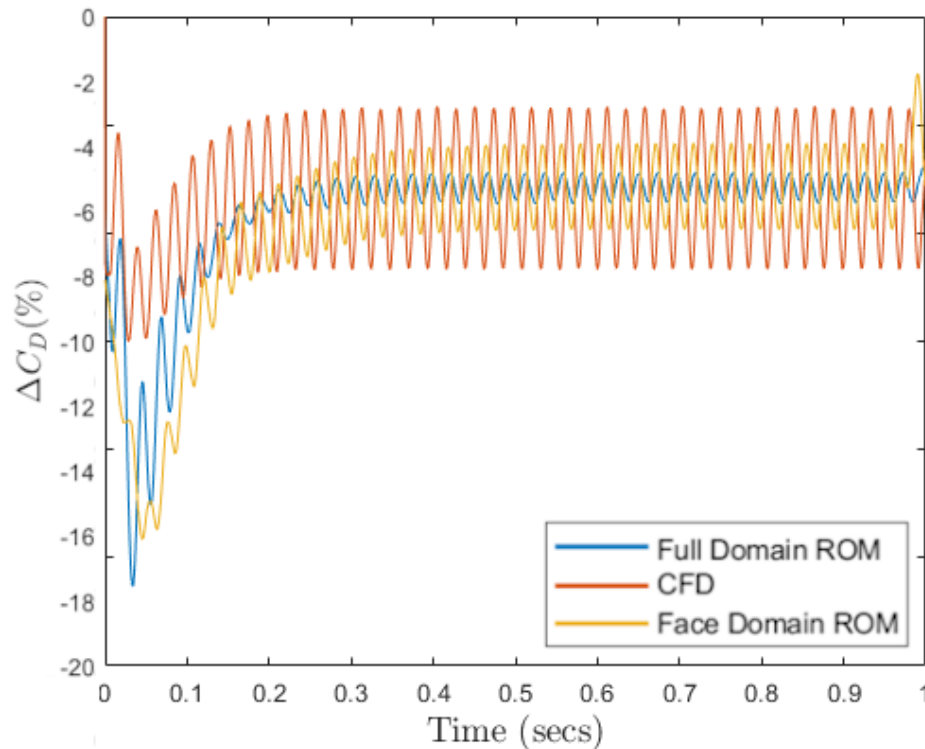


Figure 5.2: The averaged C_D from the CFD and for the full data and face data ROMs with 500 modes.

Unsurprisingly, these ROMs are very good at approximating the C_D for a

single SJA velocity as it creates a linear map from an input to an output. With a single SJA value, the relationship is easily obtainable in a single mode. These ROMs are unable to predict the C_D changes for other SJA velocities because they have not been built using the data. Therefore, a generic input map was created, with various SJA velocities, to add data across multiple SJA velocities, but also provide a framework to compare models.

5.3 Adapting the Input to Improve the Model Fit

A ROM was built using the new input map with a fixed freestream velocity of 40m/s. The number of modes was set to 500 as before. This model represents almost 100% of the information in the full order system.

Figure 5.3 shows the oscillation frequencies in the CFD simulated data. This plot was obtained by passing the pressure field through a Fast Fourier Transform (FFT) and computing the single sided amplitude spectrum. This plot can be analysed to show the frequencies of the oscillations in the pressure fields. For example, the SJA actuation frequency can be identified at 2.2kHz, where there is a peak in the magnitude of -10dB. The shedding frequency can also be identified at 50Hz, with a peak of -50dB. The other peaks are multiples of these key frequencies caused by aliasing. The other key observation from Figure 5.3 is the low frequency oscillations have a magnitude of -100dB. This shows that the change in the mean C_D experienced by the model is in the order of 10^{-2} , which was shown in Chapter 4. This plot also tells us that the SJA oscillations have a large effect on the instantaneous value of the C_D .

This ROM was used to simulate the response with a monotonically increasing input map. The CFD model was simulated with the same input map and the C_D was averaged after to give a comparison between the models. The results are shown in Figure 5.4. The peak jet velocity is kept on the x-axis, rather

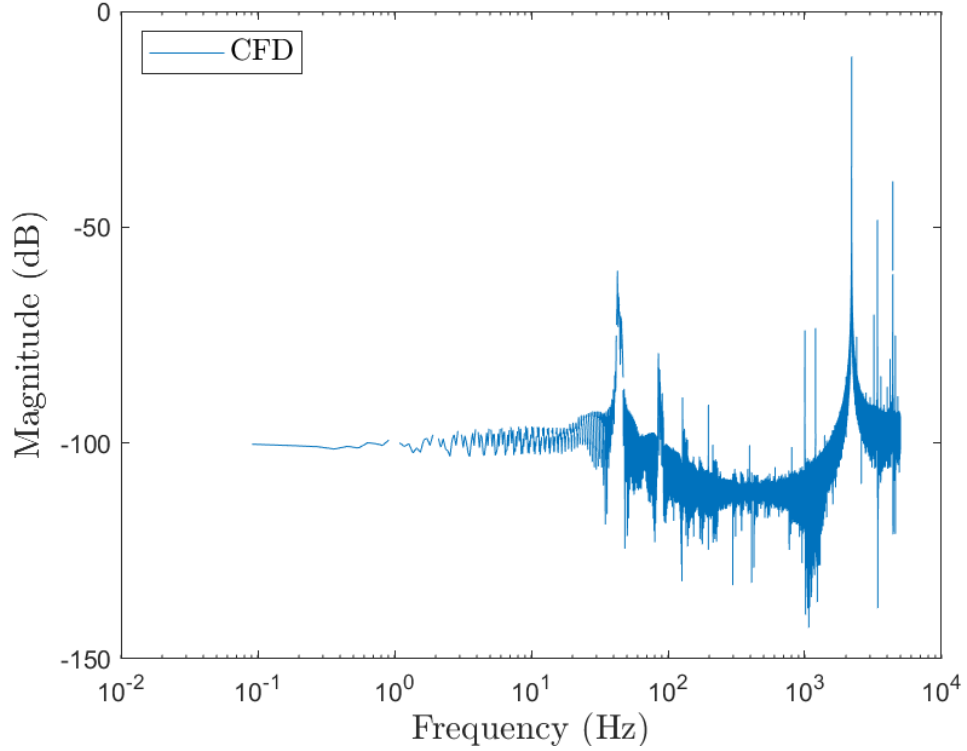


Figure 5.3: *Oscillation frequencies present in the CFD simulation data.*

than using a non-dimensional velocity with the freestream velocity, as the relationship varies with the freestream. The ROM produces an almost linear relationship, whilst the CFD data shows a non-linear relationship between the SJA velocity and ΔC_D . This leads to an over prediction of the drag reduction, for the majority of the SJA velocities, by the ROM.

The IODMD scheme attempts to create a linear map between an input and an output as demonstrated in Equation (5.1).

$$y = k_0 u, \quad (5.1)$$

where y is the output to the system, u is the input to the system and k_0 is some constant determined by the ROM. In this case, y is the C_D change and u is the SJA velocity. However, Figure 5.4 demonstrates that the CFD model has a non-linear relationship, it was found that the relationship between SJA velocity and C_D is quadratic. This was not unexpected as the drag is proportional to the square of the speed and in an ideal case, the SJAs are able to have a direct

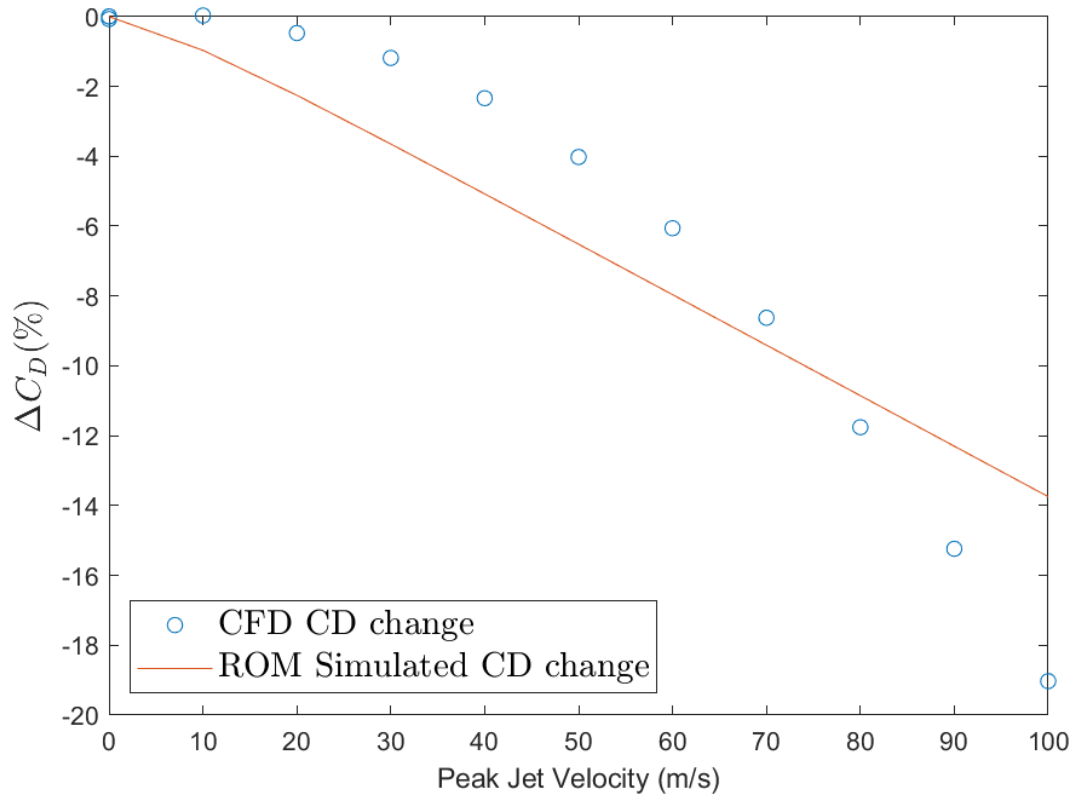


Figure 5.4: *SJA velocity vs C_D change for the linear model.*

impact on the flow. Instead of adapting the linear map to create a non-linear map, it was hypothesized that by applying a change of variables, a linear map could be used.

By replacing u with u^α in Equation (5.1), where α is a constant to be determined, the quadratic relationship between SJA velocity and C_D can be achieved. Figure 5.5 shows the system response for the new ROM using the change of variables with $\alpha = 2$. It is clear that the fit is much better than in Figure 5.4 with an average error of only 0.5%.

This agreement is further demonstrated when comparing the oscillation frequencies in the pressure field between the 2 ROMs and the CFD simulation data. The linear model, shown in Figure 5.6 is able to capture the key oscillation frequencies for the SJAs and the shedding frequency of the body, however, the low frequency oscillations, which are key to capturing the mean C_D , are much larger than the CFD simulation. The linear model also puts more emphasis on the shedding frequency's contribution to the overall pressure changes

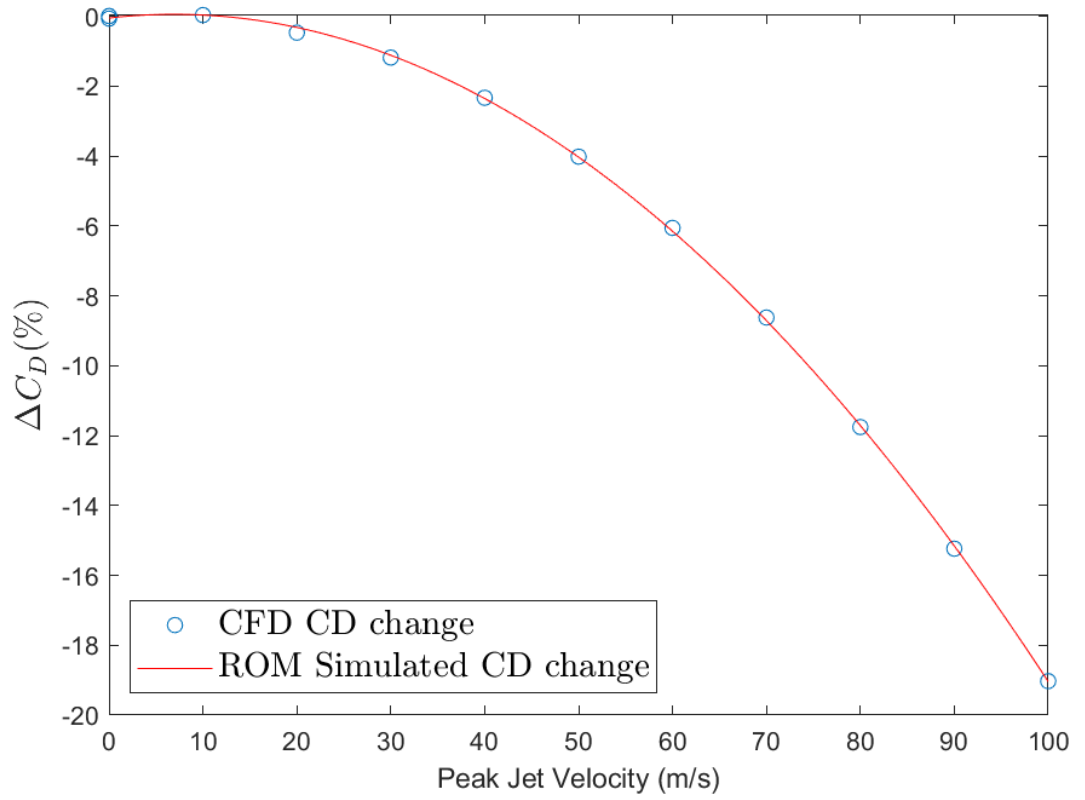


Figure 5.5: *Percentage C_D change vs SJA velocity, demonstrating the quadratic relationship.*

than for the CFD simulation.

Figure 5.7 shows the same model but with the change of variables applied to use the square of the velocity input. Again, this model is able to capture the high frequency oscillations of the SJAs and the lower frequency oscillations of the shedding vortices, however, for both key frequencies, this model under predicts the magnitude of the oscillations, which leads to inaccuracies in the instantaneous C_D . This model is able to accurately capture the low frequency oscillations in the pressure field, which leads to the accurate prediction of the mean C_D reduction.

5.4 Producing a Lower Order Approximation of the System

The ROM containing 500 modes, coupled with the change of variables, can be used to obtain a good approximation to the system for various SJA velocities

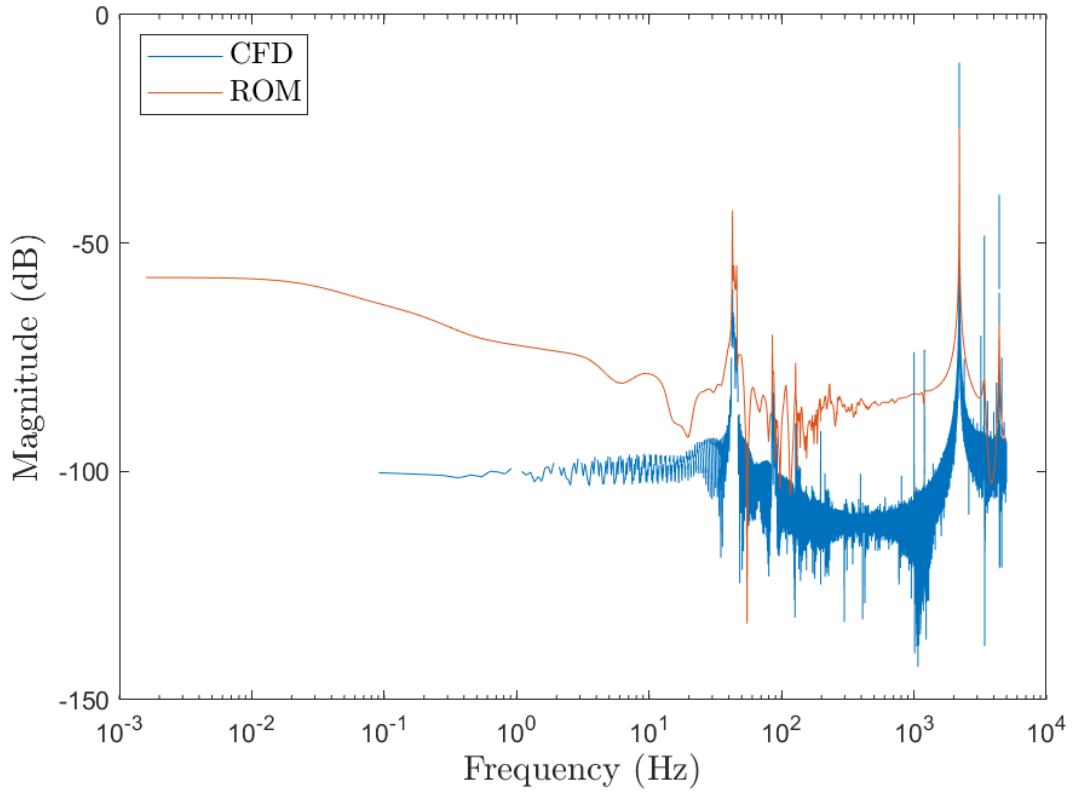


Figure 5.6: *Magnitude (in dB) vs frequency for the linear model and the CFD system.*

in real time. However, the time to simulate the model can be reduced further by reducing the number of modes in the ROM. The output is also given more consideration with two different approaches checked. The first approach just takes the C_D value directly from the CFD simulation and uses that as the output, whilst the second method uses a difference in the static pressure at the front of the body and the rear of the body, which has then been converted to a drag coefficient for comparison purposes. This has a small effect on the 2D model as shown in Figure 5.8, where both lines are almost on top of each other. The second approach does not take into account the skin friction, which is very small in the 2D CFD simulations, about 4%, and so does not have a large impact on the C_D value. By ignoring the C_D measure and using just the static pressure drop, this issue could be avoided as there is very little change in the skin friction drag, meaning that there should be a constant difference between the two approaches.

For the tests in this section, both the full domain data ROM and the face

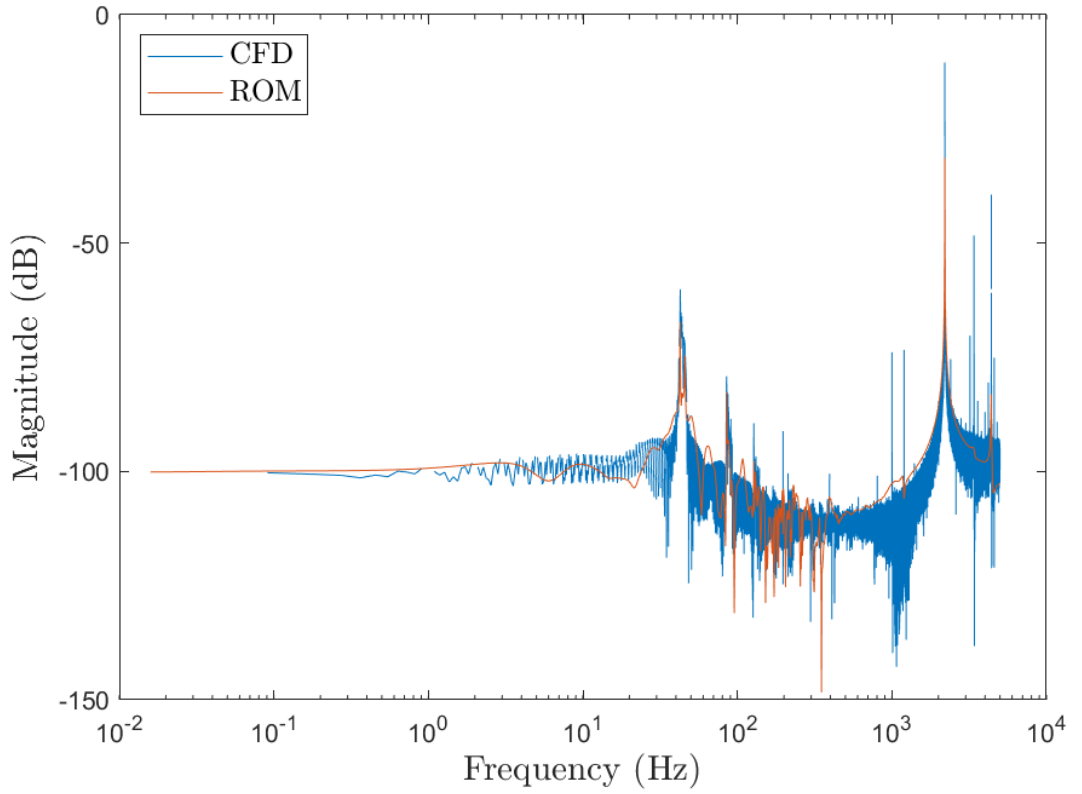


Figure 5.7: *Magnitude (in dB) vs frequency for the squared input model and the CFD system.*

domain data ROM were presented to further demonstrate the similarities between the models. These models were compared to the C_D obtained directly from the CFD simulations. Various versions of each ROM were compared to evaluate the effects of reducing the number of modes of the models. Analysis of the oscillation frequencies present in the pressure field was also compared using the same approach as was used for Figure 5.7.

For each model, the SJAs oscillate with a frequency of 2.2kHz, which can make the information hard to read. In each case, an averaged model is also presented as this represents the information that the model is trying to capture and provides a look at the responses of the system. The averaging process is applied after the models are built, meaning that the model order reduction process is not affected. A simple moving average is used, with a time window of 0.4s. This post process is used multiple times throughout this section as it allows key information to be extracted from the plots.

The input map used for all the models in this section is a monotonically in-

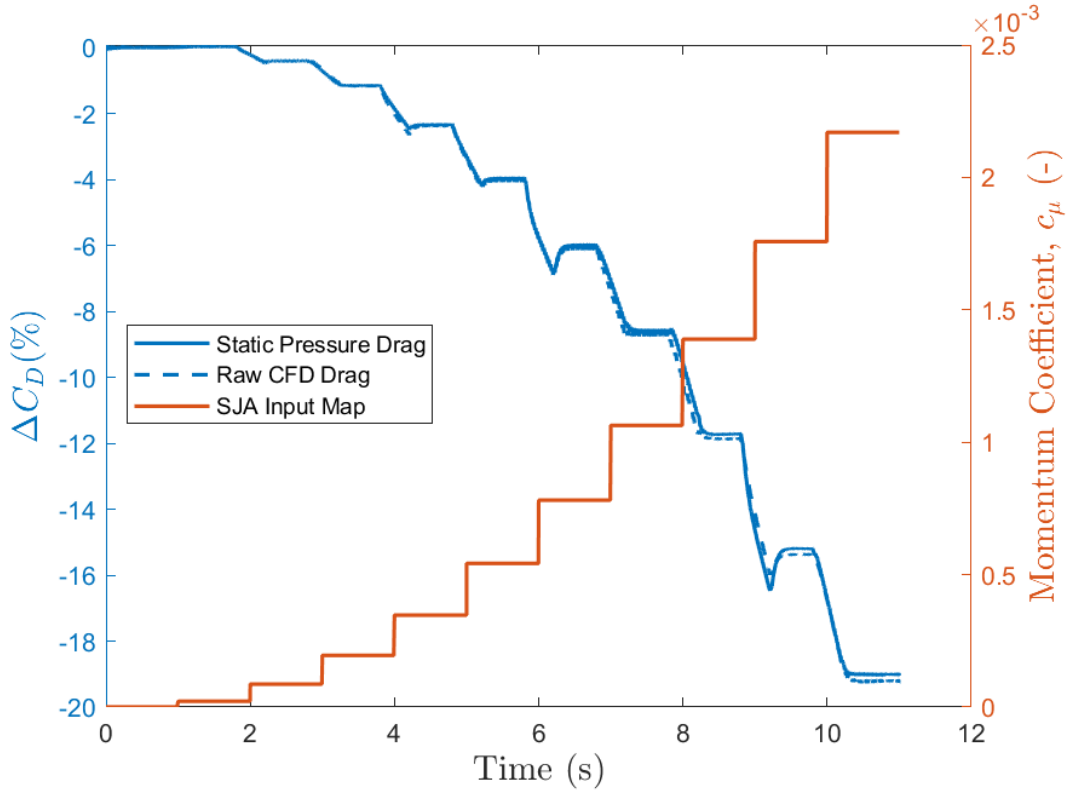


Figure 5.8: C_D obtained from the CFD simulation and computed from the static pressure increase.

creasing vector of SJA velocities, beginning at 0m/s and stepping up by 10m/s at 1 second intervals. This is indicated in each Figure to show how the changes affect the C_D .

Figure 5.9 presents the raw drag coefficient printout for the CFD simulation and both ROMs using 500 modes. The CFD simulation C_D has increasing oscillation amplitudes as the SJA velocity is increased, showing the increasing impact of the SJAs on the flow behaviour of the model. The prediction of the oscillation amplitude by both ROMs is much worse (90% error) than in Figure 5.1 (35% error). This is because the models in this section are built over multiple SJA velocities rather than the single SJA velocity that was used for the model in Figure 5.1. The model in Figure 5.1 already struggled to map the single input to various outputs, whilst the model in Figure 5.9 increases that complexity even further.

Both ROMs in Figure 5.9 experience a decay in the oscillations, whereas only the full domain data simulation experienced the decay in Figure 5.1. The

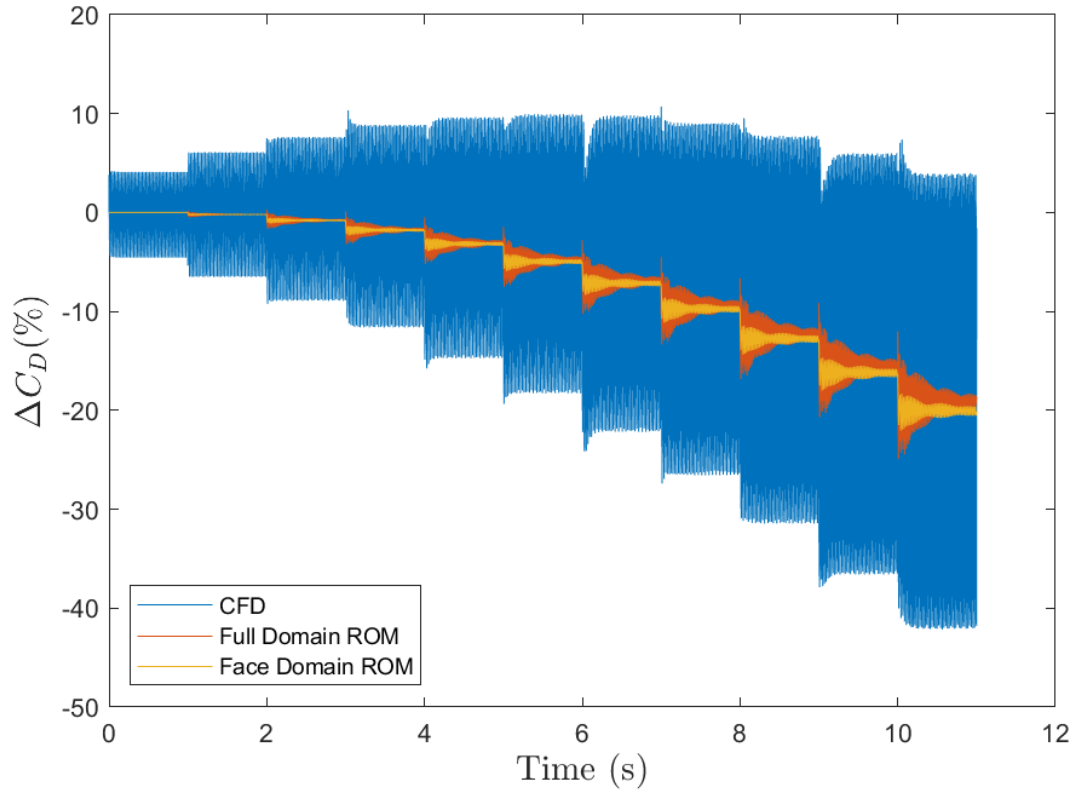


Figure 5.9: The C_D from the CFD and for the full data and face data ROMs with 500 modes.

full domain data also predicts a slightly larger oscillation amplitude when compared to the face domain data ROM. This is due to the extra data in the surrounding flow that the full domain data model is able to capture. Figure 5.10 shows the behaviour of the models over a 1 second timeframe, with the SJA velocity set to 80m/s ($c_\mu = 0.014$). It is clear that the ROM built using only the data from the faces of the model, does not capture each oscillation of the jets, but instead, the oscillations come from the vortex shedding behind the model. Despite that, the model does react to the changes in the SJA velocity as can be seen at the start and end of the timeframe where the SJA velocity is 70 and 90m/s respectively. The ROM built with the full domain dataset does capture each individual oscillation caused by the SJAs.

The oscillations in the pressure field for all cases is presented in Figure 5.11. This is similar to Figure 5.7, but the ROM built from the face domain data is overlaid on top. It is clear that the face domain ROM is also able to capture the oscillations of the shedding frequency, but again the magnitude is lower

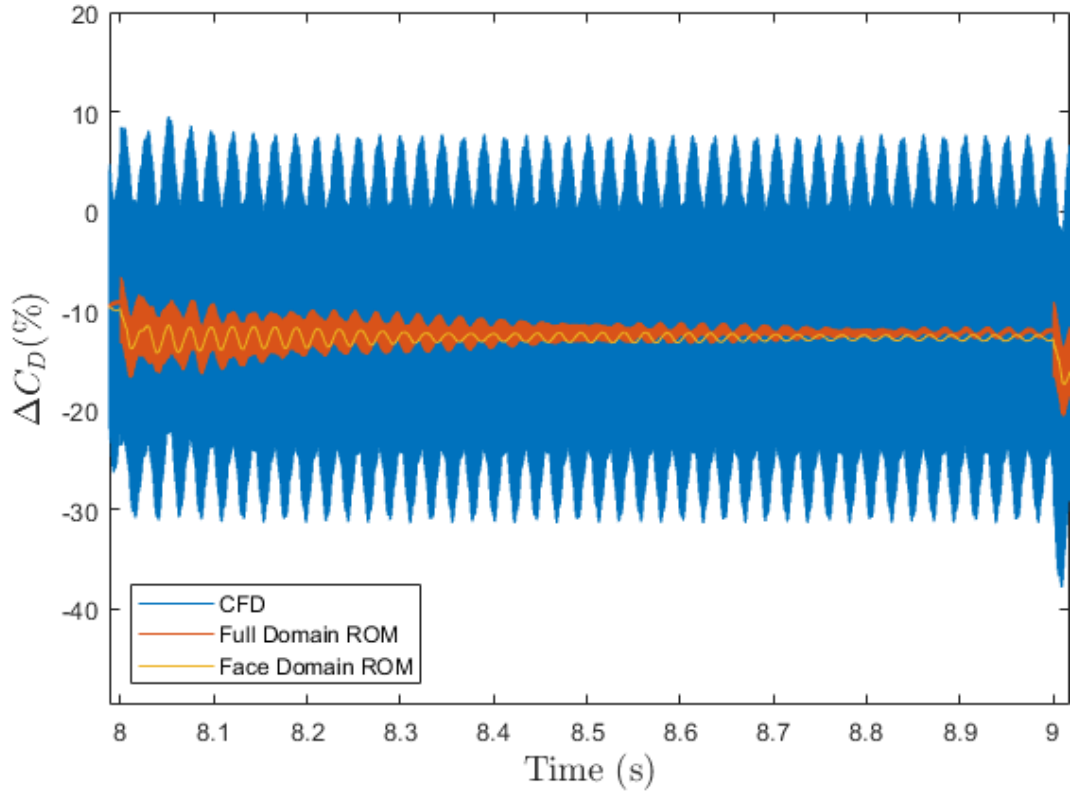


Figure 5.10: A zoomed in figure of the C_D from the CFD and for the full data and face data ROMs with 500 modes.

than for the CFD simulation data. This results in the under prediction in the oscillation amplitudes shown in Figures 5.9 and 5.10. Interestingly, Figure 5.11 suggests that the face domain ROM is also able to capture the oscillations of the SJAs, which is not apparent from Figure 5.10. The face domain ROM also under predicts the magnitude of the low frequency oscillations, providing an explanation as to why the C_D is under predicted.

Both ROMs do not match up with the majority of the peak caused by aliasing, only accounting for the shedding frequency and the SJA actuation frequency. This is likely due to these oscillations having a minor role in determining the C_D value, meaning that the IODMD scheme removes these from the ROM.

The ROMs are able to capture the high frequency oscillations in the system, but are unable to predict the magnitude of the oscillations. This means that the ROMs will be unable to fully capture the instantaneous drag coefficient for the system. However, for the purposes of the control, the most important statistic is the average drag coefficient. When discussing the drag coefficient

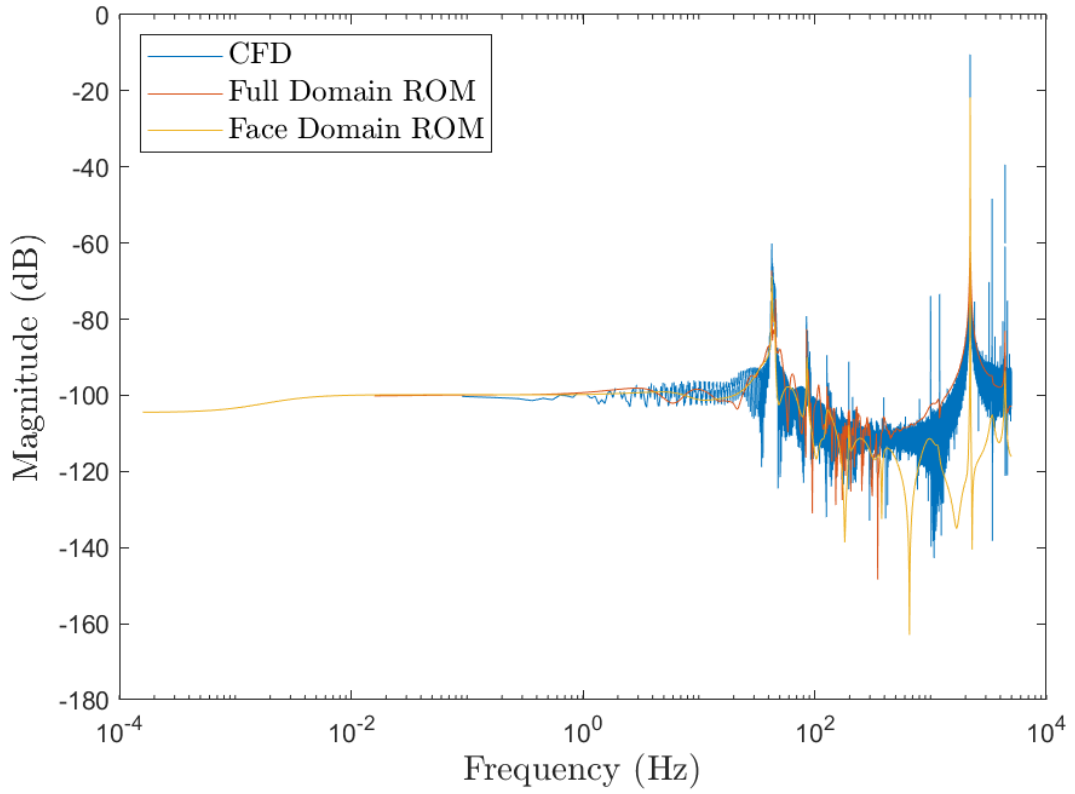


Figure 5.11: *Magnitude (in dB) vs frequency for the 500 mode Full data and Face data models, compared with the CFD system.*

for bluff bodies, the average drag is used as a measure of how aerodynamic the body is, hence, the model just needs to predict this value to be useful for control purposes. From Figure 5.12, it is evident that the models give a reasonable agreement. The full domain model produces a maximum relative error of 40%, whilst the face domain model produces a maximum relative error of 37%. The errors are impacted by the size of the drag reduction. The data in Figure 5.12 is just the average of the data in Figure 5.10. A moving average is taken, which leads to the sloped changes between SJA velocities, rather than the instantaneous changes seen in Figure 5.9.

Both of the ROMs produce similar predictions for the C_D up to SJA velocities of 70m/s ($c_\mu = 0.01$), with a maximum error to the CFD data of $\approx 40\%$ for both ROMs. For SJA velocities above 70m/s, the ROM built with the full domain dataset produces a much better approximation (7% average error) than the ROM built with just the data from the faces of the model (20% average error). The error in the ROM built from the face data remains at a similar level

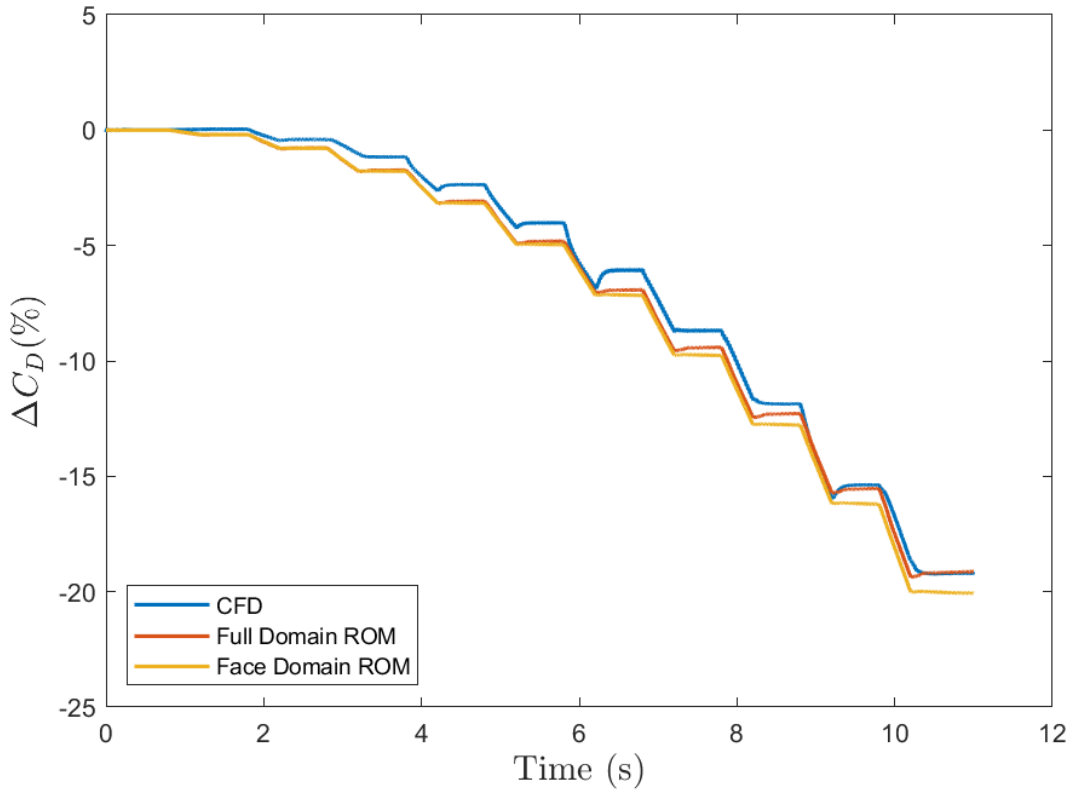


Figure 5.12: The averaged C_D from the CFD and for the full data and face data ROMs with 500 modes.

across all SJA velocities tested, whilst the full domain ROM drops to almost no error for 100m/s. This demonstrates that both ROMs at 500 modes are able to give a reasonable approximation for the average drag coefficient change for the CFD simulation. It was hypothesized that the size of the ROMs could be reduced further, hence the number of modes was reduced to 200, 20 and then 1 mode. These models capture 99.9%, 99.7% and 58.2% of the energy in the full order system. It is important to note that the approximation to the system loses a lot more information as the number of modes is reduced, but the sole focus of the ROM now is to predict the average drag coefficient for each SJA velocity. Figure 5.13 shows the data for the ROMs with 200 modes and the CFD simulation data.

It is clear that the full domain and face domain ROMs now produce a much more similar approximation to each other than for 500 modes. This is because the most important information is located on the faces of the Ahmed body, so as the full domain is restricted further, more of the modes in the full domain

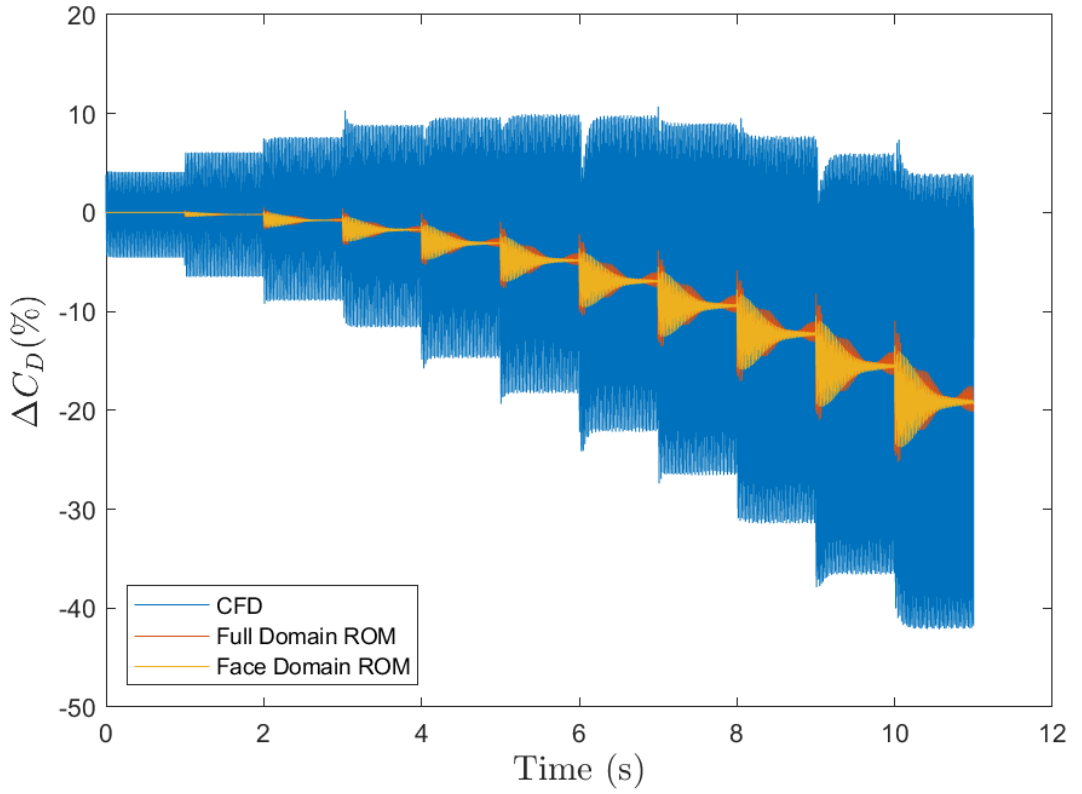


Figure 5.13: The C_D from the CFD and for the full data and face data ROMs with 200 modes.

ROM are focused on the face of the model. Neither ROM produces a good approximation to the oscillation amplitudes although it is interesting that the amplitudes are larger than for 500 modes. The decay is also quicker for the face domain data than for 500 modes. Again the key frequencies are captured by the ROMs as can be seen from Figure 5.14, but there is a large error in the magnitudes of the oscillations. It is interesting to note that the relative changes in the initial amplitudes of the oscillations are consistent for both ROMs and the CFD data, however, this is not important for this research.

Figure 5.14 presents similar results to that of Figure 5.11. The ROMs are able to capture the SJA actuation frequency, the shedding frequency and the low frequency oscillations, but cannot capture the magnitude of the high frequency oscillation. For both models, the error in the shedding frequency magnitude approximation drops to 5% from 14% at 500 modes, which could account for the slight improvement in the approximation of the C_D , seen in Figure 5.15.

Figure 5.15 presents the averaged data from Figure 5.13. A similar picture

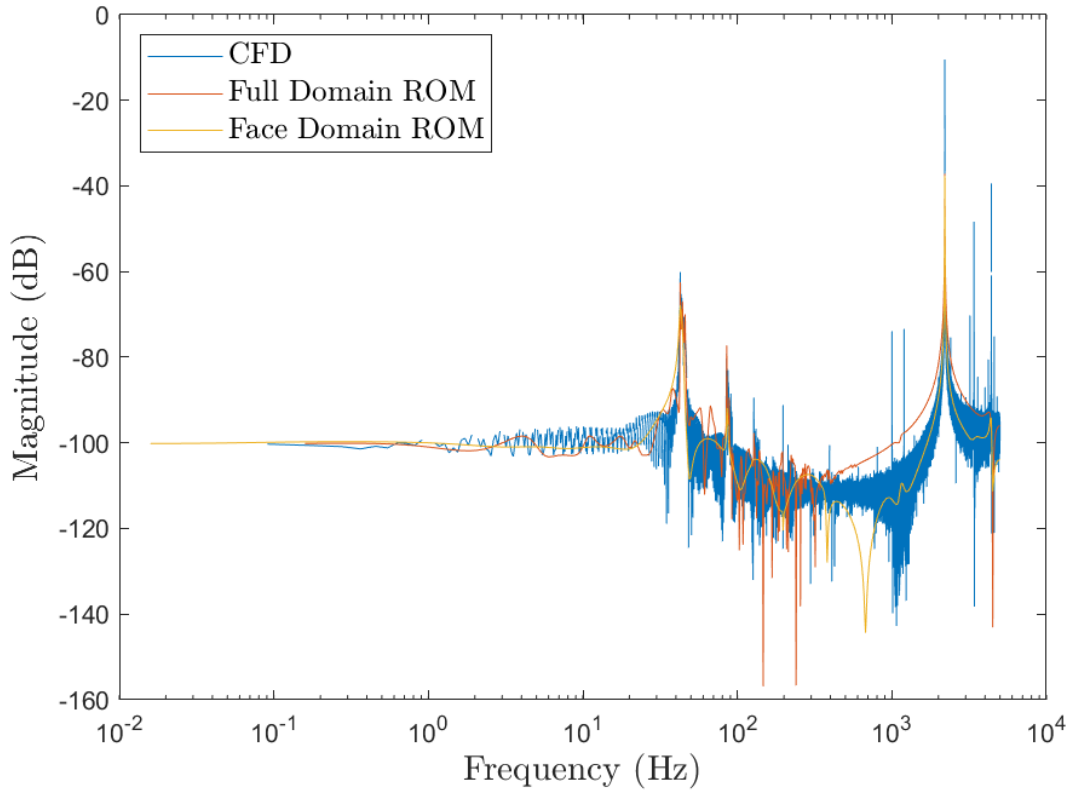


Figure 5.14: *Magnitude (in dB) vs frequency for the 200 mode Full data and Face data models, compared with the CFD system.*

appears here, with both ROMs providing a similar approximation to the system as in Figure 5.12, however, in this case, the face domain ROM is also able to predict the response to the larger SJA velocities with a higher accuracy than for the case with 500 modes. This increased accuracy is likely a coincidence as it would be expected that the ROMs with more modes would be more accurate. One explanation for this could be the removal of the reduced magnitude seen in Figure 5.11 at low frequencies. It could be that the 500 modes included a mode which caused a reduction in the low frequency magnitudes, whilst the less important modes that were not included would have balanced it out in the larger system.

The key result from Figure 5.15 is that both ROMs are still able to capture the average C_D changes caused by the SJA velocities and that a 200 mode ROM is more than sufficient for the purposes of this research. The number of modes was then dropped to 20 to test the accuracy of even smaller ROMs. The raw C_D output has not been presented for this data as it was deemed not

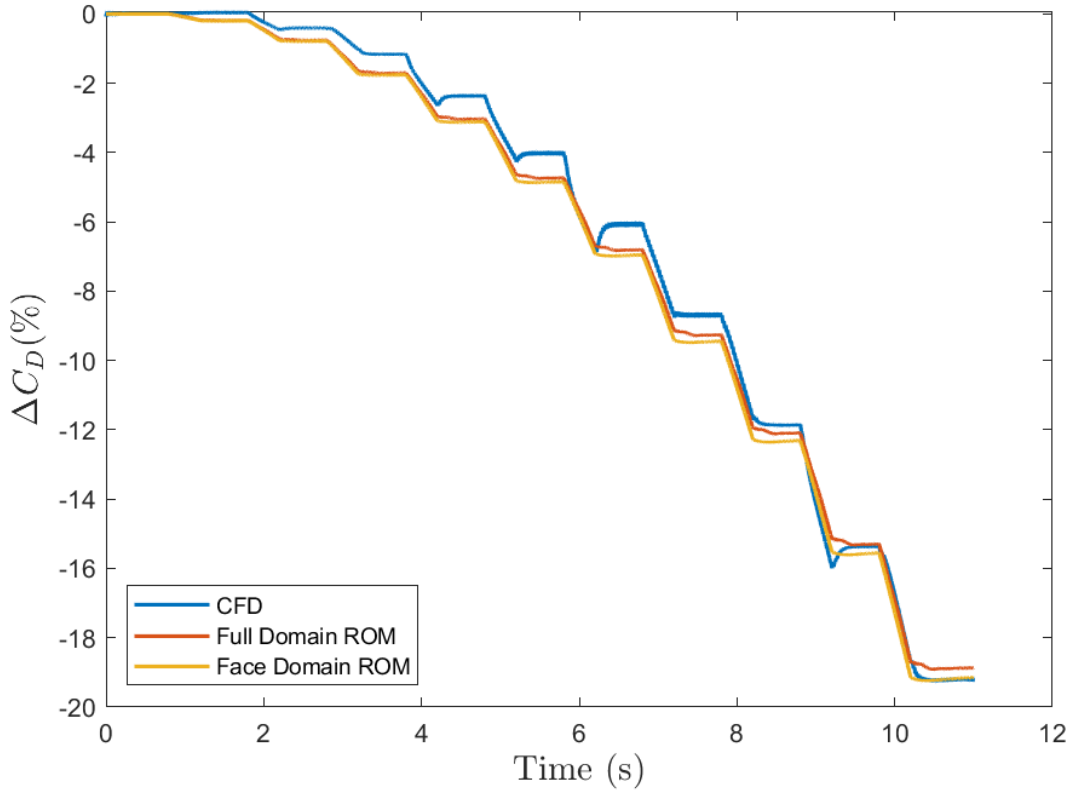


Figure 5.15: The averaged C_D from the CFD and for the full data and face data ROMs with 200 modes.

necessary, with just the frequency data and the averaged data shown instead. The oscillation frequencies have been analysed again, with the results shown in Figure 5.16.

It is clear that more information about the system has been lost from the ROMs now, with fewer frequency peaks being matched. Both ROMs now almost entirely capture the shedding frequency, with the face domain ROM having a 2% smaller error. Combining this with the accurate low frequency approximations, explains the improved approximations of the C_D again. The models still show a small peak at the SJA actuation frequency of 2.2kHz, but do not provide a good approximation to the magnitude at all.

The average C_D data along with the averaged output from the ROMs can be seen in Figure 5.17. Both of the ROM approximations are almost identical for 20 modes, suggesting that almost all of the key information for this system with 20 modes is stored on the face of the Ahmed body. The prediction is again very good with the maximum error between the CFD and either ROM

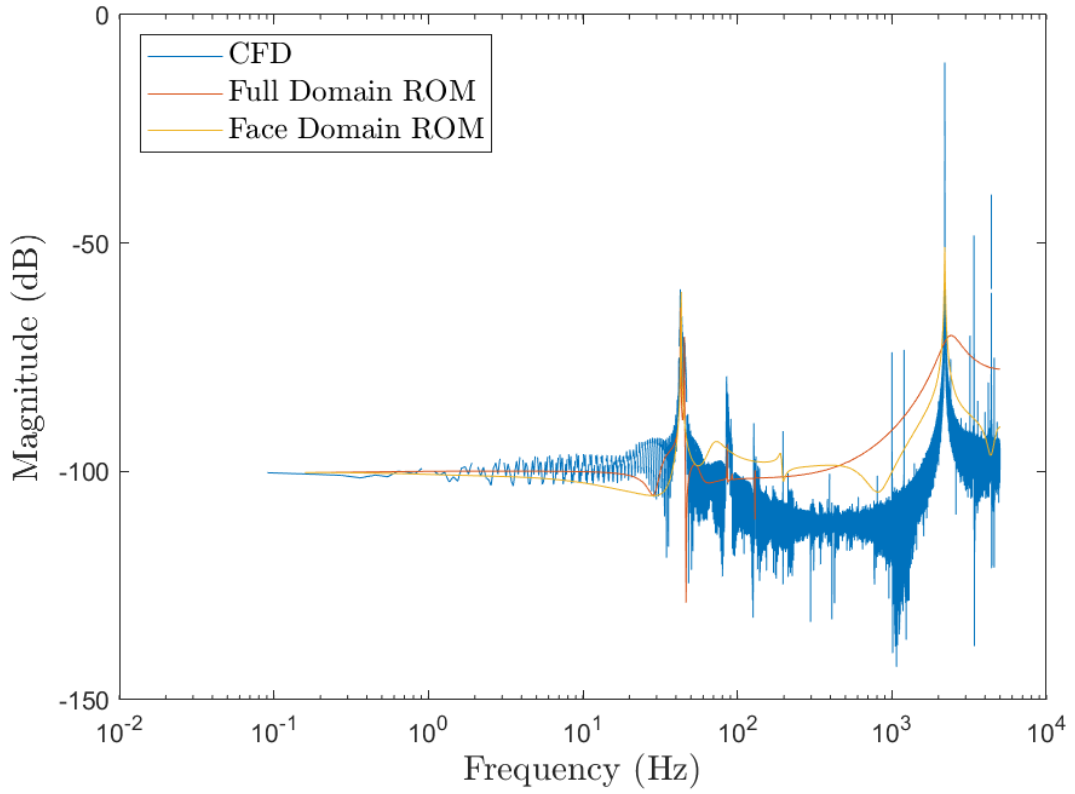


Figure 5.16: *Magnitude (in dB) vs frequency for the 20 mode Full data and Face data models, compared with the CFD system.*

at most 30%. The ROMs have maintained a similar prediction error for all of the cases tested, suggesting that the extra modes added between 20 and 500 are insignificant in predicting the average drag coefficient.

Finally, the number of modes was reduced to 1. The first mode in a reduced order model usually holds information related to the average, so it would make sense that the average C_D can be captured by a single mode. For this model, all information has been removed from the system except for the average C_D value as indicated by the single line for the ROMs on Figure 5.18. There is a slight deviation between the face domain and the full domain ROMs at high frequencies, but the low frequency approximations are exactly the same. This model does not capture the oscillations of the SJAs or the shedding frequency. However, the average C_D approximation is very good, as can be seen in Figure 5.19.

Figure 5.19 shows the simulation results for the 1 mode model. Both ROMs are now identical as the only mode used here contains information from the

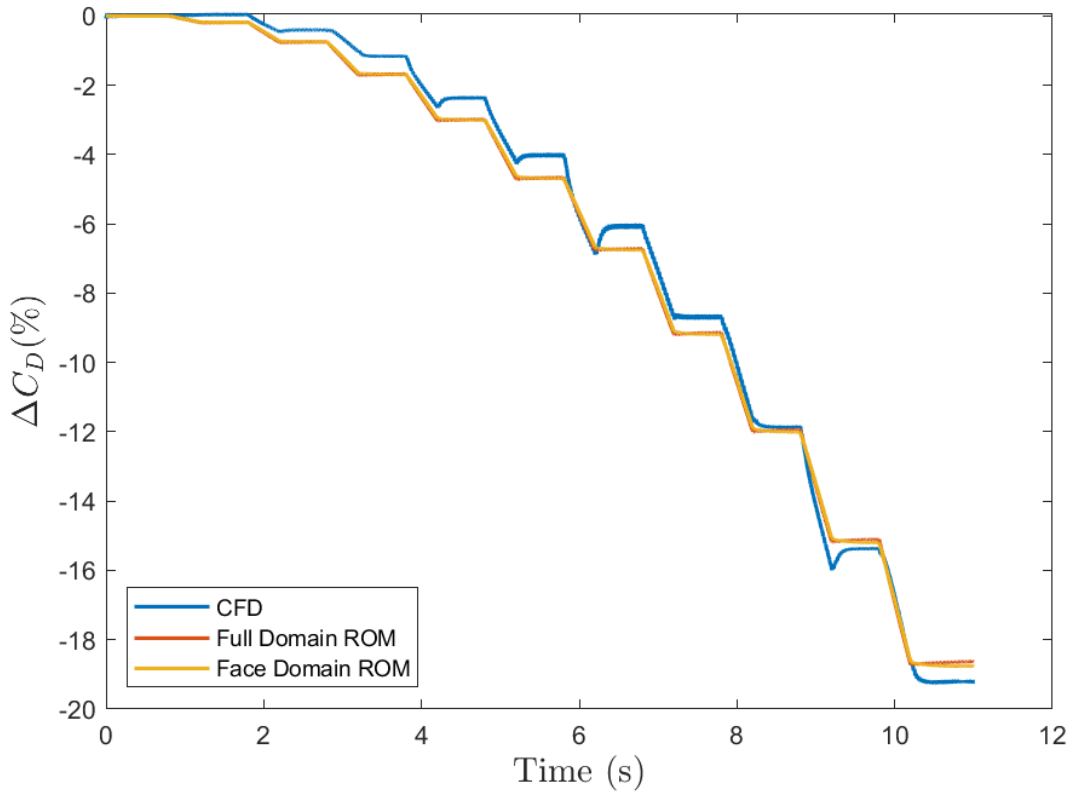


Figure 5.17: The averaged C_D from the CFD and for the full data and face data ROMs with 20 modes.

faces of the model. The approximation by the ROM is also still very good with a maximum error of 30%. This shows that there is almost no difference between the 500 mode model and the 1 mode model for predicting the average drag coefficient despite a lot of information being lost, with the single mode model only accounting for 58.2% of the variance in pressure. It was decided that the 1 mode face domain model provided sufficient accuracy in predicting the average C_D value in this setup.

For the remainder of this chapter, the 1 mode model is used as the ROM due to its accuracy and the large reduction in computation power. It is interesting to note that the model approximations showed very little variance in the average drag value as the number of modes was decreased. This shows that only 1 mode is necessary to provide a good prediction for the model C_D , whilst the added modes only enhance the instantaneous C_D prediction. It would have been a benefit to be able to approximate the instantaneous behaviour of the model, but the reduced computation power for the 1 mode model is of far

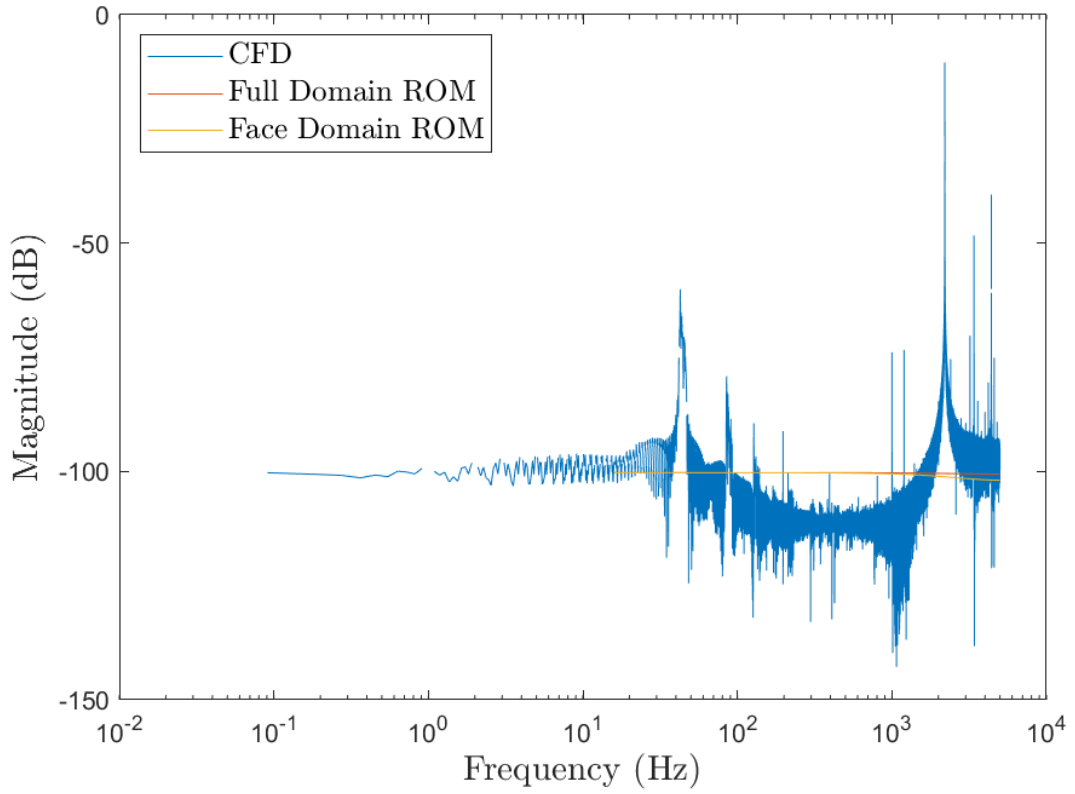


Figure 5.18: *Magnitude (in dB) vs frequency for the 1 mode Full data and Face data models, compared with the CFD system.*

greater benefit to real time flow control.

5.5 Single Mode Approximations of the Drag Coefficient

Since it was decided that the 1 mode model could be used to provide a good approximation to the system, another approach was tested that just used the CFD C_D output to fit the model to, rather than the pressure data. This reduced the complexity in building the models, as less data needed to be saved at each timestep of the CFD simulation. Both approaches provided a similar approximation to the system, Figure 5.20 shows the ROM simulation using the C_D to build the model, whilst the ROM built using 1 pressure mode can be seen in Figure 5.19. The model built using the C_D does have a slightly larger error across the majority of the SJA velocities, however, due to the massively reduced amount of data needed and the reduced computing power to build the

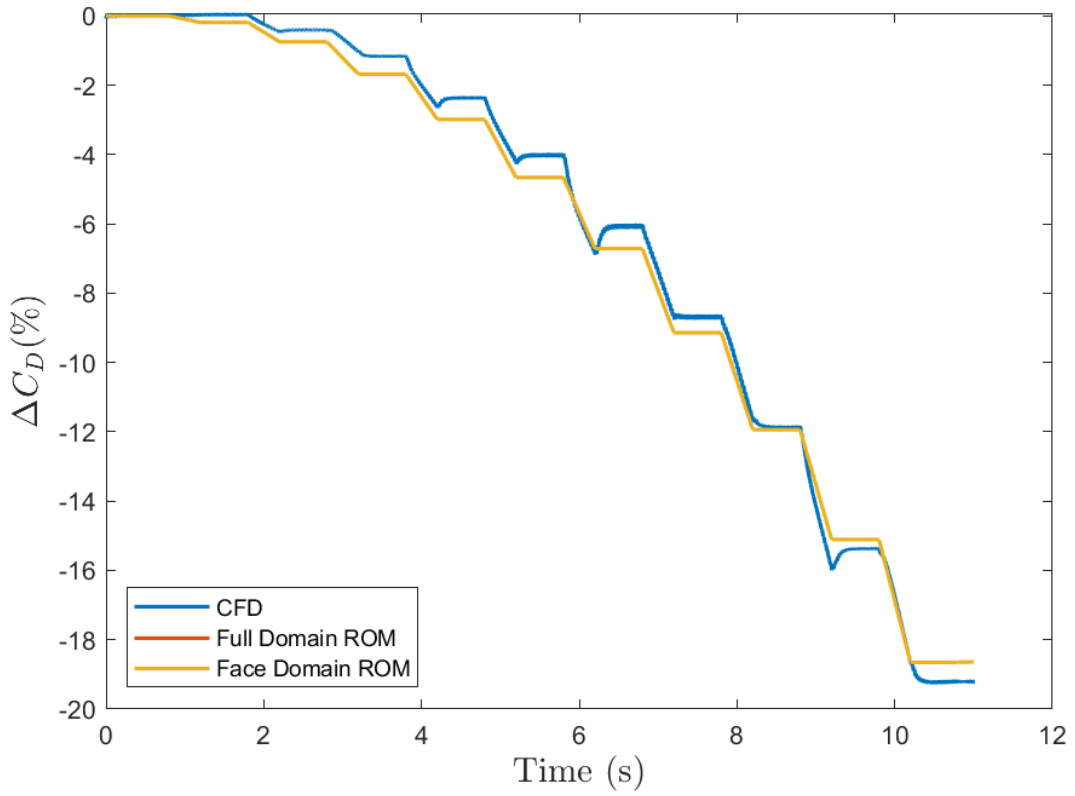


Figure 5.19: The averaged C_D from the CFD and for the full data and face data ROMs with 1 mode.

model, using the C_D is the approach that is focused on for the remainder of the research. The fit for this model can also be improved by using a reduced training set as the SJAs used for this research cannot reach velocities larger than 60m/s. However, it was determined that having a greater training set did not hinder the model too much and provided a large range of SJA inputs over which the model could be tested.

The new approach reduces the data needed as the pressure does not need to be saved over multiple timesteps. Instead, only the C_D from each timestep needs to be saved, drastically reducing the storage space required to build the model. There is also no need for the POD in the method as the C_D is only 1 mode and already contains all of the important information. This reduces the computing resources needed to process the data and build the model, meaning that larger CFD models can be reduced to a 1 mode approximation. This approach made it trivial to compute and build ROMs for the 3D CFD models in Chapter 6.

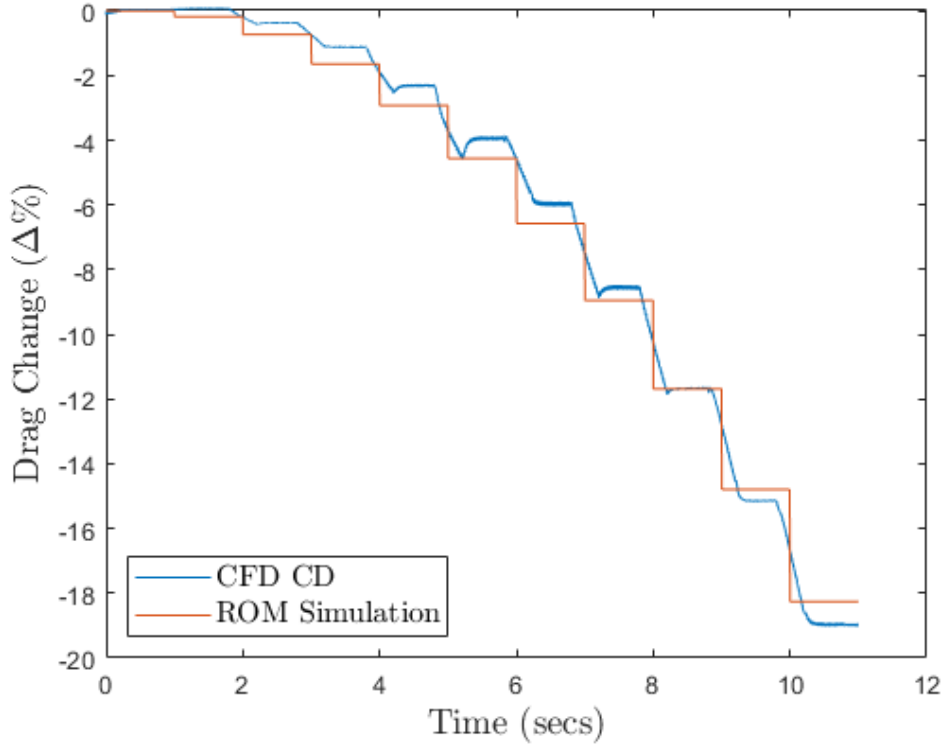


Figure 5.20: The simulation of the 1 mode model built using the C_D , compared to the C_D obtained from the CFD for the 2D 35° Ahmed body.

5.6 Modelling Freestream Effects on the Drag Reduction

The previous section showed that the 1 mode ROM can approximate the response of the C_D under actuation for various SJA velocities effectively. For the control application, the model must be able to react to changes in the freestream flow and adjust the SJA velocity accordingly. An investigation into the effects of the freestream flow on the C_D for various SJA velocities was carried out. The 2D 35° Ahmed body CFD simulation was used, with a fixed SJA velocity of 60m/s or $C_\mu = 0.008$. The freestream velocity was varied from 4.45m/s upto 40m/s, in steps of 4.45m/s. These velocities were chosen as they correspond to 10mph upto 90mph in 10mph increments. The average C_D was recorded at each freestream velocity, Figure 5.21 shows the results.

The relationship between the freestream velocity and the change in drag in Figure 5.21 is exponential. The diminishing returns on the drag reduction as

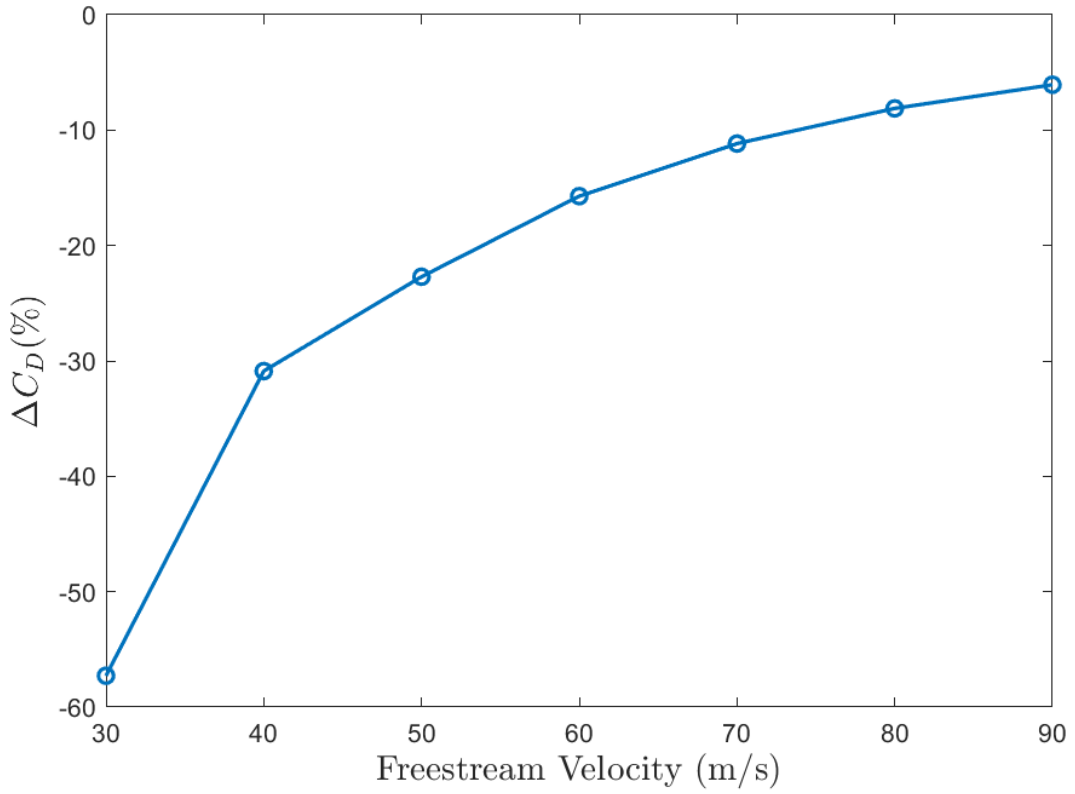


Figure 5.21: C_D change vs freestream velocity with a fixed SJA velocity of 60m/s ($C_\mu = 8 \times 10^{-3}$).

the freestream velocity is increased as to be expected, as the higher freestream velocities will overwhelm the SJAs and reduce their effectiveness. The 4.47 and 8.94mph measurements have been removed as the SJAs are too strong for these freestream velocities and the relationship breaks down. This can be seen in Figure 5.22, which shows a surface of the relationship between SJA velocity and the freestream velocity with the C_D change.

For the low freestream velocities, the large SJA values overwhelm the system and control the flow dynamics. There are various issues with this in a real world setting, such as safety etc, but from a control perspective, the power saving would be smaller than for the smaller SJA velocities anyway as the C_D reduction stagnates. This suggests that there is a limit on the drag reduction that can be achieved with this setup.

The quadratic relationship highlighted in Figure 5.4 can be seen for the freestream velocity at 40m/s. The relationship between the SJA velocity and the C_D change developed throughout this Chapter breaks down below freestream ve-

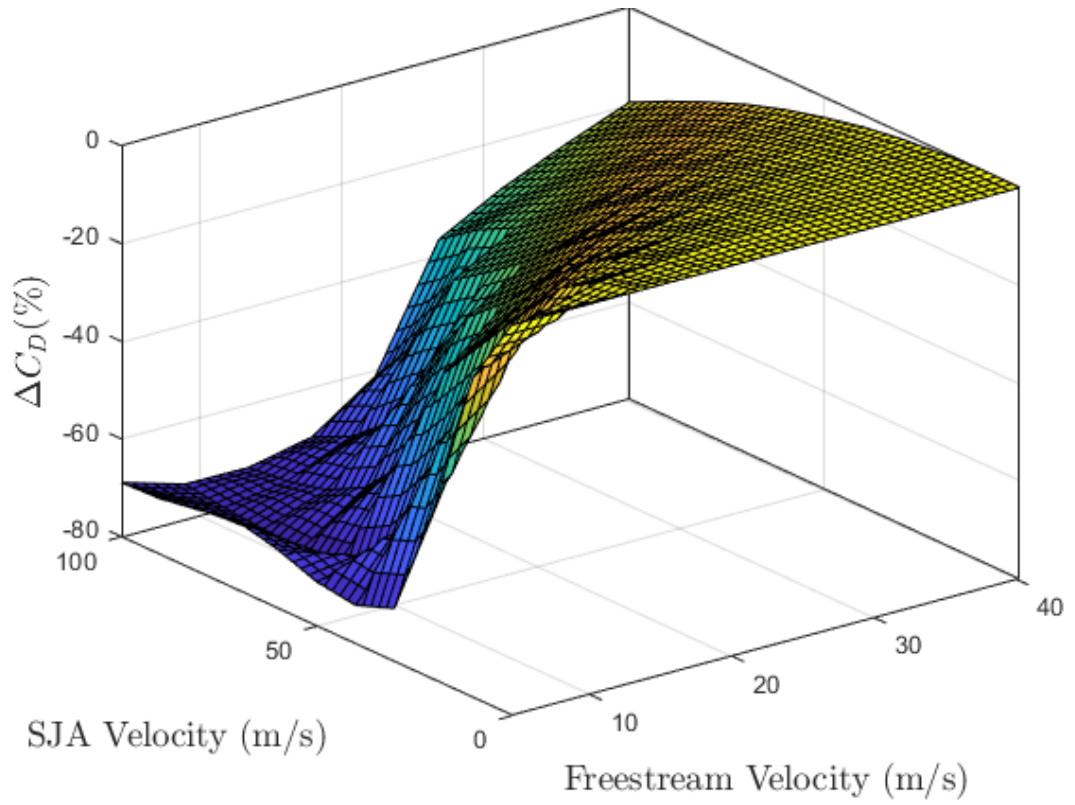


Figure 5.22: Surface of C_D change for various freestream and SJA velocities.

locities of 17.88m/s. This breakdown of the relationship was removed from the ROM as there is no net power saving for these freestream velocities. The same issue does appear for the larger freestream velocities as well, but the SJA velocities have to be so large that the issue is not considered.

The ROM approximation for the C_D change at 17.88m/s can be seen in Figure 5.23. The input map for this simulation can be seen in the Figure. This was the same monotonic increasing input map that was used previously, however, the momentum coefficients are greater due to the reduced freestream velocity. It is clear from Figure 5.23 that at 17.88m/s freestream velocity, the C_D relationship determined previously for SJA velocities above 60m/s ($C_\mu = 8 \times 10^{-3}$) begins to break down. However, the error in these predictions is still small enough to provide a useful prediction for the ESC scheme. It was deemed that this was an acceptable lower bound freestream velocity for the ROM with 40m/s freestream being the upper bound. The optimum operating freestream velocities for this model are between 22.35 and 31.89m/s.

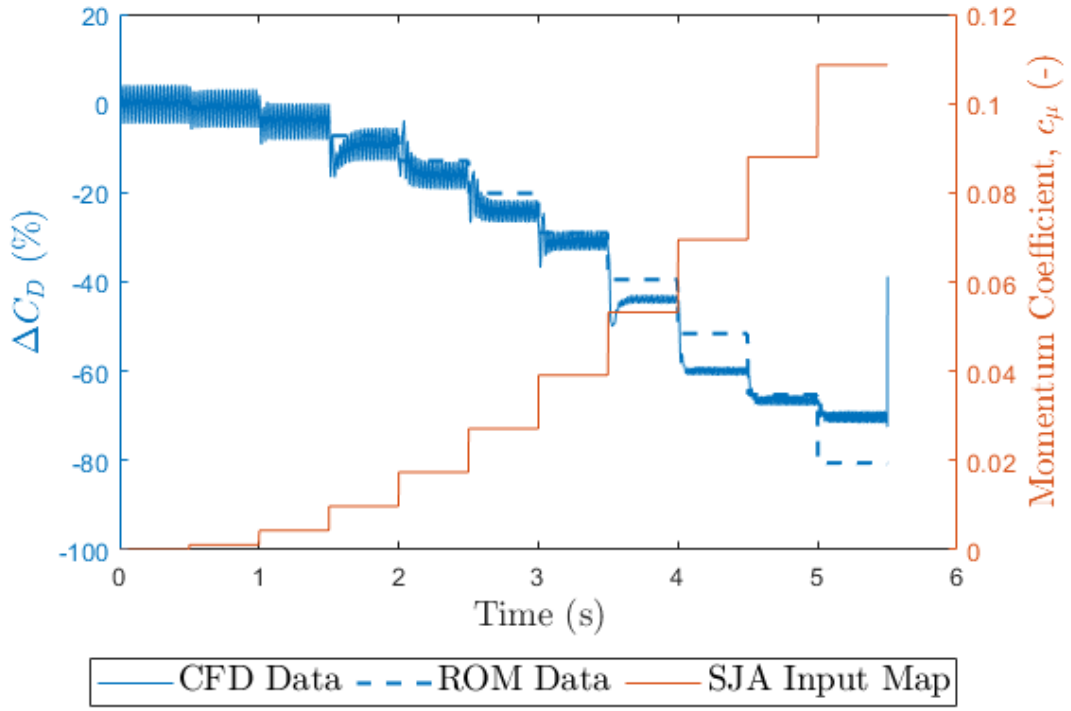


Figure 5.23: C_D vs SJA velocity for 17.88m/s freestream velocity.

At each freestream velocity tested in the operating range of 17.88-40m/s, the relationship between the SJA velocity and the C_D change is slightly different. The overall shape of the relationship is still quadratic, however, the constant k in Equation (5.1) changes. This prevents a single ROM from being able to relate the SJA and freestream velocity to the C_D change. Figure 5.24 shows the attempted fit with a single ROM using 2 inputs for the SJA velocity and the freestream velocity respectively.

To capture this data, a new input map was created that cycled through all of the freestream velocities. At each freestream velocity, the relevant SJA velocities were cycled through. This involved removing the SJA velocities that were too large for the given freestream velocity, for example, at 4.47m/s freestream, all the SJA velocities above 30m/s were removed. The input map can be seen in Figure 5.24. The mean CFD data in Figure 5.24 shows the averaged C_D change for the given SJA velocity and freestream velocity. Each freestream velocity change is easy to see as the C_D change resets to 0. The quadratic

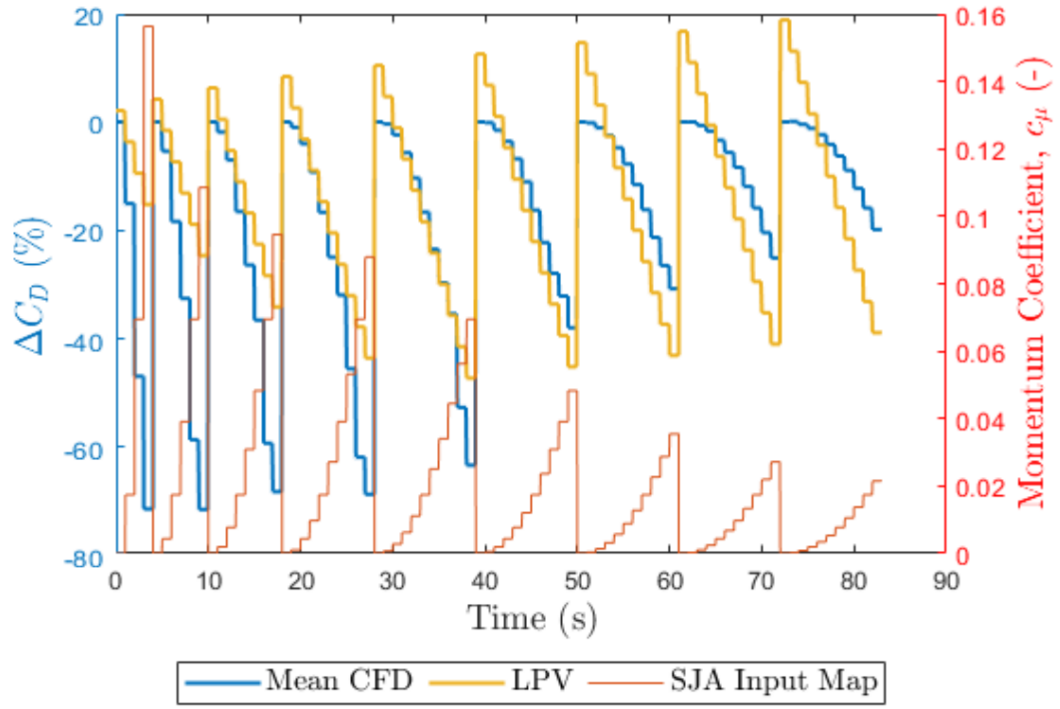


Figure 5.24: C_D change for SJA and Freestream changes for a single state space ROM.

relationship between the SJA velocity and the C_D change is evident at each freestream velocity, whilst it is also evident that the quadratic relationships are all slightly different. It is clear that the single ROM is unable to predict the C_D change for multiple inputs. This meant that a different strategy needed to be employed.

It was hypothesised that, by combining multiple ROMs in a Linear Parameter Varying (LPV) system, that a good approximation to the system could be found for varied SJA velocity input and freestream outputs. Before this LPV system was built, a linear interpolation of the C_D from the 2D CFD simulation was computed to demonstrate that a linear interpolation could be used for the system. Figure 5.25 shows the results of the linear interpolation. As expected, the linear interpolation is able to accurately approximate the C_D data on the domain that it was built.

The linear interpolated model in Figure 5.25 was used as a comparison tool for the LPV system to demonstrate the accuracy of the LPV system. To build the LPV system, a linear interpolation between the collection of ROMs computed

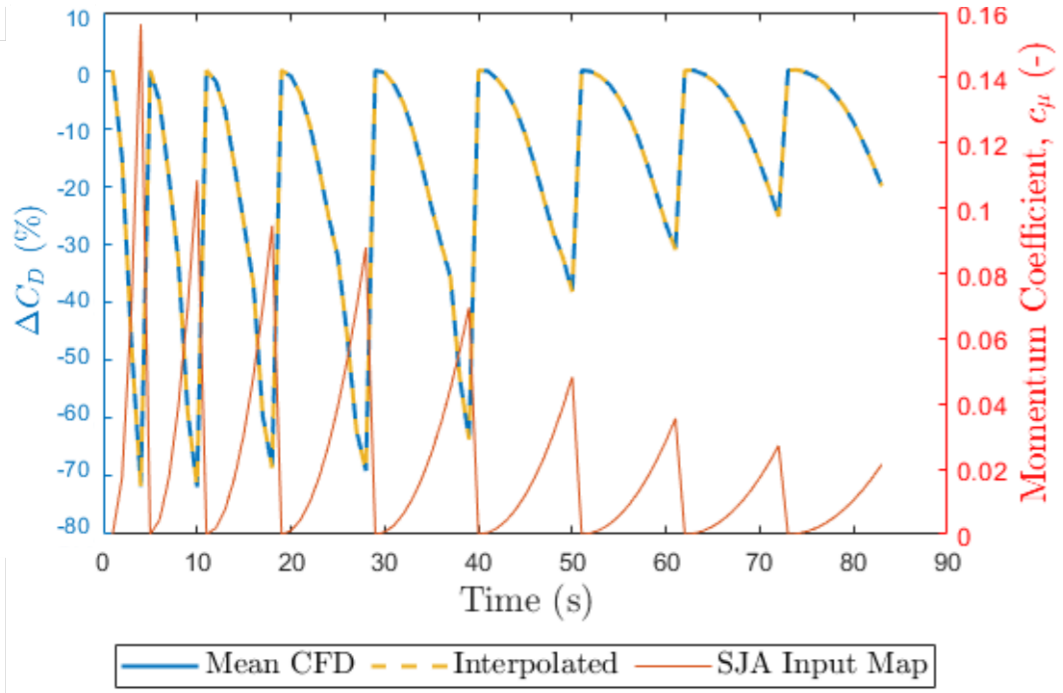


Figure 5.25: C_D change for SJA and Freestream changes with linear interpolated data.

for each freestream velocity was used. It was noted that the differences in the relationship between the SJA velocities and the C_D change was dependent on the freestream velocity and that the ROMs produced at each freestream velocity were Linear Time Invariant (LTI) systems. Therefore, a linear interpolation between the LTI models made sense. The only issue to consider here is that the LTI systems had to be coherent for the linear interpolation to make sense. It has been shown [78, 80–82] that when input-output models are used, the coefficients of the LTI model can be interpolated without issue. Hence, this approach was considered viable and appropriate.

Figure 5.26 shows the changes in C_D for the CFD simulation, the LPV system and the linear interpolation of the CFD data at a freestream velocity of 40m/s. On average over the the full simulation, the linear interpolated fit is more accurate than the LPV approximation, however, all of the models were built and tested on the same dataset. The average error for the interpolated fit is just 2.4%, whilst the error for the LPV system is 32.4%. This error is largely impacted by the lower SJA velocities where the relative error in the prediction

is very large, since the C_D values are so small. By removing the first couple of entries, the error drops to just 20%.

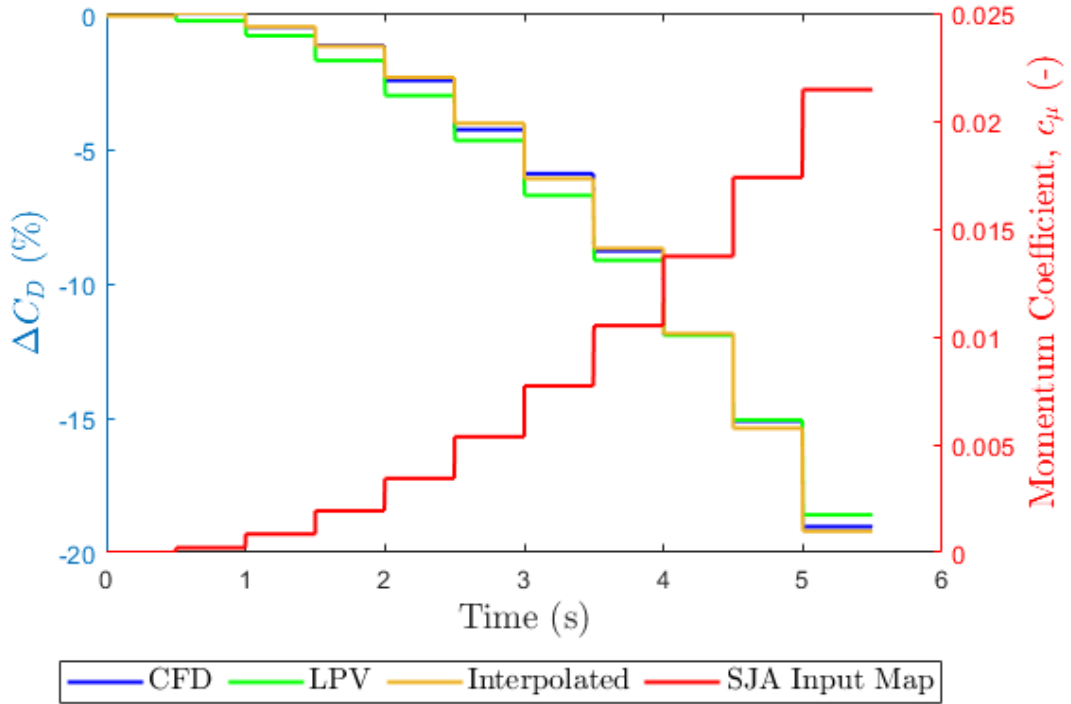


Figure 5.26: C_D change for SJA changes at 40m/s using the linear interpolated ROM compared with the CFD data and the linear interpolated data.

To check the validity of the linear interpolated model, a new input map was created with SJA velocities selected from the full range from 0-100m/s, not just the multiples of 10. This input map was then used with freestream velocities of 20m/s and 28m/s. A new CFD simulation was run for each freestream velocity with the new input map and the average C_D recorded. This was then compared to the LPV system and the linear interpolated data used in Figure 5.26. The results for a freestream velocity of 20m/s are shown in Figure 5.27. Having different training and testing input maps for these models allows the assessment of the accuracy across the whole domain.

It is clear that there are errors in the predictions for the larger SJA velocities, between 2.5 seconds and 3.5 seconds where the SJA velocities are 88 and 91m/s ($c_\mu = 6.8 \times 10^{-2}$ and 7.1×10^{-2}) respectively. This error is apparent in both the LPV system and the interpolated data, but is not as important as the SJAs are

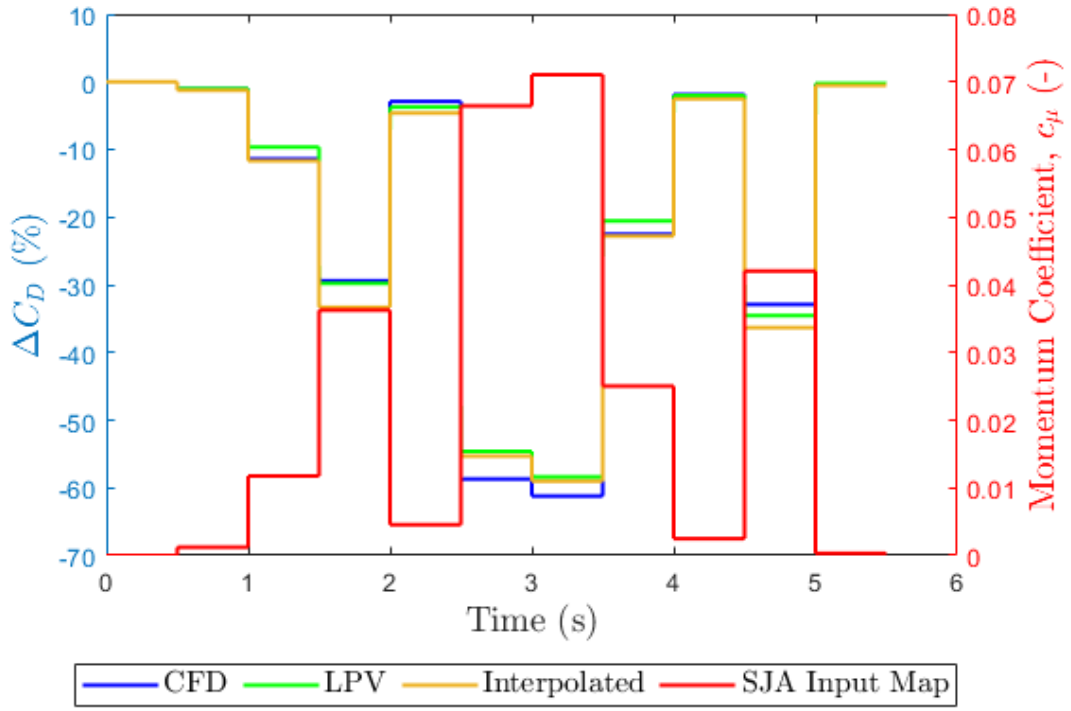


Figure 5.27: C_D change for SJA changes at 20m/s using the linear interpolated ROM compared with the CFD data and the linear interpolated data.

generally operated at lower velocities than this. The more interesting result from Figure 5.27 is that the LPV system gives a better prediction to the CFD C_D data than the interpolated data does for the majority of SJA values up to 80m/s ($c_\mu = 5.5 \times 10^{-2}$). The average error across all SJA velocities for the LPV system is 2.7%, whilst the average error for the interpolated data is 6.4%. The results for the model at a freestream velocity of 28m/s are similar, with the LPV system able to capture the C_D change for the lower SJA velocities reasonably well, but with larger errors introduced for the larger SJA velocities. This can be seen in Figure 5.28. The interpolated data provides the lowest error across all SJA velocities with an average error of 9.2% to the 11.5% for the LPV system. The LPV system has the best agreement with the CFD C_D data for SJA velocities between 40 and 70 m/s ($c_\mu = 6 \times 10^{-3}$ and 2.2×10^{-2}) with an error of 3.1% compared to 6.0% for the interpolated data. The advantage of the LPV system over the interpolated data is that the LPV system is a state space representation of the model and so can easily be simulated as well as

compared to other state space models.

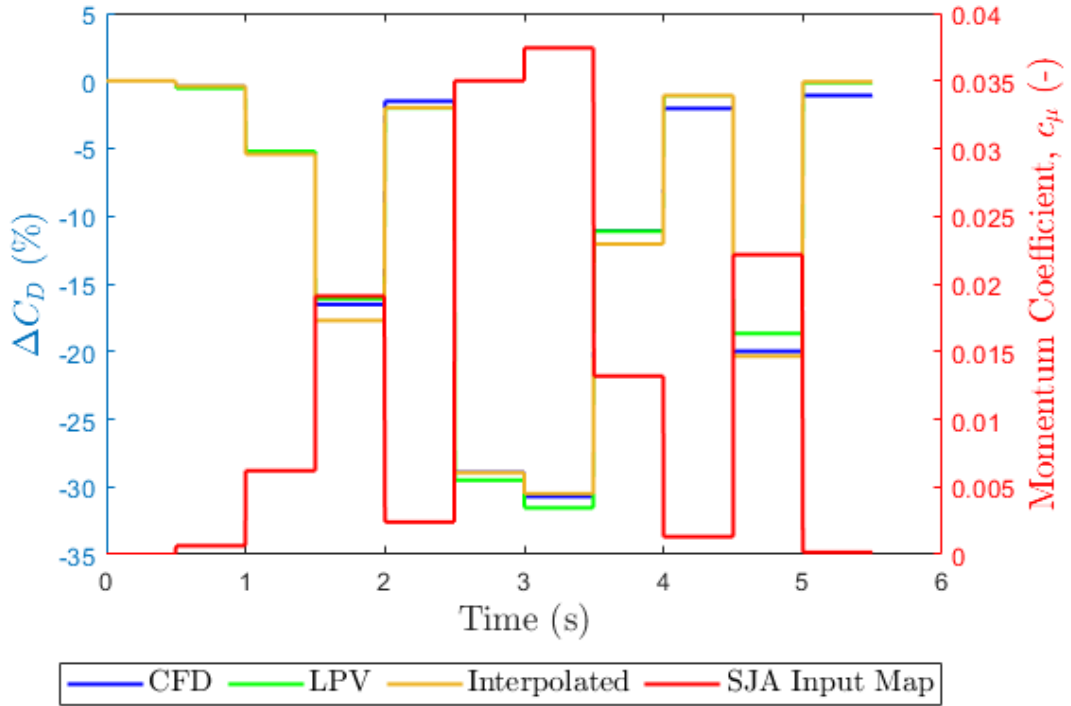


Figure 5.28: C_D change for SJA changes at 28m/s using the linear interpolated ROM compared with the CFD data and the linear interpolated data.

5.7 Net Power Savings for the Reduced Order Model

Table 5.1 shows the net power savings for the 2D 35° Ahmed body as predicted by the ROM. The results in this table can be compared to Table 4.1 to further demonstrate the accuracy of the ROMs. The general trends are the same, with the SJAs proving ineffective at low freestream velocities, with the greatest net power saving provided between 17.88m/s and 22.35m/s freestream velocities. The optimum momentum coefficient also experiences a downward trend explaining the decrease in the net power saving.

Figures 5.29 and 5.30 show the net power savings as produced by the CFD simulations alongside the power savings predicted by the ROM.

Table 5.1: A table showing the percentage net power saving predicted by the ROM for various freestream velocities for the 2D 35° Ahmed body.

Freestream Velocity (m/s)	Momentum Coefficient, $C_\mu \times 10^{-3}$ (-)	Net Power Saving (%)
4.47	4.4	0.52
8.94	4.9	2.23
13.41	4.4	4.24
17.88	4.0	5.61
22.35	3.3	5.44
26.82	2.6	3.91
31.29	2.2	3.52
35.76	1.8	3.02
40.23	1.5	2.49

Figure 5.29 shows the net power saving produced by the CFD simulation and predicted by the ROM at a freestream velocity of 22.35m/s. The ROM is able to accurately predict the net power saving for the model and also predict the optimum SJA momentum coefficient used to achieve this power saving, suggesting that the ROM should be a good fit for initialising the ESC scheme. There is a large drop in the power saving after $C_\mu = 5 \times 10^{-3}$, as the exponential power draw for the SJAs overtakes the power saved by reducing the drag around the model.

Figure 5.30 shows the net power saving produced by the CFD simulation and predicted by the ROM for a freestream velocity of 31.29m/s. Again the ROM gives a good prediction for the power saving achieved by the CFD simulation. The optimum SJA momentum coefficient is also predicted with good accuracy. The net power saving for this model has a small sweet spot in which the SJAs can produce a net power saving, therefore, it is imperative that any controller is able to accurately predict the optimum and quickly converge to this optimum to prevent large losses in performance.

5.8 Extremum Seeking Control

5.8.1 Introduction

This section explores a basic ESC scheme that aims to control the flow around an Ahmed body with Synthetic Jet Actuators (SJA) located at the top of the

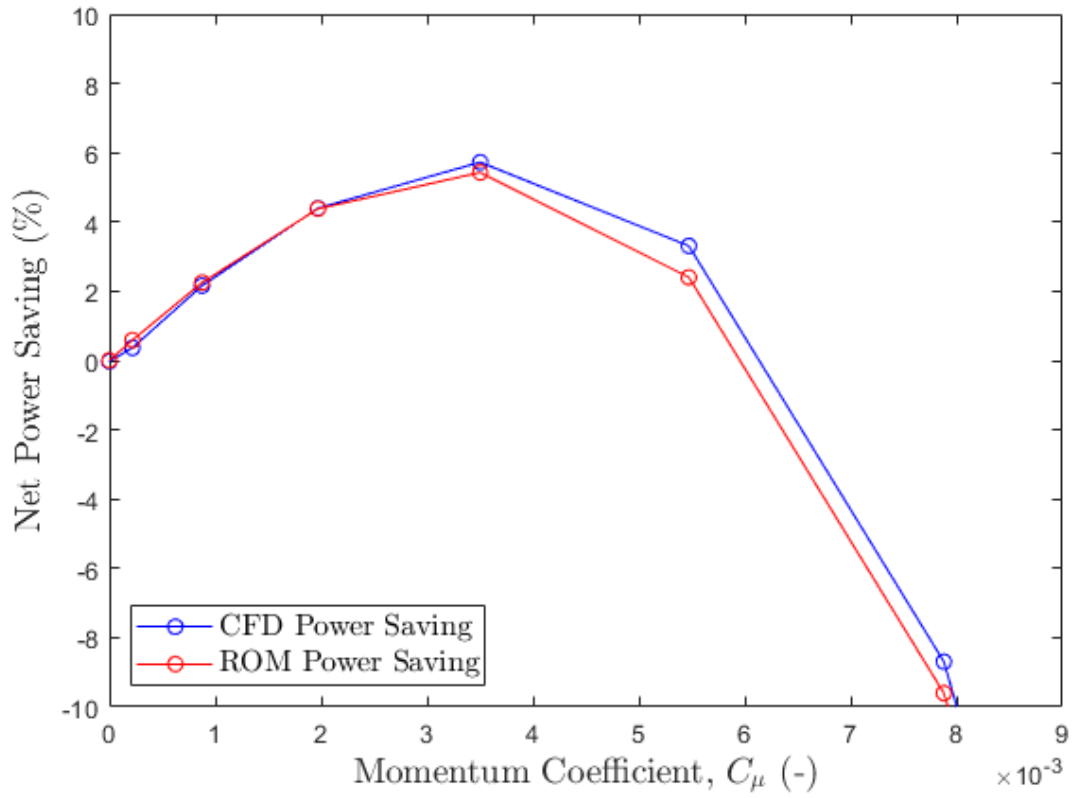


Figure 5.29: A plot showing the net power saving for the CFD model and the ROM with a freestream velocity of 22.35m/s.

rear slant. The control scheme uses the net power saved during actuation to determine the optimal input voltage, and therefore velocity, of the SJAs. A more advanced control scheme is then applied to the same model using the Reduced Order Model (ROM) developed in the previous section to provide a knowledge of the model dynamics and improve the performance of the control scheme. The new control scheme has been dubbed the ROMESC scheme. This new scheme uses an integrator reset when a change in the freestream is detected, along with an optimum SJA voltage approximation to improve the ESC scheme's performance for real time control. The optimum approximation by the ROM is not perfect, so a small tolerance is used in the freestream change detection, which allows the ESC scheme to optimise the power saving through small changes, whilst the ROM handles the abrupt changes in the flow. The ESC is also able to react to other changes in the flow and provide closed loop control as a standard ESC scheme would.

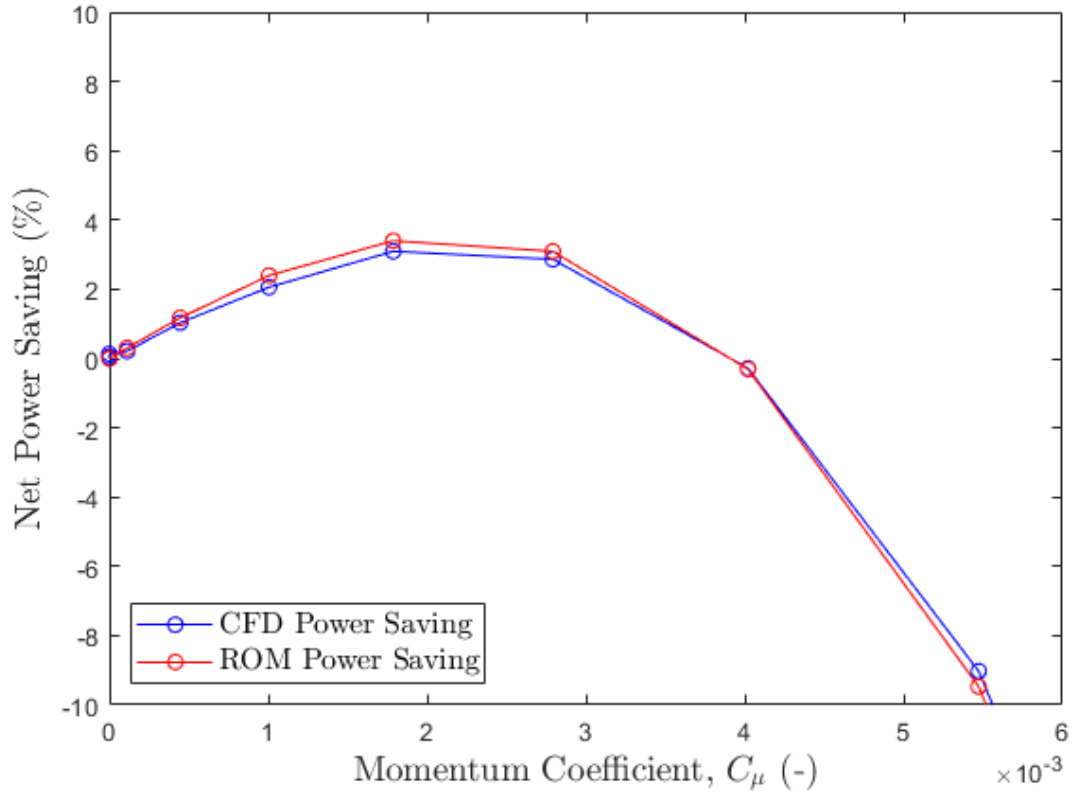


Figure 5.30: A plot showing the net power saving for the CFD model and the ROM with a freestream velocity of 31.29m/s.

5.8.2 The Basic Perturbation Based Extremum Seeking Control Scheme

The basic ESC model was applied to the 2D 35° model to create a baseline to compare the performance of the ROM model to. The net power savings for this model were shown in Table 5.1 and are used here as the objective function for the ESC scheme. Two simple tests were created for the basic ESC scheme and the ROMESC scheme, to highlight the key areas of the simulation. The first test uses a constant freestream value of 22.35m/s with an initial estimation of $c_\mu = 0$, to see how long it takes for the model to converge to the optimum SJA voltage. The second test uses a varied freestream velocity to test the responsiveness of the two models. The freestream velocity begins at 22.35m/s, then after 15 seconds, a step change to 31.29m/s is used, before finally reducing to 17.88m/s at 23 seconds. These freestream changes have been included in the Figures to allow comparison with the ESC response. In each case, the tuning

parameters are kept constant for the ESC and ROMESC models to show the performance differences.

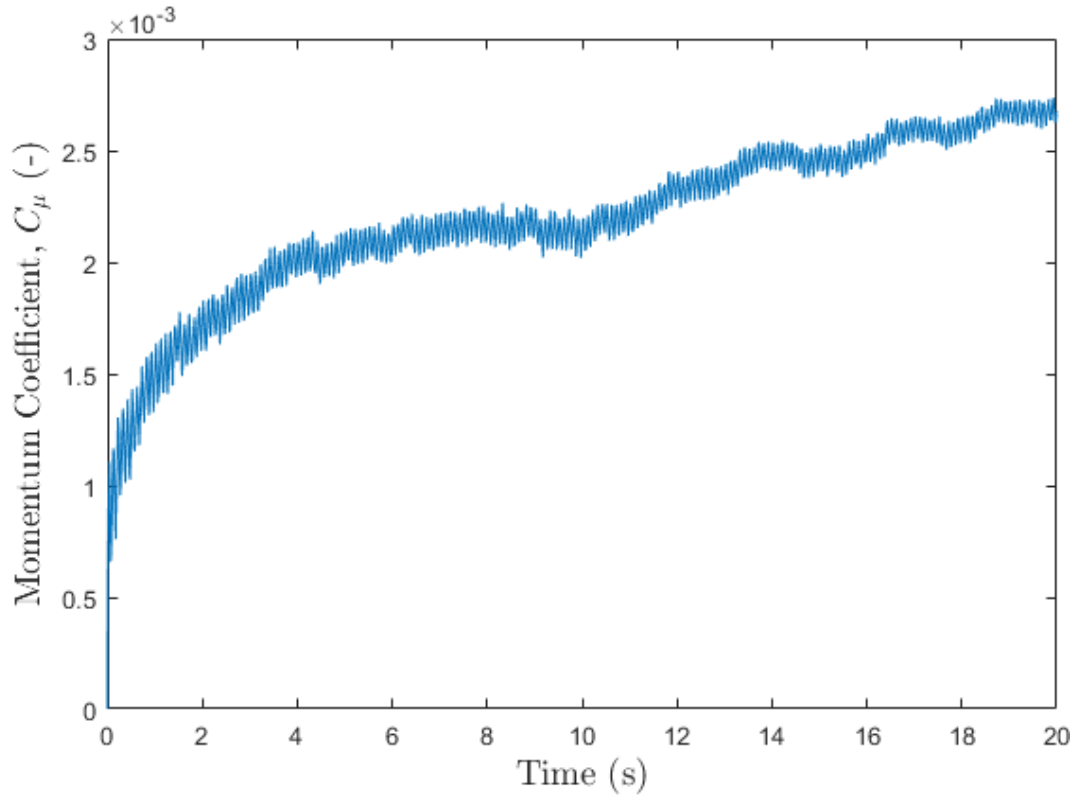


Figure 5.31: *The convergence of the momentum coefficient for the basic ESC scheme using zero c_μ initial condition for the SJAs.*

Figures 5.31 and 5.32 show the basic ESC scheme operating with $c_\mu = 0$ initially. The scheme is using a small amplitude perturbation with a high frequency, which improves the stability of the model, whilst reducing the convergence time of the scheme. The perturbation frequency of 10Hz also avoids interfering with any of the frequencies already present in the numerical model. The gain in the system was also set high, to again increase the speed of convergence of the model. It is clear from Figure 5.31 that even with the scheme tuned to prioritise convergence speed, the scheme takes a long time to converge to the expected optimum seen in Table 5.1. After 20 seconds of simulation time, the scheme still has not converged to the optimum of 3.3×10^{-3} , showing that the scheme is not very effective for real time control. This is also reflected in Figure 5.32 which shows the percentage power saved by the scheme over its runtime.

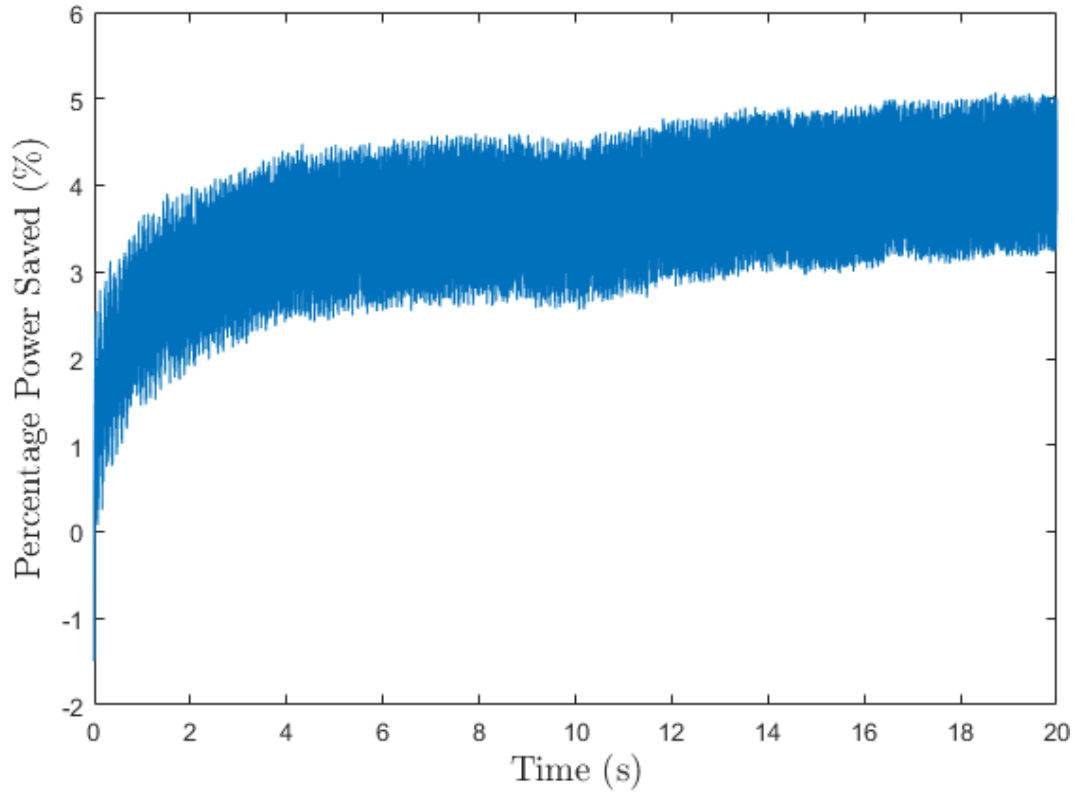


Figure 5.32: *Percentage power saved for the basic ESC scheme using zero c_μ initial condition for the SJAs.*

Figure 5.32 shows that for the full 20 second simulation time, the percentage power saved is below the optimum of 5.44% as shown in Table 5.1 and does not converge within the 20 seconds. In fact, over the 20 seconds, the scheme has saved 30% less power than if the SJAs operated at the optimum for the same length of time.

Part of the reason for the slow convergence is due to the noise present in the system. This is because the sampling rate of the ESC scheme is smaller than the frequency of timesteps in the simulation. This combined with a small variation in the drag coefficient experienced by the CFD simulation produces noise in the C_D output which affects the performance of the scheme. The ESC scheme uses high and low pass filters to attempt to reduce the impact of this noise on the system, however, it cannot be eliminated completely without losing information about the system. Other methods to combat the noise were tested, including a method of averaging the C_D over a number of timesteps before being passed to the ESC scheme, however, the ESC scheme had faster

convergence without these extra methods. The slow convergence of the basic scheme from a zero c_μ initial condition, meant that for the next test, an initial c_μ of 2×10^{-3} was used, to reduce the initial convergence time of the model. Figures 5.33 to 5.36 show the comparison in the simulation results between the basic ESC model and the ROMESC model. The freestream velocity is varied to show the responsiveness of the models. The optimum convergence can be seen in Table 5.1.

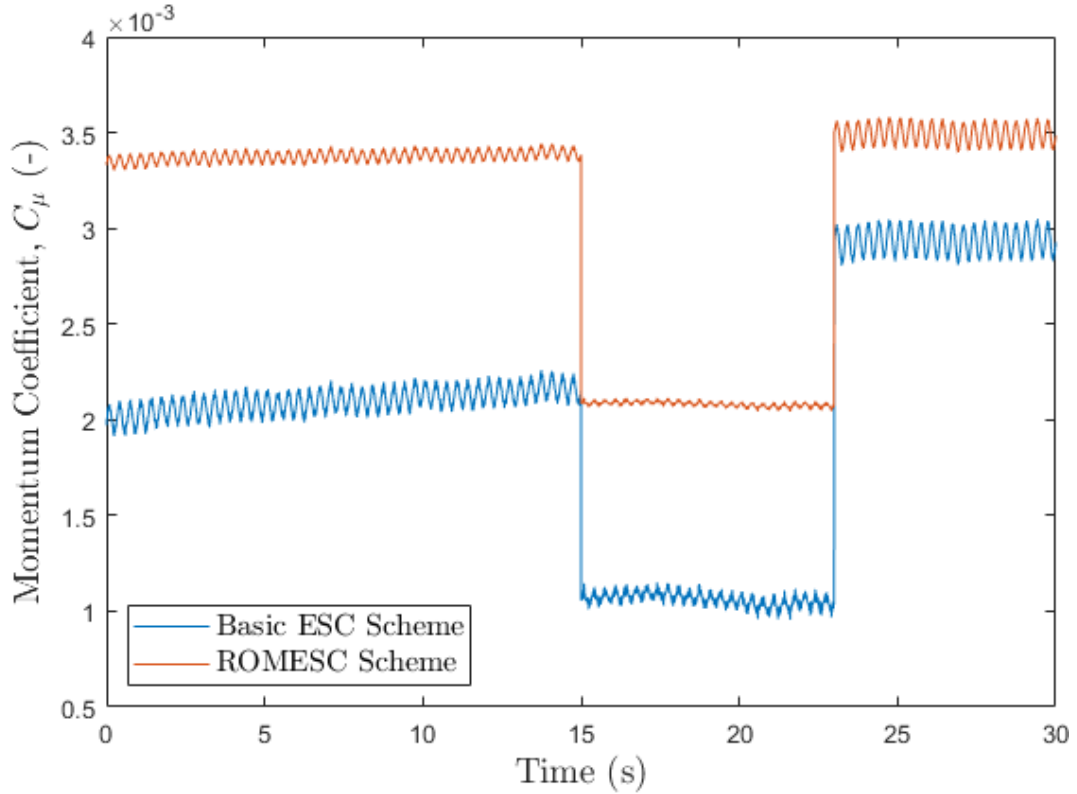


Figure 5.33: Convergence of the basic and ROM ESC schemes to an optimum SJA momentum coefficient for changing freestream conditions.

Figure 5.33 shows the SJA momentum coefficient as determined by both ESC schemes. From Table 5.1, the optimum momentum coefficient for a freestream velocity of 22.35m/s is 3.3×10^{-3} , which is clearly achieved by the ROMESC for the majority of the first 15 seconds of the simulation, whilst the basic ESC scheme never even gets close, even with the improved initial condition. The increase in the momentum coefficient for the basic ESC scheme has also slowed and is barely increasing. This is probably due to the small gradient of the net power saved function as the optimum is approached from below, however, this

just shows the increased suitability of the ROMESC to the conditions. The ROMESC is also able to deal with the changes to freestream conditions better than the basic ESC scheme. At 15 seconds, the freestream velocity is increased to 31.29m/s. Both schemes experience a drop in the momentum coefficient due to the change in freestream velocity. This is because the momentum coefficient is based on freestream velocity. From Figure 5.34, it is clear that the basic ESC scheme does not adapt to this change in freestream at all, whereas, the ROMESC scheme does. The optimum momentum coefficient for a freestream of 31.29m/s from Table 5.1 is 2.2×10^{-3} . It is clear that the ROMESC reaches this instantly and maintains this level for the next 8 seconds. However, the basic ESC scheme barely changes the voltage used as the input for the SJAs.

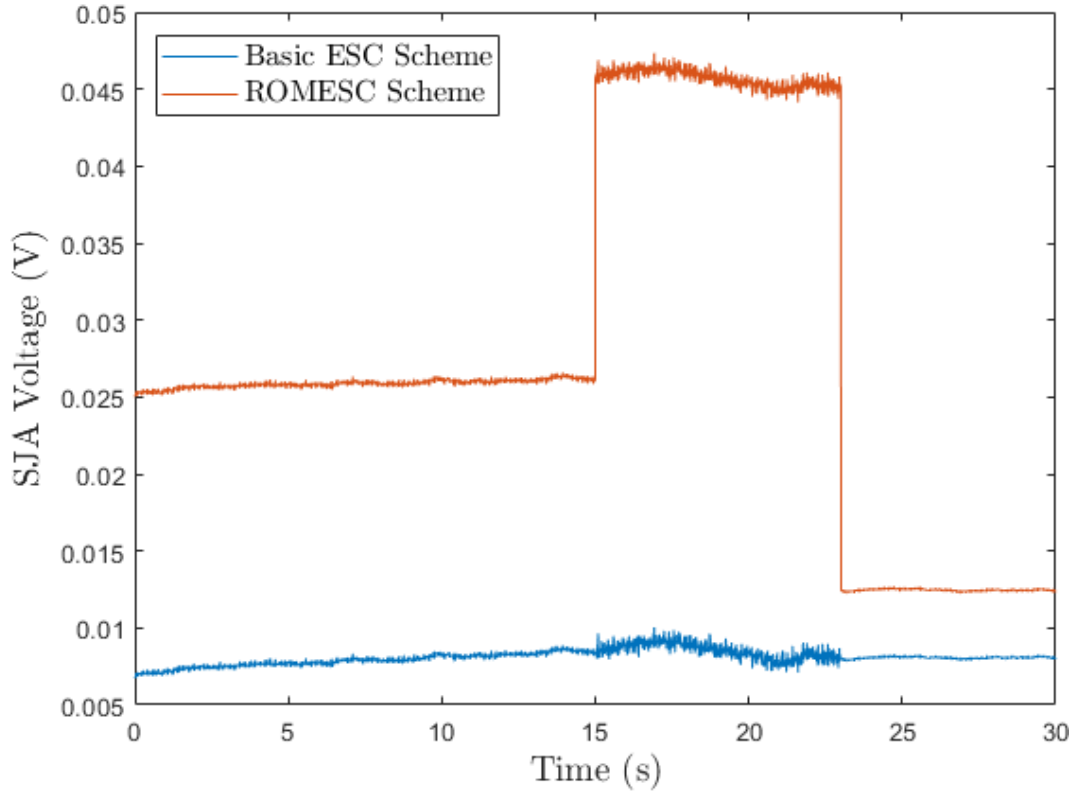


Figure 5.34: Convergence of the basic and ROM ESC schemes to an optimum SJA voltage for changing freestream conditions.

At 23 seconds, the freestream velocity is dropped to 17.88m/s and again, the ROMESC is able to adapt instantly, while the basic scheme barely responds to the change. The effects of this can be seen in Figure 5.35, which shows the change in the net power saving.

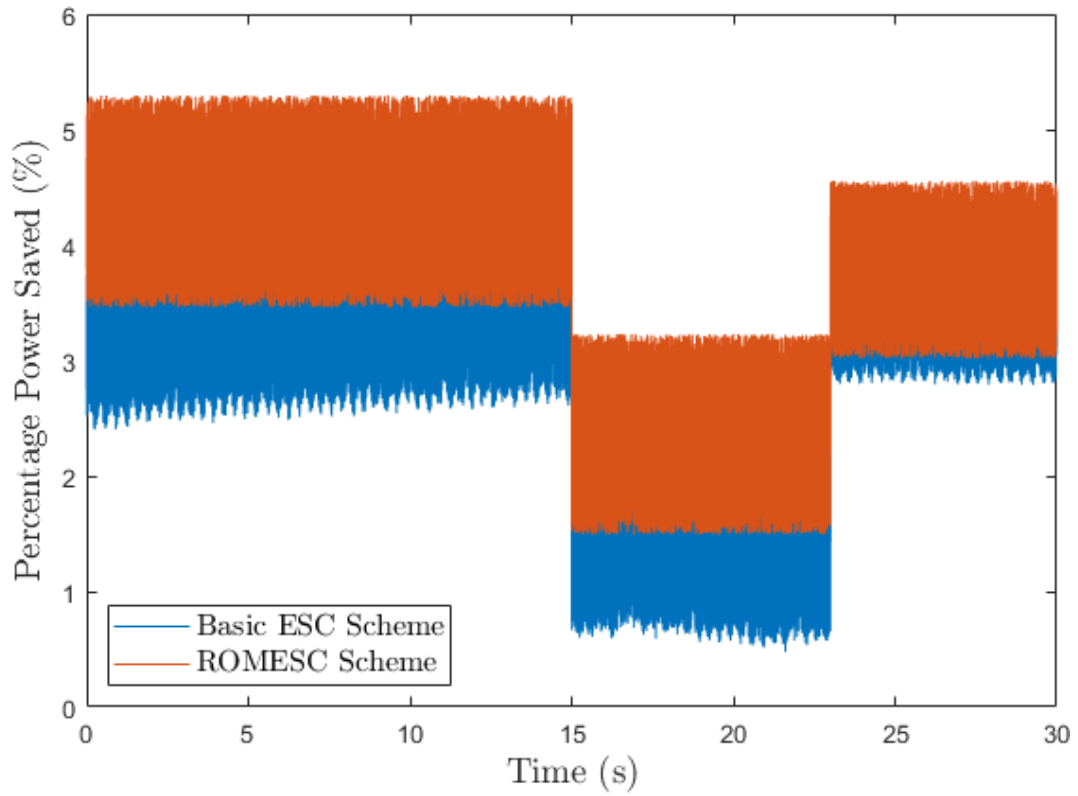


Figure 5.35: A plot showing the net power saving for both ESC schemes in changing freestream conditions.

The resultant net power saving is lower than was predicted by the ROM, as the output from the CFD simulation experienced noise due to differences in the timestep of the CFD simulation and the sampling frequency of the ESC scheme. However, it is clear that the slow convergence of the basic ESC scheme hinders the net power saving and as a result, the ROMESC produces a better power saving for longer. This effect is maintained for the full simulation with the ROMESC easily outperforming the basic ESC scheme. In fact, the ROMESC scheme saved 33% more power over the length of the simulation, than the basic ESC scheme did. By adapting the tuning parameters for the ROMESC scheme, rather than the basic ESC scheme, the performance of the ROMESC scheme may be improved further. This can be seen in Figure 5.36, which presents the net power saving results for both schemes after being passed through a lowpass filter to remove the high frequency oscillations and allows for analysis of the trends from each scheme.

The remaining variations are the noise experienced from the CFD system, but

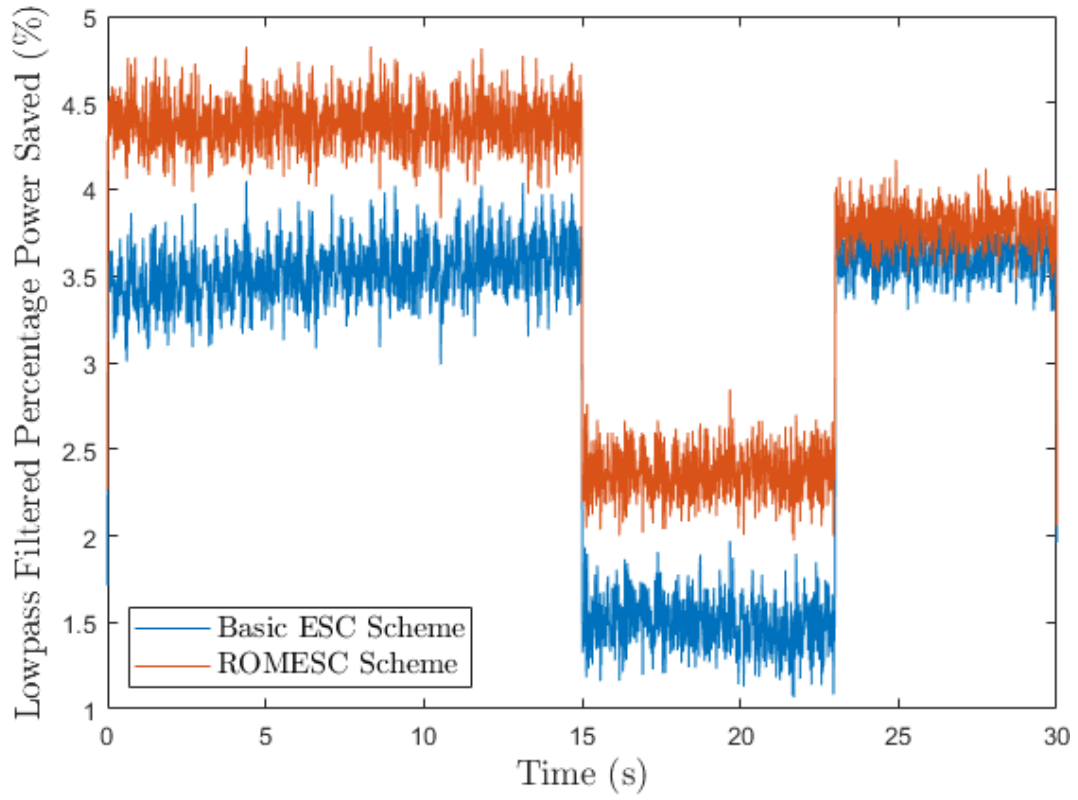


Figure 5.36: A plot showing the net power saving, for both ESC schemes in changing freestream conditions, after passing through a lowpass filter.

it is clear that the ROMESC is able to outperform the basic ESC scheme in terms of producing a net power saving.

This improved performance is not unexpected as the tests show the areas that the ROMESC was designed to improve upon. By providing a more accurate initial condition for the ESC scheme, the performance was bound to improve, similarly, it was expected to adapt to changing conditions better for the same reason. It was also expected that the scheme would reach a more accurate approximation of the global optimum condition, as information for the full system was known and described by the ROM, meaning that an optimum could be determined without the need for the simulation. The ESC scheme is still used as the model is not perfect and the ESC scheme can account for slight variations or noise in the model which the ROM cannot do.

5.8.3 Testing the Model Capabilities for a Constantly Fluctuating Freestream Velocity

The ROMESC scheme is able to adapt to step changes in the freestream flow, however, in the real world, the freestream would be constantly fluctuating. Therefore, the ROMESC scheme was tested with a fluctuating freestream velocity in the form of a sine wave. The freestream input can be seen in Figure 5.37. The frequency of the freestream sine wave was set to 0.8Hz, which was slower than any of the frequencies present in the system already, which allowed the ROMESC scheme to adapt to changes and the response to be observed. The amplitude of the freestream sine wave was set to 7m/s so that it covered the range of freestream velocities from 19.82m/s up to 33.82m/s, where the greatest effects could be seen.

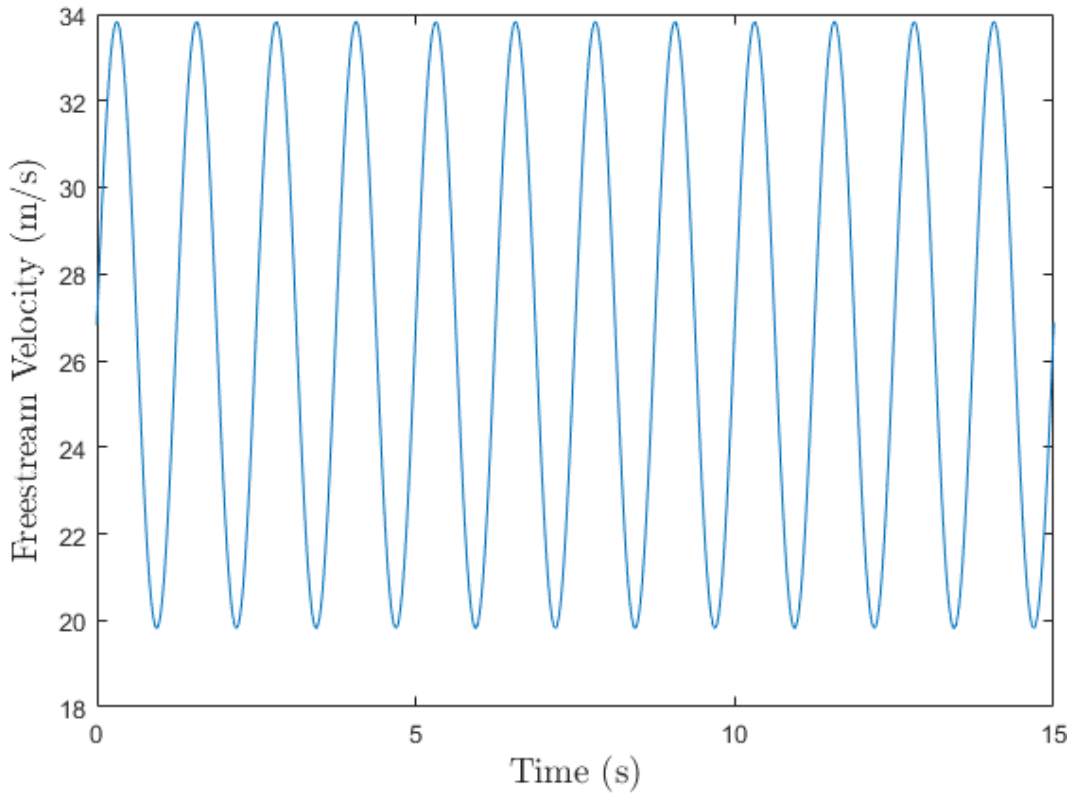


Figure 5.37: *The freestream velocity input used to test the ROMESC scheme in constantly changing freestream conditions.*

The scheme is designed to reset the integrator block whenever the freestream velocity changes. A tolerance is used to prevent the scheme from constantly

updating at every timestep in changing conditions. For this simulation, the tolerance was set to 1m/s, so every time the freestream velocity changed by more than 1m/s since the integrator was last reset, the ROM would be used to update the system. In between these resets, the ESC scheme is allowed to run and maintain or improve the power saving. This allows the ESC scheme to control the flow to correct for small errors in the model, but the tolerance is small enough that the ROM can improve the convergence for larger changes to the freestream. This tolerance could be set to a larger value to allow the ESC scheme to have a greater control if desired. Figure 5.38 shows the evolution of the momentum coefficient for the simulation.

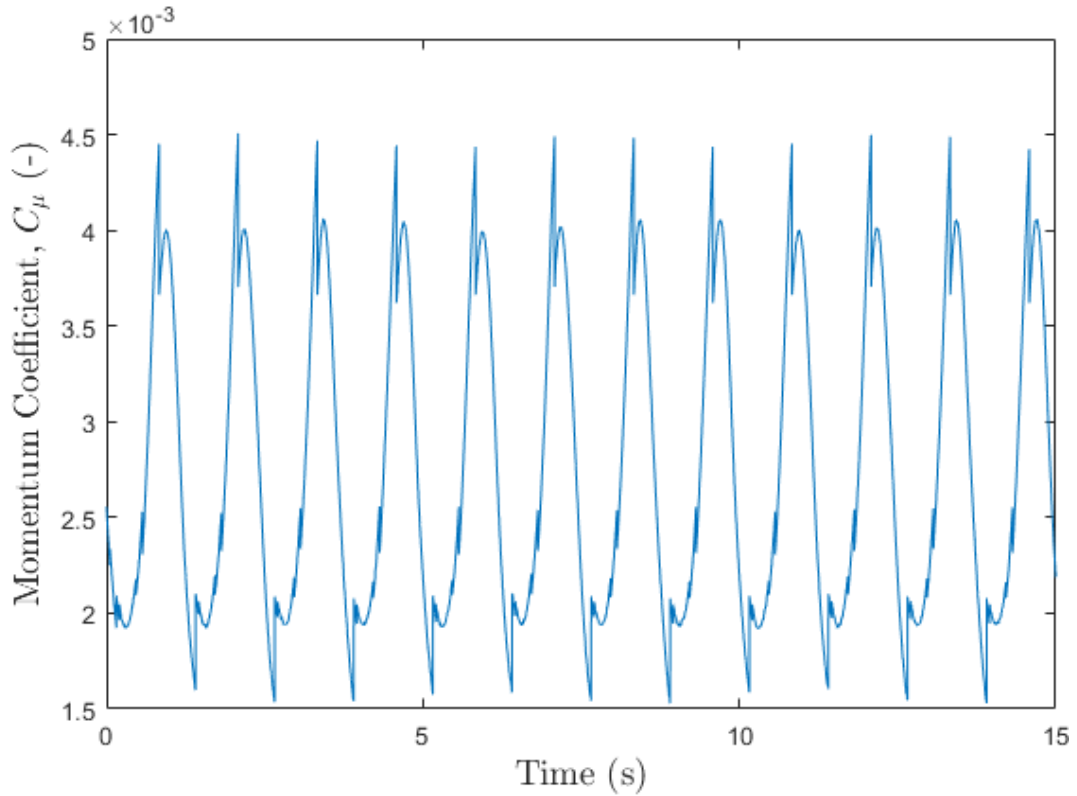


Figure 5.38: Convergence of the basic and ROM ESC schemes to an optimum SJA voltage for a sinusoidal freestream signal.

The sinusoidal like behaviour matches up with the freestream velocity seen in Figure 5.37, with the peaks in the freestream matching with the troughs in the momentum coefficients and vice versa. The sharp peaks and troughs in the momentum coefficient plot is an artefact of the timestep delay between the model needing to reset the integrator and the execution of the reset. This could

probably be removed by reordering the steps in which the model operates, but the delay does not have a major impact on the behaviour of the model. Figure 5.39 shows the raw SJA voltage from the ROMESC scheme, which is easier to see how the model is operating.

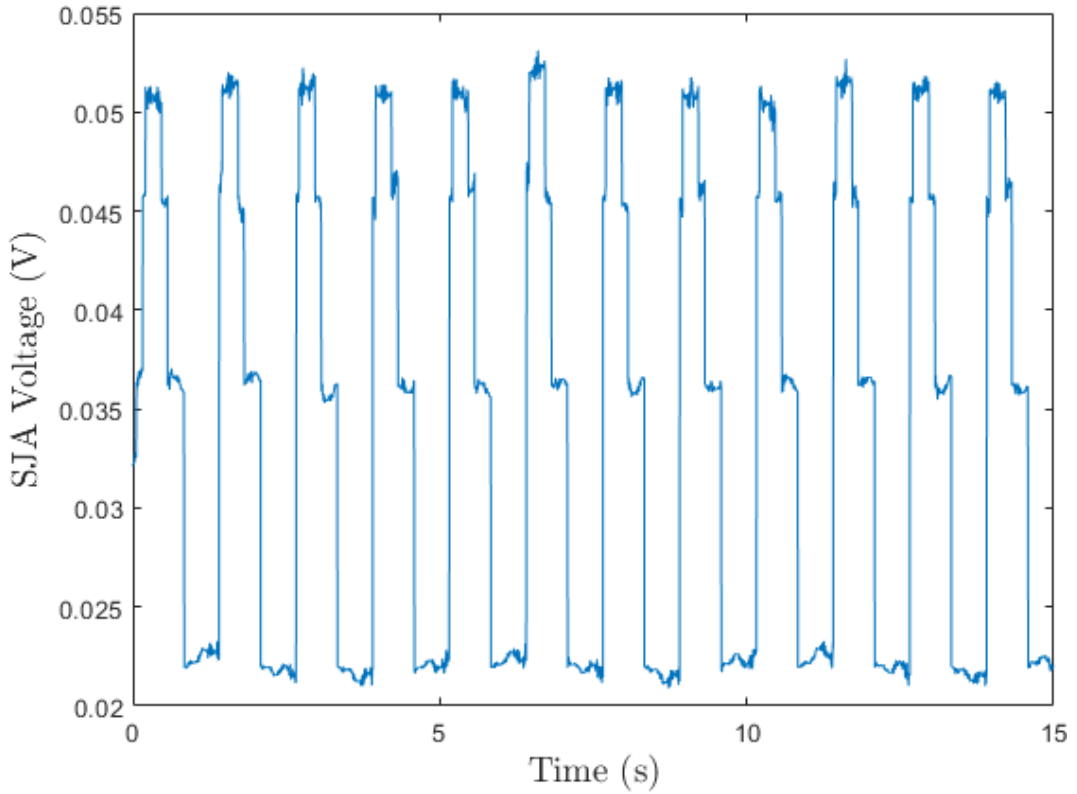


Figure 5.39: The voltage input for the SJAs as determined by the ROMESC controller for varying freestream conditions.

The sharp increases and decreases represent the resets of the integrator block and the oscillatory behaviour in between shows the ESC scheme operating. Due to the low tolerance in this simulation, the ESC scheme has very little impact, however, by increasing the tolerance, the ESC scheme could have a larger impact on the flow. The ESC scheme is stable and converges the set point is within a small neighbourhood of the optimum [85], so there should be no divergence issues to worry about. The constant resetting also improves the scheme for real time control prospects as the scheme can handle large, abrupt changes and will even adapt to large, slower changes to ensure that the ESC stays close to an optimal operating condition.

Figure 5.40 shows the percentage power saved under changing freestream con-

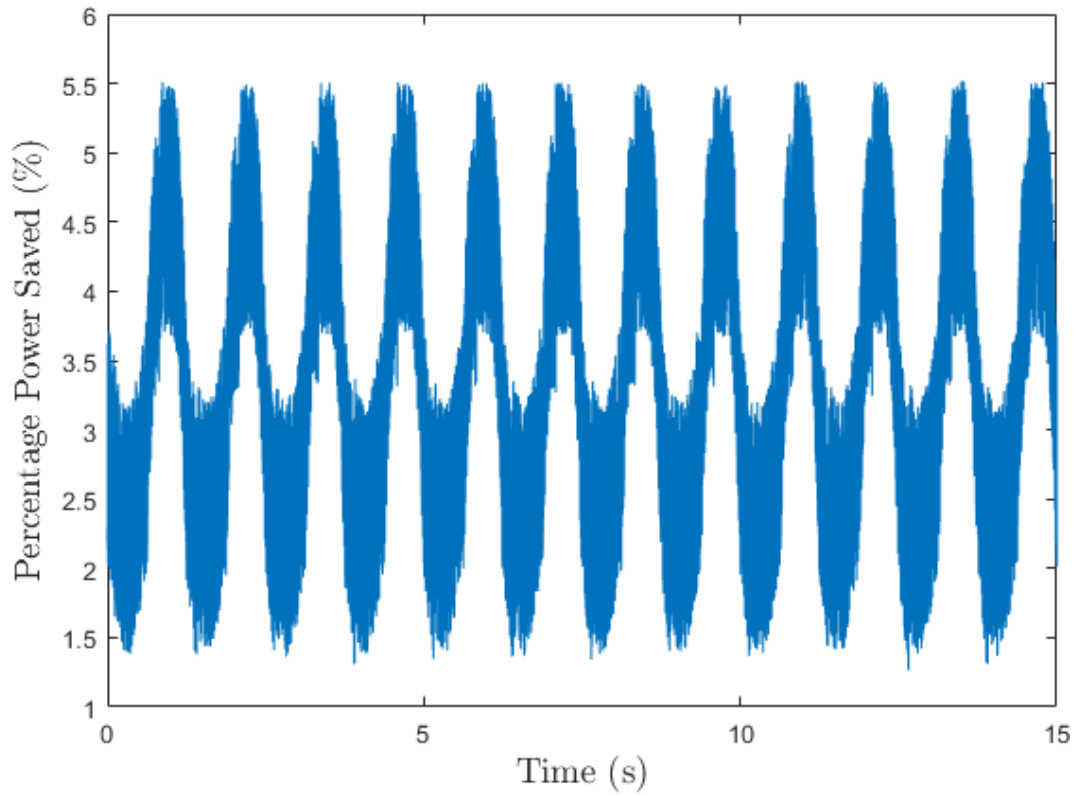


Figure 5.40: *The percentage net power saved by the ROMESC for constantly changing freestream condition.*

conditions for the ROMESC scheme. As is expected, due to the momentum coefficient in Figure 5.38, the power saving matches up with the changes in the freestream velocity. When the freestream is at its largest value of 33.82m/s, the power saving is between 1.5% and 3%, which is what is expected by using the results in Table 5.1. The smallest freestream velocity of 19.82m/s, shows a power saving between 3.7% and 5.5%, which again, is what is expected from the predictions in Table 5.1. The reason for the large variations in the power saving, is due to the behaviour of the SJAs and the noise in the CFD system, however, a net power saving is always produced.

5.9 Summary

It has been shown that the ROM is able to predict the C_D change from the equilibrium with a good accuracy of only 30%. This error is largely impact by the very small size of the approximations and in fact the errors drop to

less than 2% if the model predicts the actual drag coefficient, rather than the change from equilibrium. This was deemed a sufficiently small error for use in a controller developed later in this chapter. The ROM was also simulated using a testing input map, which was different from the training input map. This proved successful with small errors in the predictions of up to 11.5%. This result was ultimately used to predict the maximum net power saved by the SJAs at various freestream velocities, which proved to also be reasonably accurate. The linear interpolation method produced similar results to the LPV model, indicating that the LPV method was viable when approximating the full order system. One downside is that the LPV model cannot be used to reconstruct the flow field around the model, for which higher order models are probably needed. Even the 500 mode models were unable to accurately model the full order system, however, it was able to capture key frequencies in the flow, hence could possibly be applied to a frequency matching model if an alternative control was desired.

It is clear that the ROMESC scheme improves upon areas of the ESC scheme for real time control. The almost instant reset when the freestream velocity changes, eliminates the issues with convergence time for the basic ESC scheme under large, abrupt changes, while also ensuring that a largest net power saving in the SJA velocity domain is achieved. The results are very sensitive to the model and the downside to this is that an accurate model needs to be created for each new system, as the current ROM would not be suitable for any other CFD model, even if just the angle of the jets was changed.

The current ROM only accounts for changes to the freestream velocity, but this technique could easily be extended to other factors, such as wind direction and changes to the SJA behaviour. The scheme could then be used in the same way, with the new ROMs acting upon the reset, assuming that the prediction was accurate enough. A more complex model, controlling both the jet velocity and the jet pitch angle could also use the same reset technique to improve the performance for a multivariate ESC scheme.

Chapter 6

3D Results

6.1 3D Numerical Modelling

6.1.1 Introduction

The 2D CFD simulations showed a good drag reduction due to the SJA setup. This setup was applied to 3D models of the Ahmed body with the rear slant angled at 35° and 25° to the freestream flow. Various tests were completed for different jet pitch angles and actuation frequencies, which were compared to the literature where possible.

This section begins with a validation of the baseline 3D models, with results compared to the ERCOFTAC database [104]. The drag coefficient, pressure coefficients and velocity profiles are the main results compared, with qualitative analysis of the vortex structures included.

6.1.2 3D Numerical Model Validation

The drag coefficients for the Ahmed bodies with a 25° and 35° rear slant can be seen in Table 6.1. The experimental drag coefficients were obtained by Ahmed [6] and demonstrate the high and low drag regimes of the Ahmed body, either side of the critical rear slant angle. The Reynolds number, based on length, for all cases presented in the table is $Re = 2.8 \times 10^6$.

The larger drag coefficient for the 25° Ahmed body is due to the larger effect of the C-pillar vortices on the wake, causing the reattachment of the flow on to the surface of the rear slant. The separated flow causes a drop in the pressure at the top of the rear slant, leading to a large increase in the pressure drag.

The complex behaviour of the flow for this Ahmed body means it is tough to model accurately with a CFD simulation. In fact, the initial separation at the top of the rear slant is not captured by the URANS model. That is why the drag coefficient for the 25° CFD simulation is less accurate than the drag coefficient for the 35° Ahmed body simulation.

Both models were deemed to be within acceptable tolerances for the drag coefficient, however, this trend continued through the rest of the validation, with the 25° CFD model proving to be less accurate than the 35° CFD model.

Table 6.1: *The drag coefficient for the 25° and 35° Ahmed bodies, obtained by Ahmed [6] and recorded as part of this CFD study.*

	35° Ahmed Body	25° Ahmed Body
Experimental Data, Ahmed (1984)	0.26	0.3
3D CFD Simulations	0.265	0.29

The ERCOFTAC database has velocity profile data stored for both Ahmed bodies at various locations over the rear slant and in the wake. The velocity profile comparison in Figure 6.1 shows the experimental data in the centre plane over the rear slant of the 35° Ahmed body compared with the data from the CFD simulation.

It is clear that the profiles are very similar in shape over the top surface of the Ahmed body, upto $x/x_h = -0.15$. The over prediction in the flow velocity, which was present in the 2D results, does not exist here. The largest error in the profiles here is at the base of the profile located at $x/x_h = -0.15$, which is located just upstream of the top of the rear slant. The small increase in velocity here is likely due to the presence of SJA cavities located just below the top of the rear slant. The cavities were not present in the experiments conducted for the ERCOFTAC Ahmed body database, as a standard Ahmed

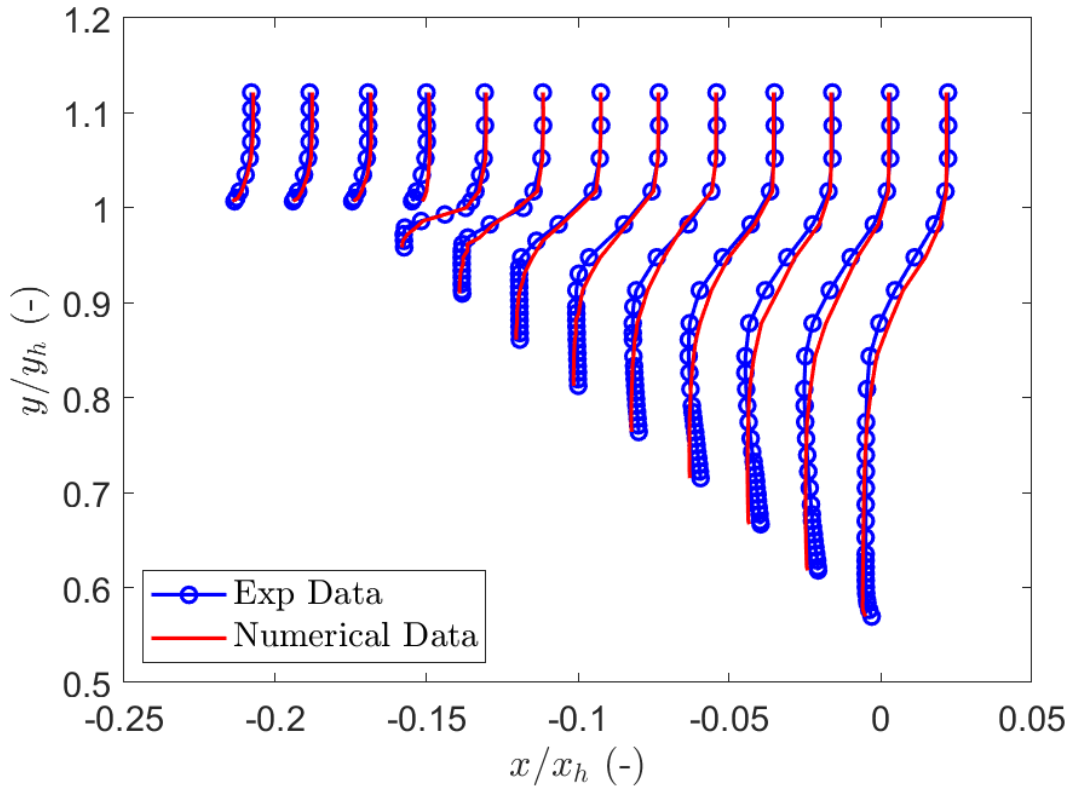


Figure 6.1: Velocity profile comparison over the rear slant of the 35° Ahmed body with velocity profile data from the ERCOFTAC database [104].

body was used. Meanwhile, the flow in the CFD model likely experienced a small increase in the flow velocity at this point due to the lack of a no-slip wall condition on the cavities. The remaining profiles also show a good agreement to the experimental data, although there are small discrepancies between the profiles at the top of the wake, with the CFD simulation results showing a small over prediction in the flow velocity in this region. Figure 6.2 presents a closer look at the last velocity profile, located at $x/x_h = 0$.

It is clear from Figure 6.2 that the over prediction of the flow is minor and at worst represents a non-dimensional velocity increase of 20%. The CFD simulation is still able to predict the recirculating region with a high degree of accuracy, even if the peak minimum velocity is slightly smaller than the experimental data. The error for each profile was computed using the formula outlined in Equation (4.1). It was found that the maximum error in the velocity profiles on the centreplane of the rear slant was 10% with an average error of just 6%. The velocity profiles for the 25° Ahmed body were also compared

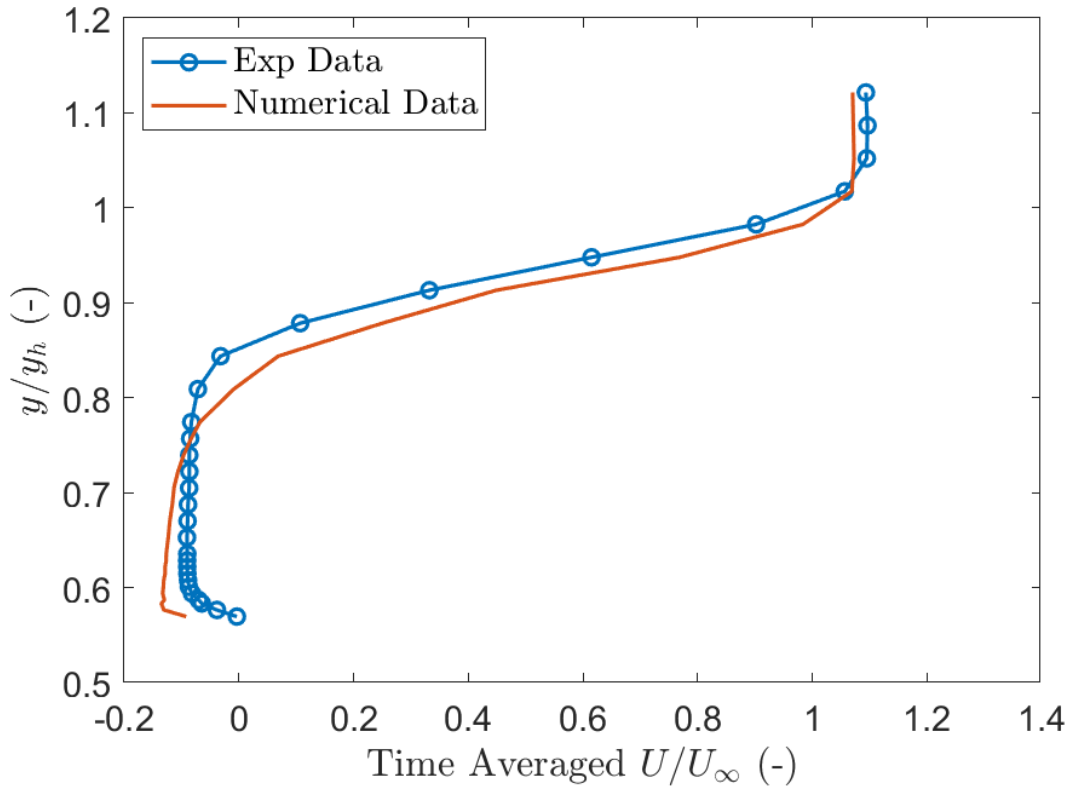


Figure 6.2: Velocity profile comparison at the end of the rear slant at $x/x_h = 0$ for the 35° Ahmed body with velocity profile data from the ERCOFTAC database [104].

and the results can be seen in Figure 6.3.

The velocity profiles for the 25° Ahmed body show a good agreement over the rear slant. The CFD simulation under predicts the flow velocity over the top of the rear slant. It also struggles to predict the flow in the top of the wake for the first few profiles at the top of the rear slant. Further down the slant the change becomes less extreme and the simulation is able to better match the experimental data. The issues here are reflected in the larger errors than for the 35° Ahmed body, with a maximum error of 15% and an average error of 10%.

The ERCOFTAC database also has velocity profile data stored for off centre planes. The planes correspond to $z/z_h = 0.25$ and $z/z_h = 0.04$ where $z/z_h = 0.5$ is the centreplane and $z/z_h = 0$ is the right hand side of the model. These profiles are key due to the increased effect of the C-pillar vortices on the velocity profiles closer to the edges of the body, at $z/z_h = 0$ and 1. Figure 6.4 shows the velocity profile comparison for the profiles located at $z/z_h = 0.25$ for the

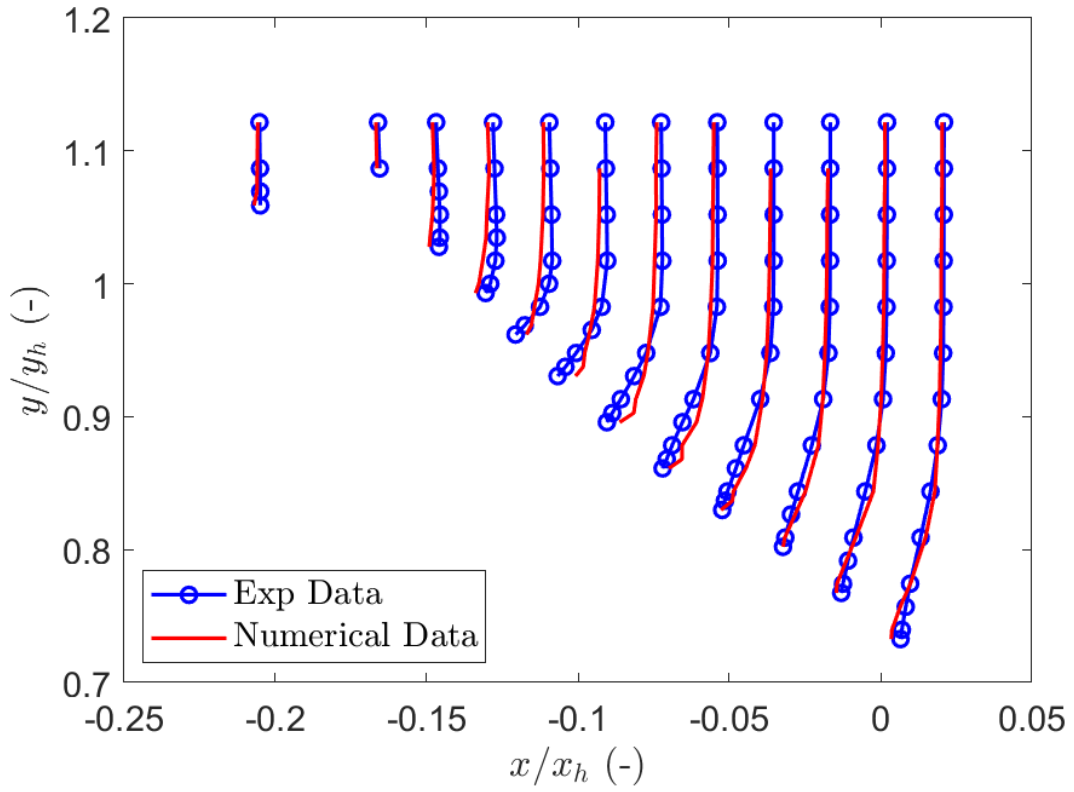


Figure 6.3: Velocity profile comparison over the rear slant of the 25° Ahmed body with velocity profile data from the ERCOFTAC database [104].

35° Ahmed body.

The behaviour of the flow is very similar to the centreplane flow with a good agreement of the flow on the top face of the body. The over prediction in the flow velocity at the top of the wake is larger than in the centreplane, which shows the effects of the C-pillar vortices on the flow in this plane. The overall agreement is still good though, which is reflected in the errors for these profiles. The maximum error is 12% with an average error of 8%. This shows a small decrease in accuracy from the centreplane flow, but still a relatively small error across the rear slant. This breaks down for the plane at $z/z_h = 0.04$ however, with the CFD simulation unable to fully capture the flow velocity in the C-pillar vortices. Figure 6.5 shows the velocity profile comparison at the edge of the body.

The CFD simulation massively over predicts the flow over the top of the rear slant and that error translates to the profiles further down the slant. This error is due to the C-pillar vortices being directed further into the wake in the

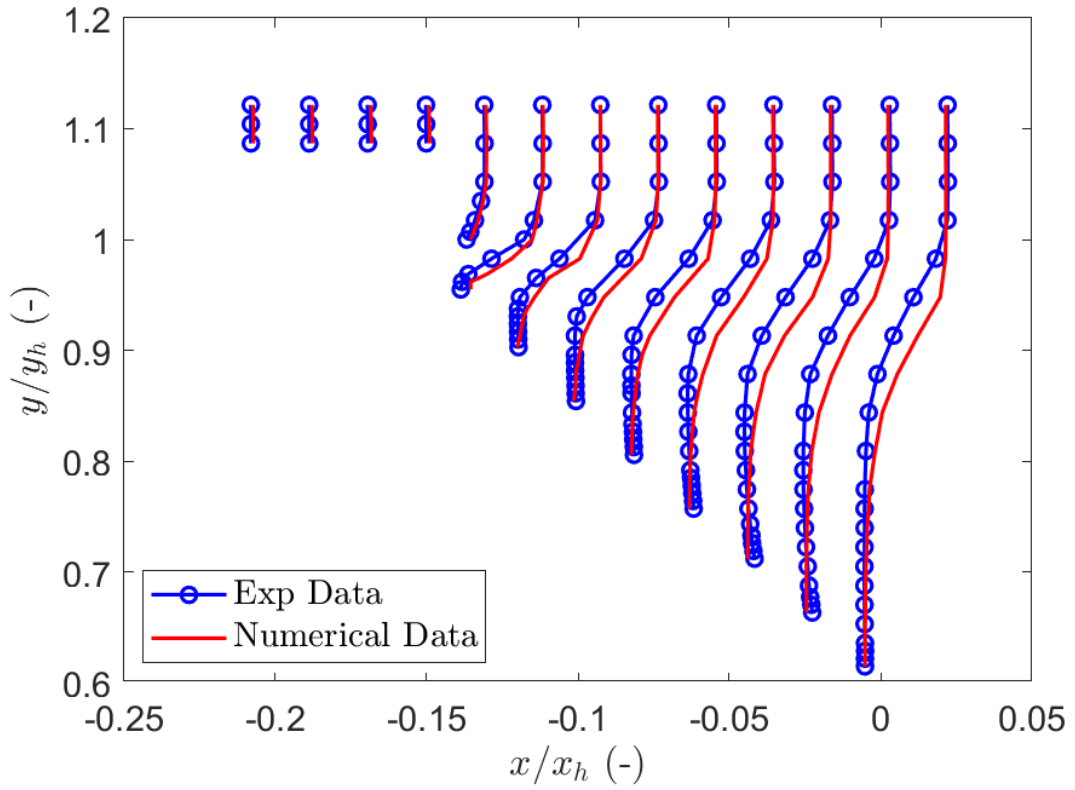


Figure 6.4: Velocity profile comparison over the rear slant of the 35° Ahmed body in the plane $z/z_h = 0.25$ with velocity profile data from the ERCOFTAC database [104].

CFD simulation than is shown in the experimental velocity profiles from the ERCOFTAC database [104]. There is less of a disparity in the profiles towards the end of the rear slant, suggesting that the error due to the C-pillar vortices has less of an impact further behind the body. The errors in these profiles are much larger as expected, with a peak error of 37% and an average error of 20%. This suggests that changes in the centreplane are likely to be more accurate to the behaviour of the experimental data, whilst changes at the edges of the body are more unreliable.

The velocity profiles in the off-centre planes for the 25° Ahmed body were also analysed. The errors in the velocity profiles at $z/z_h = 0.25$ were similar to those in the centreplane, with a slightly higher average error of 12%. Similar to the 35° Ahmed body, the errors in the profiles at the edge of the body were much larger. With a peak error of 27% and an average error of 19%, it is clear that again the CFD model is unable to capture the full flow behaviour at

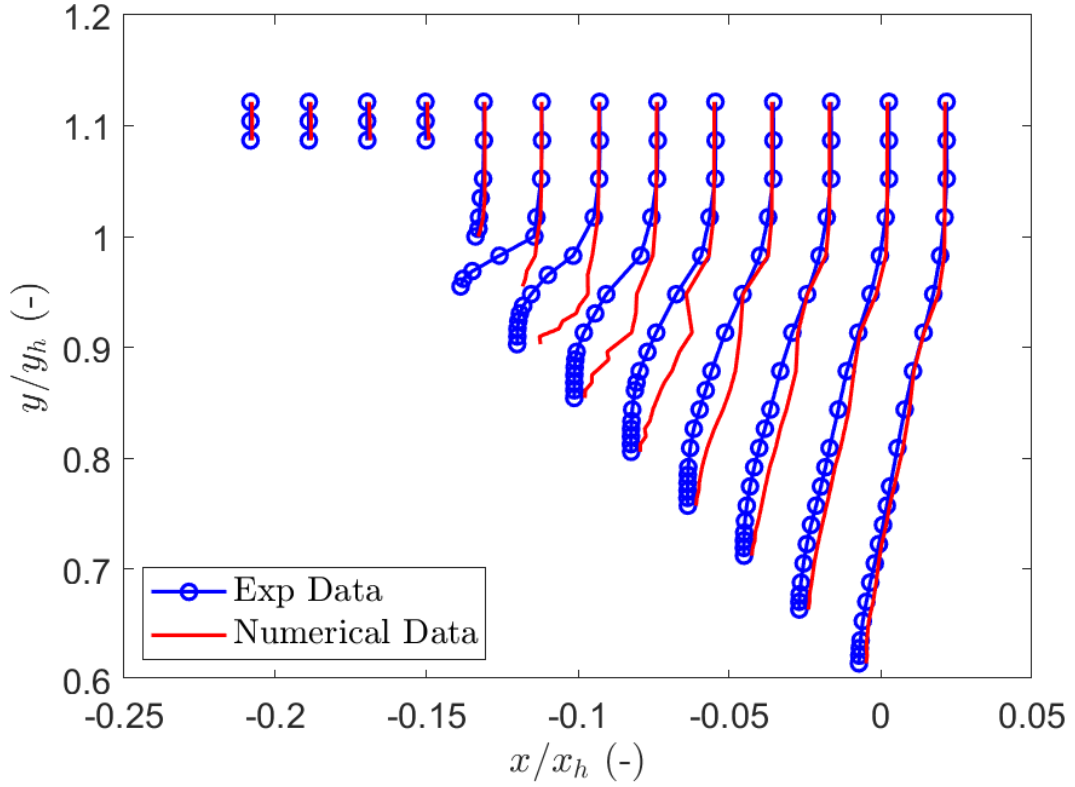


Figure 6.5: Velocity profile comparison over the rear slant of the 35° Ahmed body in the plane $z/z_h = 0.04$ with velocity profile data from the ERCOFTAC database [104].

the edges of the body, where the complex C-pillar vortex interactions happen. This was expected, as the C-pillar vortices are much stronger than for the 35° rear slant. Figure 6.6 presents the profile at $x/x_h = -0.1$ in the plane at the edge of the body ($z/z_h = 0$).

It is clear from this image that unlike the 35° case, the CFD model under predicts the flow on the rear slant. The experimental data shows a small drop in the flow velocity below $y/y_h = 0.9$, whereas the CFD data shows a steady decrease in the flow velocity below $y/y_h = 1$. The errors here are caused by the inability of the model to capture the C-pillar vortex behaviour.

Velocity profiles in the wake behind the Ahmed body were also analysed for both the 25° and 35° variants. Figure 6.7 has the profiles behind the 35° Ahmed body. There is a good agreement in all profiles, demonstrating that the flow development behind the body in the centreplane is captured with reasonable accuracy by the CFD simulation. This is backed up by the errors computed

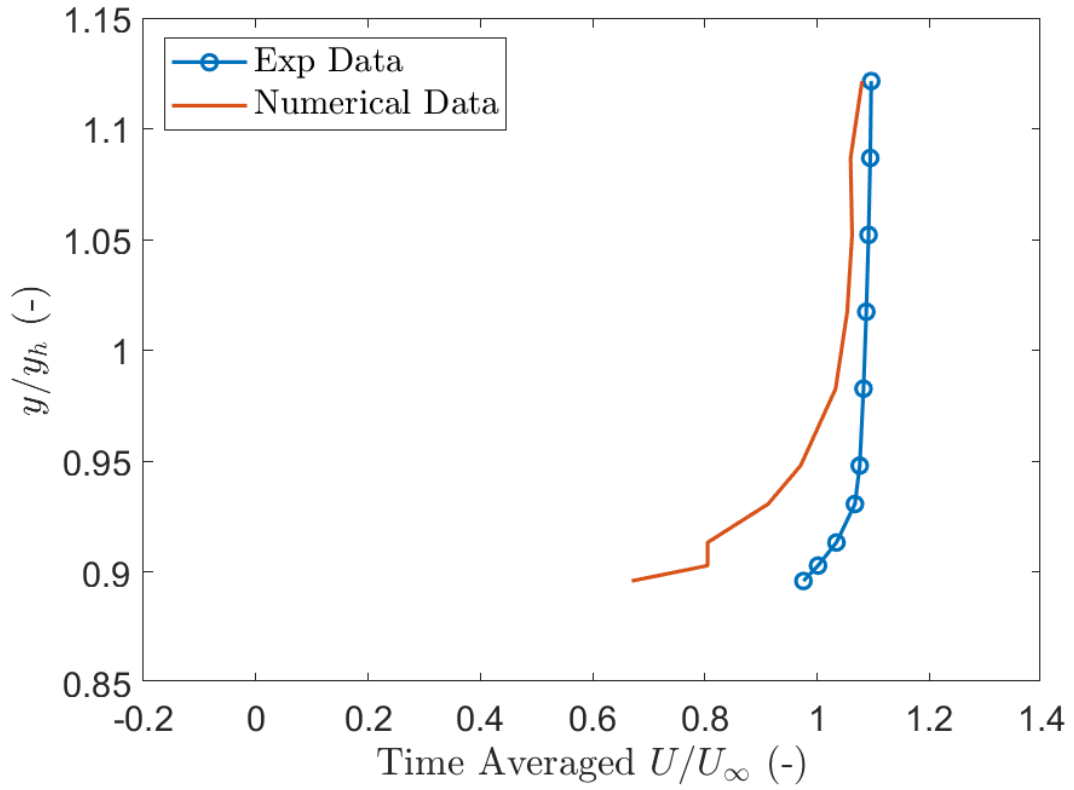


Figure 6.6: Velocity profile comparison at $x/x_h = -0.1$ on the rear slant of the 25° Ahmed body in the plane $z/z_h = 0$ with velocity profile data from the ERCOFTAC database [104].

for the wake profiles, with a peak error of 11% and a mean error of 8%. The peak error occurs in the first profile after the body in the wake at $x/x_h = 0.1$. Figure 6.8 presents this profile in more detail.

There is a large drop in the velocity behind the body at the base of the wake. This is expected as the body is not symmetric, with the velocity expected to be larger at the top of the wake due to the rear slant. The CFD simulation is unable to fully capture the drop in velocity in the lower section of the wake. This could be due to small differences in the setups of the experimental and CFD models, such as the increased ground clearance in the CFD simulation for stability reasons (0.075m rather than the 0.05m in the experiments), or the absence of feet for the model to stand on. The rest of the profile matches up with reasonable accuracy, so it was decided that the error in the base of the wake would have a minimal effect on the flow behaviour at the top of the wake, where this research is focused.

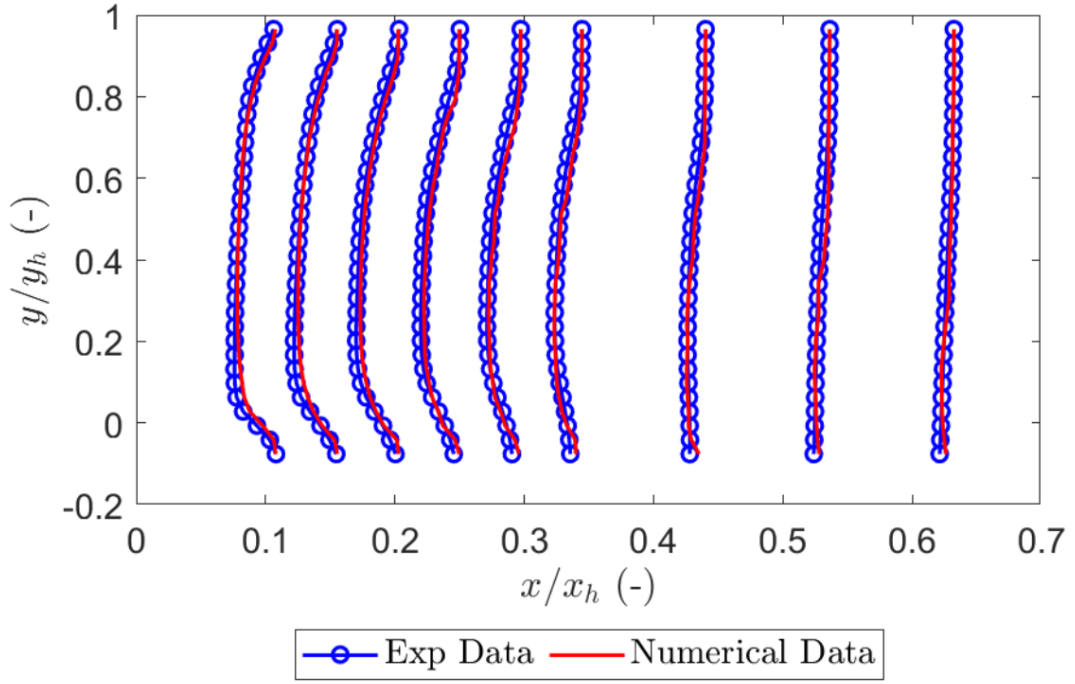


Figure 6.7: Velocity profile comparison in the wake of the 35° Ahmed body with velocity profile data from the ERCOFTAC database [104].

A similar situation arises for the 25° Ahmed body, where the CFD model is unable to capture the full extent of the drop in velocity at the base of the wake. However, it was decided that this would also have a minimal effect on the flow behaviour in the top of the wake. The errors for the 25° Ahmed body also peaked at 11%, with a mean error of 8%.

The final aspect of the validation was to qualitatively compare the vortex structures behind the body to known structures found in the literature. Figure 2.4 demonstrates most of the known vortex structures around the 25° Ahmed body. The Reynolds number used by Zhang [8] was 2×10^5 , compared to the Reynolds number of 2.9×10^6 for the CFD simulations in this research. The key vortices to focus on here are the C-pillar vortices located at either edge of the Ahmed body, along with the secondary longitudinal vortices, located more centrally than the C-pillar vortices. These vortices are key to the behaviour of the flow behind the model and play a key role in the different flow structures behind the Ahmed body for the 25° and 35° variants.

Figure 6.9 shows a vorticity iso-surface generated around the 25° Ahmed body,

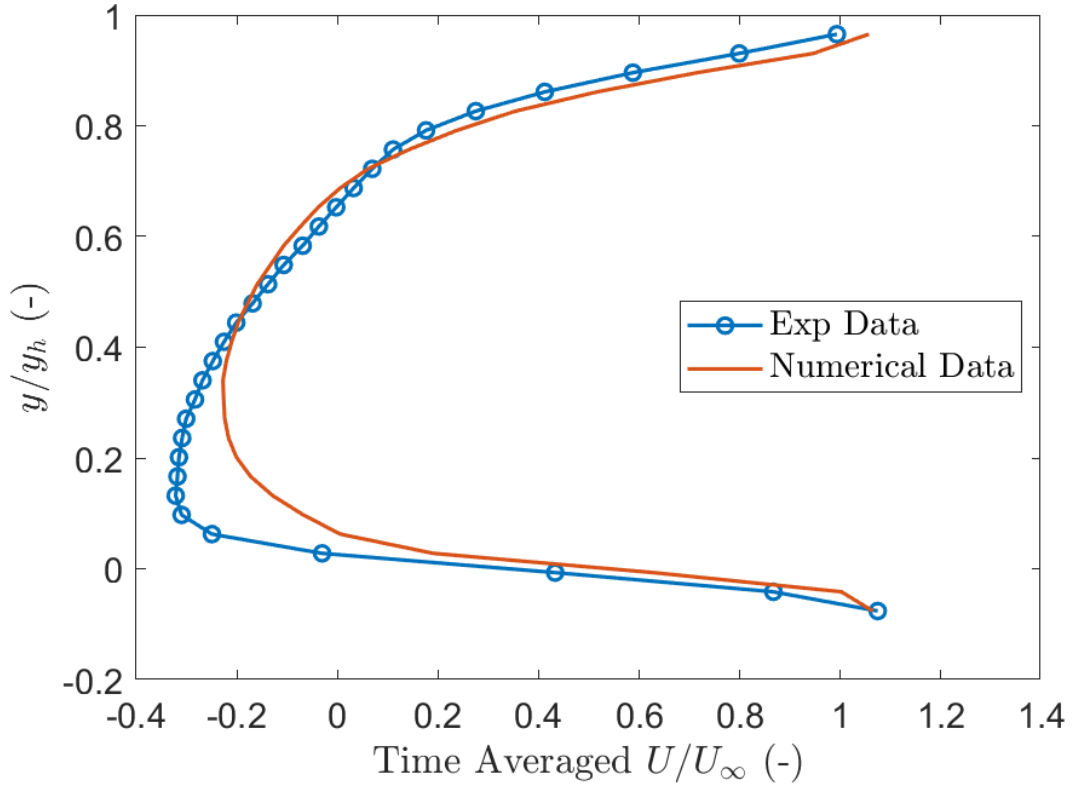


Figure 6.8: Velocity profile comparison in the wake at $x/x_h = 0.1$ for the 35° Ahmed body with velocity profile data from the ERCOFTAC database [104].

to demonstrate the vortical structures observed in the CFD simulation. It is clear to see the C-pillar vortices located behind the model and the longitudinal secondary vortices located slightly more centrally. The issues with the C-pillar vortices identified in the velocity profiles can be seen here by the separation of the C-pillar vortices from the surface of the rear slant on the lower portion of the slant.

The behaviour of the longitudinal secondary vortex is slightly different for the CFD model, with the vortex being directed more centrally, which is what leads to the small increase in mean error in the velocity profiles between the centreplane and the plane at $z/z_h = 0.25$. The longitudinal vortices located on the top face in Figure 2.4, can also be seen on the CFD model in Figure 6.9, with longitudinal vortices also located on the side faces of the model, which is to be expected but not depicted on the diagram in Figure 2.4. The CFD model does not have the small recirculation bubbles at the front of the model. This is a very small recirculation bubble in reality and the CFD simulation

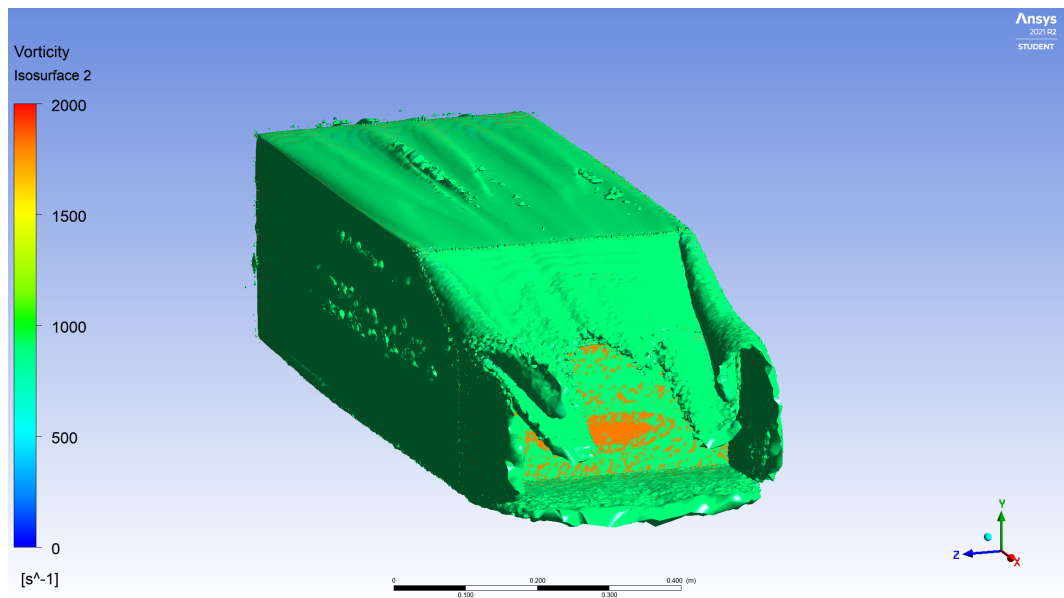


Figure 6.9: *Vorticity iso-surface showing the vortical structures around the 25° Ahmed body obtained from the CFD simulation.*

is unable to capture these small changes. There is however a decrease in the wall shear in this region, suggesting that the flow is trying to separate from the body.

The vortical structures located in the main wake region behind the body cannot be seen clearly in Figure 6.9 as there is a lot of turbulent flow in the wake area. However, the centre plane streamline plot in Figure 6.10 shows the presence of the 2 recirculation regions in the wake. The larger recirculation region is located at the top of the wake with a smaller region located in the bottom of the wake. The Strouhal number given in Figure 2.4 for these is between 0.44 and 0.54. The comparative Strouhal number given by the CFD simulation is between 0.52 and 0.62, which represents a frequency change of just 4Hz. This is at the upper edge of the region presented by Zhang [8], however this could be due to the large difference in Reynolds number. If the Ahmed body was located closer to the ground plane, the flow under the body would be restricted, leading to an even smaller lower recirculation in the wake, with a more dominant upper recirculation.

Although the structures will differ for the 35° Ahmed body, there are still comparisons that can be drawn between the 2 models. Figure 6.11 shows a

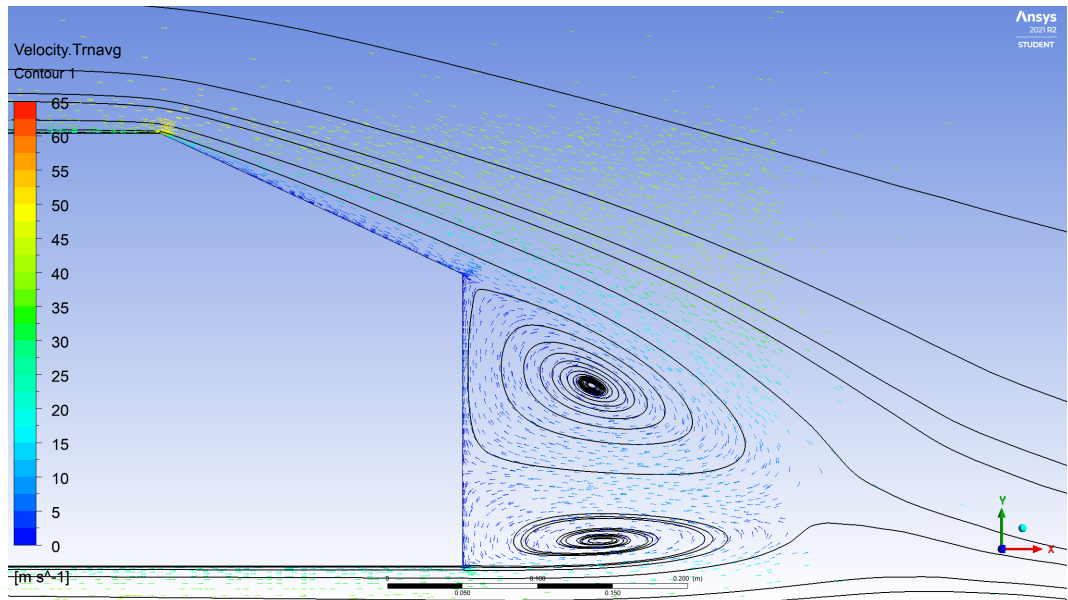


Figure 6.10: *Velocity vector plot with streamlines to demonstrate the recirculating region in the wake for the 25° Ahmed body.*

vorticity iso-surface of the 35° Ahmed body for the CFD simulation. It is not possible to determine the vortex structures in the wake due to the separated flow inducing a more turbulent flow, however, the presence of C-pillar vortices at the edges of the body is clear. There also appears to be less of a curl up effect, indicating that these vortices are weaker than for the 25° rear slant. This is corroborated by the data which shows that the pressure drag contribution from the C-pillar vortices is 10 times smaller for the 35° Ahmed body.

The longitudinal vortices on the top and side faces of the model are also present. There is again an increase in the wall shear at the front of the model instead of a recirculating region. There is also 2 recirculating regions in the wake, with the larger region located in the top of the wake and dominating the flow over the rear slant, with a smaller recirculating region located in the lower portion of the wake behind the body. This is illustrated in Figure 6.12, in which the streamlines demonstrate the 2 recirculating regions in the wake. If the body was lower to the ground plane, again, the lower recirculation region would shrink and the upper region would dominate the wake even more.

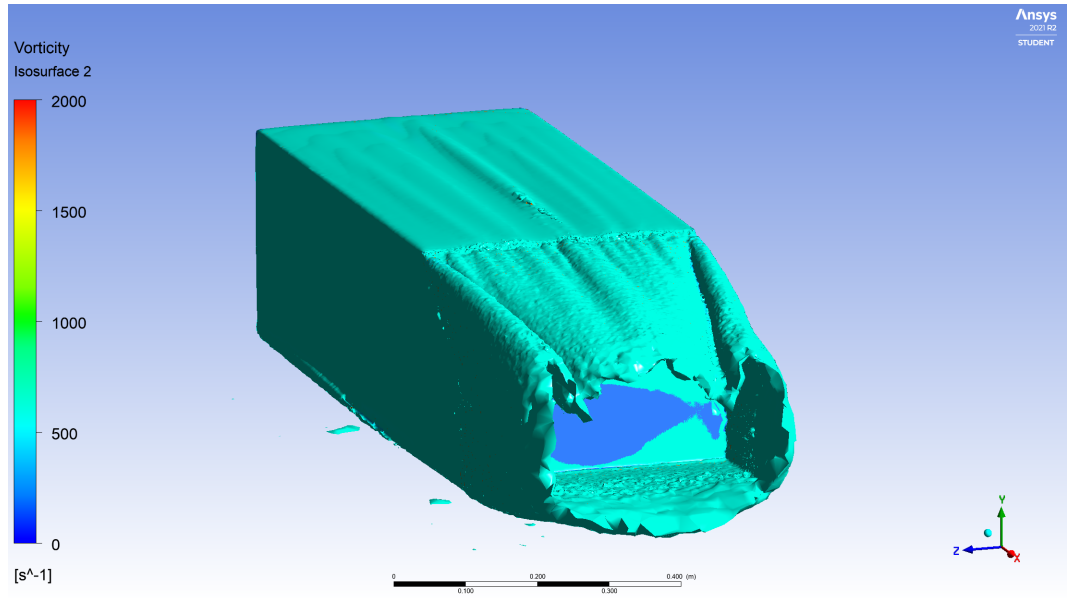


Figure 6.11: *Vorticity iso-surface showing the vortical structures around the 35° Ahmed body obtained from the CFD simulation.*

6.1.3 Controlling the Wake of a 25° Rear Slant 3D Ahmed Body Using Synthetic Jet Actuators

Park [37] presented experimental data for both the Ahmed body with the 25° rear slant and the 35° rear slant. Figure 6.13 shows the comparison between the C_p on the centreline of the rear faces for the 3D 25° Ahmed body CFD simulation as part of this research and the experimental data presented by Park [37]. This C_p is for the unactuated case and is used to demonstrate the accuracy of the rear face pressure for the CFD simulation. The black line on Figure 6.13 shows the height of the bottom of the rear slant. It is clear that both sets of data present the same trend in the rear face pressure, with an increasing pressure over the length of the rear slant and an almost constant pressure over the lower rear face. The CFD simulation provides a decent approximation to this with an average relative error of 25%.

The rear face C_p for the 35° Ahmed body can be seen in Figure 6.14. Again, the black line represents the bottom of the rear slant. The rear face C_p for the 35° Ahmed body show a much larger variation over the rear faces than the 25° Ahmed body. The pressure at the top of the rear slant is very small,

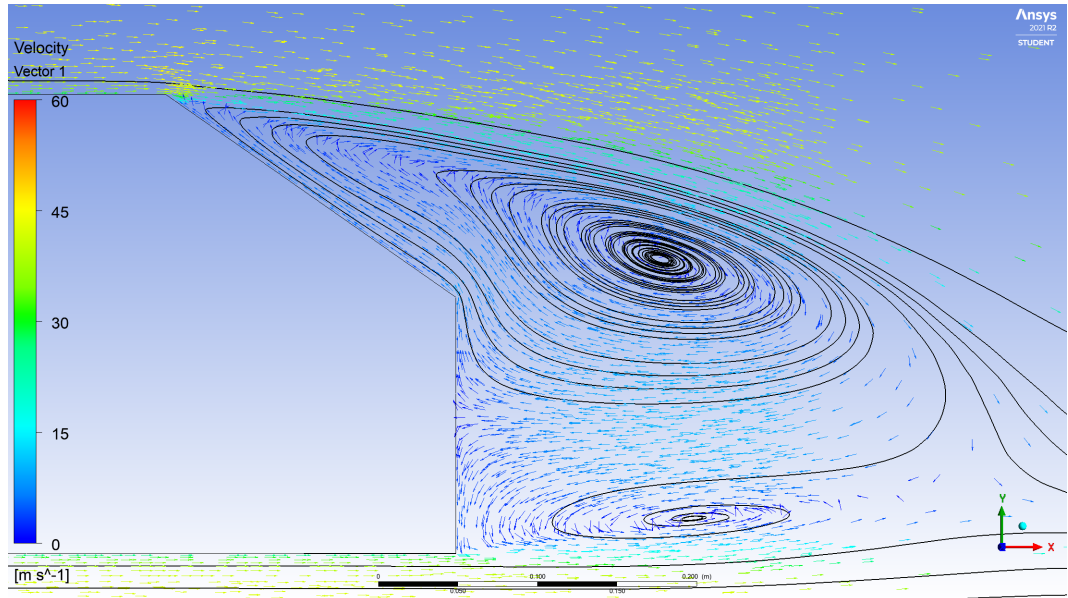


Figure 6.12: *Velocity vector plot with streamlines to demonstrate the recirculating region in the wake for the 35° Ahmed body.*

since this is where the flow separates from the Ahmed body. From $y/y_h = 0.9$ down to $y/y_h = 0.65$, there is a slight increase in the pressure on the slant, which differs from the experimental pressure. The experimental data presents a small decrease in the pressure in this region, however, the difference between the pressure values is relatively small, with only a 7% error in this region.

The transition region from the rear slant to the rear face, between $y/y_h = 0.65$ and $y/y_h = 0.5$, has a large variation in the pressure, with a significant dip at the bottom of the rear slant and a significant increase at the top of the lower rear face. The experimental data does not have enough data in this region to fully corroborate this behaviour, however, an increase in the C_p is also shown as the flow moves from the rear slant to the lower rear face. Below $y/y_h = 0.5$, the pressure from the CFD shows a small increase to $C_p = -0.1$ in the middle of the lower rear face, before experiencing a drop in the C_p at the base of the lower rear face. The experimental data shows a large decrease in C_p at $y/y_h = 0.2$, which the CFD simulation does not match. However, even with this data, the average relative error over the whole of the rear faces is only 13%.

Various SJA setups were discussed in Chapter 2, with the optimum location

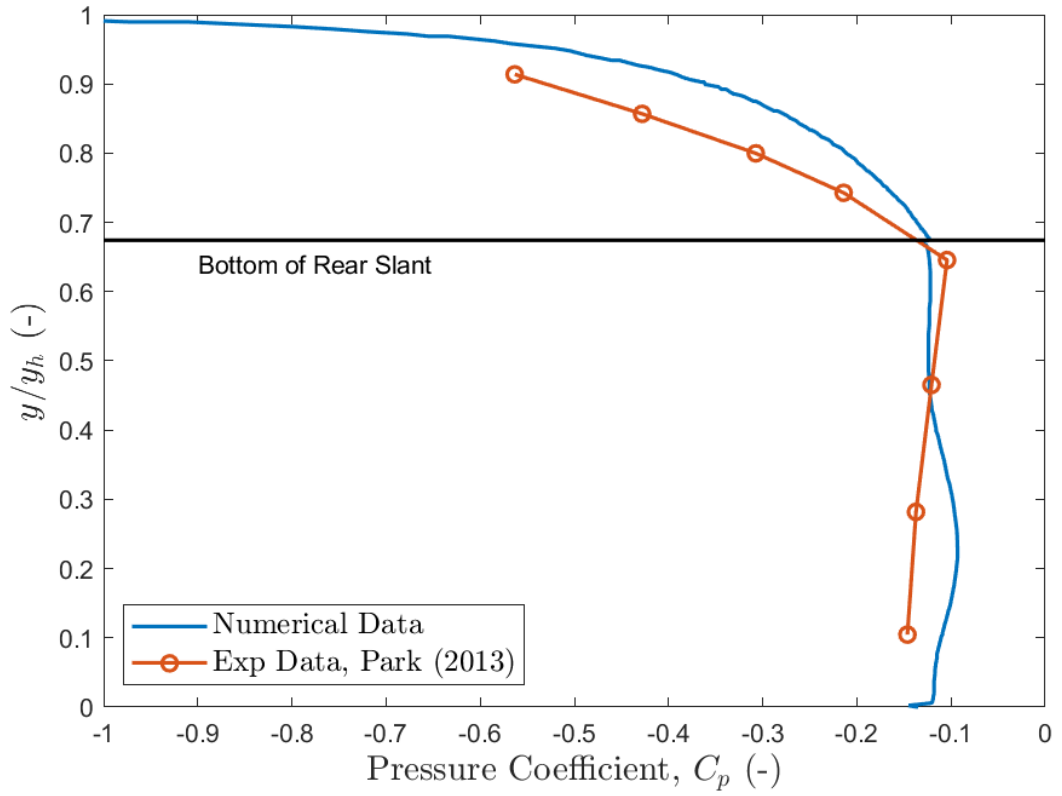


Figure 6.13: C_p plot on the rear faces of the 25° Ahmed body for the CFD simulations in this research and for the experimental C_p [37].

determined to be just below the top of the rear slant. Whilst this research focuses on jets directed into the wake to take advantage of the Coanda effect and reduce the height of the wake, jets blowing normal to the rear slant were also considered as drag reductions of up to 8% are presented in the literature [37, 67]. The SJAs blowing normal to the rear slant were tested for both the 25° and 35° Ahmed bodies. Figure 6.15 shows the results for the 35° Ahmed body.

Park et al. showed that this setup only produced an increase in the C_D , which was almost constant across most momentum coefficients. Only 2 simulations were run for this research to compare to these results as it was deemed that the results would not benefit this research. The 2 simulations used different actuation frequencies, $F^+ = 1$, which was used by Park et al. and $F^+ = 24$, which is the operating frequency of the SJAs for this research. The CFD simulation with $F^+ = 1$, produced a much smaller C_D increase than Park presented for their research, whilst the CFD simulation with $F^+ = 24$ produced

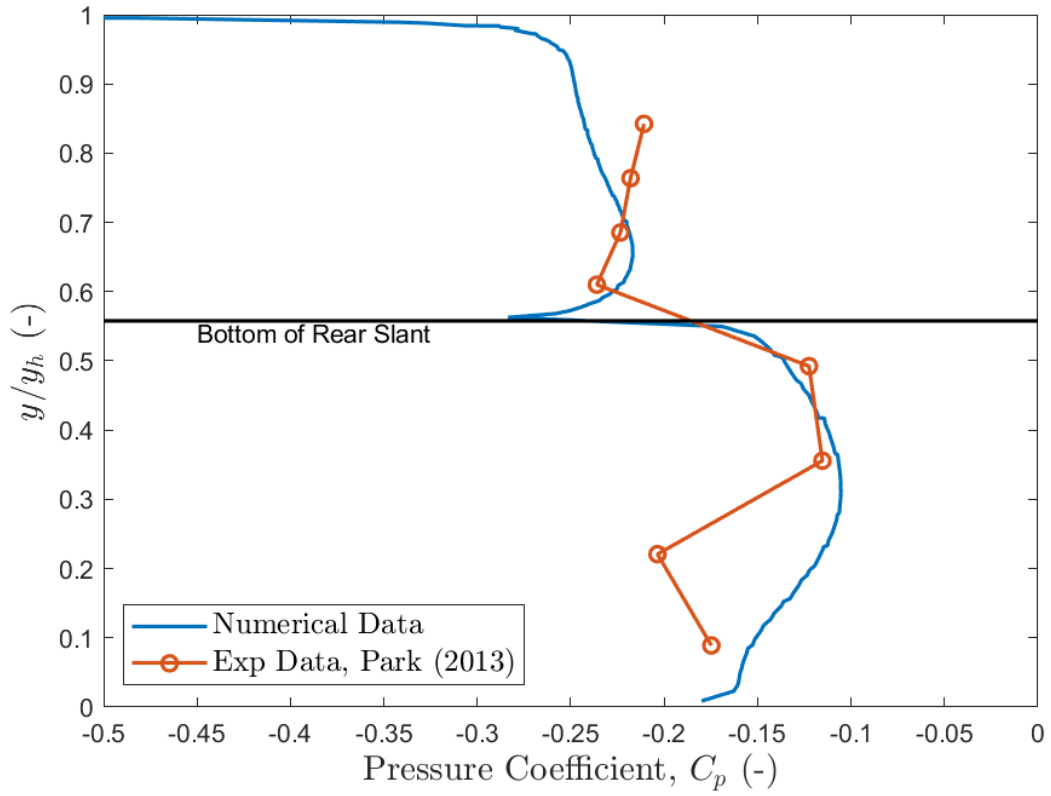


Figure 6.14: C_p plot on the rear faces of the 35° Ahmed body for the CFD simulations in this research and for the experimental C_p [37].

an almost identical drag increase to the experimental data from Park.

The difference in the results for the experimental and CFD simulations at $F^+ = 1$ are caused by the use of a turbulence model to approximate the flow field information. The numerical model is able to model the Kelvin-Helmholtz instability and the large scale structures in the wake, which provide the mean flow information. However, the small scale structures and fluctuations are not captured by the turbulence model, leading to the SJAs being ineffective in the CFD simulation. The main reason that these jets are ineffective for this setup, is because these jets rely on interacting with the small scale fluctuations to amplify their effects. This is also true for the 25° Ahmed body CFD simulation results, depicted in Figure 6.16. The normal blowing to the rear slant setup produced a drag reduction for the 25° Ahmed body of upto -4.2% for the experimental data. Again, the CFD simulation at $F^+ = 1$ did not match the experimental data, in fact, it actually produced an increase in the drag coefficient. However, the CFD simulation with $F^+ = 24$ was able to produce

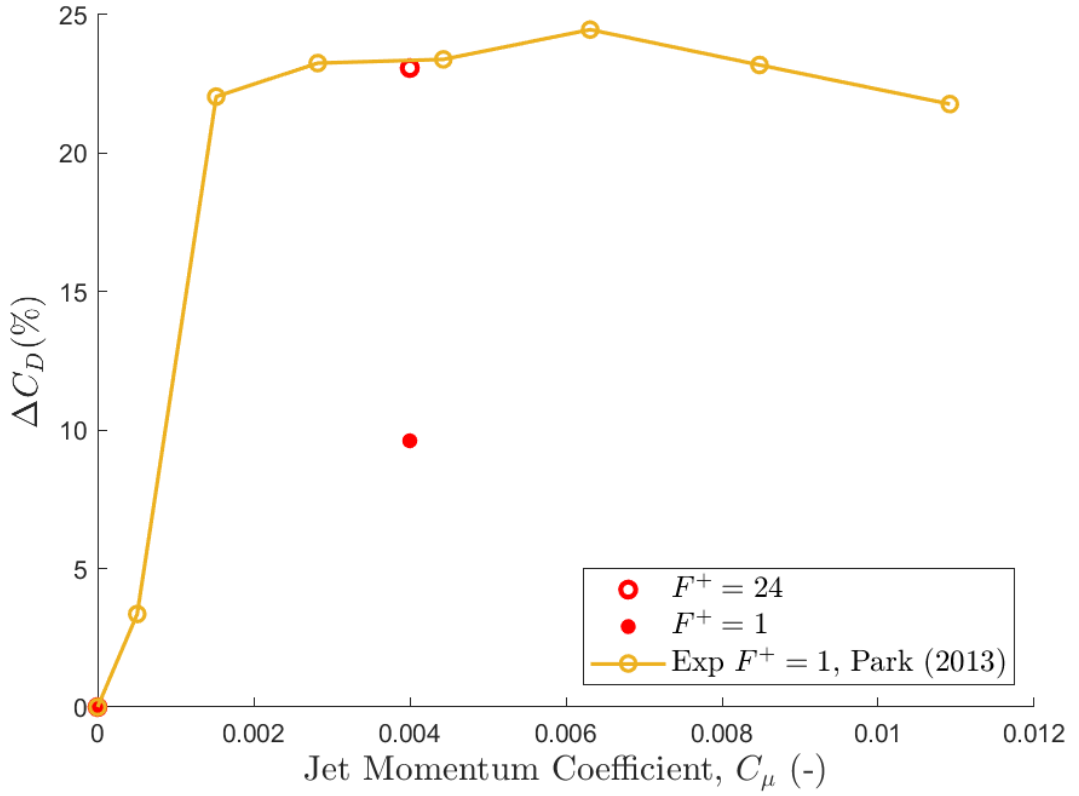


Figure 6.15: C_D change for the 35° Ahmed body with SJAs acting normal to the rear slant for the CFD simulation and the experimental C_D [37].

a reasonably close approximation. The CFD simulation still under predicted the drag reduction, with an average error of 30%.

The drag reduction for the 25° model is not large enough to produce a net power saving and the 35° model also does not produce a power saving. The CFD simulation with a much larger SJA actuation frequency was able to provide a decent agreement with the drag changes experienced by the 25° (20% error) and 35° (2% error) Ahmed bodies, suggesting that the CFD simulations can be trusted to provide realistic SJA simulations. These Ahmed bodies were tested with the jet pitch angle directed into the wake to see if the drag reductions were similar to the 2D results or improved upon the normal blowing.

The SJA setup explored as part of this research uses the jets directed along the top of the wake to attempt to delay separation to deflect the shear layer into the wake and reduce the size of the wake, thereby reducing the drag that the body experiences. This setup was applied to the 25° Ahmed body to compare the effectiveness to the setup from Park [37]. It was expected that the 25°

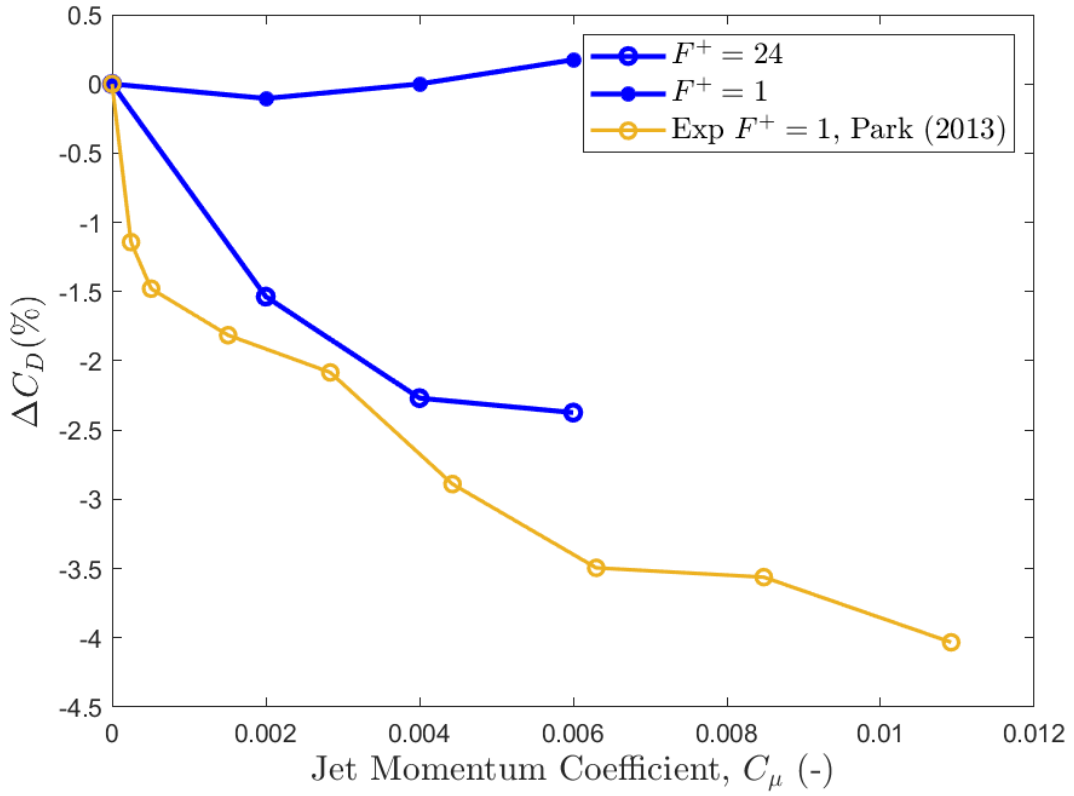


Figure 6.16: C_D reduction for the 25° Ahmed body with SJAs acting normal to the rear slant for the CFD simulation and the experimental C_D [37].

Ahmed body would not really benefit from the proposed setup.

The majority of the drag experienced by the 25° Ahmed body is pressure drag. The CFD simulations show that 83% of the total drag is pressure drag, with the remaining 17% due to skin friction drag. This is similar to the drag breakdown presented by Ahmed [6], where an approximate skin friction drag contribution of 20% was shown. In fact, it was shown that the skin friction contribution becomes more important as the angle of the rear slant increases. This is because approximately 66% of the total pressure drag is caused by the slanted face at the rear of the body. The expectation of this setup was that there would be minimal changes in the wake height and therefore minimal changes to the overall drag, which proved to be the case. There was also limited scope for delaying the flow separation for the CFD simulation of the 25° Ahmed body, due to the lack of a separation at the top of the rear slant, so this may prove more effective with a setup that is able to capture the initial flow separation.

The top image in Figure 6.17 shows the velocity contour plot in the centreplane of the 25° Ahmed body over the rear slant for the baseline case with no jet actuation. It appears that there is a small separation region in this centreplane, where the C-pillar vortices have the least effect on the flow. This is indicated by the thin blue contour along the length of the rear slant. It was noted by Zhang [8] that there should be a small separation region on the rear slant for the 25° Ahmed body with a reattachment further down the rear slant. However, this region is just a reduction in the flow velocity, rather than a separated flow region. The numerical model does not predict a separation here because a turbulence model is used, which over predicts the turbulence just before the rear slant. A higher turbulence leads to a delayed flow separation and in this case, no separation over the rear slant.

It can also be seen that there is a spike in the velocity flowing over the top of the rear slant, indicated by the orange region at the apex of the slant. Comparing this contour plot to the one for the case with jet actuation at $c_\mu = 2.5 \times 10^{-3}$ seen in the bottom image in Figure 6.17, it is clear that the velocity spike at the apex of the rear slant is much larger under actuation. This is similar to the 2D case, where the SJAs induce a larger velocity over the top of the rear slant. It is also clear that the blue contour along the rear slant has reduced in size compared to the baseline case, suggesting that the flow over the rear slant is faster under actuation. This is a direct result of the induced higher velocity over the top of the rear slant.

The increased velocity over the top of the rear slant is seen across the spanwise width of the body, which leads to an increased velocity on the slant across the width of the body. The increase is smaller at the sides of the rear slant where the C-pillar vortices have a greater control of the flow in the wake. Any changes to the C-pillar vortex behaviour can have a detrimental effect on the overall flow.

Figure 6.18 shows the pressure coefficients over the rear slant under actuation. The increased velocity over the top of the slant produces a slight drop in the

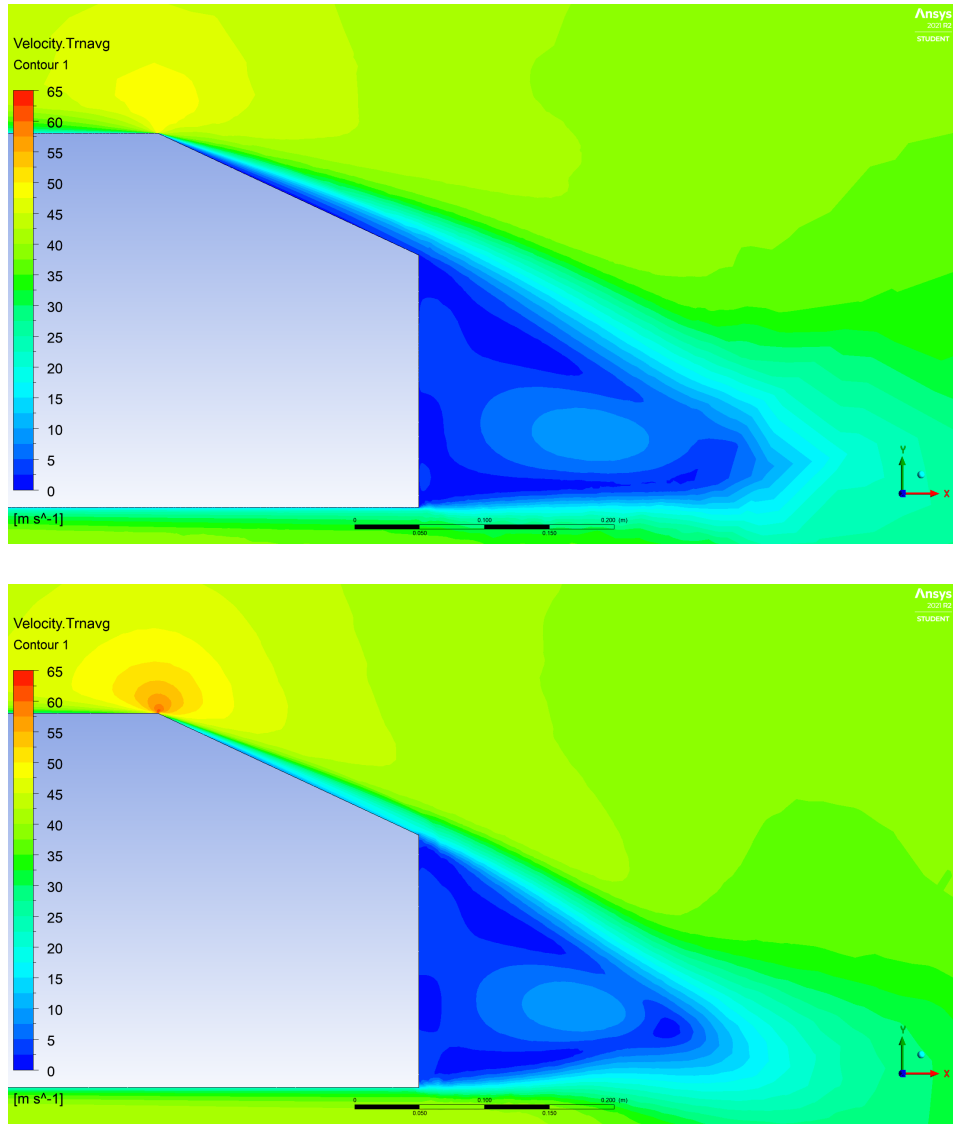


Figure 6.17: Velocity contour in the centreplane of the 25° Ahmed body with no actuation (top) and with synthetic jet actuation with $c_\mu = 2.5 \times 10^{-3}$ (bottom).

pressure coefficient near the top of the model, but the faster velocity leads to a pressure increase on the lower section of the rear slant. The effects on pressure are minor leading to the very small drop in drag of 0.5%.

The C-pillar vortices account for approximately 10% of the total pressure drag experienced by the 25° model. This is increased to 12% for the actuated flow, demonstrating that the SJAs are having a negative impact on the C-pillar vortices causing the vortices to have an increased contribution to the pressure drag on the body. The study conducted by Zhang [8] used steady jets located at the side of the rear slants to attempt to control the C-pillar vortices. This

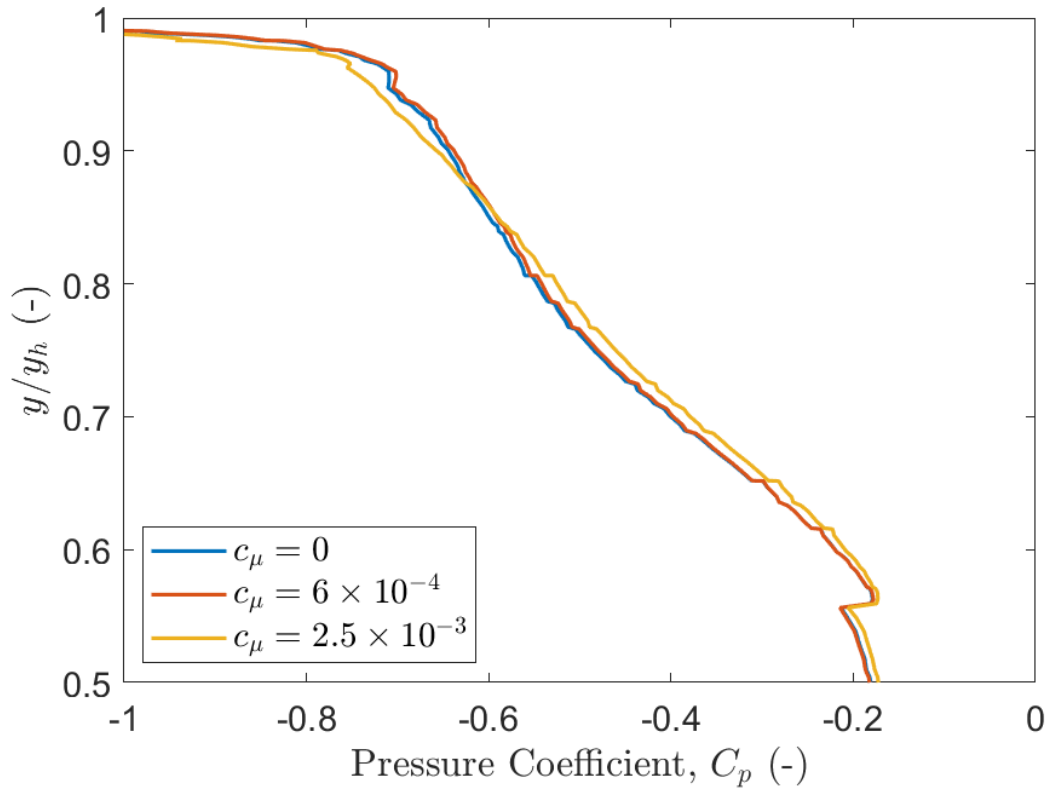


Figure 6.18: A plot of the pressure coefficient on the centre line of the rear face of the 25° Ahmed body.

was reported to have been successful, when used in conjunction with other steady jets located around the rear faces of the Ahmed body. It is possible that by implementing SJAs at the side of the rear slants, the C-pillar vortices could be controlled to reduce the drag for the 25° Ahmed body. However, for the setup tested in this research, the SJAs only seem to have a detrimental effect on the C-pillar vortices.

The results show that the proposed setup is not appropriate for the 25° Ahmed body as the jets cannot effect the key areas of the flow in the same way that they can for the 35° Ahmed body. There is less scope for delaying the separation for the 25° Ahmed body as the flow does not have a large separation, suggesting that using a jet pitch angle normal to the rear slant is better here. It is expected that the proposed setup will have a much greater impact for the 35° Ahmed body as the flow separates immediately from the top of the rear slant, meaning that the jets have a much larger scope to deflect the shear layer and reduce the wake height. The C-pillar vortices are also much weaker for

the 35° Ahmed body, suggesting that there will be a smaller detrimental effect from the SJAs interacting with them.

6.1.4 Controlling the Wake of a 35° 3D Ahmed Body Using Synthetic Jet Actuators

Littlewood [34] and Zhang [35] use steady jet actuators blowing into the wake to reduce the drag of the 35° Ahmed body. This study attempts to reduce the drag by using SJAs directed into the wake. The steady jets deflect the shear layer into the wake to reduce the size of the wake, increasing the pressure recovery on the rear faces and thereby reducing the drag on the body. The expectation is that the SJAs can have a similar effect on the wake and achieve a similar drag reduction. The 2D simulations showed that this approach works when there are no 3D effects to consider and led to a drag reduction of upto 20%. However, the 3D simulation does not demonstrate this same behaviour. Table 6.2 shows the drag coefficient for the 35° Ahmed body for different momentum coefficients. The momentum coefficients are the total momentum coefficients for the entire SJA array. The drag coefficients are determined by taking a time average of the drag once the flow has settled to an equilibrium state.

Table 6.2: *The drag coefficient for the 25° and 35° Ahmed bodies, taken from Ahmed [6] and recorded as part of this CFD study.*

	No Jet	$c_\mu = 6 \times 10^{-4}$	$c_\mu = 2.5 \times 10^{-3}$
3D CFD Simulations	0.265	0.270	0.275

With no jet actuation, the body has a base drag coefficient of approximately 0.265. This is in line with the results presented by Ahmed [6] for the 35° Ahmed body. With the jets turned on and a momentum coefficient of $c_\mu = 6 \times 10^{-4}$, the drag coefficient increases by almost 2% to 0.270. By increasing the momentum coefficient to $c_\mu = 2.5 \times 10^{-3}$, the drag coefficient increases by a further 2% to 0.275. This is a stark contrast to the 2D case, suggesting that the 3D effects are having a large impact on the flow dynamics at the rear of the body.

To investigate the impact of the jets on the surrounding flow, the velocity fields in the wake and behind the model were studied. Figure 6.19 shows the velocity contours over the rear slant for the case with no actuation and the case with $c_\mu = 2.5 \times 10^{-3}$ respectively. The blue contours in the top image in Figure 6.19 depict the wake region behind the model, with the darker blue regions showing the slowest moving flow occurs at the top and bottom edges of the wake. The lighter blue contours indicate an increase in the velocity in the centre of the wake, but this is due to the recirculating flow being directed through this region.

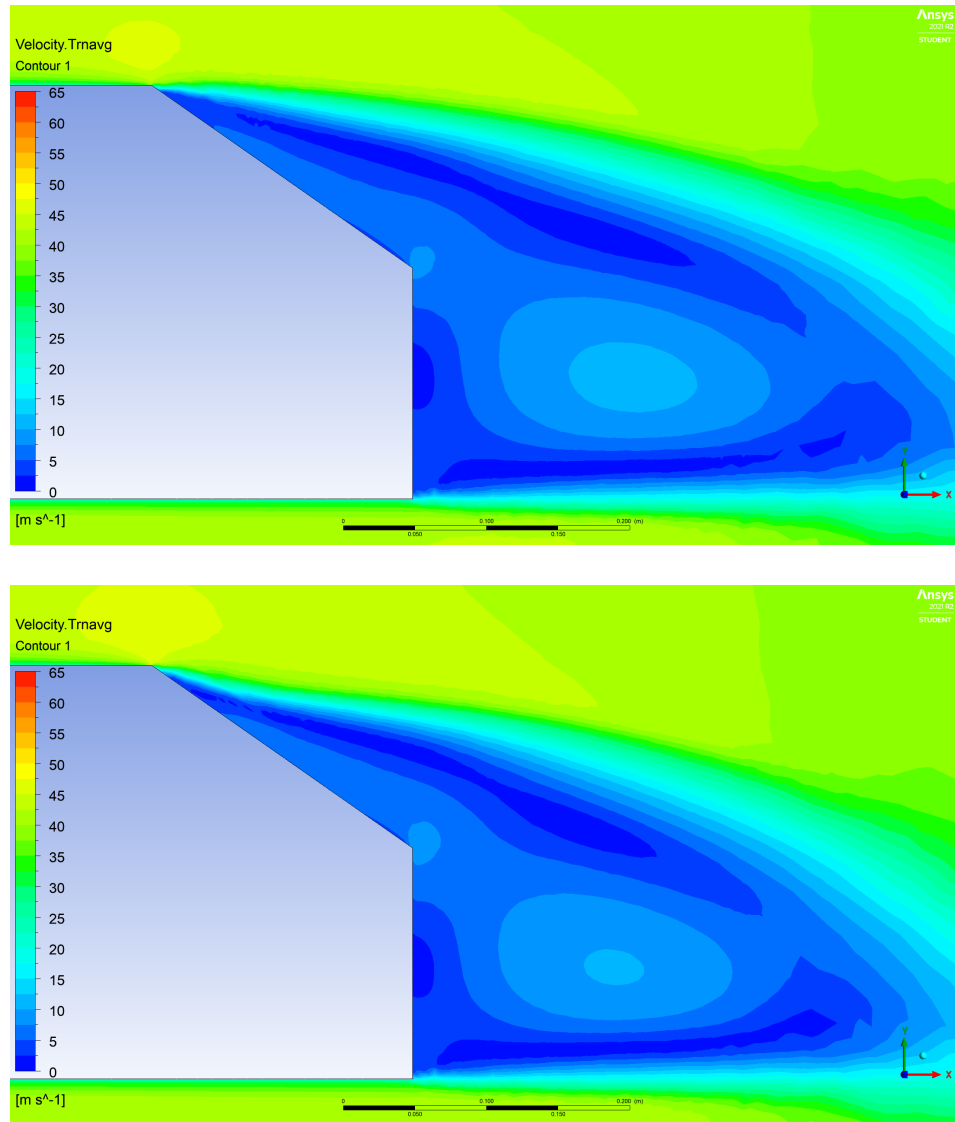


Figure 6.19: Velocity contour in the centreplane of the 35° Ahmed body with no actuation, $c_\mu = 0$ (top) and with synthetic jet actuation with $c_\mu = 2.5 \times 10^{-3}$ (bottom).

The bottom image in Figure 6.19 shows similar dark blue regions at the top and bottom of the wake. However, the light blue region is much smaller in the actuated case. This is due to the decreased wake size, restricting the recirculating flow in the centre of the wake. The wake size reduction can be seen at the top of the rear slant, where there is a clear difference in the shape of the flow as it separates from the body. This is due to the SJAs interacting with the flow in the top of the wake and deflecting the flow over the top of the rear slant into the wake. It can be seen that the yellow contour at the top of the rear slant increases in size for the actuated case, showing that the jets induce a higher flow velocity over the body, similar to the 2D case. This leads to a reduction in the length and height of the wake. This can be seen in Figure 6.20, which shows the contours of the difference in flow velocity between the 2 cases.

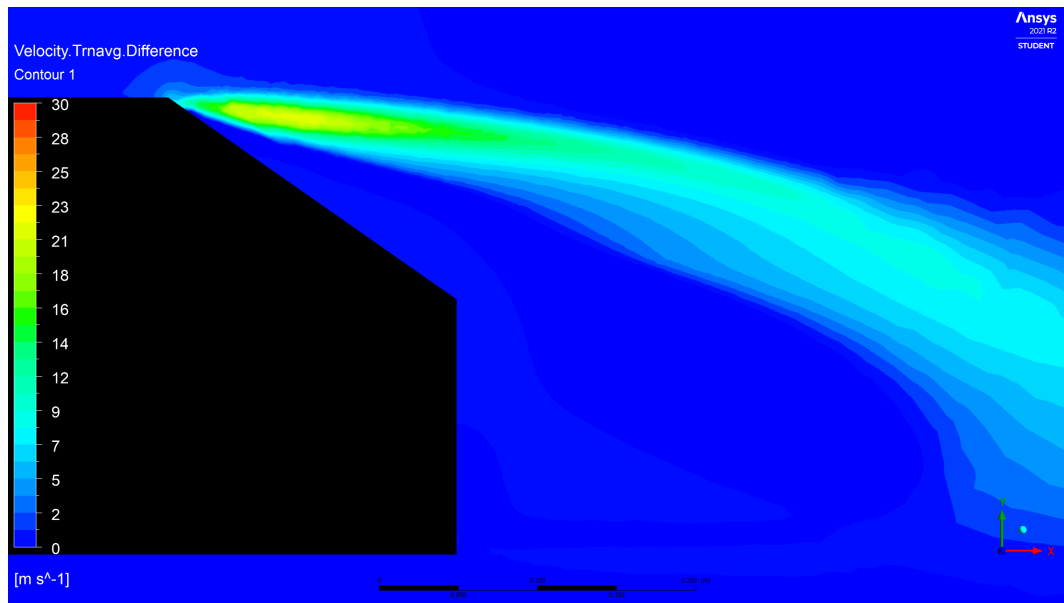


Figure 6.20: Contour of the differences in velocity for the cases in Figure 6.19.

A positive contour here, indicates a larger velocity in the actuated case. It is clear that there is an increase in the velocity at the top of the wake, showing that the height of the wake has been deflected down and the freestream flow is acting at a lower height. It is also clear to see that the jets are able to affect the whole length of the wake, even if the effect is diminished further away from

the body. It is also clear from this image that the reduction in wake height is actually an overall size reduction of the wake as the bottom of the wake is unaffected by the actuation. The width of the wake is also slightly decreased, resulting in the wake size reduction.

Since the base of the wake is unaffected, further flow control could be applied here to improve the drag reduction further. Zhang [8] used steady jets located at the top, bottom and sides of the rear faces to control the wake. To apply the same SJA setup, used at the top of the rear slant, at the bottom of the rear face, a small slant would need to be added, similar to a diffuser, as the current setup has not been tested on a squareback body. The slant could be much shorter than the slant at the top of the body, which may reduce C-pillar vortex interactions whilst still allowing for a separation delay. Other issues to consider would be the ground clearance and any objects that obstruct the flow (e.g. struts/ wheels).

To further quantify this wake height reduction and add the $c_\mu = 6 \times 10^{-4}$ case into the comparison, the velocity profiles over the wake are shown in Figure 6.21. The main result from this Figure, is that as the momentum coefficient is increased, the height of the wake is decreased.

The effects of the jets can first be seen in the profile at $x/x_h = -0.17$, which shows a very small increase in the flow velocity. This is also reflected in Figure 6.22 which shows the subsequent percentage changes between the profiles as the momentum coefficient is increased. The formula used to calculate this percentage change is Equation (4.1), which was used to compute the error between the velocity profile experimental data and CFD data. The increase in the flow velocity at $x/x_h = -0.17$ is almost 3% in each case, showing that the SJAs are inducing a small velocity upstream on the SJA location. In the immediate vicinity downstream of the SJAs, the change in the profiles is the greatest, with a peak of 11% between the profiles at $c_\mu = 0$ and 6×10^{-4} . The difference in the profiles for $c_\mu = 6 \times 10^{-4}$ and 2.5×10^{-3} only peaks at 5%, which suggests that the SJAs have a diminishing effect as the momentum coefficient

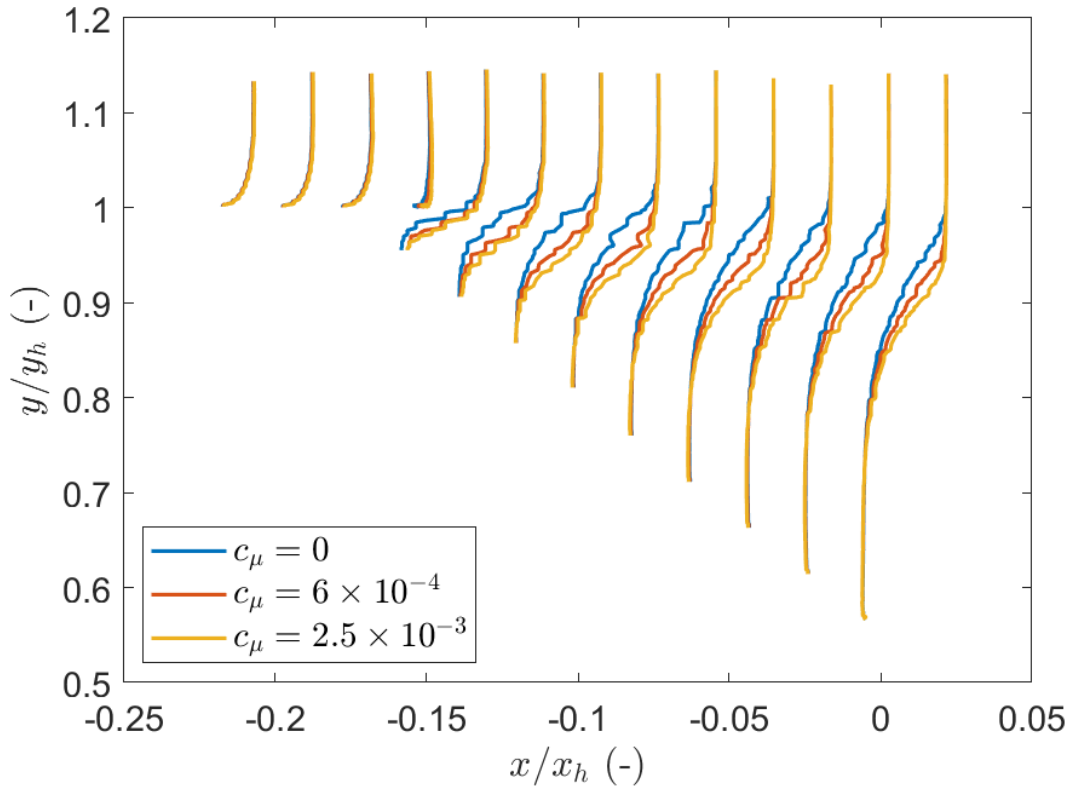


Figure 6.21: Velocity profiles in the centreplane of the rear slant for the 35° Ahmed body, showing the effects of actuation for $c_\mu = 0, 6 \times 10^{-4}$ and 2.5×10^{-3} .

is increased. This maximum change corresponds to the plots in Figure 6.21, where the greatest changes in the profiles occur between $x/x_h = -0.1$ and -0.15 .

These results show that the SJAs are behaving as expected, by interacting with the flow and reducing the wake size in the centreplane. To fully understand the flow, two off-centre planes were also considered at $z/z_h = 0.25$ and 0.04 , where $z/z_h = 0.5$ is the centreplane and $z/z_h = 0$ is the right hand side of the model.

Figure 6.23 shows the velocity contour for the baseline case with no actuation. The structure of the wake is a little different here compared with Figure 6.19, as the height of the wake drops off quicker after the body. This suggests that the height reduction in the wake will be less significant than in the centre plane. Figure 6.23 also shows the actuated case with $c_\mu = 2.5 \times 10^{-3}$ and indeed the height reduction is less significant. This is backed up by the mean difference between the cases for $c_\mu = 0$ and 6×10^{-4} being 2.14% with a peak

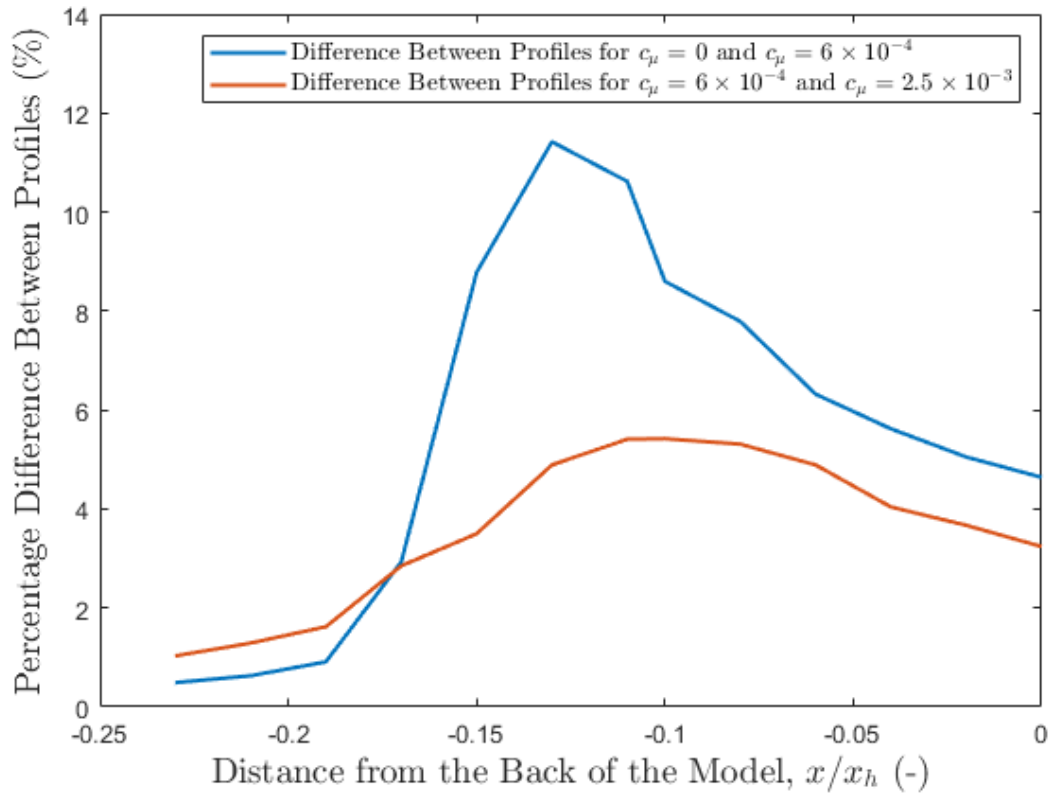


Figure 6.22: Percentage differences between the velocity profiles for the actuated flow for $c_\mu = 0, 6 \times 10^{-4}$ and 2.5×10^{-3} .

of 3.42% and the mean difference for $c_\mu = 6 \times 10^{-4}$ and 2.5×10^{-3} being 2.35% with a peak of 3%.

A similar trend is seen for the plane $z/z_h = 0.04$, where the wake in the baseline case drops off even sooner after the body as can be seen in Figure 6.24. In fact, the wake region here is dictated by the location of the C-pillar vortex located at the edge of the body. The C-pillar vortex accelerates the flow closer to the body causing the wake height to naturally be reduced. This means that the SJAs have a minimal effect on the wake height. Figure 6.25 shows the velocity profiles at $x/x_h = -0.06$. The profiles have a much changed structure than those in Figure 6.25, due to the presence of the C-pillar vortex.

The SJAs are located at $y/y_h = 0.996$, which is where the initial decrease in velocity begins. The initial decrease in velocity, between $y/y_h = 1$ and $y/y_h = 0.9$ is the effect of the C-pillar vortex and it is clear that the SJAs are unable to effect the flow in a meaningful way. The secondary decrease in the flow velocity, below $y/y_h = 0.9$ shows that the SJAs with the larger momentum

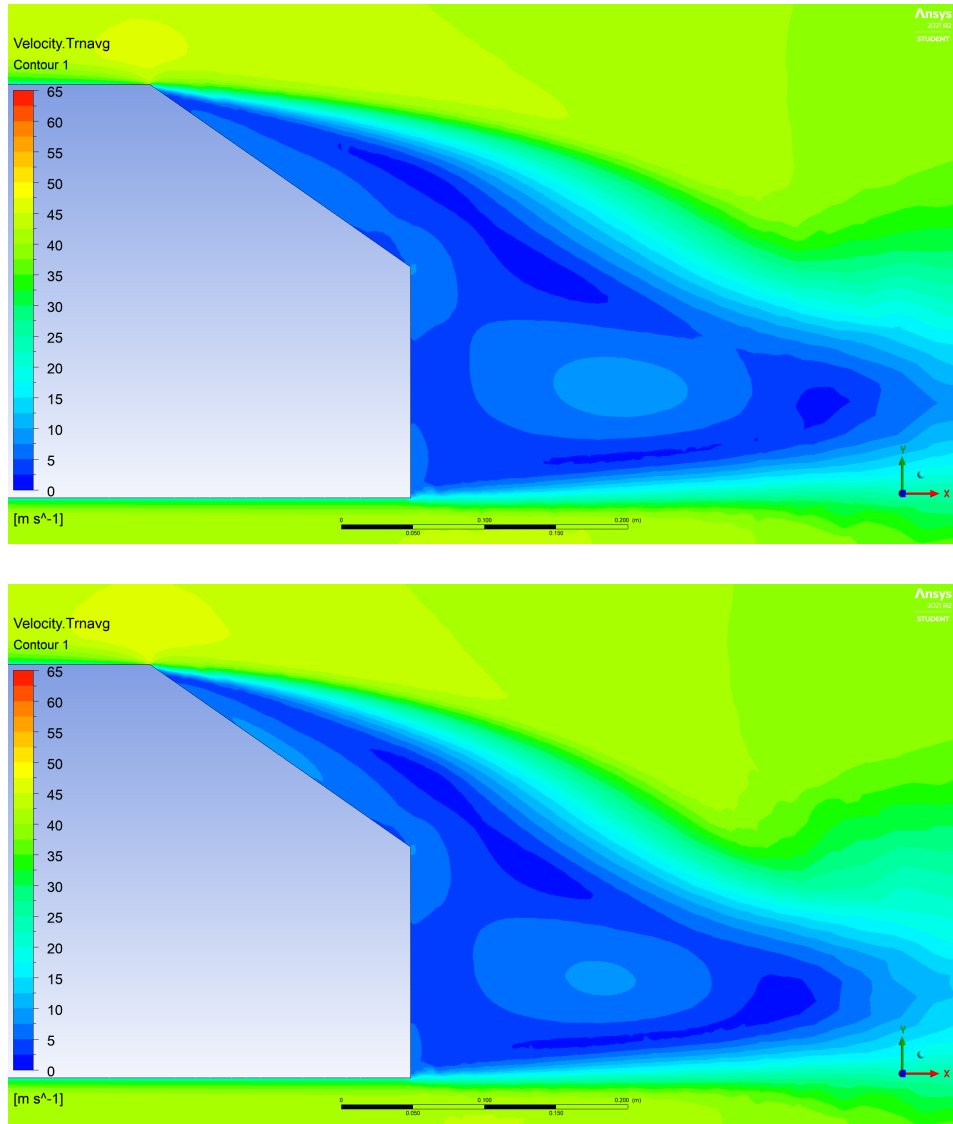


Figure 6.23: Velocity contour in the plane, $z/z_h = 0.25$, of the 35° Ahmed body with no actuation, $c_\mu = 0$ (top) and with synthetic jet actuation with $c_\mu = 2.5 \times 10^{-3}$ (bottom).

coefficient are able to reduce the height of the recirculating region on the slant. This is represented in the mean differences in the profiles, with the difference between the cases for $c_\mu = 0$ and 6×10^{-4} being 2.24% with a peak of 3.56% and the mean difference for $c_\mu = 6 \times 10^{-4}$ and 2.5×10^{-3} being 4.43% with a peak of 7.42%. The peaks in both cases are located much further down the rear slant than previously, showing that the C-pillar vortices dominate the early development of the flow at the top of the rear slant, but the SJAs begin to have a small effect further downstream, once the effect of the C-pillar vortices is diminished. This would also explain the higher momentum coefficients causing

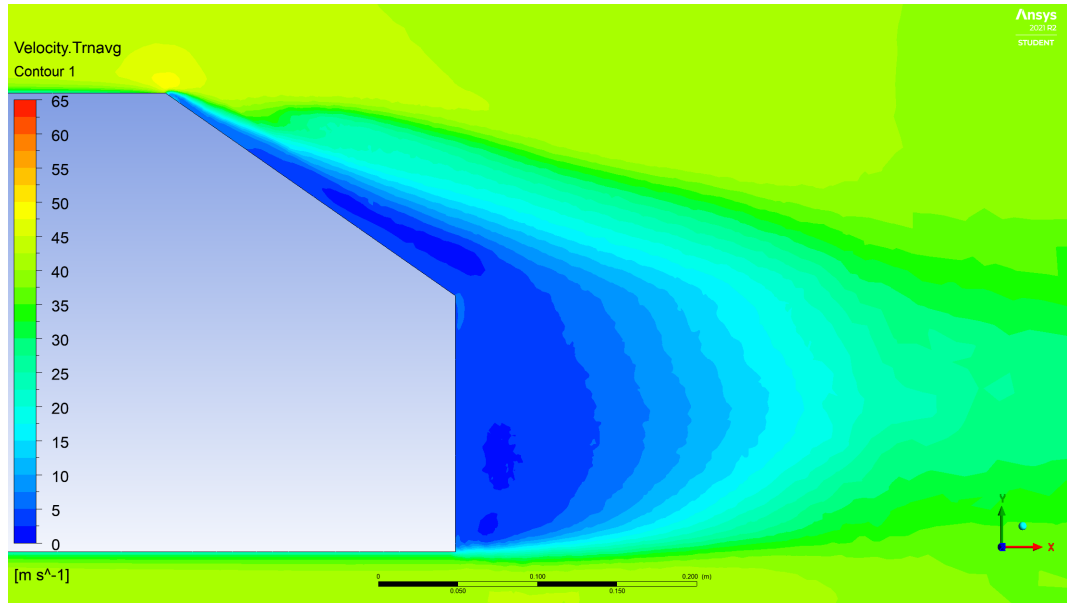


Figure 6.24: Velocity contour in the plane, $z/z_h = 0.04$, of the 35° Ahmed body with no actuation.

larger effects as the higher speeds allow the effects to propagate further down the rear slant.

To further illustrate the wake size reduction across the spanwise direction of the body, Figure 6.26 shows the top down view of the plane at $y/y_h = 0.6$, which is part way down the rear slant. The contour in this figure shows the difference between the velocity contours of the baseline case and the case with actuation at $c_\mu = 2.5 \times 10^{-3}$. The positive differences indicate a larger velocity for the actuated case. It is clear that the flow in this plane is increased across the width of the body as was demonstrated previously. One key point to notice here is the flow velocity increases around the C-pillar vortex location. This was shown previously in Figure 6.21, where the SJAs had a minimal effect on the velocity of the C-pillar vortices, however, the jets are able to increase the flow in the area underneath the C-pillar vortices and actually has a small impact on the flow on the outside of the C-pillar vortices. This is important as the C-pillar vortices have a large impact on how the flow behaves in the wake and affecting this flow can have large implications for the rest of the flow. This can also be seen in Figure 6.26, which shows that there is minimal change to the velocity in the region of the C-pillar vortices (the blue region stretching from

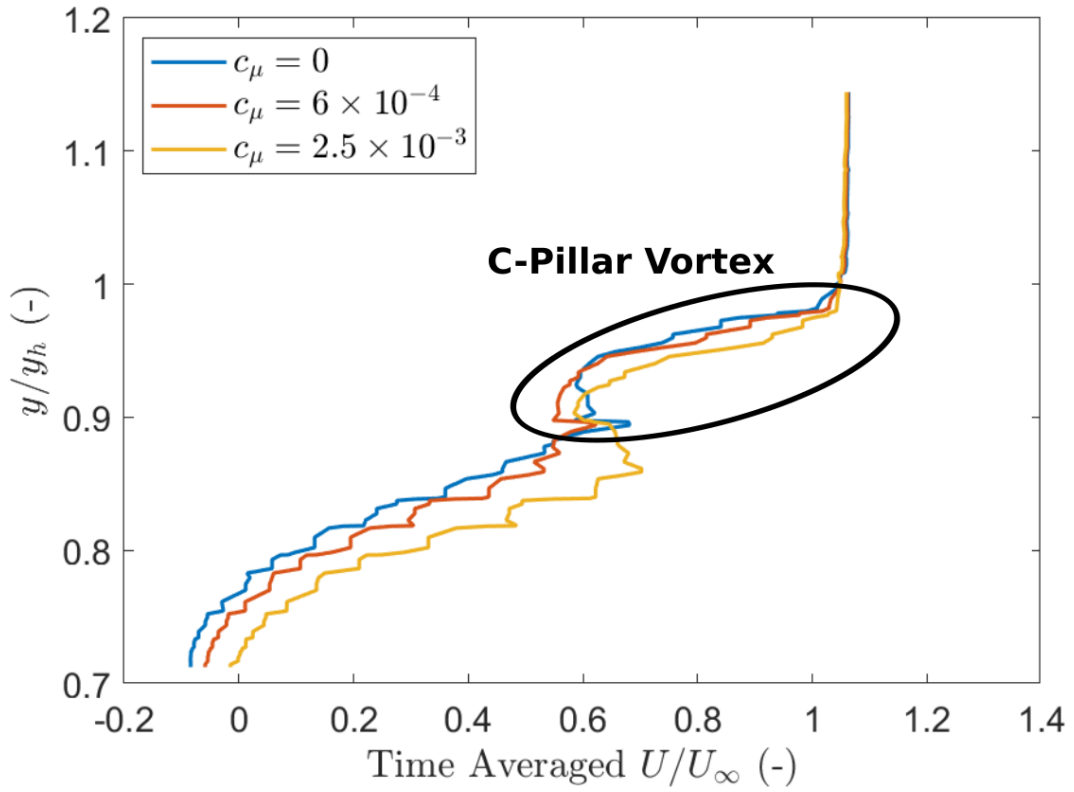


Figure 6.25: Velocity profiles in the plane, $z/z_h = 0.04$, of the rear slant for the 35° Ahmed body, showing the effects of actuation for $c_\mu = 0, 6 \times 10^{-4}$ and 2.5×10^{-3} .

just downstream of the rear corners of the body into the wake). To check the effects of the jets on the C-pillar vortices, the vortex circulation was measured before and after the jet actuation was applied. For $c_\mu = 2.5 \times 10^{-3}$, the vortex circulation increased by 0.5%, suggesting that the jets have a very small effect on the C-pillar vortices and this does not have any meaningful effect on the drag that the body experiences.

Figure 6.27 shows the pressure coefficient on the centre line of the rear faces of the Ahmed body. The 3 lines represent the unactuated case and the body under actuation with $c_\mu = 6 \times 10^{-4}$ and 2.5×10^{-3} . The C_p over the rear face ($y/y_h = 0.55$ and below) is similar across all cases. There is a negative peak in C_p at $y/y_h = 0.55$, where the rear slant ends. The changes appear above here, with a small decrease in the pressure coefficient at the top of the rear slant for higher actuation velocities. This would indicate that the SJAs are causing a drop in the base pressure on the rear slant. Therefore, despite the reduction in the wake, the drag coefficient would increase.

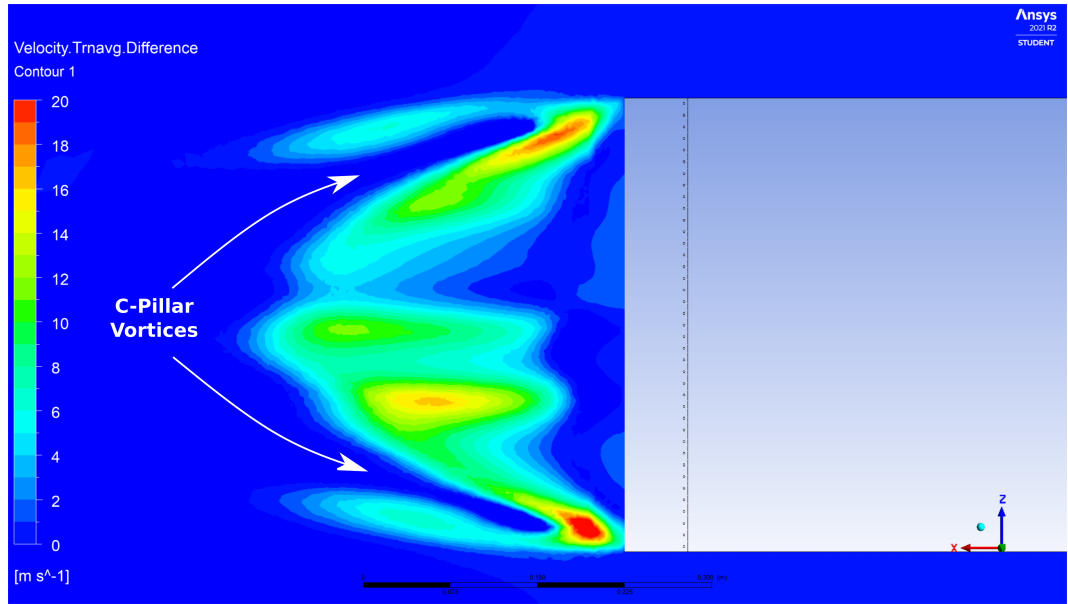


Figure 6.26: Velocity contour showing the differences in the velocity between the case with $c_\mu = 0$ and 2.5×10^{-3} in the horizontal plane $y/y_h = 0.6$.

The changes to the flow behaviour caused by the SJAs acting behind the 35° Ahmed body all suggest that the body should experience a drop in the drag coefficient, however, the drop in pressure on the rear slant causes an increase in the drag. The only explanation for this is the higher induced velocity over the top of the rear slant causes the drop in the pressure on the rear slant, which the delay in flow separation is unable to overcome.

6.1.5 Inverting the Aspect Ratio of the 35° Rear Slant Ahmed Body

The ultimate aim of this research is to be applied to road vehicles, mainly trucks, for which the Ahmed body is not the most appropriate choice. This is largely due to the differences in the aspect ratio and the shape of the rear face. A standard truck used on UK roads is much taller than it is wide, whilst the Ahmed body is the opposite. A new model was developed that flipped the width and height dimensions of the 35° Ahmed body to give a more representable aspect ratio to a truck. A 35° rear slant was still used as it allowed the proposed SJA setup to be used and the effects of the aspect ratio

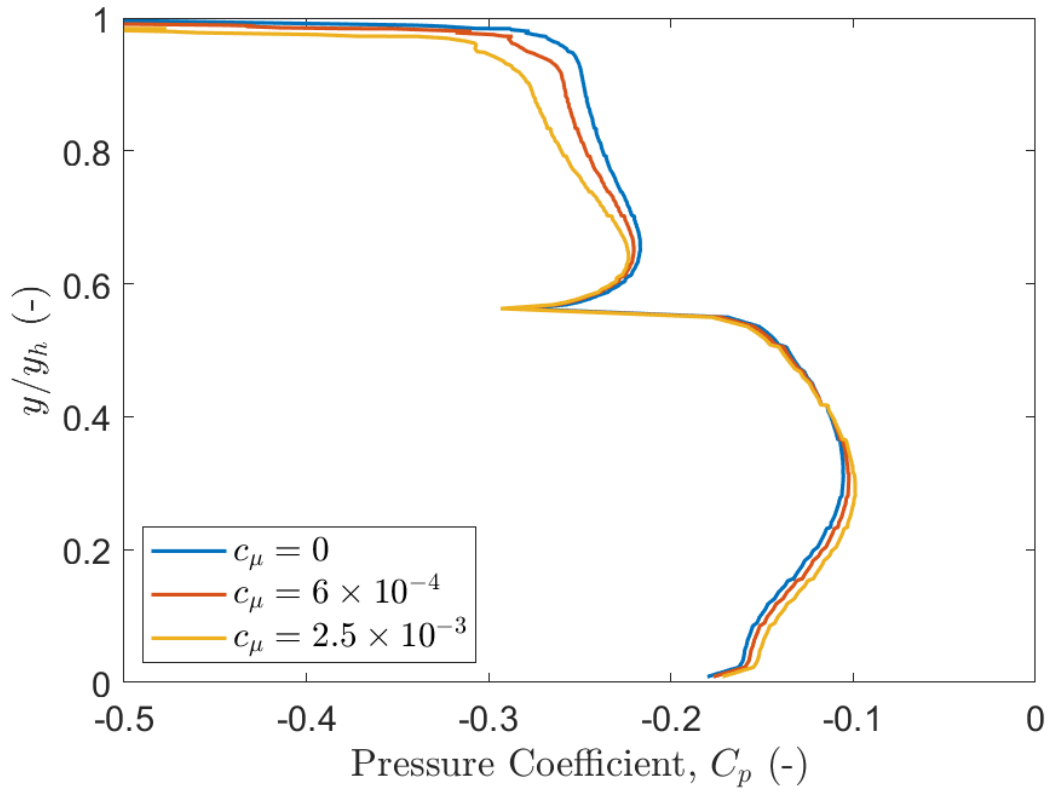


Figure 6.27: A plot of the pressure coefficient on the centre line of the rear face of the Ahmed body.

change be tested. This model was dubbed the Tall and Thin (TAT) model. There are a few differences in the wake structure and the flow around the body when compared with the standard Ahmed body. Among the differences is the drag coefficient which is much larger than for the standard Ahmed body, with a value of $C_D = 0.36$ for the baseline case with no jet actuation, compared with the $C_D = 0.26$ value obtained for the standard 35° Ahmed body. The cross-sectional areas of both models are the same, suggesting that the drag coefficient increase is caused by changes in the wake. In fact there is a lower pressure on the rear slant and an increase in the skin friction on the rear slant compared with the standard 35° Ahmed body. It was shown [16], that the narrower Ahmed body experiences a much larger circulation in the wake due to the C-Pillar vortices, which could explain the larger C_D value for the TAT body.

The wake structure can be seen in Figure 6.28 which shows the velocity contour plot in the centreplane over the rear slant for the TAT model. It is clear that

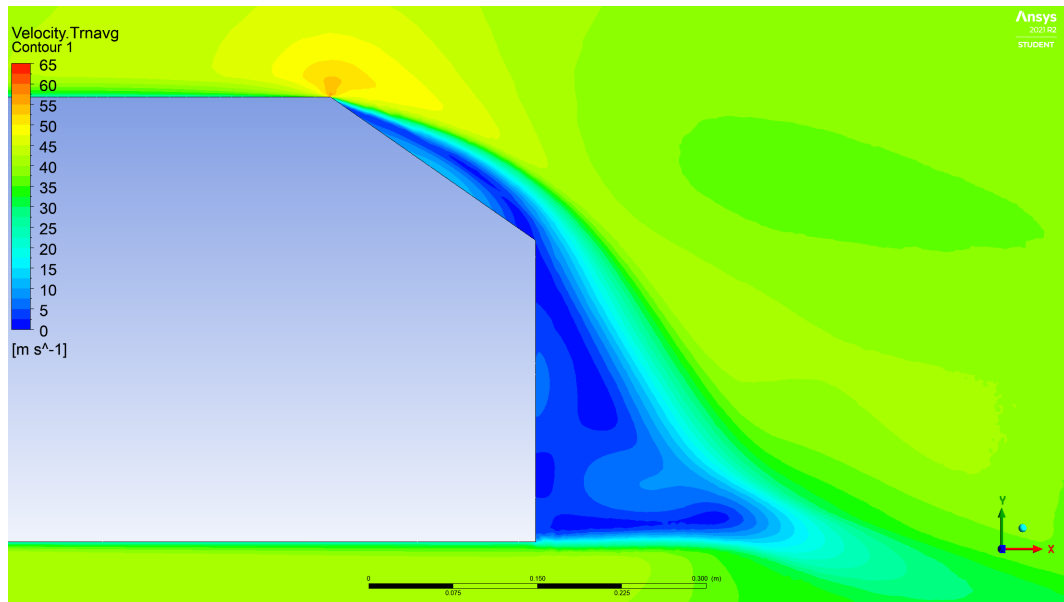


Figure 6.28: *Velocity contour in the centreplane of the TAT body with no actuation.*

the wake here is very different to the wake seen for the standard 35° Ahmed body. The flow separates at the top of the rear slant, but quickly reattaches to the rear slant, giving a small recirculation region, which has a much closer resemblance to the 25° Ahmed body than the 35° Ahmed body. This is likely due to the two C-pillar vortices being closer together and interacting with each other to redirect the flow onto the rear slant. This also explains the initial separation as the vortices are small at the top of the rear slant, but increase in diameter further down the slant leading to an increased authority over the flow. This separated region also explains the drop in pressure coefficient on the rear slant when compared to the standard Ahmed body. This separated region suggests that this body could benefit from the proposed setup, with the jets causing a delay in separation and thereby increasing the pressure on the rear slant.

The limited separation over the rear slant for the TAT model compared with the standard 35° Ahmed body also leads to a smaller wake behind the body. Again the wake structure more closely resembles the 25° Ahmed body. The reattachment of the flow also causes a larger skin friction drag component than the 35° Ahmed body, similar to the 25° Ahmed body. The proposed

setup would likely increase this skin friction drag component, however, unlike the standard 25° Ahmed body, the pressure drag decrease from the delayed separation for the TAT model is expected to be greater.

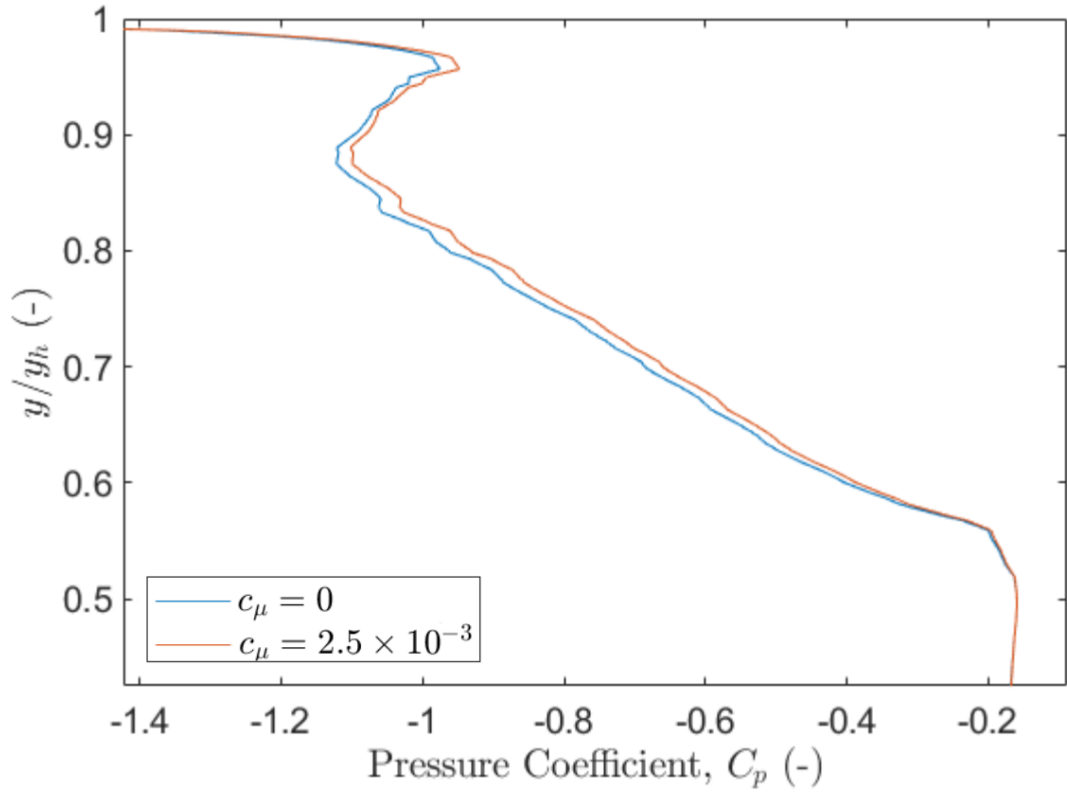


Figure 6.29: A plot of the pressure coefficient on the centre line of the rear face of the TAT body.

The base pressure drag that the TAT model experiences is larger than the pressure drag that the standard 35° Ahmed body experiences. This is evidenced from Figure 6.29 which shows the pressure coefficients on the rear face of the TAT model. There is a large drop in the pressure coefficient at $y/y_h = 0.85$ which is caused by the separated flow and the recirculating region. The body experiences a much more extreme pressure change on the rear face, more akin to the 25° Ahmed body, due to the structure of the wake, with the small difference of the pressure drop adding to the lower pressure.

Based on the comparisons to other models, it is expected that the behaviour of the SJAs will resemble the 25° Ahmed body, with the slight difference that the pressure drag reduction will be larger, due to the jets having more scope to delay the flow separation and deflect the shear layer, leading to a reduction

in the size of the recirculating region on the rear slant. However, this is not what was observed. Figure 6.30 shows the velocity contour plot for the TAT model with actuation at $c_\mu = 2.5 \times 10^{-3}$.

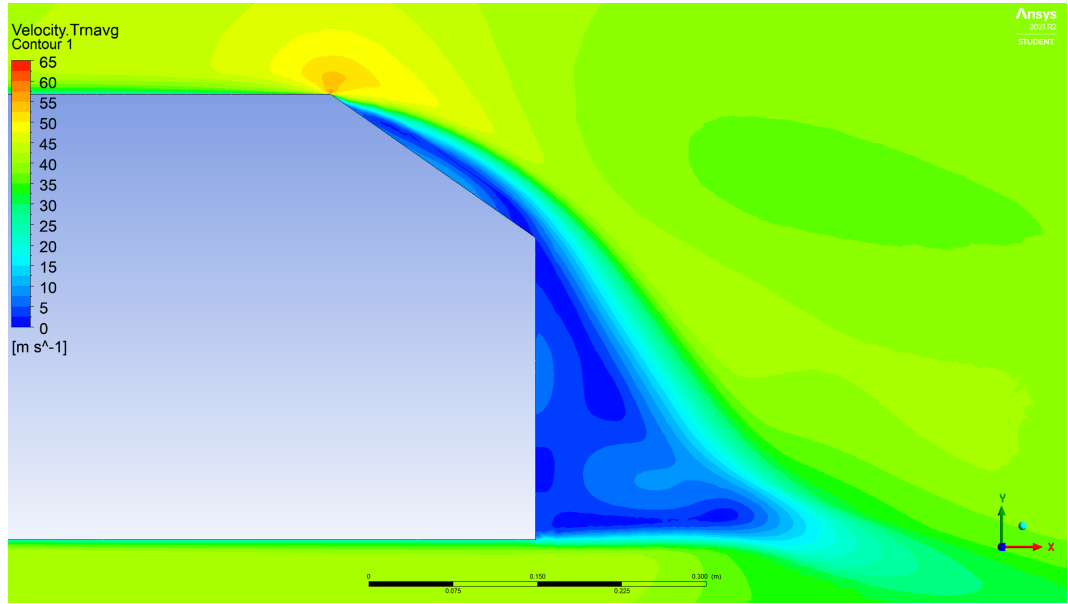


Figure 6.30: Velocity contour in the centreplane of the TAT body with synthetic jet actuation with $c_\mu = 2.5 \times 10^{-3}$.

The flow over the top of the rear slant is very similar to the unactuated case with no major induced flow increase. The recirculation region on the rear slant is also a similar size with similar flow characteristics under actuation. The only change to the flow structure that can be seen in these images is the small change to the shape, caused by a slight change to the flow separation at the top of the rear slant. There is a small shear layer deflection parallel to the flow, increasing the size of the initial recirculating region. This causes a steeper drop into the lower wake, slightly reducing the length and size of the wake.

Contrary to the standard Ahmed with the 25° and 35° rear slants, the SJAs in this case cause an increase in the overall size of the wake and recirculating region over the slant. This change leads to a small increase in the pressure on the rear slant and a decrease in the skin friction that the rear face experiences. Although both changes are minimal, both cause a small reduction in the drag coefficient and lead to a drop of 2% in the drag coefficient. Although this

decrease is only small, further tests could lead to a more optimised model and larger drag reductions. A modified jet pitch angle could also provide a benefit as the model is very sensitive to angle changes. Reducing the drag by elongating the wake and weakening the structures within is a method that has been shown to work with flaps [24], so it could be a viable approach here as well.

Again, the effects found in the centreplane are replicated across the width of the body, with the effectiveness reduced at the edges of the body where the C-pillar vortices are located.

6.1.6 Summary

It is clear that the 3D results from the simulations are not as promising as the 2D results. The SJAs behaved as expected in the cases using the standard Ahmed bodies, with a delay in separation and a reduction in the wake height, however, this did not lead to the drag reductions that were expected, due to the induced higher velocity causing a drop in the pressure on the rear slant, leading to an overall drag increase for the 35° Ahmed body. A drag reduction was achieved for the TAT model, by utilising the benefits of the proposed setup for the 35° Ahmed body and overcoming the issues that the setup had for the 25° Ahmed body. With a few optimisations, the drag reduction here could be improved and an overall power saving obtained.

6.2 3D Reduced Order Modelling

The model order reduction scheme developed in Chapter 5 was able to approximate a 2D system using a single mode obtained from a single input vector and the corresponding C_D output. The same approach was applied here for three 3D CFD models and an experimental model [67]. The 3D 35°, 25° and TAT models were all tested. Despite the less than ideal drag reductions that all models achieved, a similar relationship between the drag reduction and the

SJA velocity was present. These models use a fixed freestream velocity of 40m/s. Due to the increased computation power needed to run the 3D simulations compared with the 2D simulations, a reduced input map was used of 0m/s, 30m/s and 60m/s ($c_\mu = 0,6 \times 10^{-4}$ and 2.5×10^{-3} respectively).

Initially, the 3D 25° ROM was built using the same relationship as for the 2D ROMs, with the square of the input used, however, this did not fit with the model, so a small tweak was made to use the square root of the input instead. The change in input relationship was determined using the MATLAB curve fitting tool to get an approximate relationship before it was tested as part of the model. With the exception of this small tweak to the model, the process was the same and the results can be seen in Figure 6.31. Unlike the other 3D models, this simulation also includes a 90m/s ($c_\mu = 5.6 \times 10^{-3}$) input as well. The ROM is able to predict the average C_D reduction with a maximum error of 10%.

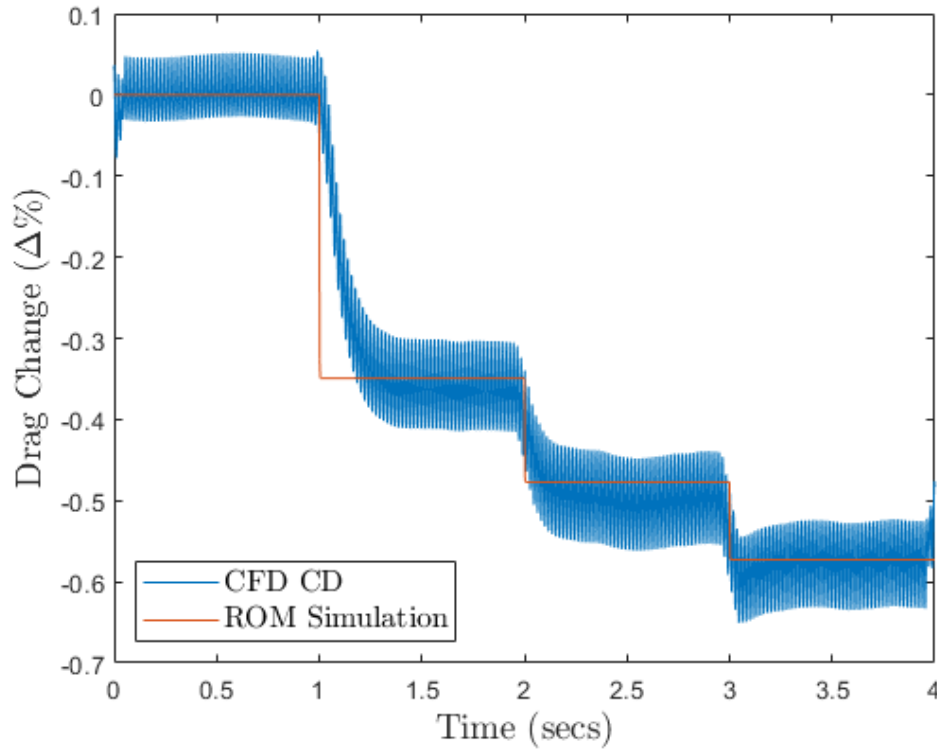


Figure 6.31: The simulation of the 1 mode model built using the C_D , compared to the C_D obtained from the CFD for the 3D 25° Ahmed body.

It was later found that the a small tweak to the exact relationship, α , was

needed for each model, with the relationship for the 3D models varying from 0.5 to 1. The ideal (no C-pillar vortices) case, such as the 2D case, uses a square relationship. It is thought that the change in relationship is caused by the decline in effectiveness of the jets at the spanwise edges of the model. In fact, the effectiveness for the jets in the 3D 35° Ahmed body is reduced proportional to the square of the distance from the centre. This results in a close to linear relationship for the 3D 35° model. The SJAs for the 25° model are largely ineffective in the current setup, whilst still affected by the C-pillar vortices and the reduction in jet effectiveness, which leads to a square root relationship. This is untested speculation and more work would need to be done to figure out the exact reasoning behind the changes in the relationship of the model.

The results for the 3D 35° ROM can be seen in Figure 6.32. This ROM used a linear map from the SJA velocity input to the C_D change output. The ROM accurately predicts the C_D change for an input of 0 and 30m/s ($c_\mu = 6 \times 10^{-4}$), but struggles a bit more with the approximation at 60m/s ($c_\mu = 2.5 \times 10^{-3}$). There appears to be a small instability in the CFD simulation between 2 and 3 seconds, which does not produce a stable average reading for the model to fit to, resulting in the error here. The simulation would need to be run for longer to provide a conclusive result of the relationship.

The TAT model was also tested and again a linear input map was used. The simulation results can be seen in Figure 6.33. The magnitude of the drag reduction was very small, however, the approximation is again very good, with a maximum error of 0.5%. The fact that the TAT model has a rear slant of 35° suggests that maybe the rear slant angle has an effect on the relationship between the SJA input and the C_D change.

These models demonstrate that the ROM built using the C_D gives very good approximations to the average C_D value obtained in these CFD simulations. All of these CFD systems used the same turbulence models and methods to simulate the results, therefore, to check that this model is not just contained

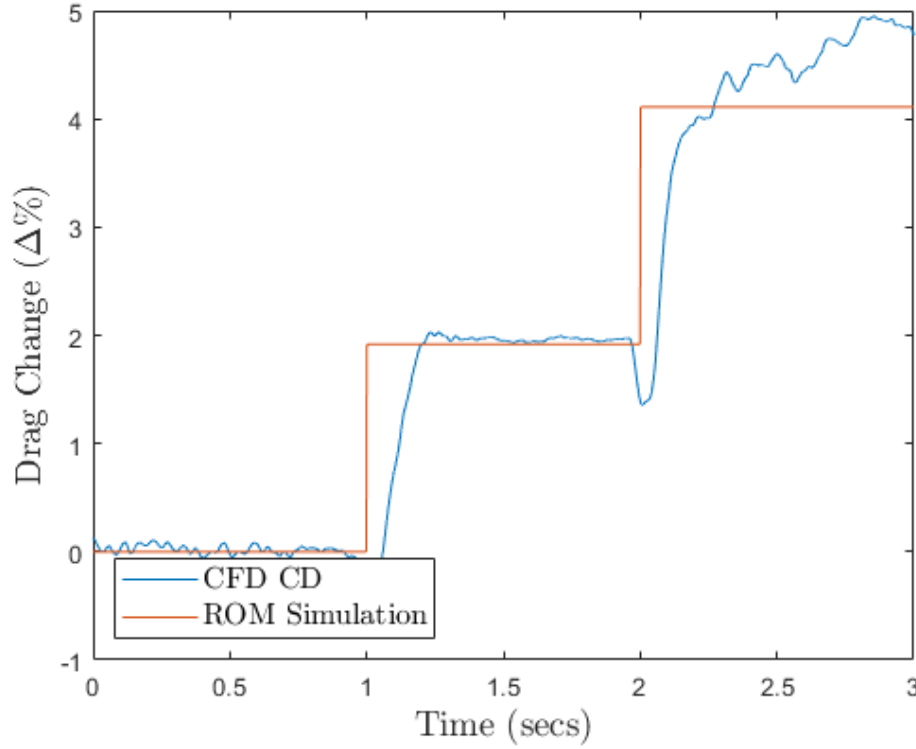


Figure 6.32: The simulation of the 1 mode model built using the C_D , compared to the C_D obtained from the CFD for the 3D 35° Ahmed body.

to these specific CFD simulations, a model was built using the C_D and SJA velocities from an experimental model [67]. Kourta [67] presents experimental results for the 25° Ahmed body with an SJA slot at the top of the rear slant. The Ahmed body used was a 70% scale of the full size model with a 25° rear slant. The freestream velocity was set to 40m/s. The momentum coefficients vary from 0 upto 4×10^{-4} . The drag reductions achieved in the paper are larger than the ones obtained in the CFD simulations for this research, but it was observed by Kourta et al. that there was a relationship between the momentum coefficient and the percentage drag reduction that they achieved. The data was extracted from the plots in their paper and a ROM was built using the data. The results in Figure 6.34 show their data alongside the ROM that was built for their model. The SJA input was replaced by $u^{0.4}$ for this model, which is close to the 0.5 used for the 3D 25° CFD simulation.

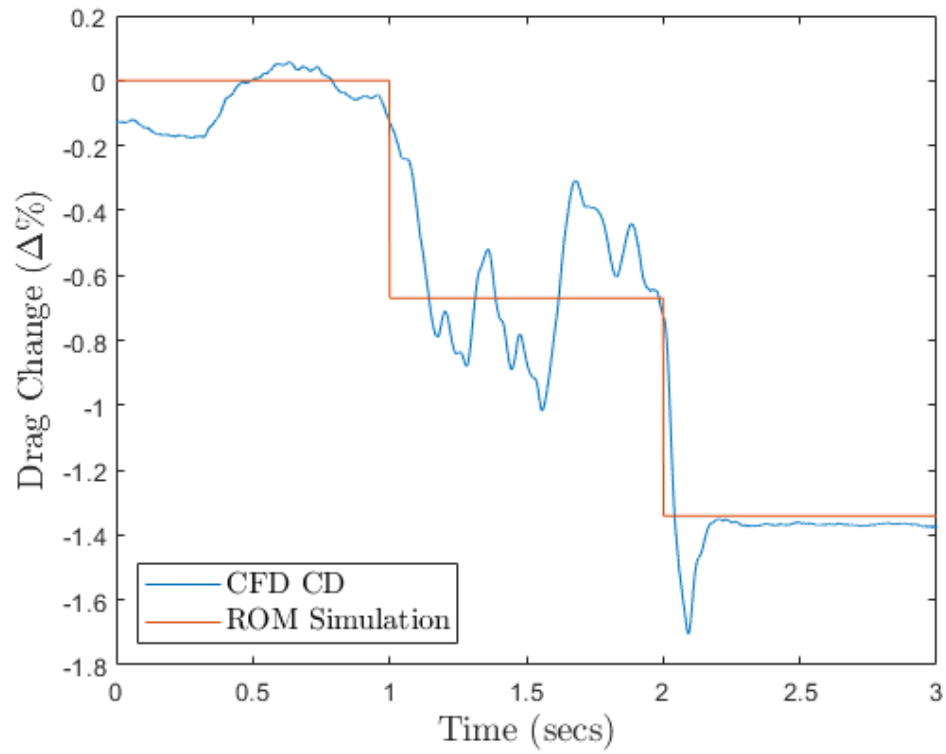


Figure 6.33: The simulation of the 1 mode model built using the C_D , compared to the C_D obtained from the CFD for the 3D TAT model.

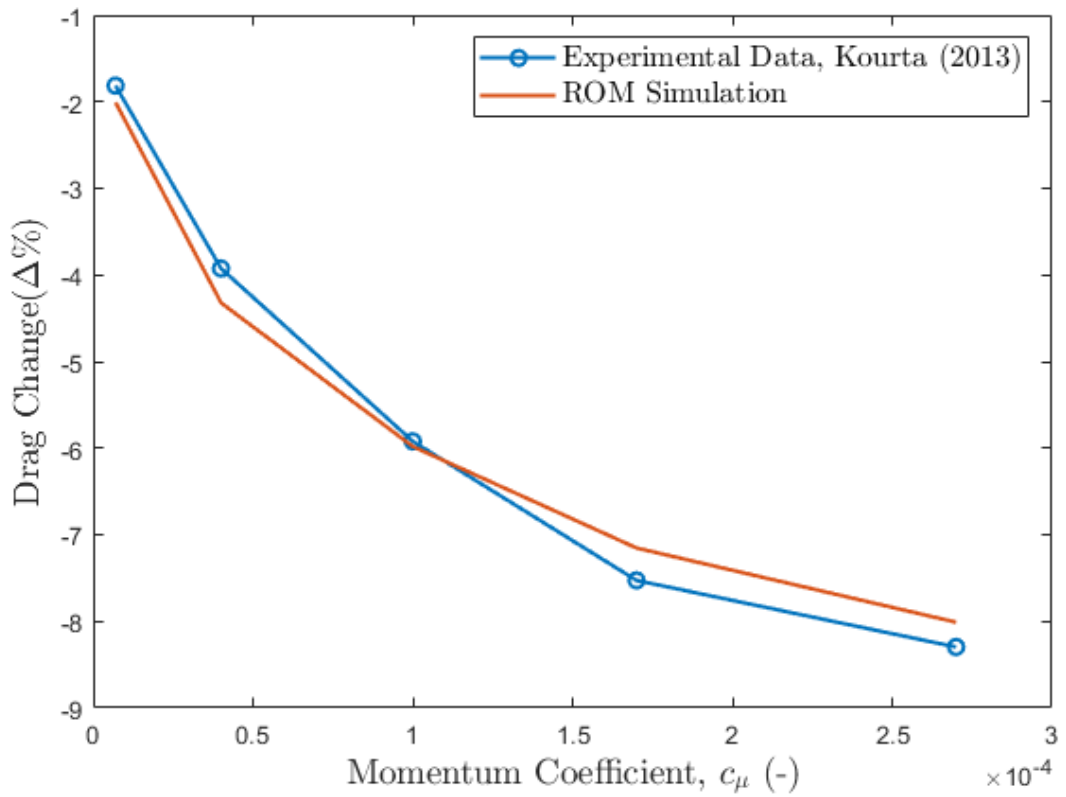


Figure 6.34: The simulation of the 1 mode model built using the C_D , compared to the C_D obtained from the experimental data obtained by Kourta [67].

The results show a very good approximation by the ROM, with an average error of 4.6%. This suggests that the 1 mode approximation to the system is viable for experimental data as well as the CFD simulations shown in this section.

6.2.1 Summary

It has been shown that the ROMs built in this section give a decent approximation to both CFD simulations and experimental data, with the maximum error in any of the models of 10%. This was deemed a sufficiently small error for the ROM as they are initially used to predict the C_D change before the Extremum Seeking Control (ESC) scheme takes over. Although the 3D ROMs produce a good approximation to the average drag in the systems and a drag reduction was observed for a couple of the models, there was no net power saving produced. The ROMs here are unable to reconstruct the pressure field, however, for real time control, the model produces a very useful approximation to the drag coefficient.

Chapter 7

Conclusion

7.1 Main Results

The aim of this research is to produce a novel control scheme, using numerical flow field information around a road vehicle, to produce a net energy saving for that road vehicle.

- **Objective 1:** *Produce a 2D CFD simulation of the 35° Ahmed body, combined with a synthetic jet actuator array to provide a drag reduction through control of the wake, with a view to producing a net power saving.*
 - The 2D CFD simulation for the 35° Ahmed body produced promising results for reducing the drag using SJAs directed into the wake. Maximum drag reductions of up to 19% were achieved at $Re = 2.9 \times 10^6$, with a maximum power saving of 2.6%. At smaller Reynolds numbers, the power saving increased to just over 5.5%.
- **Objective 2:** *Produce a low dimensional approximation of the CFD simulation, which can be used to predict the flow changes caused by operating the synthetic jet actuator array.*
 - A reduced order model (ROM) was developed and tested for the 2D CFD simulation of the 35° Ahmed body. In each case, a single mode was able to produce an approximation to the drag coefficient,

accurate to within 40%. This was deemed acceptable as this error was obtained for a jet velocity of 10m/s, where the prediction was less than 0.5%. The average error is only 30%. This causes the errors to be large, in fact, if the first few jet velocities are removed, the average error drops to 15%. If the drag coefficient is predicted, instead of the change in drag from the equilibrium, then the average error is only 1%.

- A Linear Parameter Varying (LPV) model was developed to allow the model to simulate the response to freestream velocity changes. The average error for this model was $\approx 10\%$.

- **Objective 3:** *Develop an extended Extremum Seeking Controller, using a low dimensional approximation to improve the performance of the Extremum Seeking Controller for use in real time control.*

- The ROM for the 2D CFD model was used to extend the behaviour of the Extremum Seeking Control (ESC) scheme. This allowed the ESC scheme to adapt to large, abrupt changes in the outer flow to aid in real time control.
- The new ROM used a simple approach; if a change in the freestream velocity was detected, the integrator block in the ESC scheme was reset and the LPV model provided a new approximation for the optimum operating conditions.
- The new Reduced Order Model Extremum Seeking Control (ROMESC) scheme was able to save 30% more power than the standard ESC scheme over a 30 second period.

- **Objective 4:** *Produce 3D CFD simulations of various Ahmed bodies, combined with actuator arrays to provide a drag reduction, with a view to producing a net power saving and produce a low dimensional approximation for each model.*

- The 3D CFD simulation of the 35° Ahmed body produced a small increase to the drag of 2%. This was attributed to the increase in flow velocity over the top of the rear slant producing a decrease in the static pressure on the rear slant. The SJAs were able to impact the flow, demonstrated by amplitude changes of the high frequency oscillations in the raw drag output. The wake height of the 3D model was also decreased by 2% but was not able to counteract the drag increase.
- The 3D CFD models for the 25° Ahmed body and the ‘Tall and Thin’ (TAT) model both produced small reductions in the drag of up to 3%, but neither produced a net power saving during their operation. The TAT model results were also consistent with the literature, demonstrating an increase in the base drag when compared to the standard Ahmed body aspect ratio. This was due to the decreased distance between the C-pillar vortices producing a stronger reattachment force for the separated flow on the slant. This caused the body to switch from the separated low drag regime to the high drag regime.
- A ROM was built for each of the 3D models, along with experimental measurements for the drag coefficient [67]. In each case, the approximations were accurate to within 10%. A smaller dataset was used to test the models, due to the long computation times needed to simulate the model, but the ROMs were able to capture the drag coefficient behaviour for the full size 3D models which used $\approx 4,000,000$ mesh elements.

7.2 Main Contributions

This thesis presents a novel approach to closed loop flow control of an Ahmed body using synthetic jet actuators as the control variable. This was accom-

plished using a mix of 2D and 3D CFD simulations, combined with a real time controller.

Active flow control using jets is a common practice for reducing the drag experienced by bluff bodies. The literature shows multiple examples of steady jet and synthetic jet actuators being used to reattach flow and increase the base pressure on the rear faces of a model. The steady jets use a variety of jet angles from blowing normal to the top of the body, to normal on the rear slant and finally, blowing tangential to the slant, into the wake. The examples using SJAs, typically use a reduced peak velocity when compared to steady jet blowing and only use normal to the top surface, or normal to the slant blowing. SJAs used in this way have been shown to be reasonably effective in reducing the drag on the bluff body, but the power usage by the jets results in a reduction in performance or little performance gain. Steady jets with blowing into the wake have also been shown to have large drag reductions, but the nature of steady jets results in a larger power draw and other issues such as needing onboard air supplies.

This research aimed to use a SJA array directed into the wake in an attempt to gain the larger drag reductions of the steady jets, whilst maintaining a lower power usage. By operating the SJAs at close to their resonant frequency, a large peak jet velocity could be produced and a shear layer deflection, similar to steady jet behaviour could be achieved. The 3D CFD simulations showed promise as a reduction in the wake size was experienced for both the 25° and 35° models as was hoped. However, a drop in the pressure on the rear face caused an increase in the drag coefficient. The 2D simulations showed the mechanism as was intended, with the wake size reduction leading to a pressure recovery on the rear slant and an overall drag reduction. This drag reduction, coupled with the reduced power usage of the SJAs compared with the steady jet, lead to an overall reduction in the power used for the 2D case. This allowed the 2D case to be used in a novel control scheme.

A reduced order modelling scheme was used to reduce the size of the 2D CFD

simulation. The scheme used was the IODMD scheme which has been used in the literature before to reduce small scale models to a smaller scale state space model. The implementation of the scheme here attempted to reduce a very large scale model, to a small scale model for use in real time control. There are very few cases of a model order reduction scheme being used to represent a large fluid dynamics scheme, and even fewer using the IODMD scheme. This research shows that the scheme can be used to capture key drag information from a CFD model and even further showed that the system identification approach in the scheme can be used to model the drag behaviour of the scheme with a single input of the jet velocity. This model produced an LPV representation of the drag behaviour which can be easily added to control schemes. This led to an extension of the ESC scheme being developed.

The ESC scheme used a simple approach, by predicting a more accurate initial value for the ESC scheme, the performance of the ESC scheme was greatly improved, producing a 33% greater power saving over a 30 second simulation than the standard ESC scheme. A better initial estimate led to an almost instant convergence time and the global optimum being achieved, due to the model knowing information about the entire control space. The implementation of the scheme also allows the extension of the scheme using a more complex gradient seeking behaviour, including an extension to multi input, multi output ESC schemes, assuming that an appropriate ROM is created. The model order reduction scheme could also be extended, using a similar approach that was used to add the freestream behaviour to the model.

7.3 Limitations and Future Studies

The main limitation of this work is the unexpected behaviour of the 3D CFD models, where the SJA interaction lead to an increase in the drag coefficient. This meant that most of the reduced order modelling and the ESC scheme were built using 2D data. This limited the extensions that could be made to

the reduced order model as other effects could not be analysed, such as yaw angle. The model could be extended to include the yaw angle of the bluff body with relative ease, which would allow the control scheme to adapt to changes in the direction of the wind flow as well as changes in the freestream velocity. This could also lead to a testing of SJA yaw angles for the purposes of controlling the flow.

If SJA yaw angles were taken into account, then a move to a multi variate ESC scheme could have been used, with the ROM producing the initial estimate for both the SJA peak jet velocity and the yaw angle. As for the ROM, the scheme only works for the current tested model. It was shown that there is experimental data which also satisfies the same relationship between the SJA peak jet velocity and the drag coefficient, however, there are also experiments that show a different relationship, for which a different model would need to be created. The model also only works for the range of SJA velocities and freestream velocities from which it was built, so further testing would be needed for the values outside of the test space. The data needed to build a model could be obtained quicker by using an experimental setup rather than a CFD setup, so it could be beneficial to test the effectiveness of the model on a wind tunnel simulation.

If a series of experiments could be done to complement this research, the recommendation, would be to focus on the 3D 35° Ahmed body as this is the body for which the setup was expected to produce the largest net power saving. The recommended SJA placement would be at the top of the rear slant as this is the key area targeted by this research with an aim to delaying separation and controlling the wake size. If a further experiment were run, the recommendation would be to place an array of SJAs at the base of the rear face to test if this can aid the performance of the setup.

Bibliography

- [1] H. Government, *The road to zero*, Government Document, 2018. [Online]. Available: <https://www.gov.uk/government/publications/reducing-emissions-from-road-transport-road-to-zero-strategy>.
- [2] H. Chowdhury, H. Moria, A. Ali, I. Khan, F. Alam, and S. Watkins, "A study on aerodynamic drag of a semi-trailer truck," *Procedia Engineering*, vol. 56, pp. 201–205, 2013, ISSN: 1877-7058. DOI: <https://doi.org/10.1016/j.proeng.2013.03.108>. [Online]. Available: <http://www.sciencedirect.com/science/article/pii/S1877705813004621>.
- [3] R. M. Wood, "A discussion of a heavy truck advanced aerodynamic trailer system," in *International Forum for Road Transport Technology (IFRTT) 9th International Symposium on Heavy Vehicle Weights and Dimensions*.
- [4] Atdynamics, *Fuel efficient active flow control for tractor trailers*, Web Page, 2011. [Online]. Available: <https://www.arb.ca.gov/research/apr/past/icat08-1.pdf>.
- [5] J. Pfeiffer and R. King, "Linear parameter-varying active flow control for a 3d bluff body exposed to cross-wind gusts," in *32nd AIAA Applied Aerodynamics Conference (AIAA AVIATION Forum)*, AIAA AVIATION Forum. American Institute of Aeronautics and Astronautics, 2014, doi:10.2514/6.2014-2406. DOI: [doi:10.2514/6.2014-2406](https://doi.org/10.2514/6.2014-2406). [Online]. Available: <https://doi.org/10.2514/6.2014-2406>.
- [6] S. R. Ahmed, G. Ramm, and G. Faitin, "Some salient features of the time - averaged ground vehicle wake," *Scientific Programming*, 1984.
- [7] L. Brondolo, "Comparative investigation of large eddy simulation and rans approaches for external automotive flows," Thesis, 2011.
- [8] B. F. Zhang, Y. Zhou, and S. To, "Unsteady flow structures around a high-drag ahmed body," *Journal of Fluid Mechanics*, vol. 777, pp. 291–326, 2015, ISSN: 0022-1120. DOI: [10.1017/jfm.2015.332](https://doi.org/10.1017/jfm.2015.332). [Online]. Available: <https://www.cambridge.org/core/article/unsteady-flow-structures-around-a-highdrag-ahmed-body/C40460D76A89FA37EA07F2B395C7C794>.
- [9] R. Khan and S. Umale, "Cfd aerodynamic analysis of ahmed body," *International Journal of Engineering Trends and Technology*, vol. 18, pp. 301–308, 2014. DOI: [10.14445/22315381/IJETT-V18P262](https://doi.org/10.14445/22315381/IJETT-V18P262).

- [10] P. Gilliéron and F. Chometon, “Modelling of stationary three-dimensional separated air flows around an ahmed reference model,” *ESAIM: Proc.*, vol. 7, pp. 173–182, 1999. [Online]. Available: <https://doi.org/10.1051/proc:1999016>.
- [11] A. Lahaye, A. Leroy-Chesneau, and A. Kourta, “Aerodynamic characterisation of a square back bluff body flow,” *International Journal of Aerodynamics*, vol. 4, no. 1/2, pp. 43–60, 2014, ESA Physics [physics]/Mechanics [physics]/Fluid mechanics [physics.class-ph] Engineering Sciences [physics]/Mechanics [physics.med-ph]/Fluids mechanics [physics.class-ph] Journal articles, ISSN: 1743-5447. DOI: [10.1504/ijad.2014.057804](https://hal.archives-ouvertes.fr/hal-00918917). [Online]. Available: <https://hal.archives-ouvertes.fr/hal-00918917>.
- [12] M. Tientcheu-Nsiewe, D. Tcheukam, -. Toko, F. Murzyn, A. Mehel, and B. Patte-Rouland, “Numerical study of a turbulent flow in the near-wake of an ahmed body,” *American Journal of Environmental Engineering*, vol. 2016, pp. 157–163, 2016. DOI: [10.5923/j.ajee.20160606.01](https://doi.org/10.5923/j.ajee.20160606.01).
- [13] M. Minguez, R. Pasquetti, and E. Serre, “High-order large-eddy simulation of flow over the “ahmed body” car model,” *Physics of Fluids*, vol. 20, no. 9, p. 095101, 2008, ISSN: 1070-6631. DOI: [10.1063/1.2952595](https://doi.org/10.1063/1.2952595). [Online]. Available: <https://doi.org/10.1063/1.2952595>.
- [14] H. Lienhart, C. Stoots, and S. Becker, “Flow and turbulence structures in the wake of a simplified car model (ahmed model),” *Notes on Numerical Fluid Mechanics*, vol. 77, 2002, ISSN: 978-3-642-53642-7. DOI: [10.1007/978-3-540-45466-3_39](https://doi.org/10.1007/978-3-540-45466-3_39).
- [15] F. Menter, *Test case 9.4: Flow around a simplified car body (ahmed body)*, Generic, 2002.
- [16] J. Venning, D. Lo Jacono, D. Burton, M. Thompson, and J. Sheridan, “The effect of aspect ratio on the wake of the ahmed body,” *Experiments in Fluids*, vol. 56, no. 6, p. 126, 2015, ISSN: 1432-1114. DOI: [10.1007/s00348-015-1996-5](https://doi.org/10.1007/s00348-015-1996-5). [Online]. Available: <https://doi.org/10.1007/s00348-015-1996-5>.
- [17] D. Kalyan and P. Akshoy, “Computational study of flow around a simplified 2d ahmed body,” *international journal of engineering science and innovative technology*, vol. 2, p. 6, 2013.
- [18] M. Bambhania and J. Deepak, *NUMERICAL SIMULATION OF TWO-DIMENSIONAL AHMED BODY*. 2015. DOI: [10.13140/RG.2.1.1813.9285](https://doi.org/10.13140/RG.2.1.1813.9285).
- [19] Y. Liu and A. Moser, “Numerical modeling of airflow over the ahmed body,” in *Eleventh annual conference of the CFD Society of Canada (CFD 2003) Proceedings*, ENGINEERING, CFD Society of Canada, 132Megabytes. [Online]. Available: http://inis.iaea.org/search/search.aspx?orig_q=RN:39121796.
- [20] Ansys, “Ansys fluent theory guide,” in 2020 R1. ANSYS Inc, 2020, ch. 4.

- [21] G. Franck, N. Nigro, M. Storti, and J. D'Elía, "Numerical simulation of the flow around the ahmed vehicle model," *Latin American applied research Pesquisa aplicada latino americana = Investigación aplicada latinoamericana*, vol. 39, pp. 295–306, 2009.
- [22] W. Meile, G. Brenn, A. Reppenhagen, B. Lechner, and A. Fuchs, "Experiments and numerical simulations on the aerodynamics of the ahmed body," *CFD Letters*, vol. 3, 2011.
- [23] S. Kapadia, S. Roy, M. Vallero, K. Wurtzler, and J. Forsythe, "Detached eddy simulation over a reference ahmed car model," *41st Aerospace Sciences Meeting and Exhibit*, 2004, ISSN: 978-90-481-6575-9. DOI: [10.2514/6.2003-857](https://doi.org/10.2514/6.2003-857).
- [24] J. Tian, Y. Zhang, H. Zhu, and H. Xiao, "Aerodynamic drag reduction and flow control of ahmed body with flaps," *Advances in Mechanical Engineering*, vol. 9, no. 7, p. 1 687 814 017 711 390, 2017, ISSN: 1687-8132. DOI: [10.1177/1687814017711390](https://doi.org/10.1177/1687814017711390). [Online]. Available: <https://doi.org/10.1177/1687814017711390>.
- [25] B. Khalighi *et al.*, *Experimental and computational study of unsteady wake flow behind a bluff body with a drag reduction device*, Conference Paper, 2001-01-1042, 2001. DOI: [10.4271/2001-01-1042](https://doi.org/10.4271/2001-01-1042). [Online]. Available: <https://doi.org/10.4271/2001-01-1042>.
- [26] P. Moghimi and R. Rafee, "Numerical and experimental investigations on aerodynamic behavior of the ahmed body model with different diffuser angles," *Journal of Applied Fluid Mechanics*, vol. 11, pp. 1101–1113, 2018. DOI: [10.29252/jafm.11.04.27923](https://doi.org/10.29252/jafm.11.04.27923).
- [27] H. Taiming, Z. Xiaodong, W. Zhongmin, and G. Zhengqi, "Experimental and numerical investigations of the vehicle aerodynamic drag with single-channel rear diffuser," *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, vol. 234, no. 8, pp. 2216–2227, 2020, ISSN: 0954-4070. DOI: [10.1177/0954407019893849](https://doi.org/10.1177/0954407019893849). [Online]. Available: <https://doi.org/10.1177/0954407019893849>.
- [28] D. C. Barros, J. Bor'ee, B. R. Noack, A. Spohn, and T. Ruiz, "Bluff body drag manipulation using pulsed jets and coanda effect," *Journal of Fluid Mechanics*, vol. 805, pp. 422–459, 2016.
- [29] Y. Eulalie, P. Gilotte, and I. Mortazavi, "Numerical study of flow control strategies for a simplified square back ground vehicle," *Fluid Dynamics Research*, vol. 49, p. 035 502, 2017. DOI: [10.1088/1873-7005/aa5c54](https://doi.org/10.1088/1873-7005/aa5c54).
- [30] D. Krentel, R. Muminovic, A. Brunn, W. Nitsche, and R. King, "Application of active flow control on generic 3d car models," in *Active Flow Control II*, R. King, Ed., Springer Berlin Heidelberg, pp. 223–239, ISBN: 978-3-642-11735-0.
- [31] Z. Hui, X. Hu, P. Guo, Z. Wang, and J. Wang, "Separation flow control of a generic ground vehicle using an sdbd plasma actuator," *Energies*, vol. 12, p. 3805, 2019. DOI: [10.3390/en12203805](https://doi.org/10.3390/en12203805).

- [32] S. Roy, P. Zhao, A. DasGupta, and J. Soni, “Dielectric barrier discharge actuator for vehicle drag reduction at highway speeds,” *AIP Advances*, vol. 6, no. 2, p. 025 322, 2016. DOI: [10.1063/1.4942979](https://doi.org/10.1063/1.4942979). [Online]. Available: <https://doi.org/10.1063/1.4942979>.
- [33] A. Seifert *et al.*, “Large trucks drag reduction using active flow control,” in *The Aerodynamics of Heavy Vehicles II: Trucks, Buses, and Trains*, F. Browand, R. McCallen, and J. Ross, Eds. Berlin, Heidelberg: Springer Berlin Heidelberg, 2009, pp. 115–133, ISBN: 978-3-540-85070-0. DOI: [10.1007/978-3-540-85070-0_10](https://doi.org/10.1007/978-3-540-85070-0_10). [Online]. Available: https://doi.org/10.1007/978-3-540-85070-0_10.
- [34] R. Littlewood and M. Passmore, “Aerodynamic drag reduction of a simplified squareback vehicle using steady blowing,” *Loughborough university*, 2012.
- [35] B. F. Zhang, K. Liu, Y. Zhou, S. To, and J. Y. Tu, “Active drag reduction of a high-drag ahmed body based on steady blowing,” *Journal of Fluid Mechanics*, vol. 856, pp. 351–396, 2018, ISSN: 0022-1120. DOI: [10.1017/jfm.2018.703](https://doi.org/10.1017/jfm.2018.703). [Online]. Available: <https://www.cambridge.org/core/article/active-drag-reduction-of-a-highdrag-ahmed-body-based-on-steady-blowing/5576D4E841086B2A7588A54469C29C10>.
- [36] L. Henning and R. King, “Drag reduction by closed-loop control of a separated flow over a bluff body with a blunt trailing edge,” in *Proceedings of the 44th IEEE Conference on Decision and Control*, pp. 494–499, ISBN: 0191-2216. DOI: [10.1109/CDC.2005.1582204](https://doi.org/10.1109/CDC.2005.1582204).
- [37] H. Park, J.-H. Cho, J. Lee, D.-H. Lee, and K. H. Kim, “Experimental study on synthetic jet array for aerodynamic drag reduction of a simplified car,” *Journal of Mechanical Science and Technology*, vol. 27, no. 12, pp. 3721–3731, 2013, ISSN: 1976-3824. DOI: [10.1007/s12206-013-0917-x](https://doi.org/10.1007/s12206-013-0917-x). [Online]. Available: <https://doi.org/10.1007/s12206-013-0917-x>.
- [38] M. Pastoor, L. Henning, B. R. Noack, R. King, and G. Tadmor, “Feedback shear layer control for bluff body drag reduction,” *Journal of Fluid Mechanics*, vol. 608, pp. 161–196, 2008, ISSN: 0022-1120. DOI: [10.1017/S0022112008002073](https://doi.org/10.1017/S0022112008002073). [Online]. Available: <https://www.cambridge.org/core/article/feedback-shear-layer-control-for-bluff-body-drag-reduction/F9CEAFA0DF1AE5BC61E8FF425B5D21AA>.
- [39] R. Littlewood, “Novel methods of drag reduction for squareback road vehicles,” Thesis, 2013.
- [40] D. Geropp and H. J. Odenthal, “Drag reduction of motor vehicles by active flow control using the coanda effect,” *Experiments in Fluids*, vol. 28, no. 1, pp. 74–85, 2000, ISSN: 1432-1114. DOI: [10.1007/s003480050010](https://doi.org/10.1007/s003480050010). [Online]. Available: <https://doi.org/10.1007/s003480050010>.

- [41] L. Henning *et al.*, “Extensions of adaptive slope-seeking for active flow control,” *Proceedings of the Institution of Mechanical Engineers, Part I: Journal of Systems and Control Engineering*, vol. 222, no. 5, pp. 309–322, 2008, ISSN: 0959-6518. DOI: [10.1243/09596518JSCE490](https://doi.org/10.1243/09596518JSCE490). [Online]. Available: <https://doi.org/10.1243/09596518JSCE490>.
- [42] T. Heinemann, M. Springer, H. Lienhart, S. Kniesburges, and S. Becker, “Active flow control on a 1:4 car model,” *Experiments in Fluids*, vol. 55, 2014. DOI: [10.1007/s00348-014-1738-0](https://doi.org/10.1007/s00348-014-1738-0).
- [43] S. Scott Collis, R. D. Joslin, A. Seifert, and V. Theofilis, “Issues in active flow control: Theory, control, simulation, and experiment,” *Progress in Aerospace Sciences*, vol. 40, no. 4, pp. 237–289, 2004, ISSN: 0376-0421. DOI: <https://doi.org/10.1016/j.paerosci.2004.06.001>. [Online]. Available: <http://www.sciencedirect.com/science/article/pii/S0376042104000405>.
- [44] L. Cattafesta and M. Sheplak, “Actuators for active flow control,” *Annual Review of Fluid Mechanics*, vol. 43, pp. 247–272, 2011. DOI: [10.1146/annurev-fluid-122109-160634](https://doi.org/10.1146/annurev-fluid-122109-160634).
- [45] L. Wang, Z. Luo, Z. Xia, B. Liu, and X. Deng, “Review of actuators for high speed active flow control,” *Science China Technological Sciences*, vol. 55, no. 8, pp. 2225–2240, 2012, ISSN: 1869-1900. DOI: [10.1007/s11431-012-4861-2](https://doi.org/10.1007/s11431-012-4861-2). [Online]. Available: <https://doi.org/10.1007/s11431-012-4861-2>.
- [46] M. Rouméas, P. Gilliéron, and A. Kourta, “Drag reduction by flow separation control on a car after body,” *International Journal for Numerical Methods in Fluids*, vol. 60, no. 11, pp. 1222–1240, 2008, ISSN: 0271-2091. DOI: [10.1002/fld.1930](https://doi.org/10.1002/fld.1930). [Online]. Available: <https://doi.org/10.1002/fld.1930>.
- [47] J. T. Whiteman and M. Zhuang, “Active flow control schemes for bluff body drag reduction,” no. 50282, V01AT13A002, 2016, 10.1115/FEDSM2016-7520. DOI: [10.1115/FEDSM2016-7520](https://doi.org/10.1115/FEDSM2016-7520). [Online]. Available: <http://dx.doi.org/10.1115/FEDSM2016-7520>.
- [48] G. Arwatz, I. Fono, and A. Seifert, “Suction and oscillatory blowing actuator modeling and validation,” *AIAA Journal*, vol. 46, no. 5, pp. 1107–1117, 2008, ISSN: 0001-1452. DOI: [10.2514/1.30468](https://doi.org/10.2514/1.30468). [Online]. Available: <https://doi.org/10.2514/1.30468>.
- [49] A. Seifert, T. Shtendel, and D. Dolgopyat, “From lab to full scale active flow control drag reduction: How to bridge the gap?” *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 147, pp. 262–272, 2015, ISSN: 0167-6105. DOI: <https://doi.org/10.1016/j.jweia.2015.09.012>. [Online]. Available: <http://www.sciencedirect.com/science/article/pii/S0167610515002287>.
- [50] C. P. Oriol, “Study of the bifurcation behaviour of the flow around a 2d ahmed body applied to active flow control systems,” Thesis, 2018. [Online]. Available: <http://resolver.tudelft.nl/uuid:853cb59d-9c14-45ae-ba19-5edf57f65575>.

- [51] A. Seifert, T. Bachar, D. Koss, M. Shepshelovich, and I. Wygnanski, "Oscillatory blowing: A tool to delay boundary-layer separation," *AIAA Journal*, vol. 31, no. 11, pp. 2052–2060, 1993, ISSN: 0001-1452. DOI: [10.2514/3.49121](https://doi.org/10.2514/3.49121). [Online]. Available: <https://arc.aiaa.org/doi/abs/10.2514/3.49121>.
- [52] P. Joseph, X. Amandolese, C. Edouard, and J.-L. Aider, "Flow control using mems pulsed micro-jets on the ahmed body," *Experiments in Fluids*, vol. 54, no. 1, p. 1442, 2013, ISSN: 1432-1114. DOI: [10.1007/s00348-012-1442-x](https://doi.org/10.1007/s00348-012-1442-x). [Online]. Available: <https://doi.org/10.1007/s00348-012-1442-x>.
- [53] D. E. Manosalvas, T. D. Economon, F. Palacios, and A. Jameson, "Techniques for the design of active flow control systems in heavy vehicles," in *33rd AIAA Applied Aerodynamics Conference (AIAA AVIATION Forum)*, AIAA AVIATION Forum. American Institute of Aeronautics and Astronautics, 2015, doi:10.2514/6.2015-3312. DOI: [doi:10.2514/6.2015-3312](https://doi.org/10.2514/6.2015-3312). [Online]. Available: <https://doi.org/10.2514/6.2015-3312>.
- [54] J.-F. Beaudoin, O. Cadot, J. E. Wesfreid, and J.-L. Aider, "Feedback control using extremum seeking method for drag reduction of a 3d bluff body," in *IUTAM Symposium on Flow Control and MEMS*, J. F. Morrison, D. M. Birch, and P. Lavoie, Eds., Springer Netherlands, pp. 365–372, ISBN: 978-1-4020-6858-4.
- [55] A. Glezer and M. Amitay, "Synthetic jets," *Annual Review of Fluid Mechanics*, vol. 34, no. 1, pp. 503–529, 2002, doi: 10.1146/annurev.fluid.34.090501.094913 0066-4189. [Online]. Available: <https://doi.org/10.1146/annurev.fluid.34.090501.094913>.
- [56] M. H. Hong, S. Y. Cheng, and S. Zhong, "Effect of geometric parameters on synthetic jet: A review," *Physics of Fluids*, vol. 32, no. 3, p. 031 301, 2020, doi: 10.1063/1.5142408 1070-6631. [Online]. Available: <https://doi.org/10.1063/1.5142408>.
- [57] B. Gungordu, M. Jabbal, and P. Atanas, *Development of high authority synthetic jet actuators for aerospace applications*, Generic, 2020.
- [58] A. Honohan, M. Amitay, and A. Glezer, "Aerodynamic control using synthetic jets," in *Fluids 2000 Conference and Exhibit (Fluid Dynamics and Co-located Conferences)*, Fluid Dynamics and Co-located Conferences. American Institute of Aeronautics and Astronautics, 2000, doi:10.2514/6.2000-2401. DOI: [doi:10.2514/6.2000-2401](https://doi.org/10.2514/6.2000-2401). [Online]. Available: <https://doi.org/10.2514/6.2000-2401>.
- [59] M. Amitay and A. Glezer, "Role of actuation frequency in controlled flow reattachment over a stalled airfoil," *AIAA Journal*, vol. 40, no. 2, pp. 209–216, 2002, ISSN: 0001-1452. DOI: [10.2514/2.1662](https://doi.org/10.2514/2.1662). [Online]. Available: <https://doi.org/10.2514/2.1662>.

- [60] A. Glezer, M. Amitay, and A. M. Honohan, “Aspects of low- and high-frequency actuation for aerodynamic flow control,” *AIAA Journal*, vol. 43, no. 7, pp. 1501–1511, 2005, ISSN: 0001-1452. DOI: [10.2514/1.7411](https://doi.org/10.2514/1.7411). [Online]. Available: <https://doi.org/10.2514/1.7411>.
- [61] A. Naim, A. Seifert, and I. Wygnanski, *Active Control of Cylinder Flow With and Without a Splitter Plate Using Piezoelectric Actuators*. 2002, ISBN: 978-1-62410-105-2. DOI: [10.2514/6.2002-3070](https://doi.org/10.2514/6.2002-3070).
- [62] G. Minelli, E. A. Hartono, V. Chernoray, L. Hjelm, and S. Krajnović, “Aerodynamic flow control for a generic truck cabin using synthetic jets,” *Journal of Wind Engineering & Industrial Aerodynamics*, vol. 168, pp. 81–90, 2017, ISSN: 0167-6105. DOI: [10.1016/j.jweia.2017.05.006](https://doi.org/10.1016/j.jweia.2017.05.006).
- [63] G. Minelli, E. Adi Hartono, L. Hjelm, V. Chernoray, B. Basara, and S. Krajnović, *EXPERIMENTAL AND NUMERICAL INVESTIGATION OF ACTIVE FLOW CONTROL ON A GENERIC TRUCK CABIN*. 2016.
- [64] G. Minelli *et al.*, “Active aerodynamic control of a separated flow using streamwise synthetic jets,” *Flow, Turbulence and Combustion*, vol. 103, no. 4, pp. 1039–1055, 2019, ISSN: 1573-1987. DOI: [10.1007/s10494-019-00058-4](https://doi.org/10.1007/s10494-019-00058-4). [Online]. Available: <https://doi.org/10.1007/s10494-019-00058-4>.
- [65] M. El-Alti, “Active flow control for drag reduction of heavy vehicles,” Thesis, 2012.
- [66] P. Gil, “Bluff body drag control using synthetic jet,” *Journal of Applied Fluid Mechanics*, vol. 12, pp. 293–302, 2019. DOI: [10.29252/jafm.75.253.28960](https://doi.org/10.29252/jafm.75.253.28960).
- [67] A. Kourta and C. Leclerc, “Characterization of synthetic jet actuation with application to ahmed body wake,” *Sensors and Actuators A: Physical*, vol. 192, pp. 13–26, 2013, ESA Engineering Sciences [physics]/Mechanics [physics.med-ph]/Fluids mechanics [physics.class-ph] Physics [physics]/Mechanics [physics]/Fluid mechanics [physics.class-ph] Journal articles, ISSN: 0924-4247. DOI: [10.1016/j.sna.2012.12.008](https://hal.archives-ouvertes.fr/hal-00778523). [Online]. Available: <https://hal.archives-ouvertes.fr/hal-00778523>.
- [68] M. Balajewicz, E. H. Dowell, and B. Noack, *A new approach to model order reduction of the Navier-Stokes equations*. 2012.
- [69] J. Du, F. Fang, C. C. Pain, I. M. Navon, J. Zhu, and D. A. Ham, “Pod reduced-order unstructured mesh modeling applied to 2d and 3d fluid flow,” *Computers & Mathematics with Applications*, vol. 65, no. 3, pp. 362–379, 2013, ISSN: 0898-1221. DOI: <https://doi.org/10.1016/j.camwa.2012.06.009>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S0898122112004397>.
- [70] K. Cohen, S. Siegel, and T. McLaughlin, *Control Issues in Reduced-Order Feedback Flow Control*. 2004, ISBN: 978-1-62410-078-9. DOI: [10.2514/6.2004-575](https://doi.org/10.2514/6.2004-575).

- [71] C. W. Rowley, “Model reduction for fluids, using balanced proper orthogonal decomposition,” *International Journal of Bifurcation and Chaos*, vol. 15, no. 03, pp. 997–1013, 2005, ISSN: 0218-1274. DOI: [10.1142/S0218127405012429](https://doi.org/10.1142/S0218127405012429). [Online]. Available: <https://doi.org/10.1142/S0218127405012429>.
- [72] K. Willcox and J. Peraire, “Balanced model reduction via the proper orthogonal decomposition,” *AIAA Journal*, vol. 40, no. 11, pp. 2323–2330, 2002, ISSN: 0001-1452. DOI: [10.2514/2.1570](https://doi.org/10.2514/2.1570). [Online]. Available: <https://doi.org/10.2514/2.1570>.
- [73] J. Annoni, J. Nichols, and P. Seiler, *Wind Farm Modeling and Control Using Dynamic Mode Decomposition*. 2016. DOI: [10.2514/6.2016-2201](https://doi.org/10.2514/6.2016-2201).
- [74] Mathworks, *N4sid: Help center (r2021b)*, Web Page, 2021. [Online]. Available: <https://uk.mathworks.com/help/ident/ref/n4sid.html>.
- [75] P. Van Overschee and B. De Moor, “N4sid: Subspace algorithms for the identification of combined deterministic-stochastic systems,” *Special issue on statistical signal processing and control*, vol. 30, no. 1, pp. 75–93, 1994, 0005-1098. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/0005109894902305>.
- [76] A. Jennifer, “Modelling for wind farm control,” Thesis, 2016. [Online]. Available: <https://hdl.handle.net/11299/181756>.
- [77] P. Benner, C. Himpe, and T. Mitchell, “On reduced input-output dynamic mode decomposition,” vol. 44, p. 1751, 2018.
- [78] R. Martínez-González, Y. Bolea, A. Grau, and H. Martínez-García, “An l_pv fractional model for canal control,” *Mathematical Problems in Engineering*, vol. 2009, p. 471 540, 2009, ISSN: 1024-123X. DOI: [10.1155/2009/471540](https://doi.org/10.1155/2009/471540). [Online]. Available: <https://doi.org/10.1155/2009/471540>.
- [79] Q. Zhang, “L_pv system local model interpolation based on combined model reduction,” in *SYSID 2018, 18th IFAC Symposium on System Identification*, SYSID 2018, 18th IFAC Symposium on System Identification , Stockholm, SUEDE, 09-/07/2018 - 11/07/2018 hal-01909552 Computer Science [cs]/Automatic Control EngineeringConference papers, <https://hal.inria.fr/hal-01909552/document> <https://hal.inria.fr/hal-01909552/file/sysid2018LPVintp.pdf>, pp. -. [Online]. Available: <https://hal.inria.fr/hal-01909552>.
- [80] B. Bamieh and L. Giarre, “Identification of linear parameter varying models,” in *Proceedings of the 38th IEEE Conference on Decision and Control (Cat. No.99CH36304)*, vol. 2, 1505–1510 vol.2, ISBN: 0191-2216. DOI: [10.1109/CDC.1999.830205](https://doi.org/10.1109/CDC.1999.830205).
- [81] M. Nemani, R. Ravikanth, and B. Bamieh, *Identification of linear parameter-varying Systems*. 1996, 2990–2995 vol.3, ISBN: 0-7803-2685-7. DOI: [10.1109/CDC.1995.478600](https://doi.org/10.1109/CDC.1995.478600).
- [82] Y. Bolea, J. Blesa, and V. Puig, “L_pv modelling and identification of an open canal for control,” in *2007 European Control Conference (ECC)*, pp. 3584–3591. DOI: [10.23919/ECC.2007.7068415](https://doi.org/10.23919/ECC.2007.7068415).

- [83] V. Motta and L. Malzacher, “Open-loop and closed-loop flow control based on van der pol modeling,” *Acta Mechanica*, vol. 229, no. 1, pp. 389–401, 2018, ISSN: 1619-6937. DOI: [10.1007/s00707-017-1975-4](https://doi.org/10.1007/s00707-017-1975-4). [Online]. Available: <https://doi.org/10.1007/s00707-017-1975-4>.
- [84] “Siso scheme and linear analysis,” *Real-Time Optimization by Extremum-Seeking Control*, pp. 1–20, 2003, https://doi.org/10.1002/0471669784.ch1_9780471669784. [Online]. Available: <https://doi.org/10.1002/0471669784.ch1>.
- [85] M. Krstić and H.-H. Wang, “Stability of extremum seeking feedback for general nonlinear dynamic systems,” vol. 36, no. 4, pp. 595–601, 2000, 0005-1098. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S0005109899001831>.
- [86] J. F. Beaudoin, O. Cadot, J. L. Aider, and J. E. Wesfreid, “Bluff-body drag reduction by extremum-seeking control,” *Journal of Fluids and Structures*, vol. 22, no. 6, pp. 973–978, 2006, ISSN: 0889-9746. DOI: <https://doi.org/10.1016/j.jfluidstructs.2006.04.010>. [Online]. Available: <http://www.sciencedirect.com/science/article/pii/S088997460600065X>.
- [87] P. M. Hoeijmakers, “Implementation of an extremum seeking controller for vortex shedding attenuation in a 2d cfd code.”
- [88] N. Kamagata, S. Horio, and K. Hishida, “Drag reduction for blunt body with cross flow by extremum-seeking control,” no. 48401, pp. 499–506, 2008, 10.1115/FEDSM2008-55081. DOI: [10.1115/FEDSM2008-55081](https://doi.org/10.1115/FEDSM2008-55081). [Online]. Available: <http://dx.doi.org/10.1115/FEDSM2008-55081>.
- [89] R. D. Brackston, A. Wynn, and J. F. Morrison, “Extremum seeking to control the amplitude and frequency of a pulsed jet for bluff body drag reduction,” *Experiments in Fluids*, vol. 57, no. 10, p. 159, 2016, ISSN: 1432-1114. DOI: [10.1007/s00348-016-2243-4](https://doi.org/10.1007/s00348-016-2243-4). [Online]. Available: <https://doi.org/10.1007/s00348-016-2243-4>.
- [90] G. Papageorgiou, J. Barden, J. F. Whidborne, and K. P. Garry, “Extremum seeking control for truck drag reduction,” in *2018 UKACC 12th International Conference on Control (CONTROL)*, pp. 114–119. DOI: [10.1109/CONTROL.2018.8516808](https://doi.org/10.1109/CONTROL.2018.8516808).
- [91] D. Dochain, M. Perrier, and M. Guay, “Extremum seeking control and its application to process and reaction systems: A survey,” *6th Vienna International Conference on Mathematical Modelling*, vol. 82, no. 3, pp. 369–380, 2011, 0378-4754. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S0378475410003290>.
- [92] A. Rahnema, M. Xia, S. Wang, and P. Antsaklis, “An extremum-seeking co-simulation based framework for passivation theory and its application in adaptive cruise control systems,” 2016.
- [93] Y. Li, W. Cui, Q. Jia, Q. Li, Z. Yang, and B. R. Noack, “Optimization of active drag reduction for a slanted ahmed body in a high-dimensional parameter space,” *arXiv: Fluid Dynamics*, 2019.

- [94] L. Yiqing *et al.*, “Explorative gradient method for active drag reduction of the fluidic pinball and slanted ahmed body,” 2020.
- [95] R. Kumar, S. K. Singla, and V. Chopra, “Comparison among some well known control schemes with different tuning methods,” *Journal of Applied Research and Technology*, vol. 13, no. 3, pp. 409–415, 2015, ISSN: 1665-6423. DOI: <https://doi.org/10.1016/j.jart.2015.07.007>. [Online]. Available: <http://www.sciencedirect.com/science/article/pii/S1665642315000358>.
- [96] H. H. Hu and H. H. Bau, “Feedback control to delay or advance linear loss of stability in planar poiseuille flow,” *Proceedings of the Royal Society: Mathematical and Physical Sciences (1990-1995)*, vol. 447, no. 1930, pp. 299–312, 1994, ISSN: 0962-8444. DOI: [10.1098/rspa.1994.0142](https://doi.org/10.1098/rspa.1994.0142).
- [97] D. Sipp and P. Schmid, “Closed-loop control of fluid flow: A review of linear approaches and tools for the stabilization of transitional flows,” *Aerospace Lab*, no. 6, p. 1–11, 2013, Engineering Sciences [physics]/Mechanics [physics.med-ph]/Fluids mechanics [physics.class-ph] Journal articles, ISSN: 2107-6596. [Online]. Available: <https://hal.archives-ouvertes.fr/hal-01184632>.
- [98] J. W. Hall, C. E. Tinney, J. M. Ausseur, J. T. Pinier, A. M. Hall, and M. N. Glauser, “Low-dimensional tools for closed-loop flow-control in high reynolds number turbulent flows,” in *IUTAM Symposium on Flow Control and MEMS*, J. F. Morrison, D. M. Birch, and P. Lavoie, Eds., Springer Netherlands, pp. 293–310, ISBN: 978-1-4020-6858-4.
- [99] E. Atam, L. Mathelin, and L. Cordier, *Model-Free Closed-Loop Flow Control*. 2012.
- [100] M. Engert, A. Pätzold, R. Becker, and W. Nitsche, “Active cancellation of tollmien-schlichting instabilities in compressible flows using closed-loop control,” in *IUTAM Symposium on Flow Control and MEMS*, J. F. Morrison, D. M. Birch, and P. Lavoie, Eds., Springer Netherlands, pp. 319–331, ISBN: 978-1-4020-6858-4.
- [101] R. Li, B. R. Noack, L. Cordier, J. Borée, and F. Harambat, “Drag reduction of a car model by linear genetic programming control,” *Experiments in Fluids*, vol. 58, no. 8, p. 103, 2017, ISSN: 1432-1114. DOI: [10.1007/s00348-017-2382-2](https://doi.org/10.1007/s00348-017-2382-2). [Online]. Available: <https://doi.org/10.1007/s00348-017-2382-2>.
- [102] D. Thomas, B. S. L, and N. B. R, “Machine learning control,” in *Machine Learning Control - Taming Nonlinear Dynamics and Turbulence (Fluid Mechanics and Its Applications)*, 1st ed., Fluid Mechanics and Its Applications. Springer, Cham, 2016, ch. 2, pp. 11–48, ISBN: 978-3-319-82140-5.
- [103] T. Han, S. Tang, C. Huang, and K. Dong, “Fluent/simulink collaborative simulation for missile separation from cavity,” *Advanced Materials Research*, vol. 591-593, pp. 1902–1906, 2012. DOI: [10.4028/www.scientific.net/AMR.591-593.1902](https://doi.org/10.4028/www.scientific.net/AMR.591-593.1902).

- [104] *Ercoftac ahmed body*, Online Database, Feb. 2002. [Online]. Available: https://www.kbwiki.ercoftac.org/w/index.php/Abstr:Ahmed_body.
- [105] G. Caminha, *The cfl condition and how to choose your timestep size*, Blog, 2019. [Online]. Available: <https://www.simscale.com/blog/2017/08/cfl-condition/>.
- [106] P. Holmes, J. L. Lumley, and G. Berkooz, *Turbulence, Coherent Structures, Dynamical Systems and Symmetry* (Cambridge Monographs on Mechanics). Cambridge: Cambridge University Press, 1996. [Online]. Available: <https://www.cambridge.org/core/books/turbulence-coherent-structures-dynamical-systems-and-symmetry/4D353229751BBB24C8C266B4714CA9F2>.
- [107] M. K. Isman, “Investigation of inlet effects on backward-facing step flow prediction,” *Transactions of the Canadian Society for Mechanical Engineering*, vol. 40, no. 3, pp. 317–329, 2016, ISSN: 0315-8977. DOI: [10.1139/tcsme-2016-0024](https://doi.org/10.1139/tcsme-2016-0024). [Online]. Available: <https://doi.org/10.1139/tcsme-2016-0024>.
- [108] J. Kou and W. Zhang, “Dynamic mode decomposition with exogenous input for data-driven modeling of unsteady flows,” *Physics of Fluids*, vol. 31, no. 5, p. 057 106, 2019, ISSN: 1070-6631. DOI: [10.1063/1.5093507](https://doi.org/10.1063/1.5093507). [Online]. Available: <https://doi.org/10.1063/1.5093507>.
- [109] G. Minelli, S. Krajnović, B. Basara, and B. R. Noack, “Numerical investigation of active flow control around a generic truck a-pillar,” *Flow, Turbulence and Combustion*, vol. 97, no. 4, pp. 1235–1254, 2016, ISSN: 1573-1987. DOI: [10.1007/s10494-016-9760-3](https://doi.org/10.1007/s10494-016-9760-3). [Online]. Available: <https://doi.org/10.1007/s10494-016-9760-3>.

Appendix A

Fluent UDF

```
1 #include "udf.h"
2 DEFINE_PROFILE(SJ_Inlet, thread, position)
3 {
4     // Variable declarations
5     real x[ND_ND]; // Define array for the coordinates to be
6                     // stored based on the dimension of the simulation
7     real y;
8     face_t face;
9     real time;
10    // Coordinates for the centre point of the SJA inlet
11    real x_centre = 0.8636945;
12    real y_centre = 0.2868545;
13    // Set the frequency of the SJAs
14    real freq = 2200;
15    // Set the peak jet velocity of the SJAs
16    real peakVel = 20;
17    real sjaRadius = 0.001;
18
19    // loops through all faces in the SJA velocity inlet
20    begin_f_loop(face, thread)
21    {
22        // finds the coordinates of the centre of each of the faces
23        // in the thread
24        F_CENTROID(x, face, thread);
25        // this gets the flow time from FLUENT which is used to
26        // determine the position in the cycle of the SJA membrane
27        time = RP_Get_Real("flow-time");
28        // Pythagoras to find the distance of the centroid from the
29        // centre of the SJAINlet
30        y = sqrt(pow(y_centre - x[1], 2) + pow(x_centre - x[0], 2));
31        // uses the definition for the profile of a flow in a pipe
32        // - u = Umax * [1 - (y/h)^2] * sin(freq*t*(2*Pi))
33        F_PROFILE(face, thread, position) = peakVel * (1 - (pow(y/
34        sjaRadius, 2))) * sin(freq*time*(2*3.142));
35    }
36    end_f_loop(face, thread)
37 }
```

Listing A.1: Fluent UDF to handle the synthetic jet array velocity profile in 2D