



Jaime Rodrigo Fisher Fernandez

CFD Study of a Bidirectional Synthetic Jet as an Aerodynamic Flow
Control Device

PhD

2017

Supervision

Dr. Takafumi Nishino

Prof. A.M. Savill

Degree Sought:

PhD

Date of initial registration:

02 September 2013



Jaime Rodrigo Fisher Fernandez

CFD Study of the Bi-Directional Synthetic Jet as an Aerodynamic
Flow Control Device.

PhD
2017

Supervision

Dr. Takafumi Nishino
Prof. A.M. Savill

Degree Sought:

PhD

Date of initial registration:

02 September 2013

ABSTRACT

Control of the separation and reattachment of the boundary layer has been the focus of research for a wide range of engineering applications, and it is known that a synthetic jet can be used as an active flow control device for such issues. This work presents a numerical analysis of the potential benefits of changing the inflow and outflow directions of a synthetic jet separately. The basic concept of this new flow control approach, named Bi-Directional synthetic jet (BDSJ), is to induce the Coandă effect due to the jet outflow directed downstream, and change accordingly the inflow direction for an efficient and directed suction of the boundary layer. A widely-used reference case (so-called NASA 2-D Hump) is used to validate the numerical model and to compare results. Three well known Reynolds-averaged Navier-Stokes (RANS) turbulence models are employed and they all show similar trends. Eleven different angles are tested for the inflow and outflow directions of the synthetic jet; the most effective configuration is then compared with the classical synthetic jet and Directed synthetic jet. It is shown that the best BDSJ configuration results in a shorter separation bubble length over the hump, suggesting the potential of BDSJ as a future active flow control device.

This work also attempts to answer the question; “How a Bi-directional synthetic jet, influences the aerodynamic coefficients on a aerofoil”. Simulations are carried out on an NACA 23012 at a Reynolds number of $Re = 2.19 \cdot 10^6$, where a two-dimensional structured mesh is used to evaluate the impact of the amplitude and frequency of the Bidirectional synthetic jet on the aerofoil performance over a wide range of angles of attack. Three different jet oscillating frequencies are

considered as well as three blowing ratios. The phase difference between the inflow and outflow jets, the position and the configuration of the jet exits are also evaluated. The results suggest that the amplitude of the jet is a key parameter to increase the lift coefficient and a gap between the inflow and outflow jet exits could positively influence the lift coefficient if the central location of the jets coincides with the separation point. On the other hand, the jet frequencies and the phase at which the jets operate do not seem to influence the aerofoil performance significantly.

Finally, some guidelines and recommendations are provided to develop further the actual Bi-directional synthetic jet which may lead to its optimal design and manufacture.

Keywords:

Active flow control, Aerodynamics, Boundary layer, Drag reduction, Lift Enhancement, Synthetic Jet.

ACKNOWLEDGEMENTS

I would like to acknowledge and thanks the National Council of Science and Technology of Mexico, CoNaCyT, for the financial support provided.

Dr. Takafumi Nishino, for his countless hours of help and Prof. Mark Savill for his guidance.

My lovely Family.

My Friends.

T.

CONTENT

ABSTRACT .	i
ACKNOWLEDGEMENTS	iii
CONTENT .	iv
LIST OF FIGURES	vii
LIST OF TABLES	xi
LIST OF SYMBOLS	xiii
1 Introduction	1
1.1 Background	2
1.2 Modern Research.....	6
1.3 Scope of the project.....	9
1.4 Outline of the thesis.....	9
2 Literature Review	11
2.1 Aerodynamics.....	12
2.1.1 Lift	12
2.1.2 Drag	14
2.1.3 Coandă effect	16
2.1.4 Boundary-layer.....	18
2.2 Synthetic Jets.....	23
2.2.1 Applications	23
2.2.2 Parameters.....	26
2.2.3 Flow structures with quiescent and crossflow.....	31
2.2.4 Flow separation control	33
2.2.5 Further Research	33
3 Methodology	35
3.1 Computational Fluid Dynamics.....	36
3.1.1 Numerical algorithm	37
3.1.2 Turbulence and its modelling.....	37
3.1.3 Spalart Allmaras	42

3.1.4	k-ϵ models	45
3.1.5	k-ω models	47
3.2	CFD Application on Synthetic Jets	50
4	Bidirectional Synthetic Jet	51
4.1	Test case description	54
4.1.1	Characteristics of the Synthetic Jet	55
4.1.2	Effect of jet exit velocity profile	59
4.2	Validation of Baseline simulation	62
4.2.1	Computational settings	62
4.2.2	Time step size selection	63
4.2.3	Turbulence model selection	65
4.2.4	Grid convergence	67
4.2.5	Model validation	70
4.3	Results from controlled cases	76
4.3.1	Controlled case study	76
4.3.2	Varying Mass Flow Rate (MFR)	80
4.3.3	Varying Area	85
4.3.4	Varying Velocity	88
4.3.5	Jet comparisson	90
4.4	Key findings summary	94
5	Aerodynamic Performance with a Bidirectional Synthetic Jet	95
5.1	Test case description	97
5.2	Validation	99
5.2.1	Computational domain	99
5.2.2	Flow solver and parameters	100
5.2.3	Modelling of the synthetic jet	102
5.2.4	Simulation procedure	104
5.2.5	Grid convergence	106
5.3	Results and discussion	109
5.3.1	Effect of direction	109
5.3.2	Effect of the Jet Amplitude and Frequency	110
5.3.3	Effect of the Phase of Jet Oscillations	118
5.3.4	Effect of the gap between two jet orifices	121
5.3.5	Jet exit configuration	126

5.4	Key findings summary	128
6	Design of the Bidirectional Synthetic Jet	129
6.1	Conceptualization.....	130
6.2	Design.....	132
6.2.1	Cavity	133
6.2.2	Slot exit.....	137
6.2.3	Slot inlet.....	139
6.2.4	Oscillating membrane	141
6.2.5	Magnetic frame	143
6.2.6	Functional ensemble	144
7	Conclusions	147
7.1	Research limitations and Future work.....	151
	References	152
	Appendices	165

LIST OF FIGURES

Figure 1-1	General trend in the investigation on "Synthetic jets" by 2014	3
Figure 1-2	Structure of a synthetic jet	4
Figure 1-3	Schematic diagram of a BiDirectional Synthetic.	8
Figure 2-1	Total drag plot.	15
Figure 2-2	Illustration of the Coandă effect.	17
Figure 2-3	Zones within a turbulent boundary layer.....	20
Figure 2-4	Flow around a circular cylinder.	32
Figure 3-1	Reynolds decomposition;	39
Figure 4-1	Synthetic Jets classification regarding flow direction.....	53
Figure 4-2	Discretized computational domain	76
Figure 4-3	Schematic of the inflow/outflow directions of the jet	76
Figure 4-4	Computational mesh over the hump.....	76
Figure 4-5	Mesh around the jet exit,	76
Figure 4-6	Illustration of separation and reattachment.....	76
Figure 4-7	Time step study.....	76
Figure 4-8	Velocity profile at the jet exit for Cases 2, 4 and 7.	76
Figure 4-9	Experimental velocity profiles (reproduced from Smith ⁵¹).	76
Figure 4-10	Area-Weighted Average Skin friction coefficient per time step. ...	77
Figure 4-11	Mean Skin friction coefficient – x/c	78
Figure 4-12	Comparison of C_p over the hump with no flow control.....	79
Figure 4-13	Comparison of C_p over the hump for several turbulence models. .	79
Figure 4-14	Streamwise velocity comparison at $x/c = 0.8$	79
Figure 4-15	Comparison of C_p over the hump with suction.	79

Figure 4-16	Cp over the hump for suction (with different turbulence models)	79
Figure 4-17	Separation bubble length using k- ω SST.....	80
Figure 4-18	Separation bubble length using the Realizable k- ϵ model.....	81
Figure 4-19	Separation bubble length using the S-A model.....	81
Figure 4-20	Streamwise x-velocity at 1mm above the wall surface;.....	82
Figure 4-21	Pressure coefficient comparison along the hump.....	83
Figure 4-22	Bubble length for the Varying Mass Flow Rate analyses	84
Figure 4-23	Jet exit boundaries in the hump for Varying Area	85
Figure 4-24	Bubble length for the Varying Jet Area analyses.	86
Figure 4-25	Bubble length for DSJ cases.....	87
Figure 4-26	Bubble length for the Varying Velocity analyses.....	88
Figure 4-27	Bubble length for DSJ.....	89
Figure 4-28	SJ.....	90
Figure 4-29	BDSJ 165-15 Varying Area	90
Figure 4-30	BDSJ 165-15 Varying velocity.	90
Figure 4-31	Average of varying cases, MFR, Velocity and Area for BDSJ.....	91
Figure 4-32	Average of varying cases, MFR, Velocity and Area for DSJ.	92
Figure 5-1	Schematic of BDSJ for 0° AOA	98
Figure 5-2	Discretized computational domain	99
Figure 5-3	Velocity profile at the jet exit for the seven cases analysed.....	103
Figure 5-4	Computational mesh around the profile.....	105
Figure 5-5	Computational Grid, NACA-23012.....	105
Figure 5-6	Lift coefficient comparison.	107
Figure 5-7	C _L vs AOA comparison for different BDSJ configurations	109
Figure 5-8	C _L vs AOA comparison for a BDSJ	111
Figure 5-9	C _D vs AOA comparison for a BDSJ	113

Figure 5-10	Contribution of the viscous drag to the total drag.	114
Figure 5-11	Contours of Mean C_p for 37Hz	115
Figure 5-12	C_L vs C_D comparison	117
Figure 5-13	C_L / C_D vs AOA	117
Figure 5-14	Flow separation point.....	118
Figure 5-15	Phase differences between the inflow and outflow of BDSJ.....	119
Figure 5-16	Phase sensitivity analysis on C_L and C_D	120
Figure 5-17	Close-up of the mesh elements around the jet exit	122
Figure 5-18	C_L vs AOA and C_D vs C_L for different gap cases	122
Figure 5-19	Effect of the jet gap on drag and lift coefficients 20° AOA	124
Figure 5-20	Comparison of mean stream-wise velocity contours at 20°	125
Figure 5-21	Sensitiveness of position on the slot on C_L and C_D	126
Figure 6-1	Sketch of a diaphragm pump.	131
Figure 6-2	Proposed BDSJ elliptical-circular cavity.....	135
Figure 6-3	Dual Cavity configuration.....	136
Figure 6-4	Proposed BDSJ funnel-jet exit	137
Figure 6-5	Proposed BDSJ elliptical-circular cavity.....	139
Figure 6-6	Proposed BDSJ membrane.....	141
Figure 6-7	Close up dual membrane configuration.....	142
Figure 6-8	Proposed BDSJ solenoid configuration	143
Figure 6-9	Proposed BDSJ solenoid configuration	144
Figure 6-10	BDSJ transversal cut.....	145
Figure 6-11	Double configuration cavity cut	146

LIST OF TABLES

Table 1	Turbulence models used by different authors.....	76
Table 2	Summary of grid convergence study (without synthetic jet).....	76
Table 3	Time step study.....	76
Table 4	Summary of preliminary simulations with different jet profiles (BDSJ 165-15 configuration).....	76
Table 5	No flow control configuration. Separation and reattachment values by different authors.	79
Table 6	Suction configuration. Separation and reattachment values by different authors.	79
Table 7	Classical synthetic jet configuration. Separation and reattachment values by different authors.	79
Table 8	Percentage difference of Bubble size.....	93
Table 9	Turbulence models used by different authors.....	101
Table 10	Percentage variations in C_L/C_D due to different jet-exit velocity profiles.	104
Table 11	Average differences in lift coefficient for different grids.....	108
Table 12	Percentile variations of lift coefficient against the baseline case for all frequencies and amplitudes.....	112
Table 13	Percentile variations of drag coefficient against the baseline case for all frequencies and amplitudes.	116
Table 14	Differences in lift and drag due to the BDSJ phase differences (against the baseline case “Phase 0”).....	121
Table 15	Differences in lift and drag due to the gap between two jet orifices (against the baseline case “Gap 0D”).....	123
Table 16	Drag and Lift differences in percentage for the three jet configurations	127

LIST OF SYMBOLS

c	Hump or aerofoil chord, mm
b	Jet orifice width
C_D	Drag coefficient
C_f	Friction coefficient
C_L	Lift coefficient
C_μ	Momentum coefficient
C_p	Pressure coefficient
D	Diameter of cavity or Orifice, mm
f	Frequency
f_e	Excitation frequency, Hz
f_s	Wake vortex shedding frequency, Hz
f_H	Helmholtz resonant frequency
F^+	Non-dimensional excitation frequency
g	Gravity constant
L	Characteristic length
M	Mach number
p	Pressure, Pa
Re	Reynolds Number
Re_c	Chord-based Reynolds Number
R_i	Reattachment at the iteration given
R_{i-1}	Reattachment at the previous iteration
R_f	Result at the final iteration
Stk	Stokes number
St	Strouhal Number
t	Time
$\bar{U}$	Average Velocity, m/s
U_0, U_∞	Freestream velocity, m/s
x	Streamwise coordinate
x_b	Length of the separation bubble

y	Lateral coordinate
y^+	Dimensionless distance from the wall

Greek letters

α	Angle of attack, °
Δt	Time step size
δ	Boundary-layer thickness
δ_{ij}	Kronecker delta
δ^*	Displacement thickness
θ	Momentum thickness
κ	Von Kármán constant
$k-\varepsilon$	k - epsilon turbulence model
$k-\omega$	k - omega turbulence model
μ	Dynamic Viscosity
ν	Kinematic Viscosity
ρ	Density
τ_{ij}	Reynolds stress
τ_w	Wall shear stress
ω	Angular velocity

Abbreviations

<i>2D</i>	Two-dimensional
<i>3D</i>	Three-dimensional
<i>AFC</i>	Active flow control
<i>BLC</i>	Boundary layer control
<i>BDSJ</i>	Bi-directional Synthetic Jet
<i>CFD</i>	Computational Fluid Dynamics
<i>CFL</i>	Courant-Friedrichs-Levy
<i>DNS</i>	Direct numerical simulation
<i>DSJ</i>	Directed Synthetic Jet
<i>FVM</i>	Finite volume method
<i>LE</i>	Leading edge
<i>LES</i>	Large-eddy simulation
<i>PIV</i>	Particle Image Velocimetry
<i>RANS</i>	Reynolds Average Navier-Stokes
<i>RMS</i>	Root mean square
<i>S-A</i>	Spalart-Allmaras
<i>SST</i>	Shear Stress Transport
<i>SJ</i>	Synthetic Jet
<i>TE</i>	Trailing edge
<i>TS</i>	Tollmien-Schlichting
<i>VR</i>	Jet-to-freestream Velocity Ratio
<i>URANS</i>	Unsteady Reynolds-average Navier-Stokes
<i>ZNMF</i>	Zero-Net Mass Flux

1 INTRODUCTION

This chapter provides an outline of the project, giving a basic summary and overview about classical and bidirectional synthetic jets as active flow control devices.



1.1 BACKGROUND

Excitement about flying seems to be inherent to human nature, as since the ancient Greek myth of Icarus & Daedalus (around 500 BC), investigation of flying methods has been constantly developed; the ornithopter design of Da Vinci (1488), the hot-air balloon of Bartolomeu de Gusmão (1709), and the Wright brothers' airplane (1903) to name a few historical examples. In the present day, how to fly is no longer the focus of research, but how to do it more efficiently.

There are many approaches that can be taken to improve the efficiency of aerodynamic surfaces. While avionics, thermodynamics and structural mechanics should be studied and further developed to reduce cost or yield better performance in the cockpit, engine and the structure of fuselage, fluid mechanics and specifically aerodynamics helped by CFD, need to be ultimately mastered for a comprehensive increase in lift, cargo capacity and autonomy. The present work focuses its efforts in the latter subject to explore the capabilities of active flow control applied by original means.

Whether on turbine blades or wing aerofoils, active flow control has been proven to be important for lift enhancement, drag reduction, mitigate vibration and aeroacoustics issues; enhance performance by reduction of fuel consumption and improvement of high angle of attack handling qualities. Synthetic jets play an important role in the spectrum of active flow control, and have consequently been the subject of research for the past years.

The first paper where the phrase “synthetic jet” appeared, dates back to 1954 marginally stating an application for fuel’s cooling ¹, but it was not until 1972, when an appropriate mathematical treatment together with a simple experimental validation was published². Glezer³ published in 1988 “The formation of vortex rings”, which served as a base for the future work on flow control with synthetic jets⁴.

By 2014, more than a thousand papers were written with the phrase “synthetic jets” in their title. To have a better insight about the trend of this amount in the past years, a fast approximation was made with Google Scholar with the help of library access links such as Cranfield University Library (full text), Cranfield University Pro Quest (Full text) and Open WorldCat (Library Search). The filters used were: a) no citations b) no patents, and c) only published papers. Three specific filters were evaluated: 1) the whole phrase “synthetic jet” within the title of the article, 2) the words synthetic, jet and experimental in the title of the paper, and 3) the words synthetic, jet and CFD in the title. The results of this enquiry can be seen in Figure 1-1 below.



Figure 1-1 General trend in the investigation on 'Synthetic jets' by 2014

As there is an evident area of opportunity for research in the topic of active flow control enhanced by synthetic jets through a CFD approach, one of the scopes of this work is to cover a gap in the broad band of knowledge where the interaction of three subjects: Computational Fluid Dynamics (CFD), Aerodynamics and Active flow control by the action of synthetic jets are discussed.

A synthetic jet (SJ), in the most generic approach, is a time-periodic alternate ejection and suction of fluid through an orifice. A sketch of the synthetic jet actuator⁵ is shown in the figure below, Figure 1-2, where an oscillating diaphragm is located at the bottom of a chamber, creating a synthetic jet which passes through a small orifice (often less than 2 mm) placed at the opposite face of the cavity.

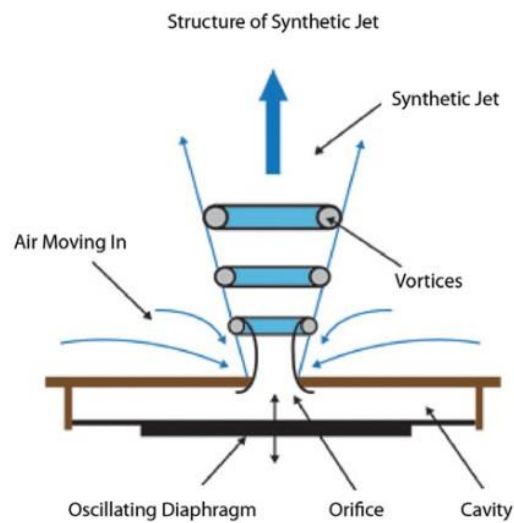


Figure 1-2 Structure of a synthetic jet

One of the reasons of synthetic jet's popularity, is because they can insert linear momentum in the system without net mass injection across the flow boundary⁶.

Boundary-layer separation control by the action of synthetic jets has been widely treated over the past years to actively improve aerodynamic performance; its success as an active flow control device has been extensively reported by several authors^{4,7,5,8,9}. In the literature, some examples of constant perturbation close to the boundary-layer for flow control purposes can be found^{10,11}, e.g., Choi et al.¹² proved that the drag, produced by the skin-friction and the wall shear stress of the turbulent boundary-layer can decrease by oscillations of the surface, which is one of the concepts used by the Synthetic Jets. According to Glezer⁶, local streamlines can be displaced by the interaction of a synthetic jet with the external cross flow and therefore prompt an apparent change in the shape of the aerodynamic surface.

The location of actuation when controlling separated flows is very important. To achieve optimal results, the actuation needs to be located at or just upstream from the point where the flow naturally separates⁹. This dependence is true for both active and passive flow control techniques, but the active control has an added benefit as their operational frequency and amplitude can be tailored for a flow situation, which means a broader range of operation and a lower dependence in actuation location. However, when the flow is not separated, (at low angles of attack for example), the effectiveness of control input is negligible¹³. A benefit of active control techniques, particularly synthetic jets, is that they can operate at various frequencies, in this instance they can take advantage of the natural flow's field receptivity to harmonic frequencies, more dramatically modifying the flow compared with passive or steady methods.

At low excitation levels, excitation at or near the unstable frequency of the separating shear layer may lead to up to 50% increase in post-stall lift¹⁴

1.2 MODERN RESEARCH

In 2004, several institutions around the world worked with NASA to study the capability of Reynolds-averaged Navier-Stokes (RANS) simulations using three validation cases with synthetic jets. Rumsey¹⁵, who compiled the results, reported that although some details of the velocity field close to the jet exit were not captured by the numerical analyses, the general trend was reproduced with reasonably good accuracy compared with experiments. Four years later, the outcomes of this international work were published including some new results obtained after the workshop¹⁶, where in the majority of analysis, computational results were compared against PIV and hot-wire measurements.

One of the conclusions regarding the numerical analysis is that when computations matched experiments better at the slot exit, they tend to match better into the field and that the simple Spalart-Almaras¹⁷ and $k-\omega$ SST¹⁶ models appeared to do reasonably well for many features of interest¹⁶, nonetheless if there are resources available, URANS and LES sometimes are required for a better description of the flow. This yields some insights in the modelling for the present study.

In one of the three validation cases used for the workshop, the effect of a synthetic jet was modelled over a wall mounted hump at 65% of the chord. The geometry was initially proposed by Seifert^{18,19,20} to study periodic excitation and steady suction or blowing to control the flow at high Reynolds numbers. This case, known as “NASA 2D Hump”, has been used for many further validation studies, e.g. Gan et al²¹, who investigated the simple suction state, Greenblatt²², who performed a number of experimental analyses, and Rumsey²³, who addressed

the oscillatory flow behaviour and compiled the work presented in the workshop¹⁵. The NASA 2D Hump has been found to be useful for validation of numerical analyses of synthetic jets because of its geometrical simplicity and well documented range of results, with a large set of computational and experimental data from different institutions, which means different approaches in turbulent models, grids size, time steps and so forth. Many different configurations of synthetic jets have been studied in the past. Also, a new type of jet flow actuator has recently been developed to achieve suction and oscillatory blowing, which however requires net mass flow to operate^{24, 25}.

Different configurations of synthetic jets have been studied; rectangular^{26,27}, circular²⁸, dual arrays in the vicinity of the leading edge²⁹, and even suction and oscillatory blowing actuators, which inject mass into the system²⁴. In more recent studies, several authors have examined the idea of Directed synthetic jet (DSJ) pursuing the same objective^{30,31,32}. The term DSJ was firstly addressed by McCormick³³, who analysed the efficiency of a curved neck of jet regarding the power input to energize the flow. Years later, a parametric analysis over the influence of the angle of the neck was made, which revealed that DSJ has less complexity and less parasitic energy cost, even though such a comparison depends on the pressure available and electrical sources³³. Razvan³¹ reported that the position of the DSJ may affect the post stall lift recovery of an aerofoil, and Lee³⁰ showed that the orientation of the DSJ is linked with the momentum coefficient^a, which increases as the jet neck angle becomes more parallel to the flow.

^a The Momentum coefficient, as defined by Amitay⁴⁵ is: $C_\mu = \bar{I}_j / \frac{1}{2} \rho_0 U_0^2 c$, where $\bar{I}_j$ is the time-averaged jet momentum per unit length during the outstroke, ρ_0 is the freestream fluid density, U_0 the freestream velocity and c is the chord or a characteristic length. Defining the averaged momentum as $\bar{I}_j = \frac{1}{\tau} \rho_j b \int_0^\tau u_j^2(t) dt$, with $\tau=T/2$ or half period of the diaphragm motion, ρ_j the jet

The synthetic jet has been studied extensively in the past few years, although there is still a lot of work to do to improve the understanding of all parameters affecting the flow. The effects of jet frequency, amplitude, slot and cavity size and so forth have been investigated to some extent. An interesting gap in the knowledge about these active flow control devices is the treatment of the inlet velocity direction of the jet. In this work, the idea of changing the inlet direction will be introduced, for which a numerical analysis would be performed to understand the potential benefits of changing the inflow and outflow vectors of a synthetic jet separately for active flow control capabilities. This novel configuration is referred to as Bidirectional synthetic jet, (BDSJ) whose target is to induce the Coandă effect downstream due to the jet outflow directed, and also subtract the upper flow with minimal perturbation for efficient suction of the upper boundary-layer.

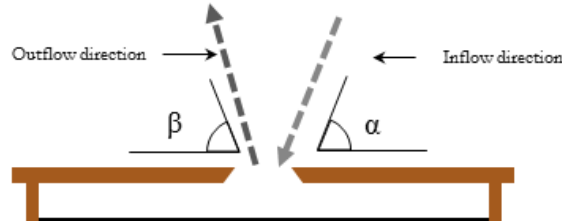


Figure 1-3 Schematic diagram of a BiDiractional Synthetic.

density, b the jet orifice width, and $u_j(t)$ is the phase averaged velocity at the jet exit plane. As can be appreciated from the above equation, the momentum added to the flow is given by the momentum of the jet, normalized by the freestream momentum, which is typically the product of the dynamic pressure with a characteristic area.

1.3 SCOPE OF THE PROJECT

The main objectives of this research project are summarized as follow:

1. Explore the idea of a Bidirectional synthetic jet and its potential benefits as an active flow control device.
2. Compare the BDSJ with classical and directed synthetic jets (if the inflow (suction) and outflow (ejection) directions of the jet are optimized separately) in terms of the separation bubble length size. Looking for the flow direction that decreases the most the separated region or enhance the lift/drag ratio.
3. Develop a basic understanding about the BDSJ, its properties and ideal intake-outflow angles as well as discover similitudes and differences against similar active flow control approaches.
4. Open a new area of research that may lead to an improvement in the efficiency of synthetic jets for the flow control in the automotive or aeronautical industry.
5. Develop a first stage design of the Bidirectional Synthetic Jet, Specifying dimensions and relationships between the different components which would lead to a better design in the future.

1.4

The present work is composed of seven chapters. The first two chapters correspond to Introduction and Literature review, where all the basic theoretical concepts are introduced and clarified to the reader so they can better understand the aspects enclosed in this thesis. In the third chapter, a further explanation to the computational tools used as well as the procedure taken to obtain the results in the following chapters is presented. In the fourth chapter, a comprehensive

validation of the usage of a Bidirectional synthetic jet over a well-known geometry is presented and developed, as well as compared with other common configurations. Chapter five explores a numerical engineering application of the Bidirectional synthetic jet as an Active flow control device over a NACA23012 profile under a subsonic flow, and some of its basic parameters are established and modelled yielding to further recommendations on the conceptual design. In chapter six, a brief exploration on the conceptual design of the device is taken, discussing ideas regarding the conceptualization and design. The last chapter concludes and further discusses the key aspects touched by the present project.

2 LITERATURE REVIEW

The literature embraced by this set of studies include more than two hundred papers revision and a careful selection of the major researches in the field.



2.1 AERODYNAMICS

2.1.1 Lift

A number of sources such as non-specialised books and magazines provide explanations of why lift occurs in aerodynamic surfaces; all of which are relevant but lack a fully comprehensive knowledge, without which; make it impossible to find an accurate solution.

One common explanation states that the flow around the aerofoil exerts a lift force, which is true, but an incomplete conclusion as there is no mention of whether the lift force influences the flow. Another popular simplified explanation is that; if two particles at the top and bottom of the profile enter the leading edge at the same time, the upper particle *must* travel faster because the curvature of the profile makes the path longer, which would result in both particles reaching the trailing edge at the same time. This is an incomplete description because the particle does not *have* to reach the exit at a particular time, moreover if we analyse deeper to see what happens at the top and bottom of a wing to a parcel of particles at the leading and trailing edge, we will see that the upper parcel of particles not only enters faster, but also leaves the profile sooner. A more detailed explanation is the Bernoulli Effect, which states that the cause of lift is a decrease in pressure on the upper surface due to an increment in the velocity, but this explanation is inaccurate for the same reasons, it is true that pressure and velocity are coupled, however it states that the low pressure is the result of the increment of velocity, and not as part of the cause, it does not explain why the high velocity gets there.

In momentum based explanations, it is generally claimed that the shape of the aerofoil produces a downward momentum by deflecting air below it, which, by

Newton's third law, the aerofoil must have an upwards force exerted on it by the air.

All previous explanations, describe with superficial accuracy the reasons for lift to exist, but if it is required to be comprehensive, it's important to understand that lift is a phenomenon that always involves a seamless combination of pressure difference and momentum transfer. According to Shevell³⁴, these explanations are not enough, stating that the Kutta-Joukowski theorem and circulation^b, give the only acceptable explanation of lift, however, McLean³⁵ states that this explanation only transfers the question from where the lift is generated to where the circulation is coming. Also let us notice that Hoffman and Johnson³⁶ computed the non-viscous Euler's equations and found lift and drag without any presence of viscous boundary-layer in a flow past a circular cylinder, concluding that lift and drag mainly results from the turbulent nature of the flow and only to a small part from viscous boundary-layers.

An example of one of the simplest equations to Mathematically treat the lift is as follows:

$$L = \frac{1}{2} \rho V^2 A C_L \quad (1)$$

Where L is the Lift force, V represents the airspeed, A the surface area^c and C_L is a lift coefficient characteristic of aerodynamic surface. In the case of wings, for instance, the value of C_L is co-related and to some extent proportional with the Reynolds Number and the angle of attack (AOA).

^b The circulation ($\Gamma = \oint_C \mathbf{v} d\mathbf{l}$), is defined as the line integral of the velocity along a closed curve.

^c The value of this area could be one of three types: Frontal area, which is the area as seen from the streamlines of flow; Planform area, which in the case of wings would be the chord times the span; and Wetted area, which considers the separation of the boundary-layer.

It is known that one of the methods to increase the value of C_L is by delaying boundary-layer separation, which is one of the aims of the present work.

2.1.2 Drag

The total drag is commonly perceived as the result of different types of drag inputs, for instance, in an airplane, the total drag is the sum of parasite drag (due to the shape or form of the non- lifting surfaces), with the pressure and viscous drag induced by the lifting surfaces. (see Figure 2-1) Viscous or frictional drag is incurred from frictional resistance of the surface passing through the air. Within a wing for example, this drag does not change significantly with AOA but increases moderately at high airspeeds. Pressure drag, however, is the result of the lift produced; if this is increased by the AOA, in the case of an aircraft, it would also generate downward velocities and vortices that will induce drag.

The theory of drag, except for the flat plate, is still weak due to the unpredictability of the separation region, as boundary-layer theory can estimate accurately the separation point but is unable to predict the pressure distribution in the separation region. Nevertheless, it is safe to say that drag is a force, therefore poses magnitude and direction and in a similar way as the lift force, is defined by an equation of the form:

$$D = \frac{1}{2} \rho V^2 A C_D \quad (2)$$

$$C_D = c_{D,pressure} + c_{D,fricction} \quad (3)$$

$$C_D = \frac{1}{\frac{1}{2} \rho V^2 A} \int_S dA (p - p_0) (\hat{n} \cdot \hat{i}) + \frac{1}{\frac{1}{2} \rho V^2 A} \int_S dA (\hat{t} \cdot \hat{i}) \tau_w \quad (4)$$

Where C_D is the drag coefficient, which, as it was pointed out, have contribution from pressure and friction, p and p_0 are the pressure at the surface dA and at far upstream respectively, $\hat{n}$ represents the normal vector and $\hat{t}$ the tangential vector to the surface of area dA , so it can be formed a unit vector to $d\hat{A}$ with the help of the unit vector $\hat{t}$, whereas τ_w represents the shear stress acting on the surface dA .

The contribution of each pressure and friction drag depends on the form of the object and the flow that surrounds it. For a flat plate for instance, if it is positioned in the stream wise direction, the contribution of drag would be mainly frictional, whereas if it is positioned perpendicular to the flow, there would be a higher pressure drag contribution to the total drag coefficient.

One of the ways on reducing the skin friction drag is by maintaining as much as possible the boundary layer in a laminar state. Turbulent boundary layer thickens more rapidly and produce more skin friction than laminar boundary layer, thus reducing the extent of turbulent flow reduces the drag.

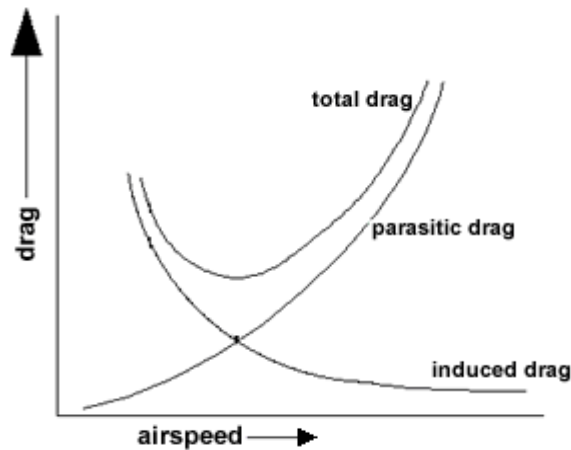


Figure 2-1 Total drag plot.

2.1.3 Coandă effect

The Coandă effect^d is a physical phenomenon in which a fluid jet tends to stay attached and follow a curved surface. This phenomenon occurs because the pressure field adjusts to provide the required force to accelerate the fluid towards the curvature's centre. In other words, the centrifugal force generated when the jet follows a curved path is compensated by a pressure gradient perpendicular to the direction of the jet (another example of how coupled pressure and velocity are). This effect might be used to maintain the jet momentum of the outlying jet in a region where the separation of the boundary-layer is more likely to occur.

The illustration in Figure 2-2, shows an example of the Coandă effect. In the figure a) an isolated free jet is shown in a quiescent air condition, as a curved surface get closer to the jet, (figure b) the jet changes direction following a concave path (figure c)

The phenomena discovered by the Romanian aerodynamicist Henri Coandă, for which the term “Coandă effect” was coined, was initially restricted to power-driven jet flows, where the total pressure of the jet is higher and its phase is the same as the surrounding fluid.

Anderson³⁷ states that viscosity is what makes a fluid stick to a surface, as explanation to the Coandă effect, while McLean³⁵ states that in the Coandă effect, the viscous forces make no direct contribution to the turning of the flow when the surface is curved.

The Coandă effect has been used in a number of applications, For example, the effect evoked by the tangential blowing at the rear of a two dimensional car

^d Named after Romanian aerodynamics pioneer Henri Coandă (1886-1972), who was the first to observe and apply the phenomena in 1910.

shaped body, may reduce the induced drag by up to 10 percent³⁸ and eliminate recirculating regions. The Coandă jet¹¹ controls circulation in an aerofoil, enhancing lift for short-distance take-off, and landing.

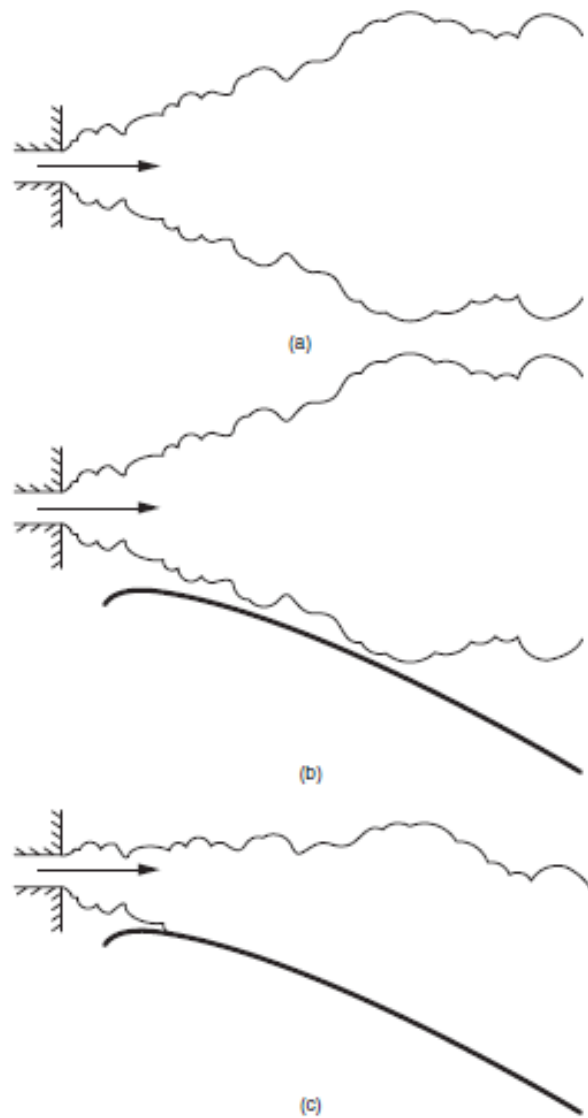


Figure 2-2 Illustration of the Coandă effect.

2.1.4 Boundary-layer

The concept of boundary-layer was first conceived by Ludwig Prandtl, an unknown 29 year old physicist at the time; whose work, published on a paper presented in 1904 at a congress of mathematicians in Heidelberg, named “On the motion of fluids with very little friction”³⁹ deal for the first time with shear stress and no-slip condition close to the surface of the element under analysis. Prandtl suggested a solution to D’Alambert paradox^e by stating it assumes wrongly a slip condition, modelling zero friction, therefore the zero drag solution^f. The boundary-layer is a thin layer close to the wall, where the effects of viscosity are highly important, as the velocity in this region tends to zero.

With this concept, new topics arise such as skin-friction, flow separation, lift, drag and so forth. In the theory of boundary-layer, significant simplifications from the Navier-Stokes equations can be made to correctly predict laminar boundary-layer profiles, as it is assumed a steady incompressible flow with constant viscosity.

Within the laminar boundary, where the flow is smooth and stable with no disturbances, only small Reynolds numbers can be found with low wall shear stress and low heat transfer. It is prone to separation under weak pressure gradients and as it walks along the leading edge it increases in thickness. Eventually, a transition occurs, and the layer becomes turbulent, where chaotic three-dimensional unsteady disturbances appear at moderate or high Reynolds numbers. In this layer high wall shear stresses and heat transfers are present.

^e D’Alambert paradox, brought to the public by the French mathematician Jean le Rond D’Alambert in 1752 stated that for an incompressible and irrotational (potential) flow the drag obtained by solving the Euler’s equation is zero.

^f Hoffman and Johnson³⁶ have claimed in 2008 the real solution to D’Alambert’s Paradox by solving computationally the Euler’s around a cylinder equations obtaining drag with no-viscous boundary-layer.

This turbulent layer offers a higher resistance to the flow separation under pressure gradients, and is the trade-off on drag reduction from the aerodynamic perspective.

The dimensionless distance from the wall y^+ is used when dealing with boundary-layers, and its equation is of the form:

$$y^+ = \frac{U^* \cdot y}{\nu} \quad (5)$$

$$U^* = \sqrt{\frac{\tau_w}{\rho}}; \quad (6)$$

$$\tau_w = \mu \left(\frac{\partial u}{\partial y} \right)_{y=0} \quad (7)$$

where U^* is known as the friction velocity or shear velocity. Within the boundary layer, we can mathematically treat differently two sections for $y^+ < 0.1\delta$. The viscous layer, for values of $y^+ < 5$, where the dimensionless velocity equals the dimensionless distance from the wall, $U^+ = y^+$, and the law of the wall, where the relationship between y^+ and U^+ follow a logarithmic formula.

$$\text{if } 30 < y^+ < 0.1\delta;$$

$$U^+ = \frac{1}{\kappa} \log y^+ + C \quad (8)$$

$$\kappa \approx 0.41; \quad C \approx 5.2$$

The layer between $5 < y^+ < 30$ is called the buffer layer, and does not have any mathematical treatment, but the velocity may be obtained by an interpolation between the two other approaches. Some authors may refer to the log layer as the “fully turbulent wall layer” and it is the section of the boundary layer which is sufficiently close to the surface that the inertia effects can be negligible, but at

the same time sufficiently far, so the molecular or viscous stresses are negligible compared to the Reynolds stresses⁴⁰.

Figure 2-3 shows the different zones within a turbulent boundary layer for a typical smooth flat plate.

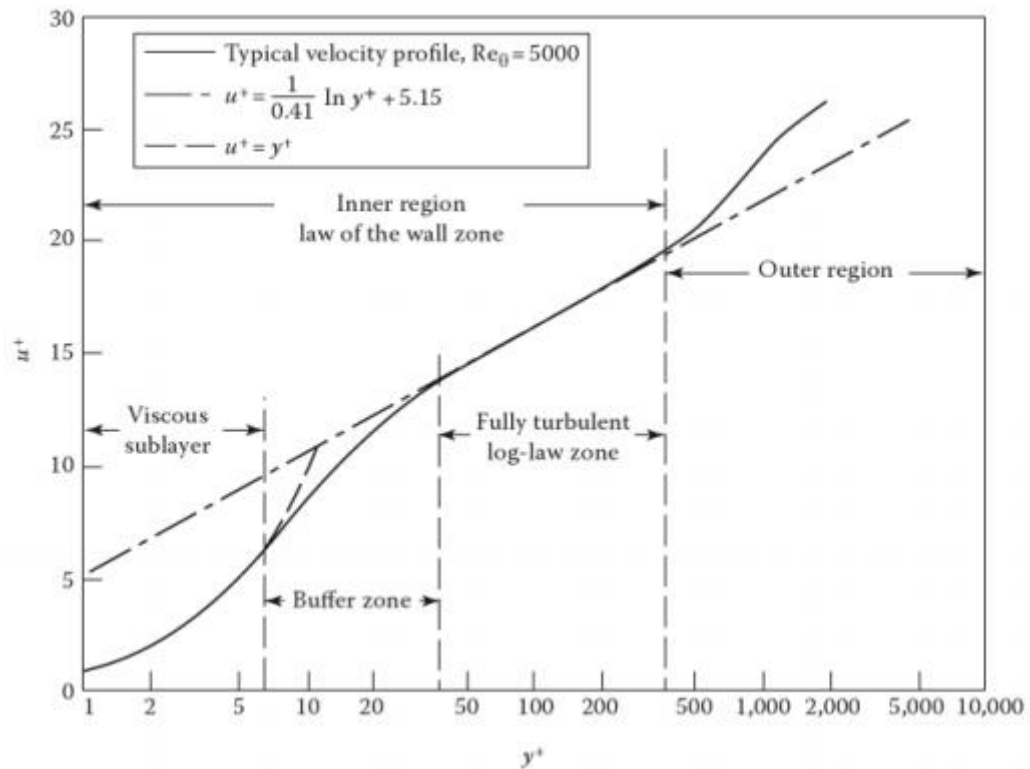


Figure 2-3 Zones within a turbulent boundary layer.

Very often, the study of a Boundary-layer is focused on maximizing lift and reducing drag. According to Hazen⁴¹ there are two important parameters to control within a Boundary-layer, which are transition^g and separation. The main objective of the synthetic jet for active flow control is to avoid separation by

^g Transition to turbulence occur because the laminar boundary layer become unstable to some types of disturbance. The location in which transition occurs depends on the nature of disturbances to the laminar boundary layer and its stability

accelerating the transition. As for the turbulent boundary-layer, the forces needed to face separation are higher.

Four different methods could be used to control the nature of the boundary-layer: Adverse pressure gradient, forced mixing, suction and jet expulsion. Adverse pressure gradient, for instance, have a strong influence in the stability of the laminar boundary layer, as it can damp out disturbances such as noise, vibration or air stream turbulence fields. This effect could be demonstrated when an aerofoil subjected to airstream in a wind tunnel changes its angle of attack. The adverse pressure gradient distribution along the profile changes dramatically, as the AOA is increased, resulting in the transition point moving forward, and a complete different boundary layer distribution. Changing the geometry could provide favourable pressure gradient, as in the deflection of the flaps or slats in an aerofoil.

Another method to control the boundary layer is forced mixing, as when mixing the outer flow with the one next to the surface with vortex generators, the separation of the turbulent boundary layer is more difficult.

Synthetic jets use a combination of suction and expulsion of air to get an effective result.

Effects of suction and expulsion of air in the boundary-layer: In an aerofoil, suction might be applied to reduce the thickness of the boundary-layer by removing the low momentum layers next to the surface, thus layers more stable appear and the transition to turbulence is postponed. In an experiment with suction slots within an aerofoil, Hazen⁴¹ concluded that applying suction reduces the streamwise momentum lows in the wake. If suction takes place entirely within the separated area, the quantity of suction required for reattachment could be

extreme. A very high rate of suction could be applied to attach the boundary-layer to the surface, but the disadvantages of maintenance and power consumption overweight the results⁴¹.

There is a big impact of the boundary-layer over the general performance of aerodynamic surfaces; therefore controlling it becomes key for most applications. A few methods are available to achieve such endeavour: e.g. Suction; as it draws-off the low-energy air below the boundary-layer, causing the higher layers above to be lowered nearer to the surface of the aerofoil, making the aerofoil effective at higher angles of attack, as delaying separation increases the angle of stall. Air ejection or blowing; as it increases the momentum of the air, causing similar advantages as suction. Vortex generators; which creates slight turbulence in the layers, which makes them more resistant to separation by adverse pressure gradient. Or the recently studied synthetic jets, which equally add momentum to the system but without net mass injection. It is known that the momentum required to keep the boundary layer attached is lower than the one required to reattach it from a separated state.

The basic outline, the derivation of the equations that defines it, and a deeper treatment on the laminar, transitional and turbulent boundary-layer can be found in the fine work by Schlichting⁴², and Kluwick⁴³.

2.2 SYNTHETIC JETS

A jet flow is defined by Mohseni⁹ as a class of fluid flow in which a stream of one fluid mixes with a surrounding fluid which might be at rest or motion. Glezer, in his paper “synthetic jets”, recognizes professor D. R. Coles^{h*} as the first who conceived synthetic shear flows⁶, and the first time a synthetic jet was treated numerically for modelling and validation was in the 2D study of Kral⁴⁴.

A synthetic jet, also referred to as a zero-net mass flux (ZNMF) jet, is formed by momentarily and periodic subtraction and ejection of the surrounding fluid. They can be produced either by a mechanical, acoustic, electromagnetic or piezoelectric drivers, often situated inside a cavity or chamber. Although mass is not added into the system, there is a momentum injection, as the flow is energized by the oscillatory movement of the membranes or diaphragms; they are intrinsically pulsatile, and its nature set them apart from the dynamics of continuous steady or pulsed jets.

2.2.1 Applications

Separation control: Artificial or synthesized jets inserted into a boundary-layer act as a turbulence enhancement, introducing momentum in the lower boundary-layer, postponing the transition and therefore the separation point, which in aerodynamic applications may result into lift improvement or drag reduction. Amitay⁴⁵ and Glezer⁴⁶ used a synthetic jet to control the flow over a symmetric aerofoil with cylindrical leading edge, moving the stall angle from 5° to 17.5° AOA showing that the location and amplitude of the jet, affect the length of the reattached flow. They reported an increase in lift by up to 100%, and decrease in

^{h*} Other pathfinders in the active flow control through synthetic jets that may be considered for their work are Ari Glezer, Douglas R. Smith, and Michael Amitay.

pressure drag by 45%, due to the action of the active control and reattachment. According to their results, once the frequency of the oscillating membrane is of the same order as the shedding frequency of the aerofoil when stall occurs, ($F^+ < 4$), any increase in actuation frequency is inversely proportional to the recovered lift to pressure drag ratio, while at higher frequencies ($F^+ > 10$) the C_L/C_D seems to become independent to the frequency. Lee⁴⁷ in his research concluded that for effective flow control, the actuator or membrane must produce strong enough vortices.

Jet Vectoring: Jet vectoring could be applied for heat or noise reduction as well as thrust deflection. Smith⁴⁸, Dahai⁴⁹ and Glezer⁵⁰ investigated experimentally the interaction of a synthetic jet with a flowing jet, which leads to the formation of a recirculating flow which creates a pressure drop between the two flows, attracting the main jet towards the synthetic jet. They found that the volume flow rate of the main fluid is the main parameter affecting the vectoring force for a given strength and jet speed of the synthetic jet.

Mixing enhancement: Chen et al.⁵¹ used a synthetic jet with the objective to improve fuel efficiency inside a turbine combustor engine by experimentally mixing and measuring two streams of cold and hot gas steam. They demonstrated that the mixing inside a simulated gas turbine combustor can be enhanced by the vortex created by the synthetic jets acting as a bluff body in the crossflow. Chiekh et al.⁵² used synthetic jet to control, vector and enhance the mixing of the flow out of a diffuser, using one pulsed jet, for vectoring, and two wall pulsing jets for enhancing mixing by more than 60%.

Thermal management: Mahalingman⁵³ studied the heat dissipation characteristics of a synthetic jet for the cooling of an integrated circuit. He found an increase of

2.5 times the average heat transfer coefficient of the channel flow where the synthetic jets were used. Pavlova⁵⁴ found that synthetic jets work better than continuous jets in removing heat from a constant heat flux surface for the same Reynolds number. They also found that for short distances from the heat source, high frequencies (of the order of 1200Hz) remove heat better than low frequencies, whereas the opposite applies at further distances.

Aero-optics: Vukasinovic⁵⁵, Gordeyev⁵⁵ and Kibens⁵⁶ studied the synthetic jet to control the wake after a flow passes a bluff-body turret, to enhance transmission of optical wave fronts through regions of separated turbulent flow. They found that for a fixed actuation level, the suppression of spectral components of optical distortion was 30% for frequencies between 0.5 and 25 kHz.

Other applications include cavity oscillations⁵⁷, electronic cooling⁵⁴, structural vibrations⁵⁸, control^{59,60,61,62} aerosol dispersion⁶³, active control applied on blade turbines⁵⁸, rotors⁶⁴, flow-induced noise reduction, etc.

Regarding the differences between synthetic jets and continuous jets, Smith⁶⁵ realized that synthetic jets grow faster than continuous jets, as they are mainly dominated by vortex pairs which entrain more fluid than continuous jet columns.

To obtain accurate results, as stated by Rumsey¹⁶, not only the frequency and amplitude have to be faithfully modelled, but often the neck length, diaphragm area, slot size, the stroke lengthⁱ, or the amount of flow that comes in a single “stroke” of synthetic jet and in general the cavity volume^j as well. However, not

ⁱ The dimensionless parameter “Stroke” length or stroke ratio, is defined as L_0/h : where $L_0 = \int_0^\tau u_0(t)dt$ $u_0(t)$ is the velocity at the jet exit, $\tau=T/2$ is half the period of diaphragm motion for a SJ, or the time of discharge, for a more generic approach which will include the BDSJ and h is the jet exit slot.

^j The volume of the cavity is more important than its shape⁹.

in all publications the cavity is perfectly emulated. Tang⁶⁶ for instance, used a velocity profile to emulate the jet velocity at the exit and Kral⁴⁴ compared three different velocity distributions along the jet exit, finding that the top hat distribution agrees better with his experiments.

2.2.2 Parameters

Lee⁶⁷ concluded in a 2D direct numerical simulation of a synthetic jet under quiescent condition that the evolution of synthetic jets strongly depends on parameters such as the Strouhal^k number while jet formation⁶⁸ and the strength⁶⁹ of consecutive vortex rings are highly affected by the Reynolds^l number. Utturkar⁷⁰, and Holman⁷¹ in their study agree with this conclusion and include that if the jet shows compressibility effects ($M > 0.3$), also the Mach number should be considered. In general, the jet profile depends on cavity and slot geometry, external flow conditions and the operational parameters of the jet. Zhong⁷² found that the **Stokes number**^m and stoke length dictate the development of vortex rings in quiescent conditions. According to Tang and Zhong in the book of Daniel New⁷³, the behaviour of the Synthetic jet depends on eight parameters, the diaphragm oscillation amplitude, diaphragm oscillations frequency, diaphragm

^k The Strouhal number, named after the German physicist V. Strouhal, comes from the nondimensionalization of oscillating patterns of the form, $u = U \cos(\omega t)$, turning into $u/U = u^* = \cos(\omega L t^*/U)$, where the argument of the cosine is the Strouhal number $St = \omega L/U$ dimensionless forces, moments, friction and heat transfer are function of both Reynolds and Strouhal number.

^l The Reynolds number is a dimensionless parameter used to describe the ratio between inertia forces and viscous forces, and it is commonly used in fluid mechanics as a parameter to predict the transition from laminar to turbulent flow or to scale fluid dynamic problems, as it is used to determine dynamic similitudes between two flow cases. The mathematical expression to measure this value is $Re = \rho LV/\mu$ where ρ is the density of the fluid, L is a characteristic length of the system, V is the velocity of the fluid, and μ is the dynamic viscosity.

^m The Stokes number is a dimensionless number that characterises particles suspended in a fluid flow. $Stk = \tau_0 u_0 / l_0$, u_0 is the fluid velocity, l_0 is a characteristic length. $Stk = \sqrt{2\pi Re_L / L^2}$

area, orifice area, orifice depth or thickness, cavity volume, fluid kinematic viscosity and fluid density, all of which are included in the dimensionless parameters previously commented. Zhang and Zhong⁷⁴ commented in their study that the behaviour of synthetic jets issued into a boundary layer, based on dimensional analysis, is determined by the Stroke Length^a, the Reynolds Number based on the Stroke length, Re_L , the dimensionless frequency and the jet to freestream velocity ratio $VR = \frac{\bar{U}_{jet}}{U_\infty}$

Frequency: It is known that, good flow control performance is found at relatively high oscillation frequencies^{9,29} (200Hz)^o. In the experiments of Mossi⁷⁵ it was demonstrated the relationship between clamping pressure and diaphragm deflection and hence natural frequency. In the open loop^p experiments of Alstom⁷⁶ it was found that the RMS pressure distributions are sensitive to changes in the forcing frequency, particularly, the shape of the distribution changes. In the study of Ferro⁷⁷ conducted on a NACA0025 aerofoil, high frequencies excitations^q near the leading edge could reattach the flow and eliminate the wake's large-scale vortex shedding which decreased drag by 45%; however low-frequency excitations turn out to be more effective, as it led to an organization and suppression of the large-scale vortex shredding with an order of magnitude smaller required momentum to reattach the flow. In a review made by Greenblatt⁷⁸ on flow control by periodic excitations, it was shown that for most studies the optimum

^a The stroke length is defined as $L = \frac{L_0}{D_0} = \frac{\bar{U}_0 T}{D_0}$, which represents the length of a rectangular fluid column, expelled by a single stroke divided by the size of the diameter slot. The study of Jabbar⁶⁹ concluded that an increase in the stroke length lead to an increase in the strength of consecutive vortex rings for a synthetic jet in quiescent flow.

^o For comparative reasons, 100 Hz was used for this study instead.

^p Closed loop refers to a system in which the input obtain feedback from its own output. In an open loop, no feedback is registered.

^q In this study the High frequency correspond to a $St=58$, while the low frequency is in the region of $St= 0.84 - 9.9$.

dimensionless excitation-frequency^r is in the range of $1 \leq F^+ \leq 2.5$, while Amitay⁴⁵ noted that an order of magnitude higher than the natural shedding frequency is the most effective for the dimensionless frequency. G.Hong, in [79] and [80], found that the effectiveness of the synthetic jet to control boundary-layer is greater when the forcing frequency is in the range of the T-S (Tollmien-Schlichting^s) frequencies. Finally, for piezoelectric-based SJA, the frequency is directly related with the power requirements and therefore should be considered as an important parameter for the comprehensive efficiency study.

Amplitude: When controlling boundary layer to reattach separated flows, the ratio of the velocity at the jet exit to the freestream speed is a key parameter to consider and most of the experimental work shows that this value should not be less than unity. The amplitude of the peak velocity at the exit and the momentum coefficient are related intrinsically by definition, although some authors⁹ argue the momentum coefficient (C_p) is more pertinent. In the study of Holman⁸¹, relative phase and actuation amplitude between two closely spaced synthetic jets was varied to determine the effect on separation location on a NACA0025 aerofoil, concluding that the actuation amplitude coupled with synthetic jet location in the streamwise direction play a significant role in the effectiveness of the flow control. Goodfellow⁸², inspired by the work of Smith⁸³ and Amitay, characterized the excitation amplitude in terms of the momentum coefficient and determined that this is the primary governing parameter to control the flow; in this study it was concluded that when the excitation applied is above a certain C_p threshold, the flow around a NACA0025 aerofoil reattaches, and the wake topology changes significantly, leading to a decrease in drag of about 50%.

^r $F^+ = f_e L / U_\infty$, the dimensionless frequency equals the excitation frequency times the distance between the separation point and the trailing edge over the freestream velocity.

^s The instability arisen within a viscous boundary layer.

In studies over aerofoils like those of Lee⁸⁴, and Kim^{85,86} good flow control properties are obtained for high^t angles of attack when the velocity at the jet exit equals three to five times the freestream velocity. In the experiments over a rectangular wing made by Margaritis⁸⁷, where a set of synthetic jets were collocated at the tip to control tip vortex, it was concluded that the blowing coefficient is more significant than the modified frequency, which had little effect on this configuration.

Slot exit: To minimize the power required to control the flow with a SJ, the slot exit should be close to the separation region. Kim⁸⁸ investigated computationally and experimentally two jet exit configurations, long rectangular slot and serial circular under quiescent and cross flow conditions; differences were found in the velocity decay slopes along the jet stream distance, playing the frequency a distinctive role as well. The interaction of the vortex generated by singular holes increases the turbulence spreading rate. The effect of slot orientation (spanwise and streamwise) was treated by Smith⁸⁹ and Aram⁹⁰ while Orkwis⁹¹ evaluated two sets of synthetic jets in and off-phase separated by certain distance. Xiong⁹² found that regarding the strength of the vortex pair in the streamwise direction, the slot width, followed by the cavity height and slot depth are the most influencing factors. Mangesh⁹³ mentioned that there is a noticeable effect of cavity dimensions on the velocity at the exit and the geometry of the orifice has an effect on the flow structures produced⁹⁴. Aram⁹⁰ investigated the effect of the orientation of two rectangular synthetic jets over the hump, finding that streamwise oriented slots obtain a smaller separation bubble size. Spencer⁹⁵ conceptualized a variable diameter synthetic jet, based on the swimming of a jellyfish, in which vortex rings

^t In these studies, AOA below 17° are not shown, but the trend suggest little lift increment is achieved for the lowest amplitudes.

are generated by the contraction and expansion of its body, finding an increase in momentum when the diameter contracts accordingly with the pulsing jet. An attempt to understand the impact of orifice orientation in crossflow was made by Buren⁹⁶, who found a relationship between the orientation of the slot, vortex formation and velocity field. In a numerical optimization of a synthetic jet applied on an aerofoil made by Montazer⁹⁷, it was found that the optimal length of the synthetic jet slot dynamically decreases as the angle of attack increases.

Cavity: As shown by the study of Krishnan²⁶, a flow axially symmetric is expected when a jet is coming out of a long cylindrical tube, whilst a spherical symmetry is expected when a jet comes from out of a small and thin orifice. In the Study of Kosasih⁹⁸, the generation of the vortex ring was experimentally and computationally compared on three different cavity shapes (half ball, tube and cone) with three diameters each (3, 5 and 8 mm), finding that for a conical cavity with the smaller orifice the highest velocity is produced, and no vortex ring could be generated with their biggest orifice diameter for any cavity shape. In the study of Feero⁹⁹, similar cavities were investigated but with constant volume, concluding that cylindrical cavity shapes maximize momentum flux, but more complex cavities reduce power consumption and improve efficiency.

Flow Field: The flow field surrounding the synthetic jet may be subdivided into near and far fields. In the near field the periodic development of the train of vortical structures begins and interacts with the surrounding flow; in this region both rectangular SJ and planar continuous jet exhibits similar characteristics²⁶. In the far field, there is an axisymmetric region where the jet starts to decay and the vortical structures break down regardless of the shape (similarly for both rectangular and circular jets).

Arrays: Orkwis⁹¹ set up a 2D analysis of a set of Synthetic Jets in and off-phase separated by a certain distance which when evaluated, an improvement in the momentum coefficient behaviour against a single jet was found.

One of the objectives of the present work is to verify whether some or most of these parameters can be equally used to measure the performance of the bidirectional synthetic jet, as this is an original hypothesis. As in the BDSJ the inflow and outflow direction changes, the behaviour and interaction with either quiescent air or cross flow is affected. The vortical structures may differ, as their behaviour could be the sum of a pulsed subtraction jet with a pulsed ejection jet.

2.2.3 Flow structures with quiescent and crossflow

A synthetic jet is produced by the exchange of momentum in a train of vortices when ejection and suction of fluid across an orifice are alternatively occurring across an orifice, resulting in zero net mass flux. The flow during the suction stroke is nominally axisymmetric, whereas the flow during the ejection stroke is narrowed to the centreline of the jet. During the ejection, the flow separates at the edges of the orifice and forms vortex rings or vortex pairs which constantly moves away from the centreline of the jet. The influence of the reversed flow induced by the suction on the vortex pair is strongly dependant on its distance of from the orifice as well as its own strength.

When the synthetic jet is integrating with crossflow, the structures produced and the effect is considerably different. An illustration for the interaction of synthetic jets with a cross flow is shown in Figure 2-4, taken from Amitay⁶, which shows streamlines of smoke around a circular cylinder. On figure a) the flow is stable with no perturbances and appears to be separated around 80° , in figure b) two adjacent rectangular synthetic jets actuators, operating in phase, are placed at 60° , although the difference is subtle, it can be appreciated a relatively small deformation of smoke in the top of the cylinder. In figure c) the jet is placed at 180° , and the flow appears to be attached almost in the whole surface of the cylinder. Finally, in figure d) the jets are still placed at 180° but the bottom one is leading by $2\pi/3$ out of phase, resulting in a vectoring effect

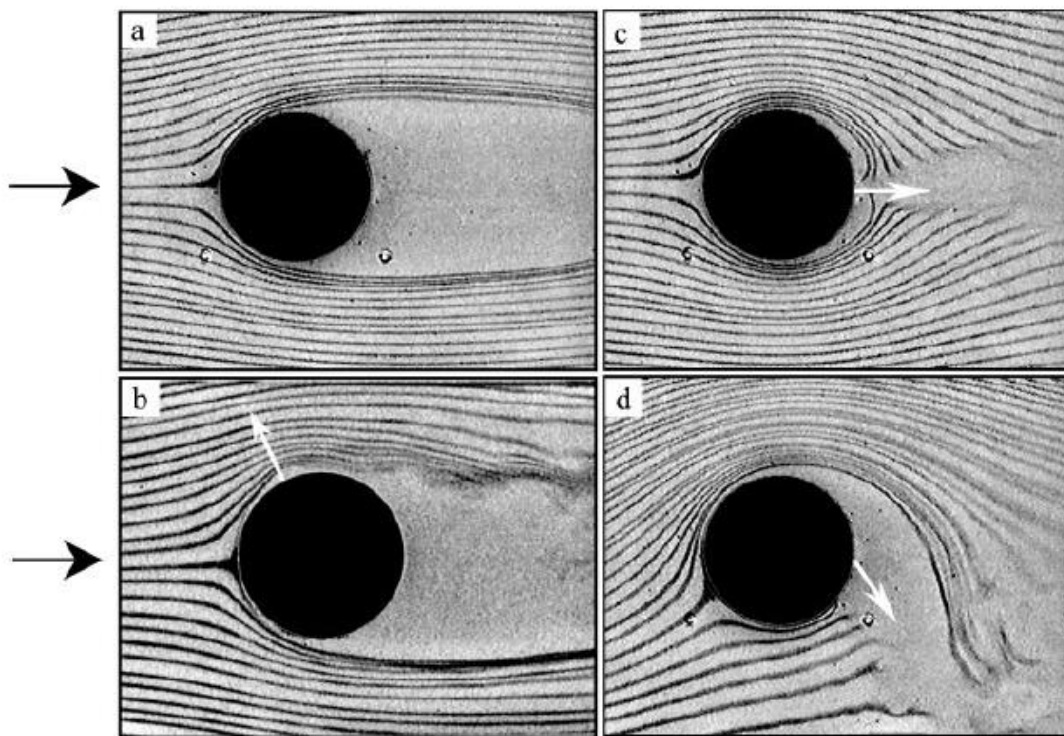


Figure 2-4 Flow around a circular cylinder.

2.2.4 Flow separation control

The experimental investigation of Ma^{100} on the flow control around a circular cylinder with synthetic jets concluded that the natural frequency of the wake vortex (around a cylinder submerged on water) can be changed by an increase in the excitation frequency. According with their PIV results, the synthetic jets can alternate the vortex shredding mode behind the cylinder, switching between symmetric and antisymmetric modes.

2.2.5 Further Research

As it was seen in the previous sections, even though some of the parameters that modify the behaviour of the synthetic jet are still not fully developed or quantitatively studied or measured, most of the basic factors that characterize a synthetic jet are already to some extent studied either analytically, experimentally or numerically. However, one key component which has been overlooked is the influence of the direction of the jet exit compared with that of the inlet. In other words, the effect of the angle of the intake flow with respect of the outflow stroke has not been mentioned in any of the previous studies, therefore there is a need to address this particular gap in knowledge.

3 METHODOLOGY

The theory and computational methods utilized by this study are shown in this chapter.



3.1 COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics, commonly known as CFD, is the computationally aided analysis of fluid flow systems that may include mass and heat transfer, as well as chemical reactions. This technique has become very powerful and indispensable for aerodynamic analyses, as the results obtained by the approximations and assumptions taken, emulate with enough accuracy most engineering problems. Running a CFD code while being unaware of the theoretical background of it may jeopardise the simulation and lead to unrealistic results. Making poor choices of input parameters may result in a solution that is not as accurate or representative as it should be, and not understanding the limitations of a code can lead to a misinterpretation of the results.

First, it is important to understand which equations the code is solving, how these equations model the physics of the problem, and how the code solves them. Regarding a CFD code, it is not only the basic equations of motion, but also the model of the turbulence and transition, which determine the applicability of the method to the problem, and the physical fidelity of the results. In addition, the influence of the grid size or time-step will determine whether adequate numerical accuracy can be obtained. A complete development of the equations and methods of CFD is out of the scope of this chapter, where only key equations and concepts will be brought to the reader. However, a comprehensive introduction and further studies on CFD methods can be found in literature from, for example, Malalaskera¹⁰¹, Ferziger¹⁰² and Hirsch¹⁰², while a deeper treatment on turbulence modelling can be obtained from Wilcox¹⁰³ or Lesieur¹⁰⁴.

3.1.1 Numerical algorithm

There are mainly three methods to solve numerically partial differential equations in the form of algebraic equations. Finite differences, finite element and spectral methods, but many well-established CFD codes employ the finite volume method (a special finite difference formulation) as a numerical algorithm to solve the equations in question due to its intrinsic similarity with the underlying physical conservation principle, as this method is also known as the control-volume approach. In a nutshell, the finite volume method consists of three steps; control volume integration of the governing equations over the entire domain; discretization or formation of a system of algebraic equations, and solution of these equations by an iterative method. Firstly, the solution domain is subdivided into a finite number of small control volumes, which define the boundaries, not the nodes as in the finite difference method. The integral form of the conservation equation is then applied to each control volume and to the whole domain. The discretized equations set up at the centre of each control volume are then solved using an iterative method after the boundary conditions are applied.

3.1.2 Turbulence and its modelling

There is not an exact answer about why turbulence happens, but there are many models that attempt to predict its characteristics. There are three key points that stand out regarding turbulence: it is always three dimensional, fluctuating and chaotic. The Reynolds Number (inertial forces over viscous forces) gives an insight on whether the flow may be laminar or turbulent. For a flow around a cylinder, for instance, at $Re < 5$, the flow is laminar, attached and

steady, all the viscous forces are dominant, at $5 < \text{Re} < 40$ the flow separates, for $40 < \text{Re} < 200$, the separated flow becomes periodic, from $200 < \text{Re} < 350,000$ ¹⁰⁵, the laminar separation turns into a turbulent periodic wake and finally, for Reynolds values over 350,000 the turbulent separation is chaotic and the inertia forces are dominant. Turbulent flow is dominated not only by the object shape and dimension (large scale) but also by the motion and evolution of small eddies (small scales, which are challenging to compute), as unsteady^u aperiodic motion appear, and the flow exhibits chaotic spatial variations with a strong dependence from initial conditions and a wide range of scales.

To correctly calculate all the scales, extremely fine grids would have to be used to solve all the time-dependant Navier-Stokes equations for a sufficiently long time interval so that the fluid properties reach a statistical equilibrium. Spalart¹⁰⁶ in 2000, estimated that a direct numerical simulation for a full large-airplane configuration at flight Reynolds number will require at least 10^{16} grid points and $10^{7.7}$ time steps to reach a statistically steady solution. Coleman¹⁰⁷ stated that the number of grid points required for a 3D study will be of the order of $\text{Re}^{9/4}$. With the technology of 2017, this would require a prohibited amount of resources to compute, and for most engineering applications detailed solution of small eddies may be ignored, and that is why turbulence modelling appear.

Osborne Reynolds in 1894 applied a mathematical technique, known today as Reynolds decomposition (shown in Figure 3-1), to separate the velocity values in the N-S equations into its average and fluctuating part, giving birth to the Reynolds-average Navier-Stokes equations. This technique attracted little attention on the time of being developed¹⁰⁸, as nobody found a concrete

^u A Von Karman Vortex is an example of an unsteady laminar flow.

application for his method. However, years later and with the introduction and improvement of computational tools, the RANS equations form the foundation of turbulence modelling, as with this technique it is possible to calculate the mean flow without resolving the motion of turbulent eddies.

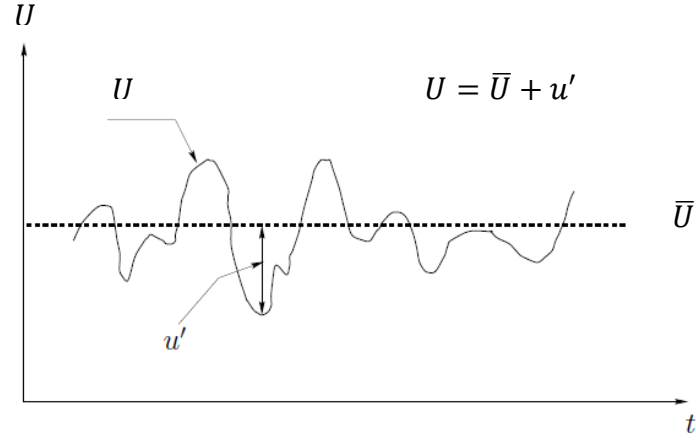


Figure 3-1 Reynolds decomposition;

By considering the relationship $\mathbf{U} = \bar{\mathbf{U}} + \mathbf{u}'$ and substituting it into the Navier-Stokes equations, new unknowns are obtained, which are known as Reynolds stresses, and turbulence modelling deals with shaping the Reynolds stress $(-\rho \overline{u'_i u'_j})$.

The following equation is known as the Reynolds-averaged Navier-Stokes equation (RANS).

$$\rho \frac{\partial U_i}{\partial t} + \rho U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} (2\mu S_{ji} - \rho \overline{u'_i u'_j}) \quad (9)$$

There are several methods to model turbulence. RANS^v based models can be subdivided into the first order (zero, one and two equation) and second order

^v The difference between RANS and URANS rely on the concept of time step. RANS use a local optimized time step for each cell and get fast steady state results whereas URANS uses a global

models (algebraic and Reynolds stress models). All first order models, also known as eddy viscosity models, are based on an assumption made by Boussinesq^w in 1877 which is an analogy between laminar and turbulent flow, and states that the turbulent shear stress can be expressed as:

$$\textit{Laminar Flow:} \left\{ \begin{array}{l} \tau_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu \delta_{ij} \frac{\partial u_j}{\partial x_j} \\ q_i = \frac{k}{c_p} \frac{\partial T}{\partial x_i} \end{array} \right\} \quad (10)$$

$$\textit{Turbulent Flow:} \left\{ \begin{array}{l} \tau_{ij}^t = -\rho \overline{u_i' u_j'} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \rho k \\ q_i^t = -\rho \overline{u_i' t} = \frac{k_t}{c_p} \frac{\partial T}{\partial x_i} \end{array} \right\} \quad (11)$$

$$\textit{Laminar Flow:} \left\{ \begin{array}{l} \tau_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu \delta_{ij} \frac{\partial u_j}{\partial x_j} \\ q_i = \frac{k}{c_p} \frac{\partial T}{\partial x_i} \end{array} \right\} \text{The previous}$$

The Boussinesq hypothesis has been widely challenged. Several authors question seriously his validity, Schmitt¹⁰⁹ for instance, studied numerically and experimentally the wake on tetrahedral blunt body, and found high discrepancies between DNS, LES and experiments with models using the hypothesis. However,

time step, therefore is time accurate. For some studies, and if the time step is small enough, RANS can capture small fluctuations, or unsteady behaviour, in the mean quantities and therefore the solution can be considered time accurate.

^w This hypothesis is only valid when the scale of the eddies is much smaller than the scale of the mean flow, and when the velocity fluctuations are highly isotropic.

for engineering applications and simple flows, this approximation is often sufficient.

With the previous assumption, as the Reynolds stresses are expressed in terms of the turbulent viscosity and kinetic energy, to close the problem, a method to find these unknowns is the focus of turbulence modelling. Choosing the right turbulence model could be one of the main issues while doing computational fluid dynamics analysis. Tang et al.⁶⁶ found in their analysis that RNG k- ε and standard k- ω models produced the best match between computational and experimental results for their synthetic jet, while You¹¹⁰ used LES to match results on flow separation. In this work, based on experience and a comparison on what the scientific community used for similar flows, three turbulence models were utilized; the one-equation Spalart-Allmaras model, the two-equation Realizable k-epsilon model and Shear Stress Transport (SST) k-omega model.

3.1.3 Spalart Allmaras

The S-A is a transport partial differential equation that models the turbulent viscosity. It was developed and published by Spalart and Allmaras in 1992 with the objective of modelling aerodynamic flows with one single equation, improving the approximation taken by the algebraic mixing length models, and providing a simpler alternative to the two equation turbulence models. The formulation and coefficients of this model, inspired by the work of Baldwin¹¹¹, were defined using dimensional analysis, selected empirical results^x and Galilean^y invariance. In its formulation, Spalart¹¹² used distance to the closest wall, and the model yields relatively smooth laminar-turbulent transition. It is known to be economical and accurate for attached wall-bounded flows, 2D-mixing layers, wake flows, and flows with mild separation and recirculation, but somewhat weak on massively separated flows, jet flows and free shear^z flows or decaying turbulence. This model, compared with the standard k- ϵ and k- ω has demonstrated improvements in the prediction of flows with adverse pressure gradient, but not as much as the SST k- ω model.

This model is based on the relationship:

$$\frac{\tau_{ij}}{\rho} = -\overline{u'_i u'_j} = 2\nu_t S_{ij} \quad (12)$$

^x The two dimensional empirical results used in the development of the Spalart Allmaras equation were 2D mixing layers, boundary layer flows on a flat plate and wakes.

^y The Galilean Invariance, sometimes called Newtonian relativity as well, states that the laws of motion in all inertial frames are the same.

^z A flow is termed free shear, when is not bounded by a solid surface, e.g. the far wake, mixing layer or jets.

where the eddy viscosity ν_t is a function of the wall distance as in $\nu_t = \tilde{\nu}f_{v1}$ and a variable viscosity $\tilde{\nu}$, and if away from the wall, the relationship becomes $\nu_t = \tilde{\nu}$.

The transport equation for the eddy viscosity is:

$$\begin{aligned} \frac{\partial \rho \tilde{\nu}}{\partial t} + \frac{\partial \rho \tilde{\nu} u_j}{\partial x_j} = \frac{1}{\sigma} \left[\underbrace{\frac{\partial}{\partial x_j} \left\{ \rho(\nu + \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial x_j} \right\}}_{\text{conservative diffusion}} + \underbrace{C_{b2} \rho \left(\frac{\partial \tilde{\nu}}{\partial x_j} \right)^2}_{\text{non-conservative diffusion}} \right] \\ + \underbrace{C_{b1}(1 - f_{t2})\rho \tilde{S} \tilde{\nu}}_{\text{production}} - \left(\underbrace{\frac{C_{w1} f_w}{\kappa^2}}_{\text{near wall turbulence destruction}} - \underbrace{\frac{C_{b1}}{\kappa^2} f_{t2}}_{\text{transition damping of production}} \right) \rho \left(\frac{\tilde{\nu}}{d} \right)^2 + \underbrace{f_{t1} \rho \Delta U^2}_{\text{transision source of turbulence}} \end{aligned} \quad (13)$$

Here the subscripts are: b=basic, t=trip (start of transition), v=viscous and w=wall, κ is Von Karman constant and d the distance to the closest wall. The basic coefficients for free shear flow are:

$$\sigma = 2/3, \quad \kappa^{aa}=0.4187, \quad C_{b1}=0.1355, \quad C_{b2}=0.622,$$

The constants and auxiliary functions for the destruction of turbulent eddy viscosity in the boundary layer are;

$$C_{w1} = \frac{C_{b1}}{\kappa^2} + \frac{1+C_{b2}}{\sigma}, \quad C_{w2} = 0.3, \quad C_{w3} = 2, \quad f_w = g \left[\frac{1+C_{w3}^6}{g^6+C_{w3}^6} \right]^{1/6}$$

$$g = r + C_{w2}(r^6 - r) \quad r \equiv \frac{\tilde{\nu}}{\tilde{S} \kappa^2 d^2}$$

And the constants and modelling functions for the near wall region are;

$$\tilde{S} = S + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{v2}, \quad S = \sqrt{2S_{ij}S_{ij}}, \quad S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) = \text{mean vorticity tensor}$$

$$f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}^3}, \quad f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}}, \quad C_{v1} = 7.1 \quad \chi = \frac{\tilde{\nu}}{\nu},$$

^{aa} Von Kármán Constant, some recent studies conclude this value may be closer to 0.40.

$$f_{t1} = c_{t1}g_t \cdot \exp\left(-c_{t2}\frac{\omega_t^2}{\Delta U^2}[d^2 + (g_td_t)^2]\right), f_{t2} = c_{t3} \cdot \exp(-c_{t4}\chi^2),$$

$$g_t = \min\left[0.1, \Delta U/(\omega_t\Delta x_t)\right]$$

$$c_{t1} = 1, \ c_{t2} = 2, \ c_{t3} = 1.2, \ c_{t4} = 0.5$$

3.1.4 k- ϵ models

The standard k- ϵ model was initially propose by Lander and Spalding (1974) and a corrected (low-Re) version by Lander and Sharma¹¹³ but actually developed from the studies of Chou¹¹⁴ (1945), Davidov (1961), Harlow¹¹⁵ (1968), and Jones¹¹⁶ (1972), being the “retuned” version of Lander the most popular two equation model of all. In this work, the realizable version proposed by Shih¹¹⁷ is used, which has a new formulation for the turbulent viscosity, and a new transport equation for the dissipation of turbulent kinetic energy (ϵ), which was derived from an exact equation for the transport of the mean-square vorticity fluctuation. This model tends to be more accurate in predicting spreading rate of round and planar jets, rotating flows boundary layers under strong adverse pressure gradients, recirculation and separation.

To model the turbulent viscosity using the realizable k- ϵ , the velocity and length scales are defined as:

$$\vartheta = \sqrt{\kappa} \quad \ell = \frac{\kappa^{3/2}}{\epsilon} \quad \mu_T = C_\mu \vartheta \ell$$

where ϑ is the velocity scale, ℓ is the length scale, μ_T is the turbulent kinematic viscosity, C_μ is a dimensionless constant for the standard model, but for the realizable approach is a function of the main strain and rotation rates, angular velocity of the system and turbulent fields^{bb}:

$$C_\mu = \frac{1}{A_0 + A_s \frac{\kappa U^*}{\epsilon}}, \quad U^* \equiv \sqrt{S_{ij}S_{ij} + \tilde{\Omega}_{ij}\tilde{\Omega}_{ij}}, \quad \tilde{\Omega}_{ij} = \Omega_{ij} - 2\epsilon_{ijk}\omega_k$$

^{bb} This improves performance for planar round jets (predicting more accurately its spread), for boundary layers under strong adverse pressure gradient or separation and for the rotation or recirculation regions or on bodies with strong curvature.

$$\Omega_{ij} = \overline{\Omega_{ij}} - \epsilon_{ijk}\omega_k \quad A_0 = 4.04, \quad A_s = \sqrt{6} \cos \phi, \quad \phi = \frac{1}{3} \cos^{-1}(\sqrt{6}W),$$

$$W = \frac{S_{ij}S_{jk}S_{ki}}{\tilde{S}^3}, \quad \tilde{S} = \sqrt{S_{ij}S_{ij}}, \quad S_{ij} = \frac{1}{2} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right)$$

Here $\overline{\Omega_{ij}}$ is the mean rate of rotation tensor viewed from a rotating reference frame with an angular velocity ω_k .

The model equation for the transport of kinetic energy and dissipation for the realizable k- ϵ model is:

$$\frac{\partial(\rho\kappa)}{\partial t} + \frac{\partial}{\partial x_j}(\rho\kappa u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial \kappa}{\partial x_j} \right] + G_k + G_b - \rho\epsilon - Y_M + S_k \quad (14)$$

$$\begin{aligned} \frac{\partial(\rho\epsilon)}{\partial t} + \frac{\partial}{\partial x_j}(\rho\epsilon u_j) \\ = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + \rho C_1 S_\epsilon - C_2 \rho \frac{\epsilon^2}{\kappa + \sqrt{\nu\epsilon}} + C_{1\epsilon} \frac{\epsilon}{\kappa} C_{3\epsilon} G_b + S_\epsilon \end{aligned} \quad (15)$$

$$\begin{aligned} & \frac{\partial(\rho\kappa)}{\partial t} + \frac{\partial}{\partial x_j}(\rho\kappa u_j) \\ &= \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial \kappa}{\partial x_j} \right] + G_k + G_b - \rho\epsilon - Y_M + S_k \frac{\partial(\rho\epsilon)}{\partial t} + \frac{\partial}{\partial x_j}(\rho\epsilon u_j) \\ &= \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + \rho C_1 S_\epsilon - C_2 \rho \frac{\epsilon^2}{\kappa + \sqrt{\nu\epsilon}} + C_{1\epsilon} \frac{\epsilon}{\kappa} C_{3\epsilon} G_b + S_\epsilon C_1 \\ &= \max \left[0.43, \frac{\eta}{\eta + 5} \right], \quad \eta = S \frac{\kappa}{\epsilon}, \quad S = \sqrt{2S_{ij}S_{ij}} \end{aligned}$$

$$\begin{aligned}
& \frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial}{\partial x_j}(\rho\varepsilon u_j) \\
&= \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_T}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S_\varepsilon - C_2 \rho \frac{\varepsilon^2}{\kappa + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{\kappa} C_{3\varepsilon} G_b + S_\varepsilon C_1 \\
& C_1 = \max \left[0.43, \frac{\eta}{\eta + 5} \right], \quad \eta = S \frac{\kappa}{\varepsilon}, \quad S = \sqrt{2 S_{ij} S_{ij}} \\
&= \max \left[0.43, \frac{\eta}{\eta + 5} \right], \quad \eta = S \frac{\kappa}{\varepsilon}, \quad S = \sqrt{2 S_{ij} S_{ij}}
\end{aligned}$$

G_k and G_b are the generation of turbulent kinetic energy due to the mean velocity gradient and buoyancy, respectively, and the turbulent Prandtl number Pr_t equals 0.85.

$$\begin{aligned}
G_k &= -\rho \overline{u'_i u'_j} \frac{\partial u_j}{\partial x_i} = \mu_T S^2 & G_b &= \beta g_i \frac{\mu_T}{Pr_t} \frac{\partial T}{\partial x_i} & \beta &= -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right)_p \\
Y_M &= 2\rho \epsilon M_t^2 & M_t &= \sqrt{\frac{\kappa}{a^2}} & a &= \sqrt{\gamma R T}
\end{aligned}$$

3.1.5 **k- ω models**

The standard Wilcox¹⁰³ **k- ω** model is known to give reliable results on wall and pressure gradient effects, and is preferable over the **k- ε** for wall-bounded flows such as boundary layers and channel flows. In this two-equation model, the dissipation per unit turbulent kinetic energy, ω includes modifications for low-Reynolds number effects, shear flow spreading and compressibility.

In this work, the Menter's¹¹⁸ **k- ω SST** model is used, which uses the **k- ω** near the wall region and the **k- ε** in the region away from walls using a blending function to bridge between the two. According to industrial experience, by Menter¹¹⁹, the

SST models differs from the standard mode as there is a gradual change from the standard **k- ω** model in the inner region of the boundary layer to a high-Reynolds-number version of the **k- ϵ** model in the outer part of the boundary layer, and modified turbulent viscosity formulation to account for the transport effects of the principal turbulent shear stress. In the **k- ω** model, the second transport equation is of the form:

$$\frac{\partial}{\partial t}(\rho\kappa) + \frac{\partial}{\partial x_j}(\rho\kappa u_j) = P - \beta^* \rho \omega \kappa + \frac{\partial}{\partial x_j} \left[\left(\mu + \sigma_\kappa \frac{\rho\kappa}{\omega} \right) \frac{\partial \kappa}{\partial x_j} \right] \quad (16)$$

$$\frac{\partial}{\partial t}(\rho\omega) + \frac{\partial}{\partial x_j}(\rho\omega u_j) = \frac{\gamma\omega}{\kappa} P - \beta^* \rho \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\mu + \sigma_\omega \frac{\rho\kappa}{\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\rho\sigma_d}{\omega} \frac{\partial \kappa}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (17)$$

$$\frac{\partial}{\partial t}(\rho\kappa) + \frac{\partial}{\partial x_j}(\rho\kappa u_j) = P - \beta^* \rho \omega \kappa + \frac{\partial}{\partial x_j} \left[\left(\mu + \sigma_\kappa \frac{\rho\kappa}{\omega} \right) \frac{\partial \kappa}{\partial x_j} \right] \frac{\partial}{\partial t}(\rho\omega) + \frac{\partial}{\partial x_j}(\rho\omega u_j)$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho\omega) + \frac{\partial}{\partial x_j}(\rho\omega u_j) &= \frac{\gamma\omega}{\kappa} P - \beta^* \rho \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\mu + \sigma_\omega \frac{\rho\kappa}{\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\rho\sigma_d}{\omega} \frac{\partial \kappa}{\partial x_j} \frac{\partial \omega}{\partial x_j} \text{where} \\ &= \frac{\gamma\omega}{\kappa} P - \beta^* \rho \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\mu + \sigma_\omega \frac{\rho\kappa}{\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\rho\sigma_d}{\omega} \frac{\partial \kappa}{\partial x_j} \frac{\partial \omega}{\partial x_j} \text{where} \end{aligned}$$

$$P = \tau_{ij} \frac{\partial u_i}{\partial x_j}, \quad \tau_{ij} = \mu_t \left(2S_{ij} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} \rho \kappa \delta_{ij}, \quad S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$

And the turbulent eddy viscosity is computed from

$$\mu_t = \frac{\rho\kappa}{\hat{\omega}}, \quad \hat{\omega} = \max \left[\omega, C_{lim} \sqrt{\frac{2\bar{S}_{ij}\bar{S}_{ij}}{\beta^*}} \right], \quad \bar{S}_{ij} = S_{ij} - \frac{1}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij}$$

According to an evaluation made by Catalano¹²⁰ comparing several RANS model, including S-A and **k- ϵ** the SST **k- ω** show the best compromise between the physical capabilities and the numerical stiffness. They used a 2D RAE 2822 aerofoil, and an RAE M2155 wing representing a study of transonic flow with strong shock-boundary layer interaction with an induced separation and a third

study of a wing section of the A310 aircraft representing high lift flow, with strong adverse pressure gradient and stream lines curvature effects.

3.2 CFD APPLICATION ON SYNTHETIC JETS

In the present work, all previous techniques explained have been carefully studied and applied to model a new approach to active flow control. The flow solver used in this study, ANSYS FLUENT (v14 & v15), employs a control volume method to discretize the incompressible Reynolds-averaged Navier-Stokes (RANS) equations with second order accuracy into a series of algebraic equations that are numerically solved. The SIMPLE^{cc} algorithm is used for pressure-velocity coupling for steady state calculations, while the PISO^{dd} algorithm is used for transient calculations with second-order fully implicit scheme for the time integration, as this scheme compares favourably with other schemes in terms of robustness, accuracy and required CPU processing time for transient flows¹²¹. It is worth noting that Tang⁶⁶, Bettini¹²² and He¹²³ also use the same solver for their studies, as they found it able to emulate key features of synthetic jets.

^{cc} Semi-Implicit Method for Pressure Linked Equations; The fundamental purpose of the SIMPLE and PISO algorithms is to derive a pressure correction equation by enforcing mass continuity over each control volume. In the SIMPLE algorithm, after the boundary conditions are set, an initial guess to the gradients of velocity and pressure are computed, then the momentum equation is discretised and solved, after which new values of velocity are obtained and with that the pressure correction equation is solved. Then the pressure and velocity fields are updated with under-relaxation factors. Then all other transport equations are solved, and the procedure iterates until convergence occurs¹⁵⁵.

^{dd} Pressure-Implicit with Splitting of Operators; Initially created to solve unsteady compressible flows is sometimes seen as an extension of SIMPLE with a further corrector step to improve it. PISO implies an increase in computational resources, as the algorithm solves twice the pressure correction equation, but it has been found to be fast and efficient.

4 BIDIRECTIONAL SYNTHETIC JET^{ee}

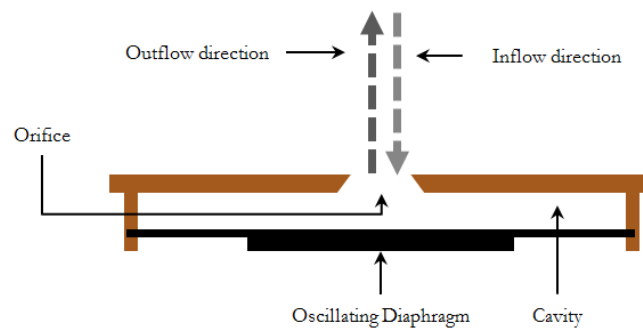
This chapter presents a numerical analysis of the potential benefits of changing the inflow and outflow directions of a synthetic jet separately for active flow control. The main target of the Bi-Directional synthetic jet (BDSJ) is to induce the Coandă effect downstream due to the jet outflow directed, and subtract the upper flow with minimal perturbation for efficient suction of the upper boundary-layer. A widely-used reference case (so-called NASA 2-D Hump) is used to validate the numerical model and to compare results. Three common turbulence models are employed and they all show similar trends. Eleven different angles are tested for the inflow and outflow directions of the synthetic jet; the most efficient configuration is then compared with the classical synthetic jet and Directed synthetic jet. It is shown that the best BDSJ configuration results in a shorter separation bubble length over the hump, suggesting potential benefits of the BDSJ as a future active flow control device.

^{ee} Part of the content of this chapter was published with the title “Numerical Analysis of a Bidirectional Synthetic Jet for Active Flow Control” in AIAA Journal, DOI:10.2514/1.J055081. Ref.^[156]

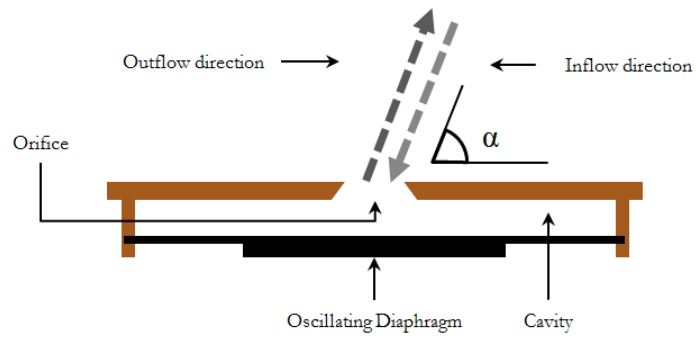


In the present chapter, a novel active flow control concept is numerically tested, called Bi-Directional synthetic jet (BDSJ) for actively controlling the flow over aerodynamic surfaces, using RANS simulations of flow over the NASA 2D Hump model. A sketch of the Bi-Directional synthetic jet and its comparison with the classical synthetic jet and the directed synthetic jet can be found in Figure 4-1. A set of CFD studies on a synthetic jet applied to the widely analysed “2D NASA Hump” (case 3 of the 2004 workshop¹⁵) is used to compare results. The effects of the Bi-Directional synthetic jet are simulated numerically by three different approaches, prescribing the profile and direction of oscillatory flow at the jet exit boundary, modifying the speed at the outlet of the jet, and altering the apparent area of the jet exit.

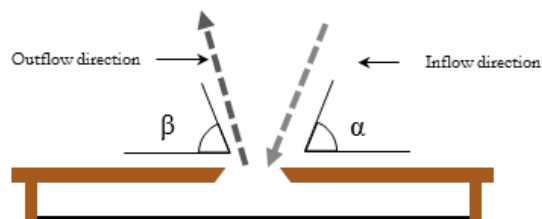
The actual device, details of which will be discussed in section 6, may involve a rapidly oscillating magnetic membrane and an internal tube connecting two jet holes to entrain air into one cavity and eject it from the other, changing the inflow and outflow directions of the jet. In the present chapter, however, only one jet hole was considered for simplicity.



a) *Classic Synthetic Jet*



b) *Directed Synthetic Jet*



c) *Bi-Directional Synthetic Jet (illustrative figure)*

Figure 4-1 Synthetic Jets classification regarding flow direction.

4.1 TEST CASE DESCRIPTION

The geometry of the hump emulates the upper surface of a 20% thick Glauert-Goldschmied airfoil.

The working fluid is air at 20°C with a viscosity of 1.84×10^{-5} kg/ms and a density of 1.185 kg/m³, giving a Reynolds Number (based on the hump chord length) of approximately one million, and a Strouhal Number of 30 ($St = Lf/U_\infty$). The reference pressure for the domain is set at 101,325 Pa. For all the analyses presented in this chapter, unless explicitly stated, the inlet velocity is $U_\infty = 34.6$ m/s (Mach 0.1), normal to the boundary, the turbulent intensity at the inlet is 5% and the turbulent length scale^{ff} 1 mm as suggested by common practice. Zero streamwise-gradient conditions are given at the outlet, with a backflow turbulent intensity of 10% and backflow turbulent length scale of 1 mm, while the no slip wall is set with a roughness height of 0 m. Each computation is initiated with a uniform velocity $(u, v) = (U_\infty, 0)$ given to the entire computational domain. For transient analyses, the time step size is set at 5×10^{-5} seconds, which was found to be sufficiently small, as showed in section 0, i.e. the solution did not change with further reducing the time step size.

The final two-dimensional grid of the experimental hump¹²⁵ comprises 0.6 million elements, with the first 20 layers from the wall at a growing rate of 1.1. The size of the smallest elements close to the wall of the hump is 3×10^{-5} [m], and 2 mm

^{ff} The turbulent length scale is a physical quantity related to the size of the large eddies that contain the energy in turbulent flows; There is an approximate relationship for fully developed duct flows¹³⁶, $l = 0.07 * L$, where L is geometric relevant dimension. When the inlet involves a turbulent boundary layer the length scale may be approximated as $l = 0.4 \delta_9$.

besides the jet exit there are two spherical refinement of 2×10^{-5} [m]. The highest skewness^{gg} value is less than 0.7 before a smoothing operation, after which the minimum orthogonal quality reached values higher than 0.7. For all the analysis, the average y^+ is smaller than one.

4.1.1 Characteristics of the Synthetic Jet

Some authors state the need to perfectly model the cavity of the synthetic jet to obtain accurate results. For instance, Xiong⁹² says that regarding the strength of the vortex pair in the stream-wise direction, the slot width, followed by the cavity height and slot depth are the most influencing factors. According to the results of Chaudhari⁹³, there is a noticeable effect of cavity dimensions on the velocity at the exit, and according to Watson⁹⁴, the geometry of the orifice have a big effect on the flow structures produced. Rumsey¹⁶ stated that to obtain accurate results, not only the frequency and amplitude have to be faithfully modelled, but also the neck length, diaphragm area, slot size and in general the cavity volume.

However, not in all publications the cavity is perfectly emulated. Tang⁶⁶ for example, emulated the jet using a velocity profile, and Kral⁴⁴ made a 2D RANS simulation with three different velocity distributions along the jet exit, where the top hat agreed better with experiments.

In the present work, the cavity was substituted by velocities profiles, as the actual mechanical device still does not exist, and the main purpose of the investigation is to compare all the different types of jets under equal

^{gg} As the grid is mainly made of triangles and tetrahedral elements, the mesh quality parameter skewness is determined as $S = (\text{size of the optimal cell} - \text{cell size}) / \text{optimal cell size}$.

circumstances. It has been reported that different cavity shapes do not affect significantly the flow characteristics at the jet exit¹³⁵; therefore the actual cavity is not modelled in this study. Still there may be effects not modelled due to the omission of the cavity, but a fair comparison is still reached, as the effect would be neutralized by applying the same principle to different types of synthetic jets.

The synthetic jet is modelled by manipulating the velocity boundary conditions across the jet exit (located at $x/c=0.65$), similarly to the study by Viken et al.¹³⁰ Eleven different angles are considered for the inflow/outflow directions of the jet, as described in Figure 4-16 (note that the figure has been rotated 18° to match the angle over the hump).

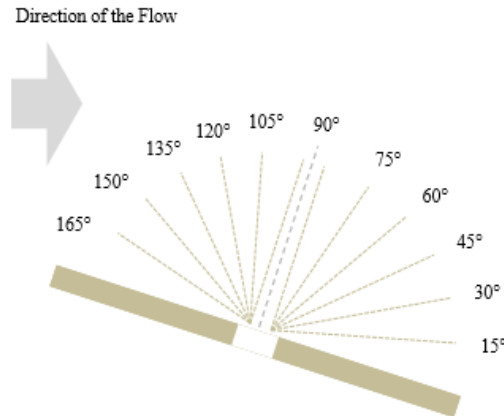


Figure 4-2 Schematic of the inflow/outflow directions of the jet

The jet velocity (in the prescribed inflow/outflow directions) has a parabolic profile across the jet slot and fluctuates in time, defined as

$$V_j(\xi, t) = V_j(\xi) \cdot \sin \omega t \quad (18)$$

where ξ is the distance (across the jet slot) from the upstream end of the jet slot, $V_j(\xi)$ is a parabolic profile defined later in Section 0, ω is the frequency (100 Hz),

and t is the time. The horizontal (x) and vertical (y) components of the velocity, considering the prescribed jet direction, are specified through User Defined Functions (UDF) in FLUENT (See Appendix A in page 165). These velocity components across the jet exit are monitored throughout the simulation to assure that the jet direction and mass flow rate are as intended.

For all simulations with a synthetic jet, the following three steps were employed as a strategy for fast convergence:

1. Run the simulation for 15,000 iterations in steady state, which is sufficient for the continuity residual to decrease to 10^{-6} .
2. Run the simulation for 6,000 time steps with 20 iterations per time step to obtain statistically converged solutions. In this transient time, not only the residuals were measured as a convergence criteria, but also average skin friction and average pressure coefficient.
3. Run the simulation for another 2,000 time steps for taking time statistics, resulting in a total of 8,000 time steps for each simulation (corresponding to about 33 flow through times based on the hump chord).

All these steps were followed automatically by the set of instructions in a journal file shown in Appendix B.1.

It is known that when computations match better at the slot exit, they tend to match better in the flow field outside¹⁶, explaining why it is important to refine the mesh close to the jet. Figure 4-18 shows the mesh around the exit; more than 90 elements are used across the slot with a size of 0.02 mm for each element (element size at jet exit / jet exit size = $0.01\overline{11}$). The arrows indicate the jet direction at b) 90° and c) 30°.

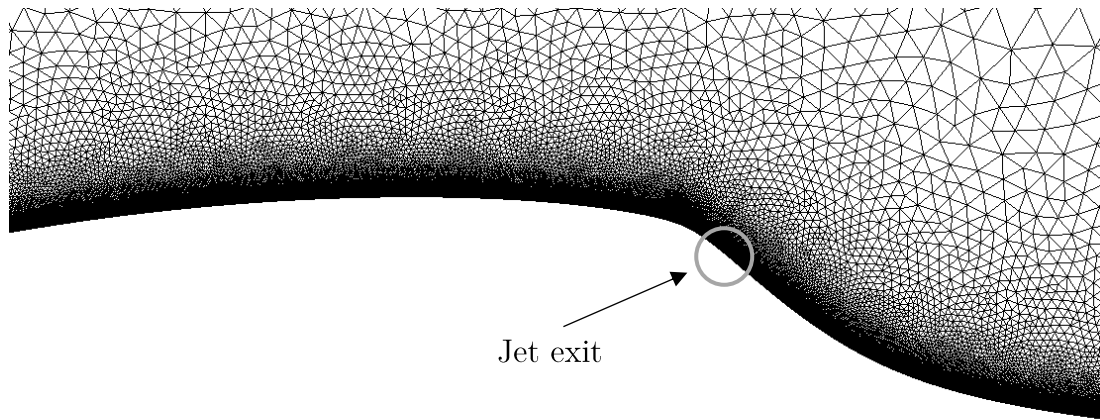
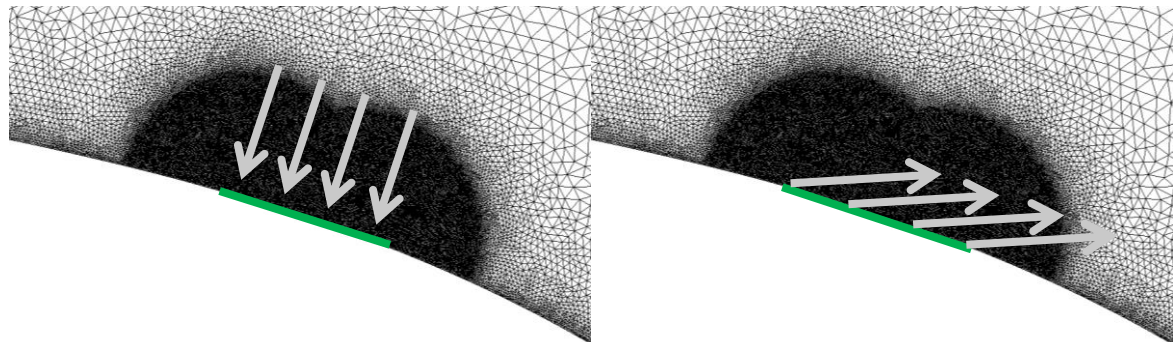


Figure 4-3 Computational mesh over the hump.



b) Inflow at 90°

c) Outflow at 30°

Figure 4-4 Mesh around the jet exit,

Amitay⁴⁵ used a variety of angles of the actuation jet in an unconventional aerofoil, so that he modified the angle of incidence of the jets with the air. In the present work, a similar numerical approach is followed for the modelling of the synthetic jet.

For boundary-layer studies, structured meshes are often used among the research community due to the numerical convenience, as no significant changes in the flow are expected in the direction parallel to the wall, and a structured mesh can give accurate results without carrying as many elements as an unstructured will do. Moreover, for numerical convenience in the code interpolations close to the wall, it is recommended that the first two layers away from the wall should be prismatic or structured elements. However, for modelling convenience as there was a need for continuous refinement and adaption near the exit, an unstructured mesh was used over the surface of the hump, leaving the structured mesh for the rest of the domain.

4.1.2 Effect of jet exit velocity profile

To confirm the effect of the velocity profile specified at the jet exit, a set of preliminary simulations was performed (for the BDSJ 165-15 configuration, i.e. inflow and outflow angles are 165° and 15°) with the S-A model, changing the profile from a top hat to a fully parabolic one. The profile is defined as:

$$V_j(\xi) = V_{j,max} \left(1 - \left(\frac{\xi}{D} - 0.5 \right)^2 S \right) \quad (19)$$

where $V_{j,max}$ is the maximum jet velocity at the middle of the jet slot, D is the diameter (width) of the jet exit, and S is a non-dimensional shape factor that defines the shape of the parabolic profile. Note that the value of $V_{j,max}$ depends on S to keep the mass flow rate constant. The results are summarized in . The

separation and reattachment points were estimated from the location of the local minimum values of the mean friction coefficient over the wall of the hump.

Table 1 Summary of preliminary simulations with different jet profiles (BDSJ 165-15 configuration).

Case	S	$V_{j,max}/U_\infty$	Separation	Reattachment	Bubble length	Difference in bubble length
			x/c	x/c	x/c	(Case _x - Case ₁) / Case ₁
1	0.0	0.86	0.7175	1.227	0.5095	0.00%
2	0.2	0.88	0.7175	1.228	0.5102	0.14%
3	0.6	0.91	0.7175	1.228	0.5104	0.18%
4	1.0	0.96	0.7175	1.229	0.5116	0.41%
5	2.0	1.03	0.7175	1.231	0.5133	0.74%
6	3.0	1.13	0.7175	1.230	0.5123	0.56%
7	4.0	1.29	0.7175	1.231	0.5132	0.73%

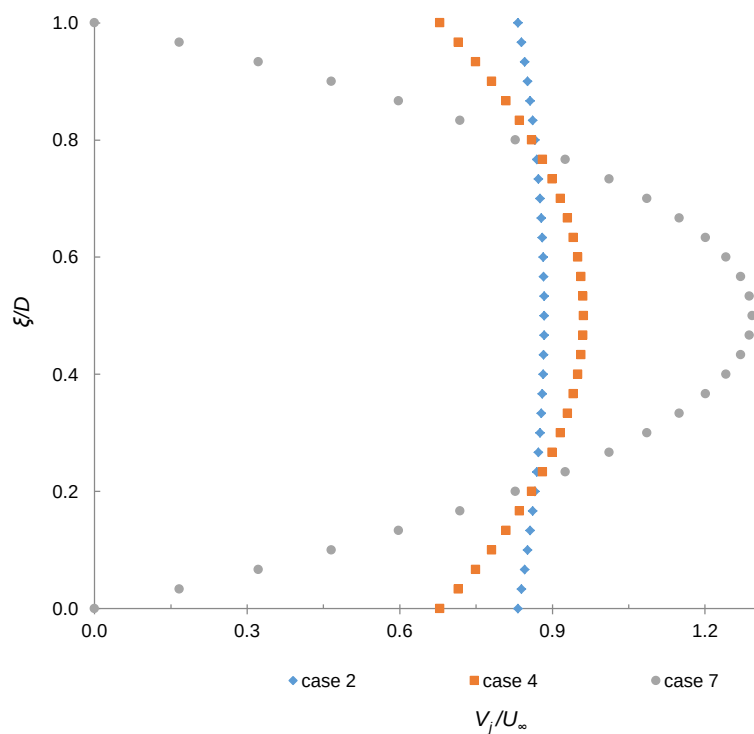


Figure 4-5 Velocity profile at the jet exit for Cases 2, 4 and 7.

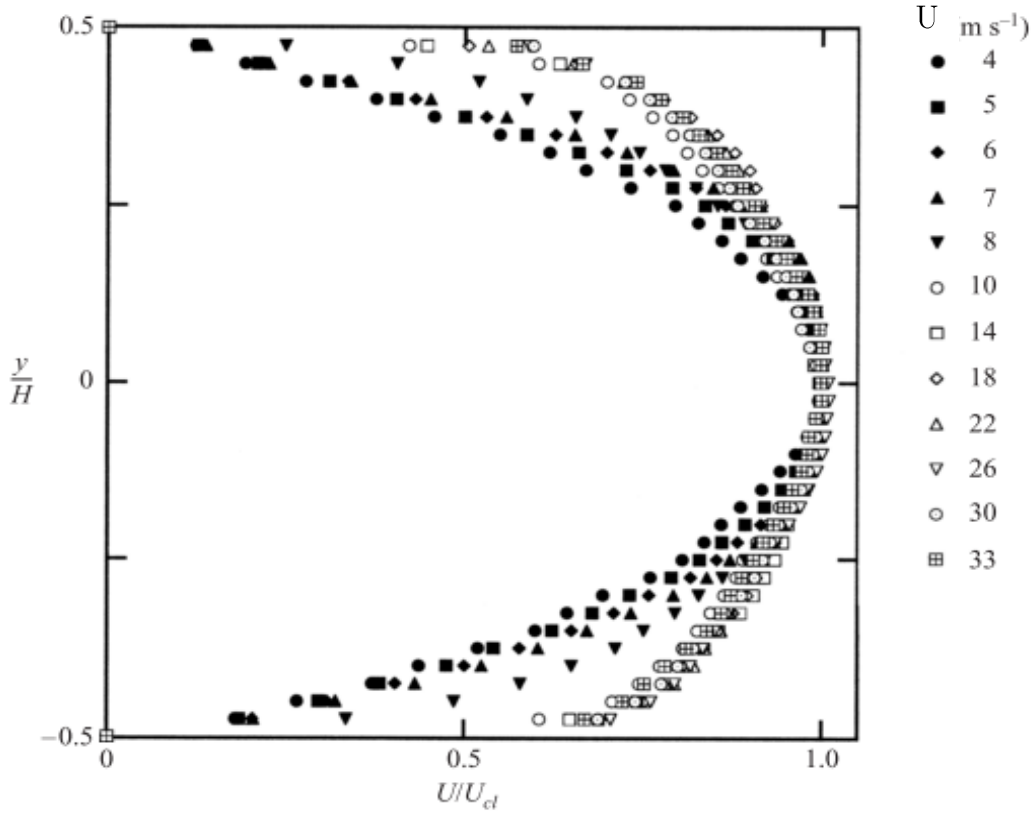


Figure 4-6 Experimental velocity profiles (reproduced from Smith⁵¹).

Figure 4-21 shows the velocity profiles across the jet exit for Cases 2, 4 and 7.

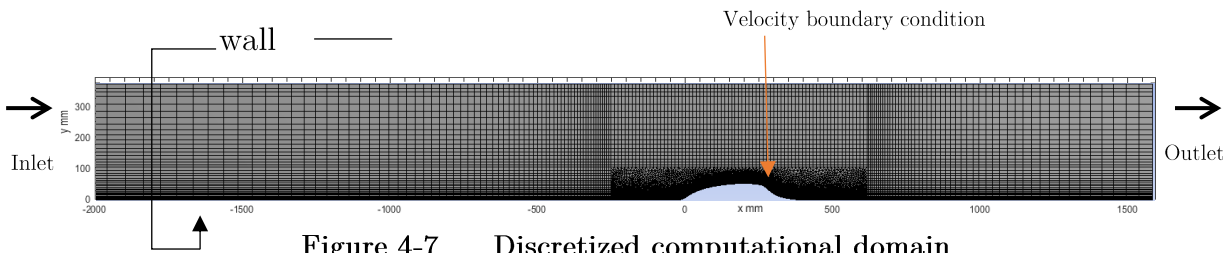
Figure 4-22, reproduced from Smith⁴⁸, shows different time averaged stream-wise velocity profiles for a typical air flow inside a duct and is shown for comparison.

As can be seen from , the effects of different jet profiles on the separation bubble size are insignificant. Hence the jet velocity profile of Case 4 ($S = 1$ and $V_j/U_\infty = 0.96$) is used for the rest of the study.

4.2 VALIDATION OF BASELINE SIMULATION

4.2.1 Computational settings

. The computational domain mimics the test section of the experiments¹²⁴, where the body was mounted between two glass endplates, and both leading and trailing edges matched smoothly the wind tunnel splitter-plate. The dimensions of the computational test section are 382 mm high and 3,531 mm long. The model leading-edge is placed at 4.61 chord lengths (1935mm) from the inlet, and the length of the hump is 420 mm, which has a 1.8 mm jet slot at 65 % of the chord ($x/c=0.65$). Detailed data about the experimental configuration can be found in Gatski¹²⁵. The 2-D computational domain, together with the final mesh employed in this study, is shown in Figure 4-15. The details of a grid convergence study will be described later in section 0



Although there are some side-wall effects on the domain, the results of the experiments are nominally two-dimensional as stated by D. Greenblatt, et al.¹²⁶, and according to the results of Karthikeyan Duraisamy et al¹²⁶, there is no significant difference between the outputs of a two dimensional mesh and a three dimensional for this type of analyses. Therefore, two-dimensional simulations are

conducted in this study, as a number of analyses to discover trends are required, and a three-dimensional simulation would need extensive computational resources.

4.2.2 Time step size selection

The influence of the time step was measured through a sensitivity analysis refining the size of the step up to the point where the maximum CFL number in the whole domain is lesser than one. It was found that refining the size of the time step to lower values than 1×10^{-4} seconds makes no effect on the final solution, even though the CFN for this time size is up to 20. Also, the total time of the solution was estimated through a sensitivity analysis. Figure 4-20 shows that, after half a second of simulation, there is almost no change in separation and very little in reattachment. The last column in Table 3 below displays the converged percentage of the reattachment for each calculation. R_i is the reattachment at the given iteration, while R_{i-1} and R_f are the position of the reattachment values for the previous and last iterations respectively. For the first six sets of results the change was considerable, but after that, the reattachment value changed by less than 1%.

The separation and reattachment points were obtained by means of the mean skin-friction coefficient graph along the wall of the hump. It is important to bear in mind that, the statistics of the mean quantities are collected in the interior of the cells and not on the surface of the wall. Therefore, when the mean quantity of a cell or a node value is plotted on the wall surface, values in nearby cells attached to the wall¹³⁶ are actually plotted.

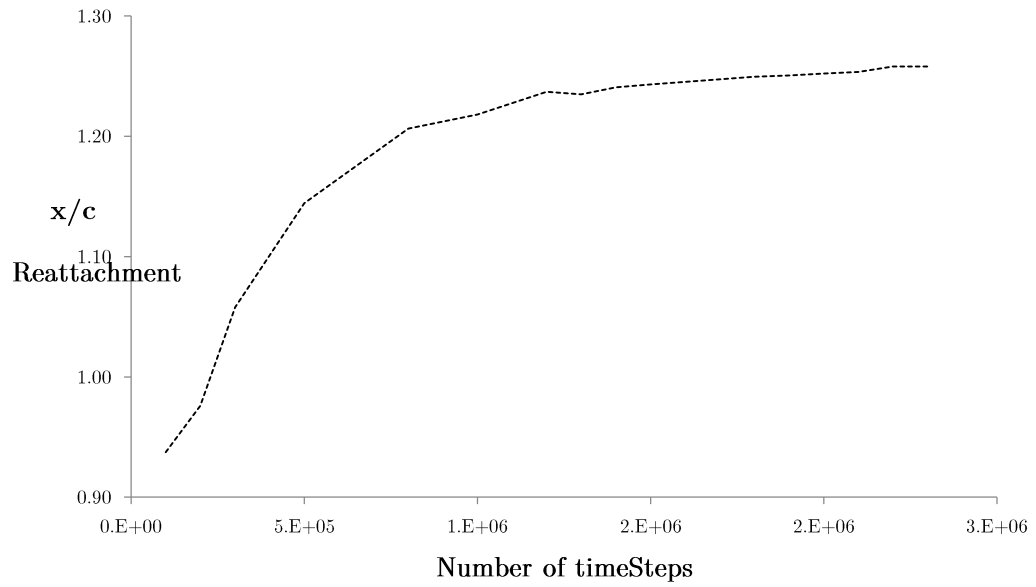


Figure 4-8 Time step study

Table 2 Time step study

Time step size = 5×10^{-7} seconds

time/tstep	Separation x/c	Reattachment x/c	$(R_i - R_{i-1})/R_i$	$(R_i - R_f)/R_f$
100,000	0.665	0.937		
200,000	0.666	0.976	3.9585%	22.43%
300,000	0.667	1.058	7.7630%	15.90%
800,000	0.666	1.206	5.1328%	4.10%
1,000,000	0.666	1.218	0.9733%	3.16%
1,200,000	0.666	1.237	1.5184%	1.67%
1,300,000	0.666	1.235	-0.1735%	1.84%
1,500,000	0.666	1.243	0.1954%	1.18%
1,900,000	0.666	1.251	0.1028%	0.58%
2,000,000	0.666	1.252	0.1255%	0.45%
2,100,000	0.666	1.253	0.0912%	0.36%
2,200,000	0.666	1.258	0.3745%	-0.01%
2,300,000	0.666	1.258	-0.0114%	0.00%

4.2.3 Turbulence model selection

Although it is known that RANS turbulence models cannot predict quantitatively correctly the reattachment point downstream of the hump, they still tend to give qualitatively correct solutions that are sufficient for comparative studies. To find a trend in parametric studies, RANS simulations are still the most appropriate approach as large eddy simulations (LES) and detached eddy simulations (DES) would require prohibitive computational resources to solve a large number of cases; e.g, Morgan¹²⁷ made a comparative analysis of ILES versus RANS and obtained better agreement with experiments than RANS but at the expense of more than 20 million elements. A recent effort to overcome such difficulties was made by, e.g. Larsson et al¹²⁸.

A summary of earlier studies using RANS turbulence models for this type of flow is given in .

Table 3 Turbulence models used by different authors.

Studied by	Model Used	SJ over a
Krishnan et al.(p.145) ¹²⁶	S-A & SST	2D&3D/Hump
Morgan et al. (p.151) ¹²⁶	Realizable k- ϵ	2D&3D/Hump
Karthikeyan et al. (p.163) ¹²⁶	S-A	2D&3D/Hump
Rumsey ¹²⁹	S-A, SST, & EASM-k ω ^h	2D/Hump
Katam et al. (p.175) ¹²⁶	SST	2D/Hump
Viken. et al. ¹³⁰	S-A	2D/Hump
Spall et al. (p.185) ¹²⁶	Standard k- ϵ Wilcox k- ω RST, S-A, SST	2D/Hump
Shmilovich et al. (p.191) ¹²⁶	S-A & SST	2D/Hump
Marongiu et al. (p.193) ¹²⁶	Myong and Kasagi k- ϵ & SST Menter k- ω for baseline k- ϵ & S-A for steady suction	2D/Hump

^{hh} Explicit Algebraic Stress Model in k- ω form.

Shariff et al. (p.199) ¹²⁶	Metacomp's 2 eq. cubic k- ϵ	3D/Hump
Balakumar (p.201) ¹²⁶	SST	2D/Hump
Laouedj et al. ¹³¹	SST	3D/Flat Plate
Lyons et al. ¹³²	S-A	3D/Hump
Kim et al. ⁸⁸	SST	3D/Flat Plate
He et al. ¹²³	k- ϵ SST, S-A	2D/Hump
Bettini & Cravero ¹²²	RNG k- ϵ	2D/Hump

In this study, the Spalart-Allmaras, the Realizable k- ϵ and k- ω SST turbulence models are used as they have shown to be reasonably reliable for this type of flow, besides the fact that more than 90 % of the researchers who contributed to the third case of the 2004 workshop used at least one of these models¹⁵. To demonstrate that the trend (of the performance of different types of synthetic jets) observed does not depend on the turbulence approach, these three different models are used for comparison.

It should be remembered that the standard k-epsilon model is only valid in fully turbulent flows¹³³; hence here the improved realizable version is used. For some studies, the k- ω SST model has shown good agreements with experiments for the unsteady flow fields¹³⁴, and according to computational data obtained by Lyons¹³² and its comparison to experimental data, the Spalart-Almaras turbulence model predicts the mean-field statistics well, and better than DES, DDES, and ZDES.

For the k- ω SST, and the k- ϵ realizable models, the average computational time was 4.5 hours with 32 parallel Intel E5-2660 (Sandy Bridge) CPUs (2x16 cores at 2.2 GHz), while for S-A model, it only took 3.5 hours, even though both S-A and k- ω SST cases required the same amount of time steps and iterations. This difference is obviously because solving only one equation for SA is faster than two for the k models.

4.2.4 Grid convergence

To achieve grid convergence, grids of different sizes were created and compared until the predictions of separation and reattachment points converged. The final two-dimensional mesh of the hump comprises 0.62 million elements, with the first 20 layers from the wall stretching at a growing rate of 1.1. The size of the smallest elements close to the hump surface is 0.03 mm. Also, within 2 mm from the jet exit there are two spherical zones with refined mesh elements of 0.02mm (0.011 of the jet slot width; see Figure 4-18). shows the data obtained from three different grids with increasing the number of elements near the wall, therefore reducing the y^+ value. The CFLⁱⁱ number at the jet exit oscillates between 0 and 86 while its average value at the inlet is 0.028.

It is important to keep in mind that a converged solution is not necessarily an accurate one; however, it is the first step to reach the expected result. The mesh refinement was mainly done close to the surface of the hump and over the separated region, such that the average y^+ decreased to values below one.

ⁱⁱ The Courant-Friedrichs-Lewy condition: defined as $c = \frac{u\Delta t}{\Delta x}$, represents the amount of grid cells a fluid particle travel in one time step.

Table 4 **Summary of grid convergence study (without synthetic jet).**

Number of Elements	Average y^+	Separation x/c	Reattachment x/c
Experiment	-	0.665	1.11
150,000	4.07	0.657	1.254
620,000	0.82	0.655	1.232
1,500,000	0.36	0.655	1.230

For all these simulations, the scaled^{jj} residuals were reduced by at least six orders of magnitude, while the skin friction coefficient^{kk} and pressure coefficient were monitored to confirm the convergence.

According to Karthikeyan et al¹²⁶, for this type of RANS simulation, the solutions are fairly grid independent when the wall normal spacing is less than 2.5 wall units. Moreover, grid refinement in the separated region does not improve the predictions. As can be seen from Figure 4-20, once the y^+ reaches a value close to one, very little changes appear with grid refinement i.e. mesh convergence is achieved with 0.62 million grid points. Therefore, this value is used for all following simulations, as the mesh with 1.5 million points shows almost no change in reattachment and none in separation.

Separation is defined as the point close to the wall with zero velocity that occurs when a boundary layer is traveling against an adverse pressure gradient and after which unsteady flow conditions or small eddies start appearing and dominating the boundary layer, making it fully turbulent.

^{jj} The scaled residuals are an indicator of convergence, and is the imbalance or difference between two consecutive results of a transport equation, over the largest absolute value of the equation in the first 5 iterations.

^{kk} The friction coefficient equals the wall shear stress over the dynamic pressure.

$$C_f = \frac{\tau_{wall}}{\rho V^2 / 2}$$

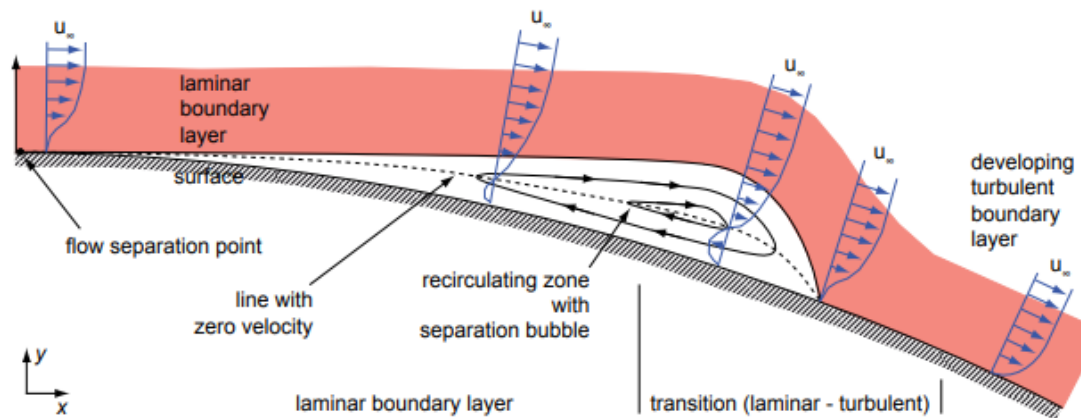


Figure 4-9 Illustration of separation and reattachment.

Reattachment on the other hand, is also a point with zero velocity close to the wall, however, downstream of reattachment, the velocity profile is parabolic again, as illustrated in Figure 4-19.

4.2.5 Model validation

The pressure coefficient C_p along the dimensionless distance over the hump (x/c) when no flow control is applied, can be seen in Figure 4-25. It is clear the pressure tends to increase as the flow stagnates at the beginning of the hump ($x/c=0$), and decrease around the separation section ($x/c=0.65$). A number of experimental and computational analyses for this flow were compared already in the 2004 workshop¹²⁶ but in this plot only the experiments from Greenblatt and the 2D SA results from Karthikeyan¹²⁶ are shown as a reference. It is clear how the reference CFD results, the experiments (PIV measurements¹³⁸) and the current CFD results with the S-A model agree well at $x/c < 0.65$, and at $x/c > 0.65$ all analyses indicate some minor discrepancies

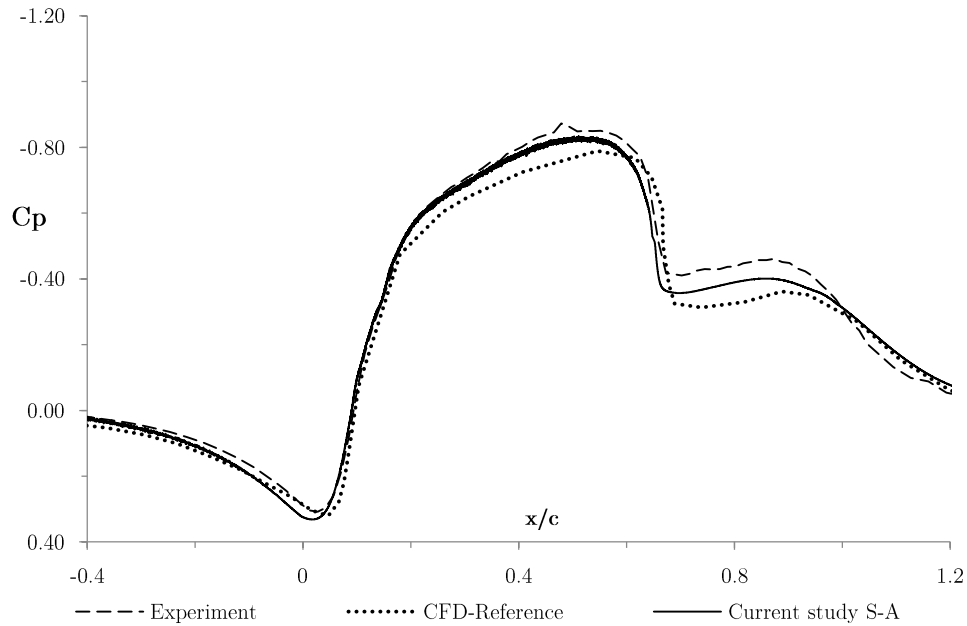


Figure 4-10 Comparison of C_p over the hump with no flow control.

Figure 4-26 compares the results of different turbulence models, showing that all three models give similar outcomes.

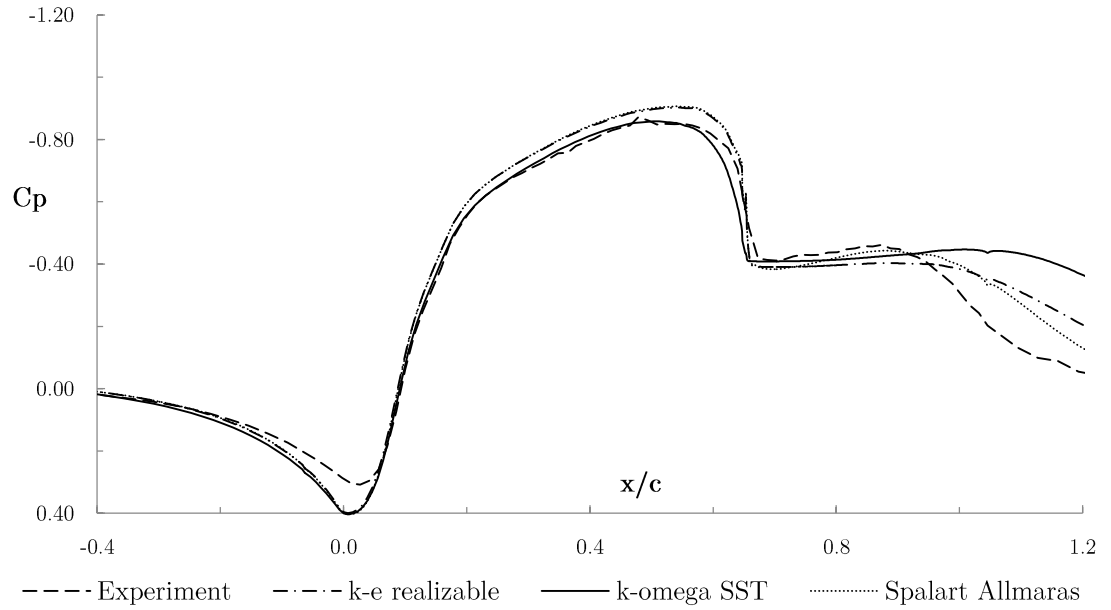


Figure 4-11 Comparison of C_p over the hump for several turbulence models.

A comparison of streamwise velocity profiles (along the vertical direction) at $x/c = 0.8$ with no flow control is presented in Figure 4-27. The current CFD results (with the S-A turbulence model) are compared with the reference CFD results¹²⁶ as well as the experiments¹³⁹ at $x/c=0.8$, which, according with experiments, is within the zone where full recirculation is expected. All analyses agree reasonably well regarding the velocity profile.

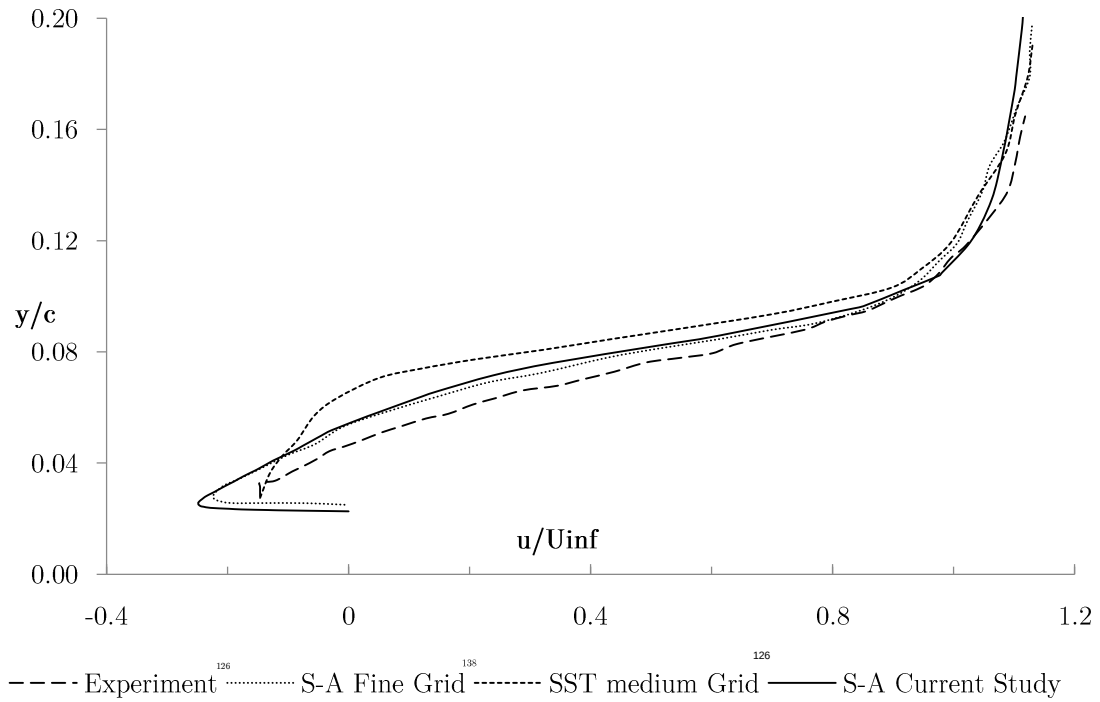


Figure 4-12 Streamwise velocity comparison at $x/c = 0.8$.

Once the simplest case is validated, flow control with suction is applied and the outputs compared. A comparison of C_p over the hump for the case with constant suction through the jet exit at a rate of 0.01518 kg/s is shown in Figure 4-28. Before the separation, the results are in good agreement with the experiments and reference CFD results, but after the separation there are some discrepancies which were expected as reported previously.

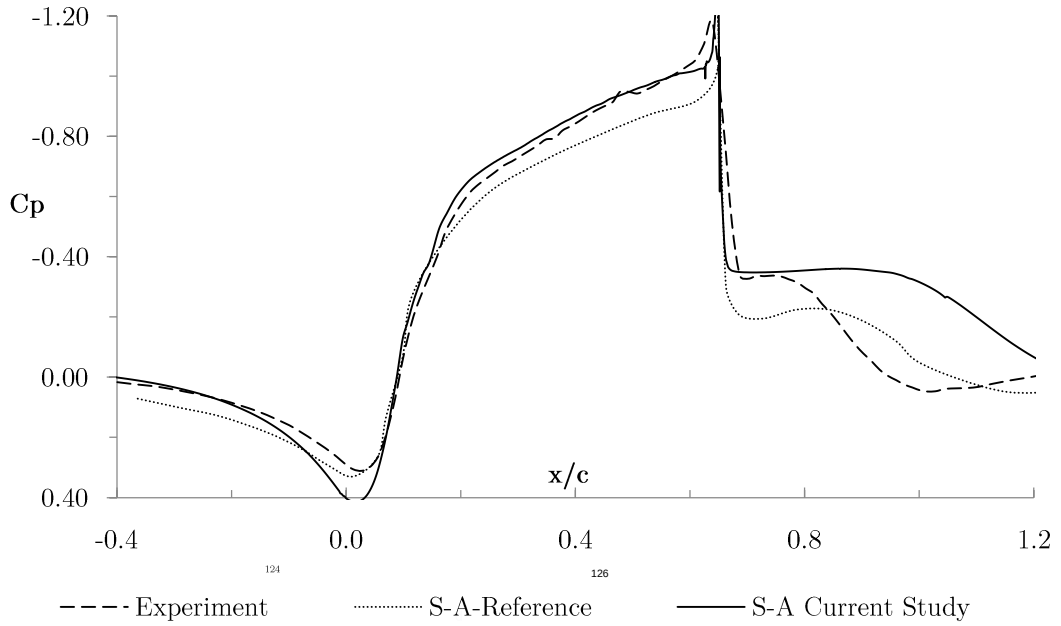


Figure 4-13 Comparison of C_p over the hump with suction.

The following three tables (5 to 7) provide comparisons on the separation and reattachment values for three configurations: the case with no flow control or “Baseline” case, where the synthetic jet is turned off; the suction case, where a constant suction condition is given at the jet exit; and, the classical synthetic jet case, where a transient sinusoidal oscillatory behaviour is prescribed at the jet exit.

Table 5 No flow control configuration. Separation and reattachment values by different authors.

Study	Author	Separation x/c	Reattachment x/c	Bubble length x_b/c
Experiment	D. Greenblatt et al. ¹²⁴	0.670	1.110	0.440
DES	Saric et al. ¹⁴⁰	0.663	1.121	0.458
LES	Saric et al. ¹⁴⁰	0.667	1.114	0.447
Typical RANS	Rumsey et al. ^{129, 141}	0.661	1.240	0.579
k- ω SST	Present study	0.676	1.247	0.571
S-A		0.715	1.259	0.544

Realizable k- ϵ 0.718 1.233 0.516

Table 6 Suction configuration. Separation and reattachment values by different authors.

Study	Author	Separation x/c	Reattachment x/c	Bubble length x_b/c
Experiment	D. Greenblatt et al. ¹²⁴	0.686	0.940	0.254
DES	Saric et al. ¹⁴⁰	0.674	1.105	0.431
LES	Saric et al. ¹⁴⁰	0.671	0.947	0.276
Typical RANS	Rumsey ^{129, 141}	0.680	1.030	0.350
k- ω SST		0.715	1.011	0.296
S-A	Present study	0.671	0.995	0.324
Realizable k- ϵ		0.689	0.852	0.163

Table 7 Classical synthetic jet configuration. Separation and reattachment values by different authors.

Study	Author	Separation x/c	Reattachment x/c	Bubble length x_b/c
Experiment	D. Greenblatt et al. ¹³⁹	0.680	0.980	0.300
DES	Saric et al. ¹⁴⁰	0.662	1.110	0.448
LES	Saric et al. ¹⁴⁰	0.671	1.050	0.379
Typical RANS	Rumsey et al. ^{129, 141}	0.670	1.220	0.550
k- ω SST		0.713	1.345	0.638
S-A	Present study	0.714	1.271	0.534
Realizable k- ϵ		0.691	1.179	0.489

These results suggest that the current CFD procedure can be used to predict the behaviour of this type of flow qualitatively correctly; hence, similar procedures can be used to assess the (qualitative) performance of the novel synthetic jet configurations as well. It should be borne in mind that our objective here is to

assess the trend of the influence of different jet configurations and not the exact locations of the separation and reattachment points.

Several turbulence models were tasted to see which one can get closer to the experimental profile when suction is applied. Figure 4-29 shows these results and is clear all turbulence models did reasonably well.

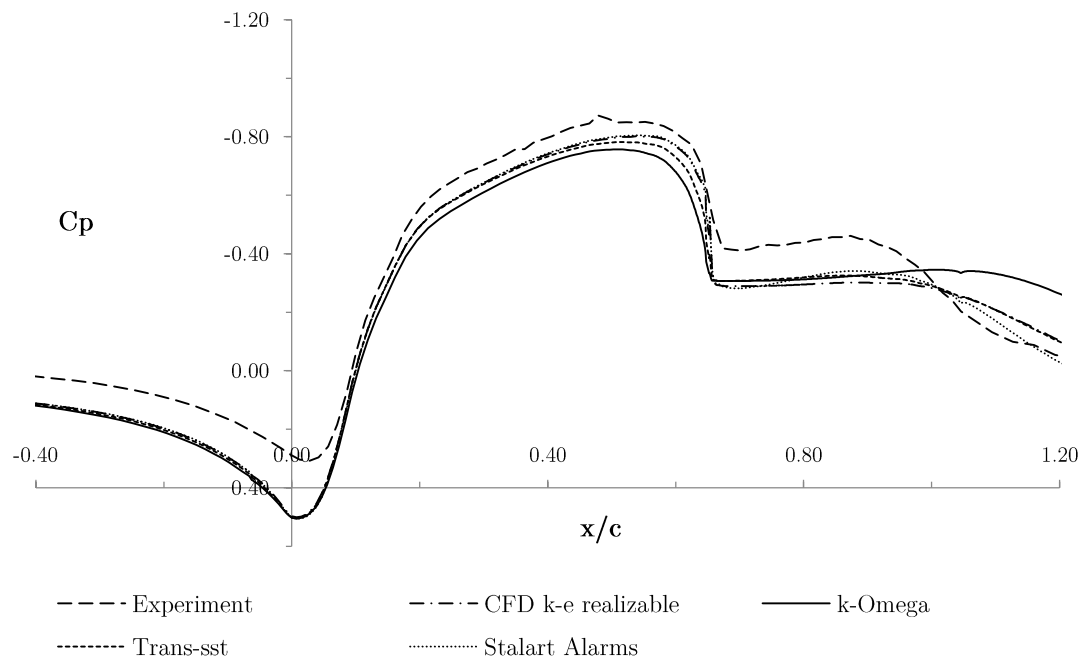


Figure 4-14 C_p over the hump for suction (with different turbulence models)

4.3 RESULTS FROM CONTROLLED CASES

4.3.1 Controlled case study

As this is a transient analysis, the bubble length is found from a statistically averaged value. The separation bubble length in this study is obtained by calculating a convex value in the mean skin friction coefficient along the surface of the wall. as according with the experimental result of Naughton¹³⁷ and its comparison with computational predictions, the RANS computations accurately predict the separation region. The area-weighted average skin friction coefficient is used as a convergence parameter to help determine the accuracy of the final solution. A typical graph of the area-weighted average skin friction coefficient along the time steps has the form of Figure 4-23. It is clear from this figure that convergence is achieved since iteration 3,000.

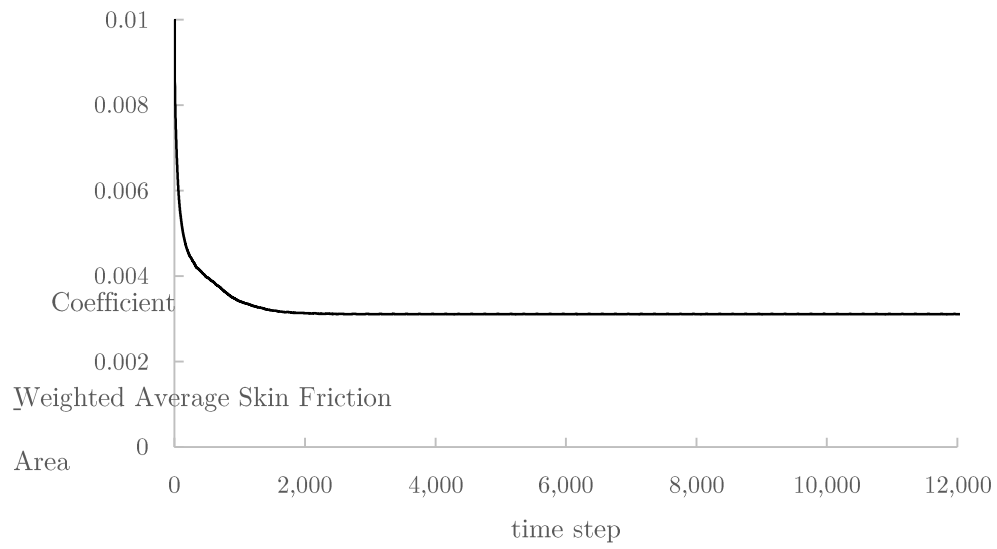
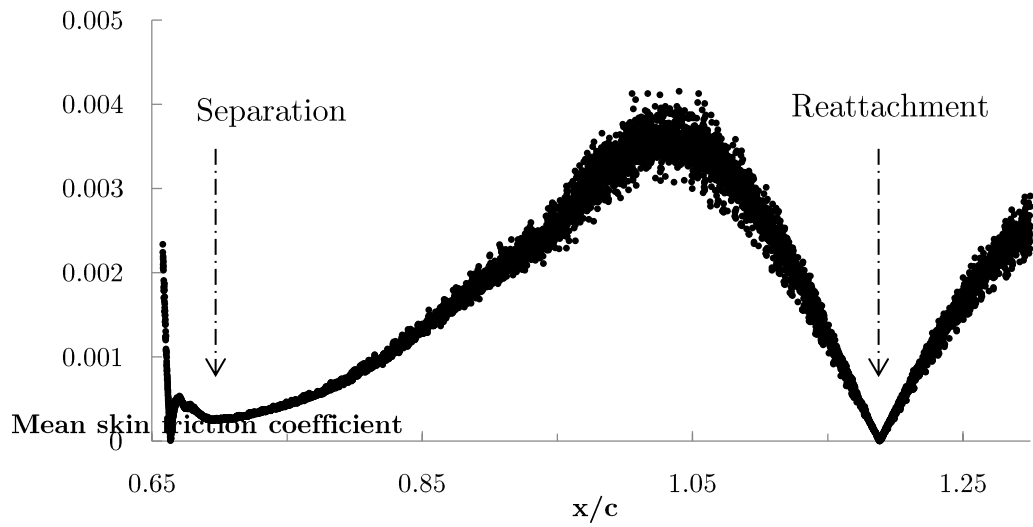
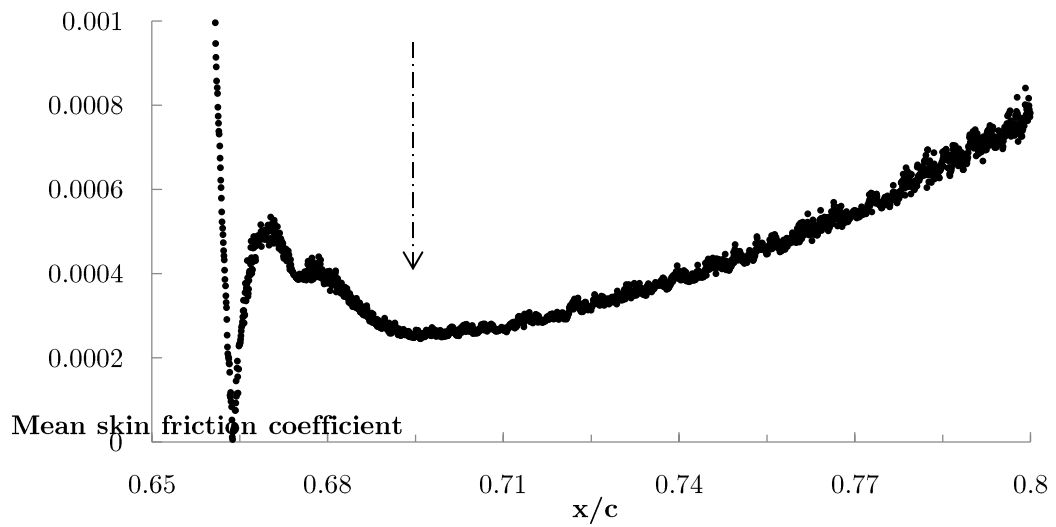


Figure 4-23 Area-Weighted Average Skin friction coefficient per time step.

Figure 4-24 shows the mean average in time of the skin friction above the hump for the BDSJ at 15° - 15° solved with the realizable $k-\epsilon$ model. This figure represents a typical behaviour for all other configurations and turbulent models. In this case the exact value of separation is $x/c=0.69588$ and $x/c=1.1883$ for reattachment.



a) Separation and reattachment



b) Separation

Figure 4-24 Mean Skin friction coefficient – x/c .

An appreciation of the bubble length can be seen in Figure 4-24-a, and on Figure 4-24-b a close-up to the separation is observed. Note that there is a sudden drop in the mean skin friction coefficient at approximately $x/c = 0.66$. this drop is due to the jet exit influence, and do not correspond to the start of the separation. The separation region is dynamically changing over time, if the values in the Figure

4-24 were not mean values and this graph was changing with the time step, an oscillatory behaviour would be seen on both separation and reattachment regions. The separation and reattachment were considered on the concave shape shown in the Figure.

DSJ analyses

Due to the nature of this analysis, when a velocity profile is defined at the boundary of the jet exit, at angles away from 90° , the mass flow rate is affected by the direction, as the relative area of the exit becomes smaller. To avoid this unrealistic behaviour, but still compare the flow characteristics expected, three approaches were taken to the most relevant cases, where the direction of the input and output are equal.

Several simulations were performed to cover various combinations of inflow/outflow jet angles, with the three different turbulence models. Also, three different approaches regarding the jet exit conditions were employed to fully understand the characteristics of the Bi-Directional synthetic jet in comparison with the classical synthetic jet. These three approaches are: (i) Varying mass flow rate, i.e. keeping the same area (width) of the jet slot and the same amplitude of the velocity as the classical synthetic jet case; (ii) Varying velocity, i.e. keeping the same area (width) of the jet slot and the same mass flow rate as the classical synthetic jet case; and (iii) Varying jet area, i.e. keeping the same amplitude of the velocity and the same mass flow rate as the classical synthetic jet case. Note that, in this CFD study, it is impossible to fix these three parameters (mass flow rate, velocity and jet slot width) all together when the jet direction is changed (since the jet exit boundary is always aligned with the hump surface). For the

“Varying jet area” case, the middle point of the jet slot is fixed while changing the slot width, to make a fair comparison.

In all three cases, the Varying values are modified by a factor proportional to the sinusoidal value of the angle at which this jet is operating. e.g. at 15° , a factor of 0.25 is used, while at 75° all values are modified from the 90° case by a factor of only 0.96.

4.3.2 Varying Mass Flow Rate (MFR)

A series of simulations were performed with three turbulence models, six jet outflow directions and eleven jet inlet orientations, giving a total of 198 simulations. The following three figures (Figure 4-30, 17 and 18) summarize the results.

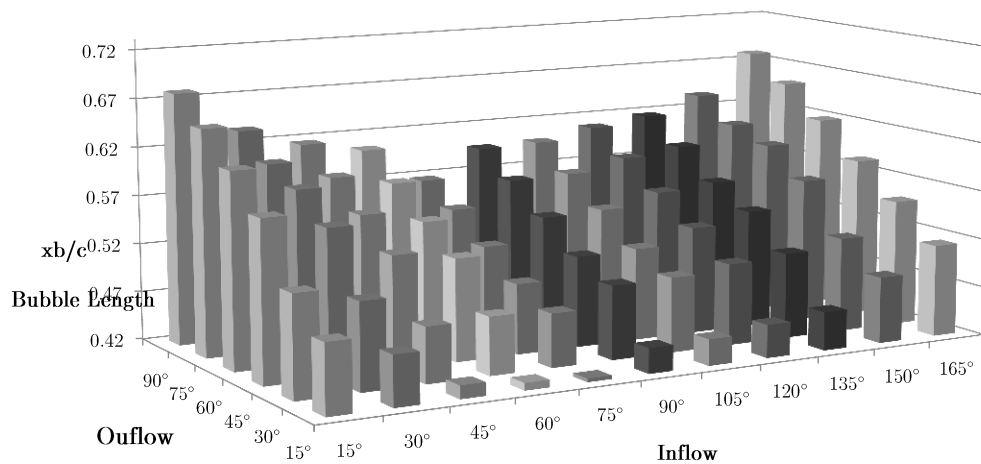


Figure 4-30 Separation bubble length using $k-\omega$ SST.

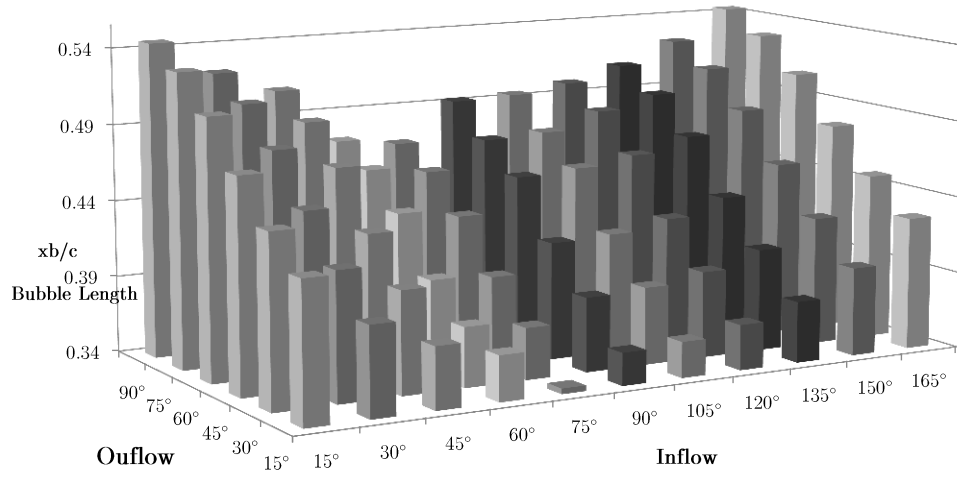


Figure 4-31 Separation bubble length using the Realizable $k\text{-}\epsilon$ model.

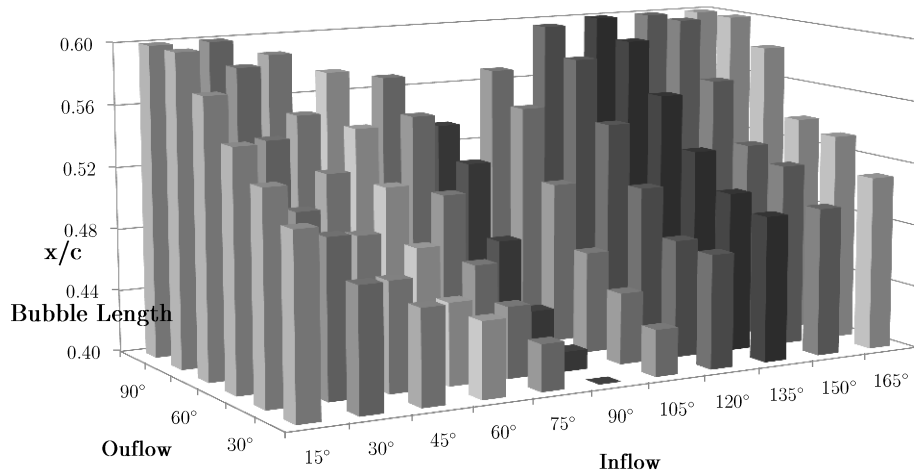


Figure 4-32 Separation bubble length using the S-A model.

As can be seen from these three figures, the $k\text{-}\omega$ SST and Realizable $k\text{-}\epsilon$ models tend to predict the configuration of inflow angle 75° and outflow angle 15° (75-15) as the one with the shortest separation bubble size, whereas the S-A

model predicts a slightly different configuration (90-15) as the minimum size configuration. Also, the first two models show somewhat different trends before and after 90° for the inflow angle.

As a constant velocity boundary condition was set at each inflow and outflow angle, when the direction of the jet is perpendicular to the surface, a rectangular stroke length which base is equal to the jet exit and height equal to the jet exit velocity can be drawn. At 90-90 there is a maximum amount of flow coming out of the hole. As the vector of the jet velocity inclines, the rectangular stroke length drawn squeezes, due to the stretching of the relative base, e.g. at 45° the relative base would be 30% smaller, ($\cos 45^\circ = 0.7$). This shrinking effect, due to the computational simulation is the cause of the “unrealistic” behaviour in the jet, and is treated and compensated in the following section by applying a set of different boundary conditions.

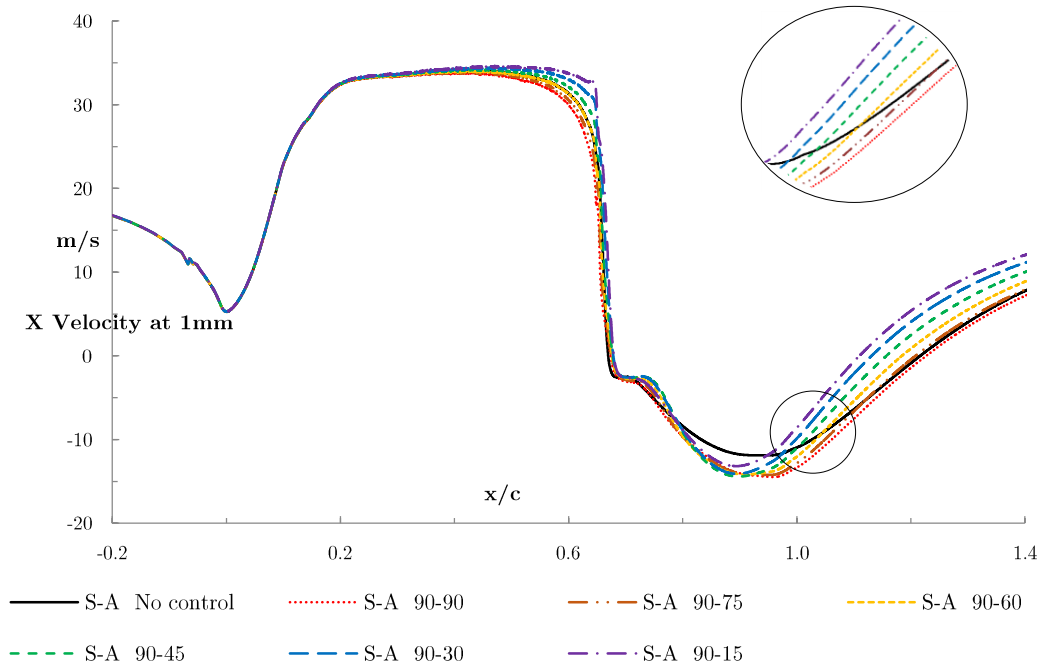


Figure 4-33 Streamwise x-velocity at 1mm above the wall surface;

The x-velocity along a line 1mm above the wall surface is plotted in Figure 4-33 (the first and second numbers of each case describe the inflow and outflow angles of the jet, respectively), for the S-A model with a fixed inflow angle of 90° . The “no control” case has also been plotted here for comparison; it is clear from this graph that the smaller the outflow angle the sooner the reattachment takes place. Figure 4-34 shows a comparison of pressure coefficient distributions along the wall surface for the same set of cases. A continuous change of the pressure distribution can be seen with the change of the jet outflow angle.

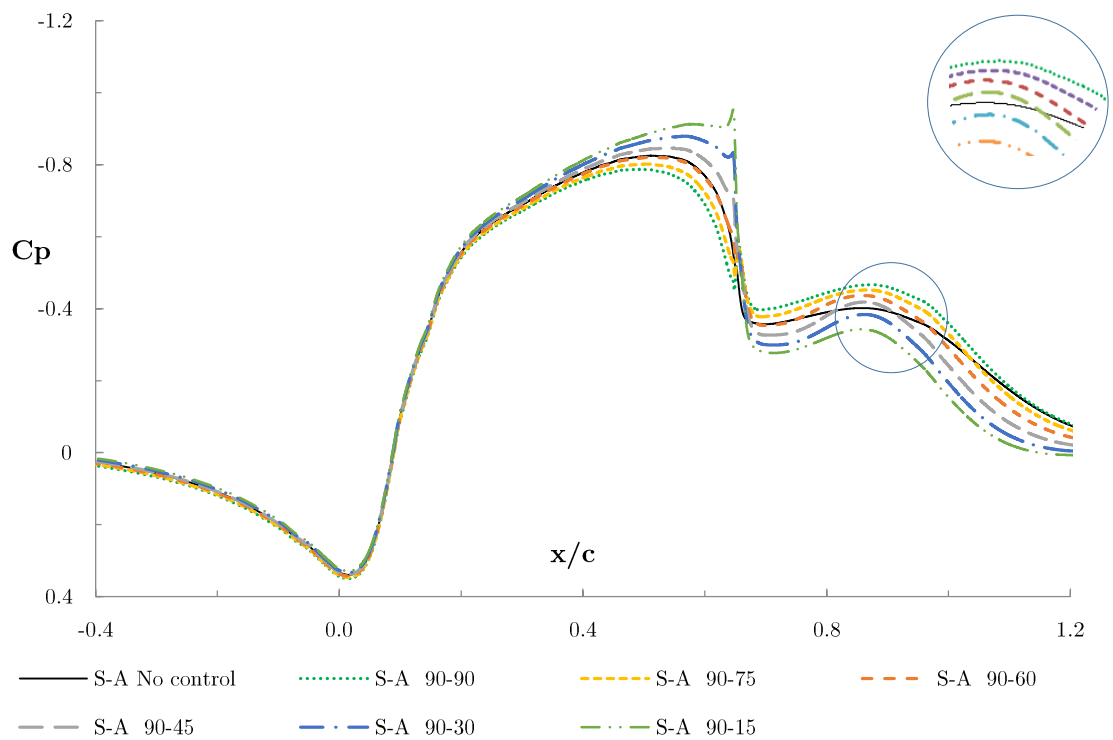


Figure 4-34 Pressure coefficient comparison along the hump.

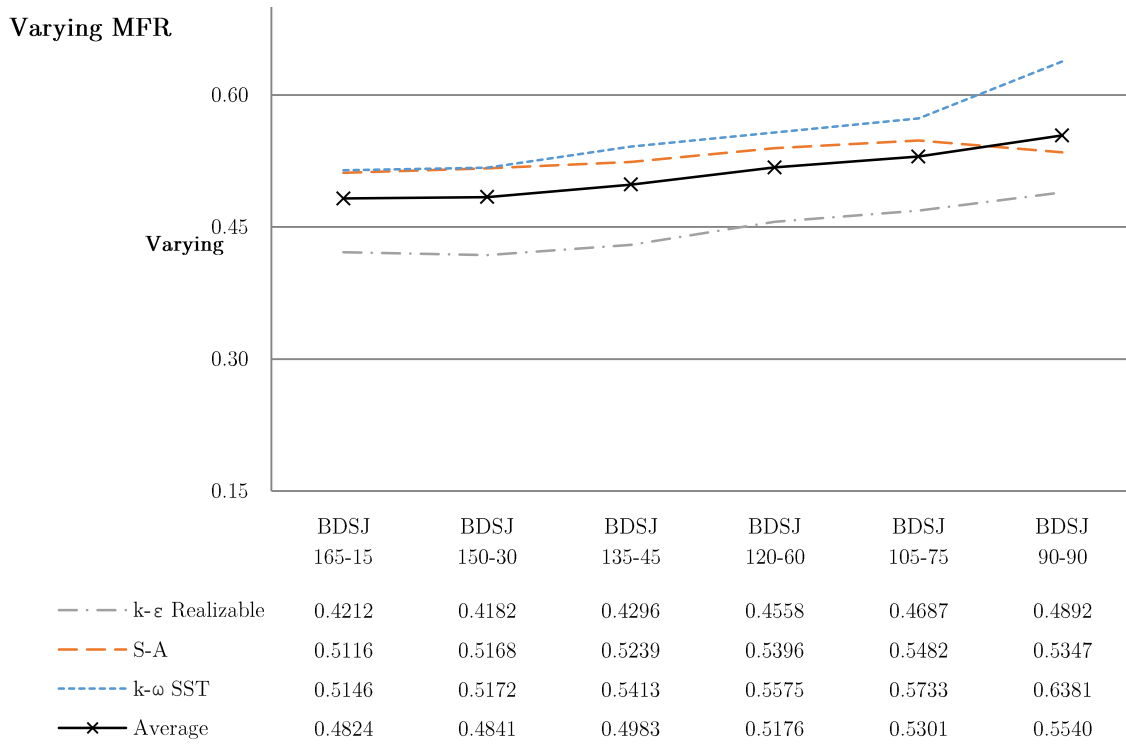


Figure 4-35 Bubble length for the Varying Mass Flow Rate analyses

In Figure 4-35 the separation bubble size is compared for the cases of the BDSJ with 15, 30, 45 and 60 degrees of inclination, with the classical synthetic jet 90° perpendicular to the surface.

Figure 4-35 shows comparisons of the separation bubble length between the most representative cases of the BDSJ with inlet-outlet jet directions of 165°-15°, 150°-30°, 135°-45°, 120°-60°, 105°-75°, and the classical synthetic jet case (90°-90°), for the three turbulence models. The results for the three different turbulence models are all plotted for comparison, together with their average results. As can be seen from the figure, the general trend is that the BDSJ performs better than the classical synthetic jet, i.e. the separation bubble length reduces as the difference between the inflow and outflow angles is increased.

4.3.3 Varying Area

The procedure followed for the Varying area cases, consisted in modifying the two-dimensional area of the jet exit, in such a way that both mass flow rate and velocity at the jet exit are kept constant for different jet angles. The proportional values of modification in all three cases are the same, which are the values of $\sin(15^\circ)$, $\sin(30^\circ)$, $\sin(45^\circ)$, $\sin(60^\circ)$, and $\sin(75^\circ)$. Figure 4-36 shows the boundaries at the jet exit over the hump, which were alternatively applied to increment or diminish the area to keep the other parameters constant.

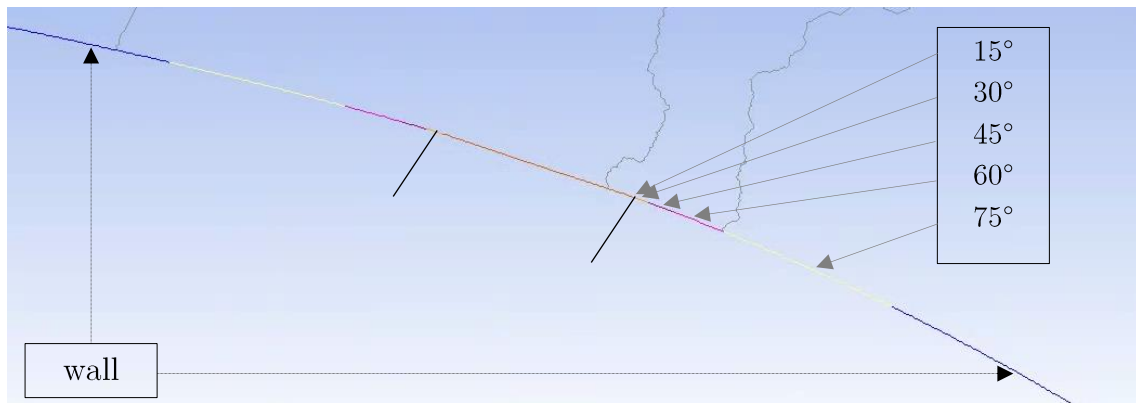


Figure 4-36 Jet exit boundaries in the hump for Varying Area

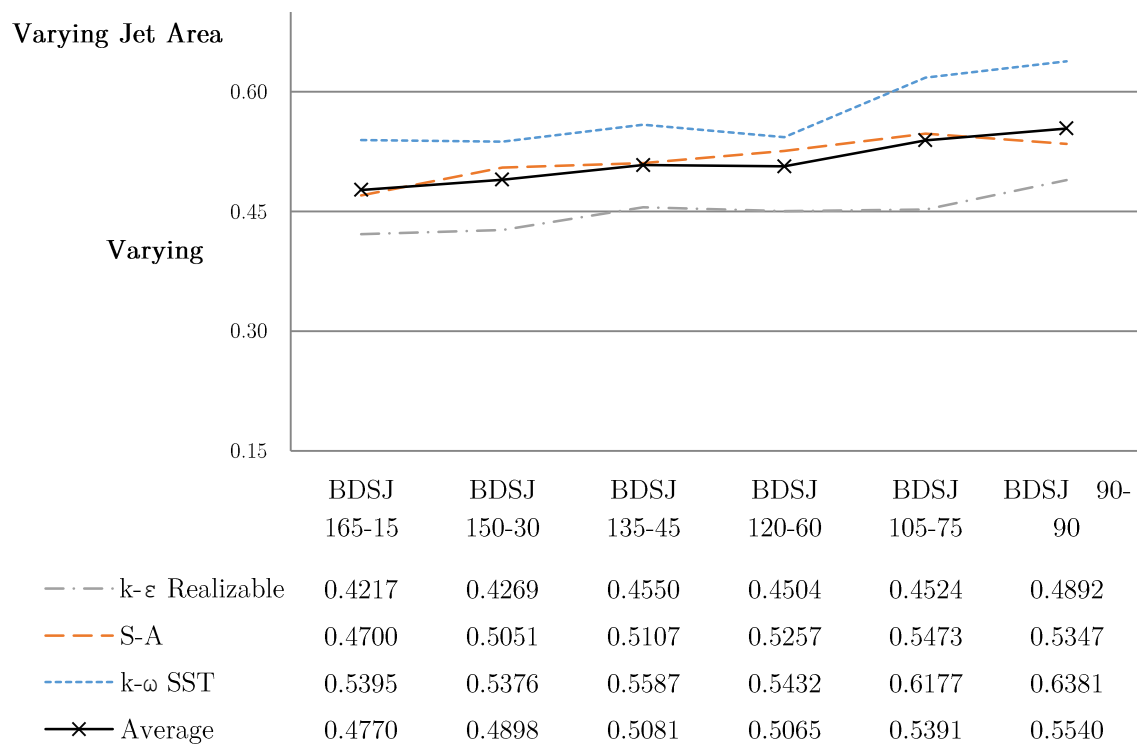


Figure 4-37 Bubble length for the Varying Jet Area analyses.

In a similar manner, for comparative purposes, the boundary conditions were adapted to include how a Directed Synthetic Jet would behave under such conditions. Figure 4-38 shows the dimensionless bubble length for the Directed Synthetic Jet cases.

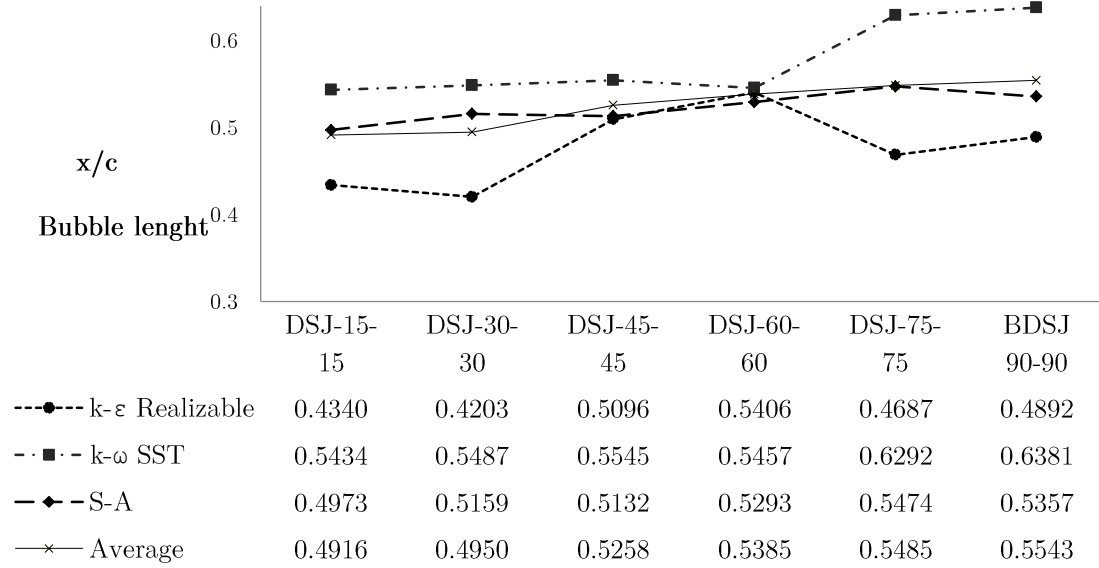


Figure 4-38 Bubble length for DSJ cases.

A slight increasing trend is noted as the exit angle approaches zero, the average values and plot in the previous Figure show a clear increasing slope, suggesting a higher effectivity for drag reduction on directed synthetic jets whose exit is closer to parallel to the surface.

4.3.4 Varying Velocity

In this section, the velocity at the jet exit was modified by the same factor previously commented to keep the mass flow rate and the area of the jet exit constant for different jet angles. It is worth noting that a higher jet velocity may result in a higher friction loss within the internal part of the Synthetic jet and hence a higher power required operating the device.

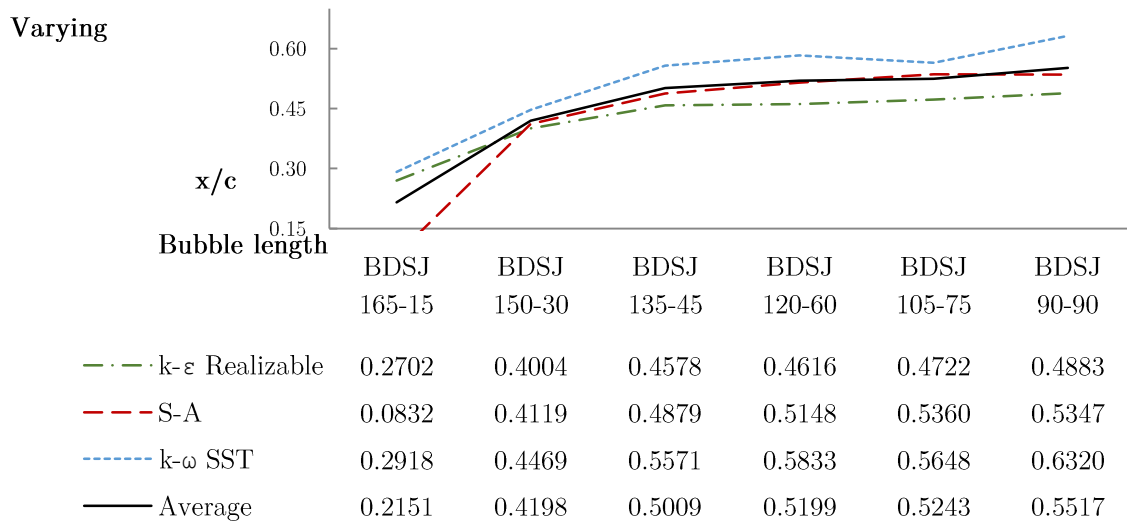


Figure 4-39 Bubble length for the Varying Velocity analyses

Figure 4-39 illustrates the results of changing the jet velocity as mentioned. In this plot is very clear an increase in the size of the separation bubble for all cases. This is because the jet velocity is four times higher at the smallest end (BDSJ 165-15), injecting more momentum into the boundary layer, which allows the Coandă effect to take place and attach the boundary layer. The results on Figure 4-40, show a steady decrease in size of the bubble length for the DSJ as its angle of incidence increases, just as in the previous figure, suggesting that the effect of the exit velocity may have higher impact on the flow control capabilities of the

synthetic jets. Taking into consideration this new knowledge when designing the BDSJ, a multiplication of velocity system could be included inside the design, to take advantage of this possible effect. However, more research needs to be done to further conclude on this matter.

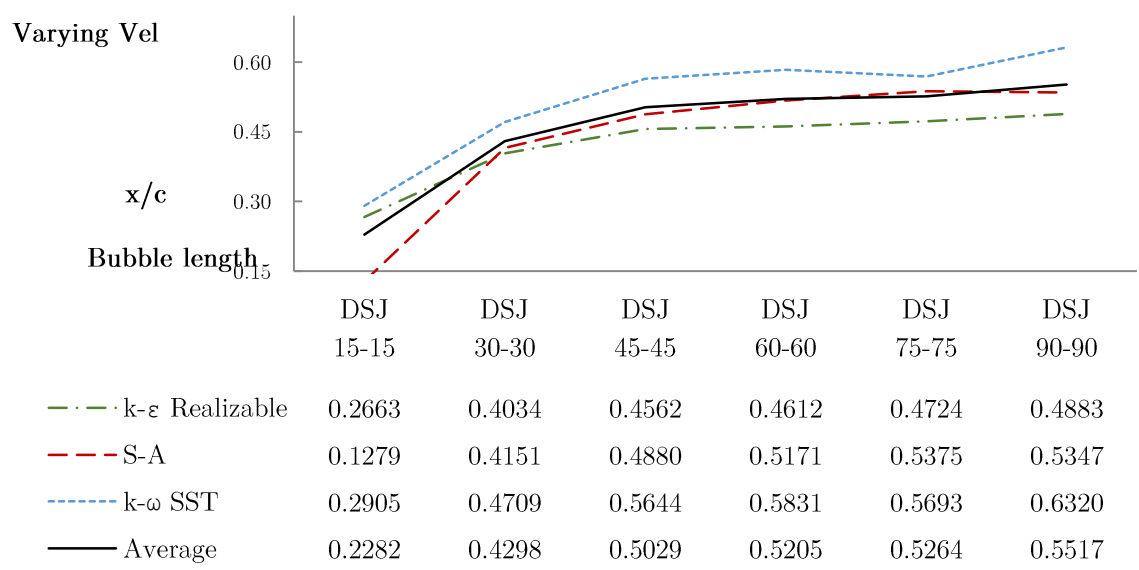


Figure 4-40 Bubble length for DSJ

4.3.5 Jet comparisson

Figure 4-41 shows comparisons of the contours of the time-averaged streamwise velocity over the hump for the cases with the classical synthetic jet whereas Figure 4-42 and Figure 4-43 compare the BDSJ 165-15 for the varying area and varying velocity cases respectively. These contours illustrate the effect of the different synthetic jet conditions on the size of the separation bubble not only in the streamwise but also in the wall-normal directions.

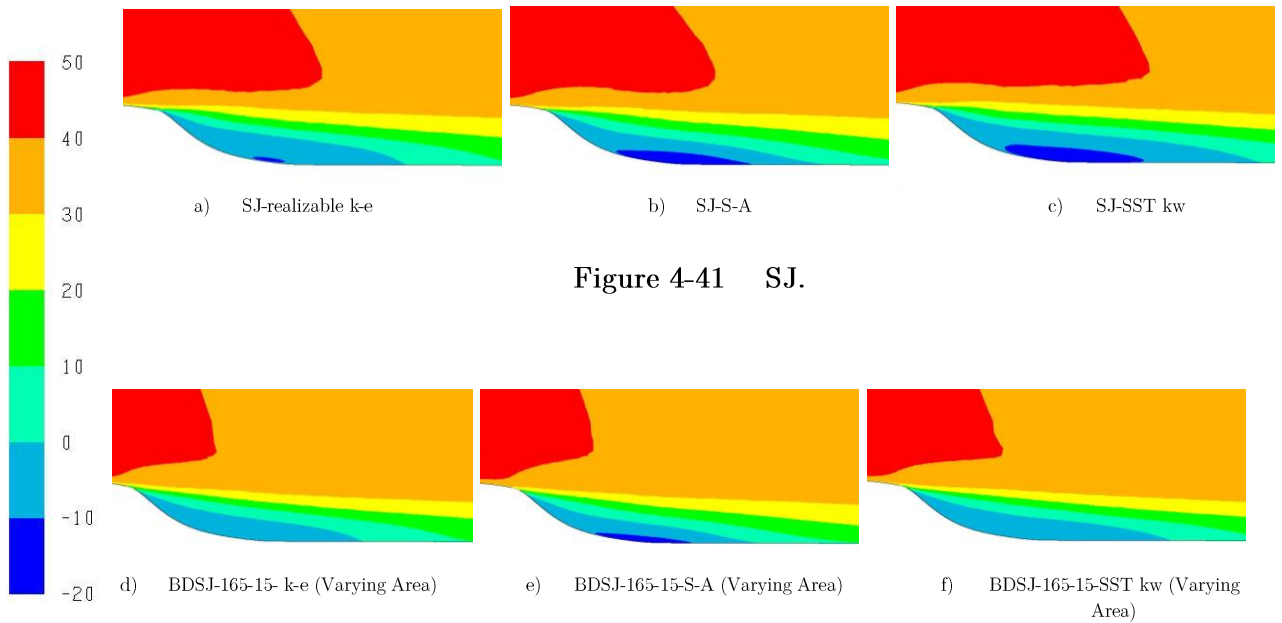


Figure 4-41 SJ.

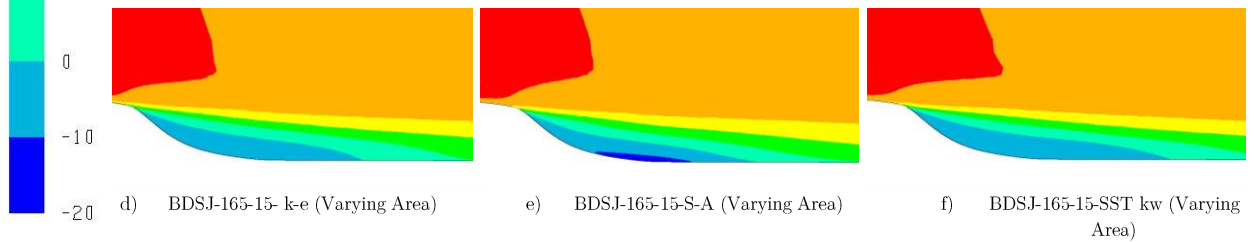


Figure 4-42 BDSJ 165-15 Varying Area

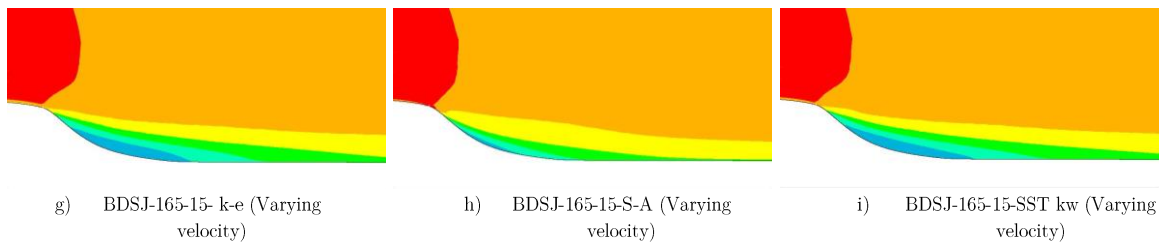


Figure 4-43 BDSJ 165-15 Varying velocity.

In Figure 4-44, the average of the previous three approaches for every turbulence model can be found in a graphical manner to understand in a comprehensive manner the influence of each contribution. The decrease in the bubble size is evident for the cases where the intake-outtake is closer to parallel to the direction of the flow.

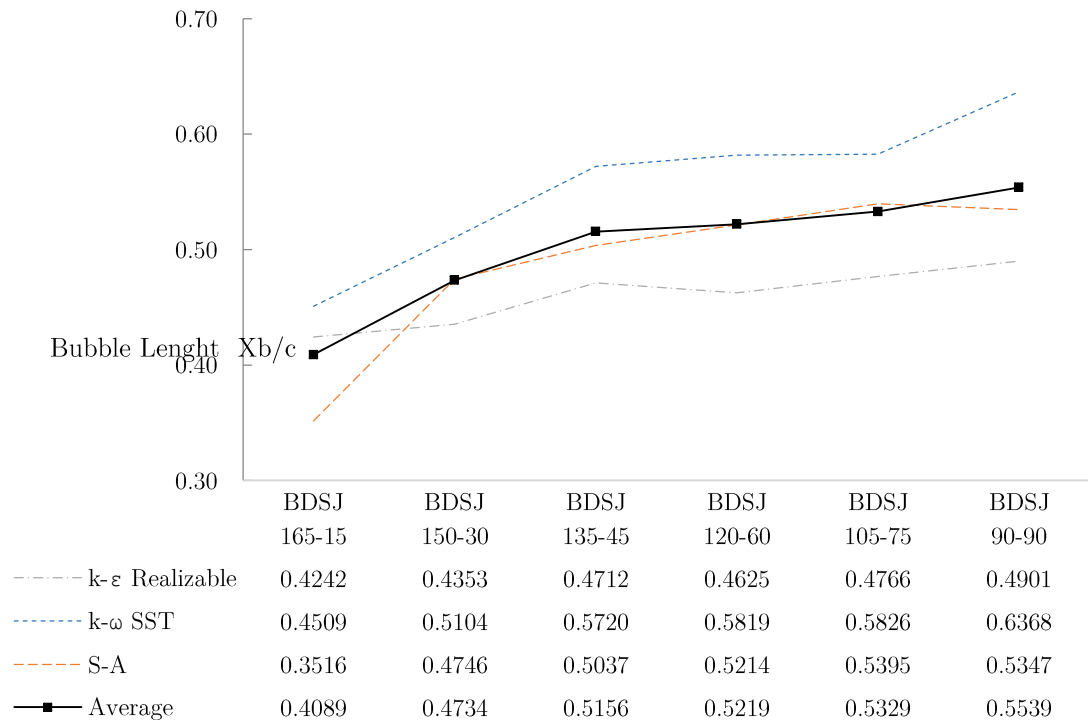


Figure 4-44 Average of varying cases, MFR, Velocity and Area for BDSJ.

The same procedure can be followed for the DSJ cases, which results are shown in Figure 4-45. In a similar way to the BDSJ case, the closer the exit angle to the perpendicular, the smaller the bubble size. To visualize whether the impact of changing the inflow direction affects the results,

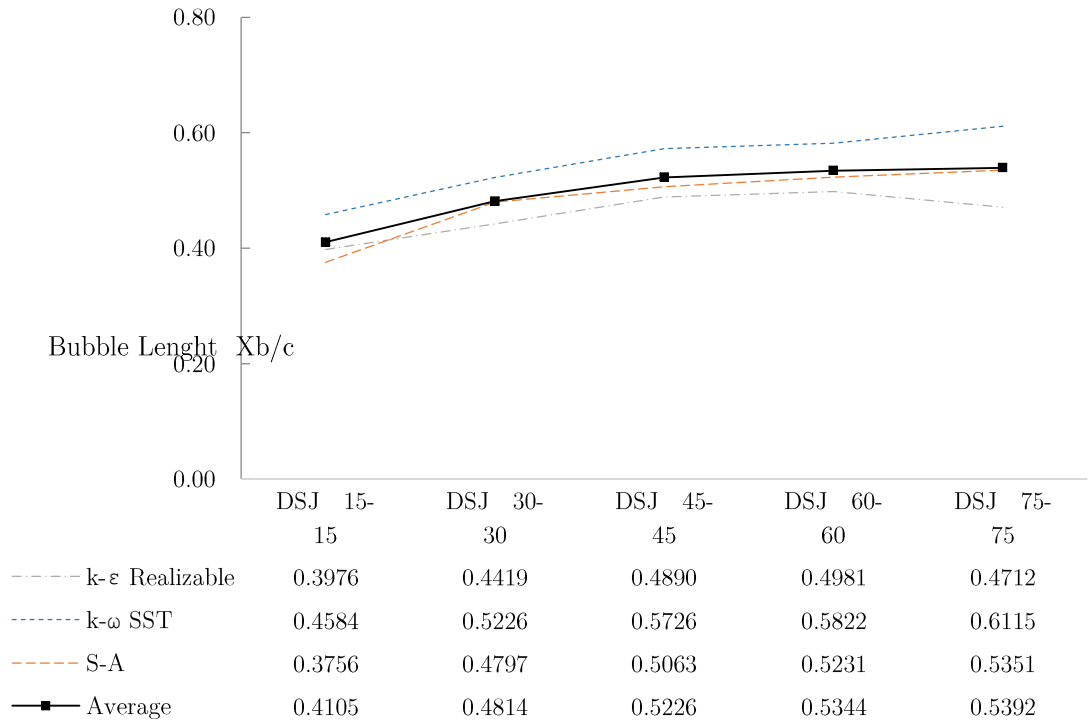


Figure 4-45 Average of varying cases, MFR, Velocity and Area for DSJ.

To measure the impact of only changing the direction of the inflow, a comparison of the BDSJ with the DSJ could be included. Table 8 shows the percentage difference between the BDSJ and DSJ of the average values of the three turbulence models for each approach (Varying MFR, Area and Velocity). From these data, we can see small but constant differences in the size of the separation region, with the BDSJ decreasing this size in most of the cases by up to 6%, suggesting a better overall performance of BDSJ compared with DSJ.

Table 8 Percentage difference of Bubble size

Average for the Three turbulence models			
	$(\text{BDSJ-DSJ})/\text{BDSJ}$		
	MFR	Area	Velocity
Out-15	1.76%	-0.17%	-6.11%
Out-30	-0.40%	-2.40%	-2.40%
Out-45	-0.27%	-3.47%	-0.39%
Out-60	-0.91%	-6.33%	-0.11%
Out-75	-2.69%	-0.49%	-0.39%

4.4 KEY FINDINGS SUMMARY

From the previous sections of this chapter a few key point are worth to stand out.

- Single varying the velocity at the jet exit as a boundary condition is not enough to evenly simulate the performance and distribution of the flow in the BDSJ, and corrections have to be made. In the present chapter correction with varying velocity, mass flow rate and cross sectional area were used to compensate this effect.
 - The amplitude or velocity at the jet exit is particularly influential in the final size of the separation bubble length.
 - When comparing the separation bubble size downstream of a hump for cases such as a) actuation of a SJ b) actuation of a DSJ and c) actuation of a BDSJ, the shortest time average bubble length is obtained by the BDSJ.
-

5 AERODYNAMIC PERFORMANCE WITH A BIDIRECTIONAL SYNTHETIC JET^{rr}

This chapter presents a numerical investigation to answer the question – “How a Bi-directional synthetic jet influences the aerodynamic performance of a aerofoil”. Simulations are carried out on a NACA 23012 at a Reynolds number of 2.19×10^6 , where a two-dimensional structured mesh is used with RANS turbulence models to evaluate the impact of the amplitude and frequency of the Bidirectional synthetic jet on the aerofoil performance over a wide range of angles of attack. Three distinct jet oscillating frequencies: $F_j^+ = 0.159, 1$ and 3.24 are considered as well as three blowing ratios, $V_j/U_\infty = 1, 3$ and 5 . The phase of the inflow-outflow, the position and the configuration of

^{rr} Part of the content of this chapter is submitted to the AIAA Journal with the title “CFD Study of a Bidirectional Synthetic Jet over a NACA 23012 Airfoil”.

the jet exits around $0.12c$ are also evaluated. The outputs suggest that the amplitude of the jet is a key parameter to control while looking for the highest lift coefficient and a gap between exits could positively influence the lift if the central location of the two jets coincides with the separation point. On the other hand, the range of frequencies used and the phase at which the jets operate do not seem to have effect on the final output.

5.1 TEST CASE DESCRIPTION

One of the long-standing challenges in aeronautical engineering is to enhance the efficiency of wings and blades, i.e. to maximize lift and minimize drag. Since the performance of aerodynamic surfaces greatly influences fuel consumption from take-off to landing, an improvement in this regard leads to the reduction of cost and environmental impact. In order to achieve this goal, the bidirectional synthetic jet has been proposed in the previous chapter as a potential new active flow control device.

The efficiency of a synthetic jet as an active flow control device over an aerofoil has been studied by several authors. For instance, Amitay⁴⁵ has found that both the location and the strength of the jet affect the separation bubble length over an unconventional airfoil. You and Moin¹¹⁰ confirmed the effectiveness of synthetic jets using large-eddy simulations (LES), where the onset of separation over a NACA 0015 aerofoil was effectively delayed by using a synthetic jet, resulting in a significant increase in the lift coefficient. They also pointed out the importance of the prediction on the flow inside the synthetic jet actuator, as the directional variations of the jet during the oscillatory periods affect the boundary-layer. Kim and Kim¹⁴² found that the highest lift was achieved when the synthetic jet location coincided with the separation point. They also suggested that the control surface performance and stall characteristics could be improved by resizing separation bubbles. Other types of synthetic jets have also been studied; for example, Esmacili et al.⁵⁹ numerically analysed with Reynolds-averaged Navier-Stokes (RANS) simulations the effectiveness of a tangential synthetic jet as a separation control device over a NACA 23012 aerofoil at $Re=2.19 \times 10^6$, where the

jet blowing ratio and the inclined angle of the jet were studied. In summary, the classical synthetic jet over an aerofoil has been studied extensively in recent years using both experiments and computations. However, the characteristics of the BDSJ as an active flow control device over an aerofoil have not been investigated experimentally or even computationally.

In the present chapter, the BDSJ is used over a NACA-23012 aerofoil to study numerically the influence of its amplitude and frequency on the lift and drag coefficients. Also, the effects of spatial (streamwise) gap and temporal phase differences between the inflow and outflow jets on the aerodynamic coefficients are investigated. The nomenclature used for the BDSJ in this chapter is similar to the previous chapter; as shown in Figure 5-1, the difference here is the rotation of the reference with respect to the baseline 0° AOA, as the base^{ss} of Figure 5-1 will be changing with AOA.

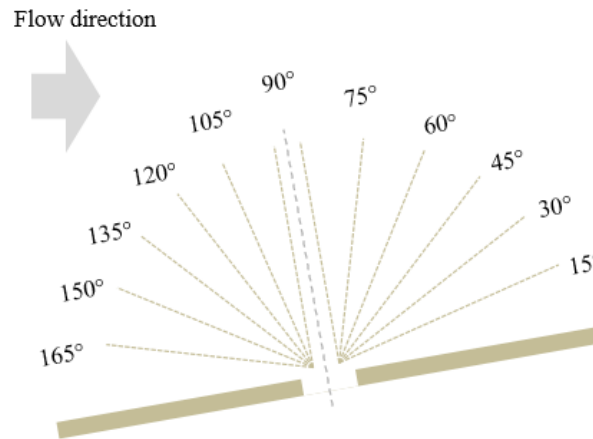


Figure 5-1 Schematic of BDSJ for 0° AOA

^{ss} As the BDSJ is collocated at 12% of the chord of a NACA23012, the inclination over the horizontal is 18.5°

5.2 VALIDATION

5.2.1 Computational domain

The jet was simulated over a two-dimensional NACA 23012 profile with chord length (c) equal to 1 m and the distance from the aerofoil to the inlet and outlet boundaries are both 20 chords. The profile has two 1.1 mm cavity slots at 12 % of the chord ($x/c=0.12$). These two slots correspond to the inflow and outflow ports of the BDSJ and can be moved towards both upstream and downstream by up to five times the slot width (D). The computational domain, together with the final mesh employed in this study, is shown in Figure 5-2. Details of a grid convergence study will be described in section 5.2.5 .

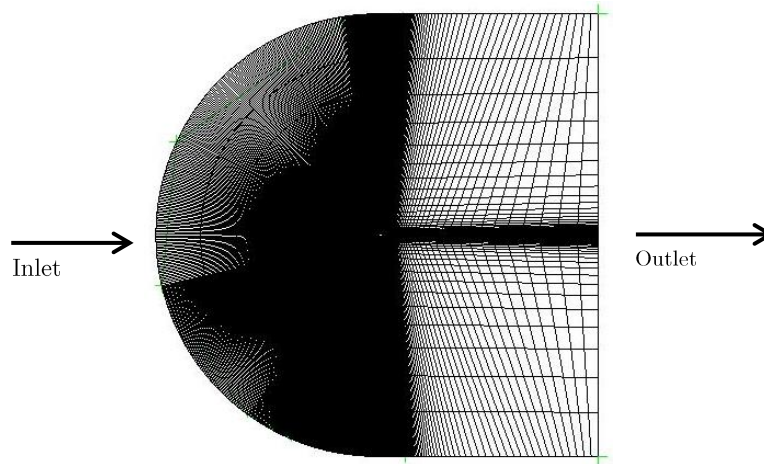


Figure 5-2 Discretized computational domain

5.2.2 Flow solver and parameters

The software used in this study is the next version to the previous chapter, ANSYS FLUENT v15, in which a pressure-based solver is used with the SIMPLE algorithm for pressure-velocity coupling for steady state calculations, while the PISO algorithm is used for the transient calculations¹³⁶ with a second-order fully implicit scheme for time integration.

The working fluid is air at 20°C with a viscosity of 1.78×10^{-5} kg/ms and a density of 1.1935 kg/m³, giving a Reynolds Number (based on the airfoil chord length) of 2.19×10^6 . The reference pressure for the domain is set at 101,325 Pa at the outlet boundary, the inlet velocity is $U_\infty = 32.64$ m/s (Mach 0.1) and the jet frequency (when applied) varies from 5.90Hz to 37.09Hz and 120Hz, which in dimensionless form are $F^+ = 0.159, 1$ and 3.24 respectively. These set of frequencies were used for a number of simulation on the 2004 workshop, and are used here to keep continuity and coherence with the previous validated study. Here F^+ is defined as $F^+ = fL/U_\infty$, where f is the frequency in hertz, L is a characteristic length, which in this case is the distance from the jet exit to the trailing edge, and U_∞ is the freestream velocity. The jet amplitude varies from $U_j/U_\infty = 1, 3$ and 5 . Each computation is initiated with a uniform velocity $(u, v) = (U_\infty, 0)$ given to the entire computational domain. For transient analyses, the time step size is set as $\Delta t = 4.49 \times 10^{-4}$ seconds for $F^+ = 0.159$, $\Delta t = 7.15 \times 10^{-5}$ seconds for $F^+ = 1$ and $\Delta t = 2.21 \times 10^{-5}$ s for $F^+ = 3.24$, all resulting in 377 time steps per synthetic jet oscillation cycle. The sensitivity study has shown that this time step size is sufficiently small. The three frequency used were chosen based on the work of

Regarding the turbulent model, earlier studies, as shown in , have used different turbulence models on a synthetic jet over an aerofoil. As in the previous chapter three RANS models (Spalart-Allmaras, $k-\omega$ SST and Realizable $k-\epsilon$) were compared and found to give similar results. In the present chapter, for simplicity and less computational requirements, the one equation Spalart-Allmaras model is chosen to model the Reynolds stresses in the flow. The turbulence intensity at the inlet is set to be 5% and the turbulence length scale 1 mm.

Table 9 Turbulence models used by different authors.

Authors	Aerofoil	Re	Turbulence Model
Lee et al. ¹⁴³	NACA-64A210	667,000 and 1,000,000	$k-\omega$ SST
Lee et al. ²⁹	NACA-64A210	667,000 and 1,000,000	$k-\omega$ SST
Kim et al. ¹⁴²	NACA-23012	2,190,000	$k-\omega$ SST
Kim et al. ⁸⁵	NACA-23012	2,190,000	$k-\omega$ SST
Monir et al. ⁵⁹	NACA-23012	2,190,000	S-A
Nae ¹⁴⁴	NACA-0012	1,800,000	Standard $k-\epsilon$
Ishibashi et al. ¹⁴⁵	NACA-0012	100,000	DES
Macovei et al. ¹⁴⁶	NACA-0012	135,000	$k-\omega$ SST
You, D. et al. ¹¹⁰	NACA-0015	896,000	LES
Duvigneau et al. ¹⁴⁷	NACA-0015	896,000	$k-\omega$ SST
Kotapati et al. ¹⁴⁸	Flat plate with bounded leading edge	60,000 and 100,000	LES

5.2.3 Modelling of the synthetic jet

It has been reported that different cavity shapes do not significantly affect the flow characteristics at the jet exit¹³⁵; therefore the actual cavity is not modelled. However, even though some authors^{85,44} have used a top-hat velocity profile at the jet exit for its simplicity, here the influence of the jet-exit velocity profile on the aerofoil lift curve is verified. The parabolic profile employed for the jet velocity across the jet exit is the same defined in equation (12) with the main difference being the dimensions of D and S for this geometry.:

$$V_j(\xi) = V_{j,max} \left(1 - \left(\frac{\xi}{D} - 0.5 \right)^2 S \right)$$

Figure 5-2 shows the profiles for the seven cases analysed. As the mass flow rate needs to be constant for a fair comparison, the value of $V_{j,max}$ has been adjusted depending on S . The results for different angles of attack (AOA) are summarized in Table 10.

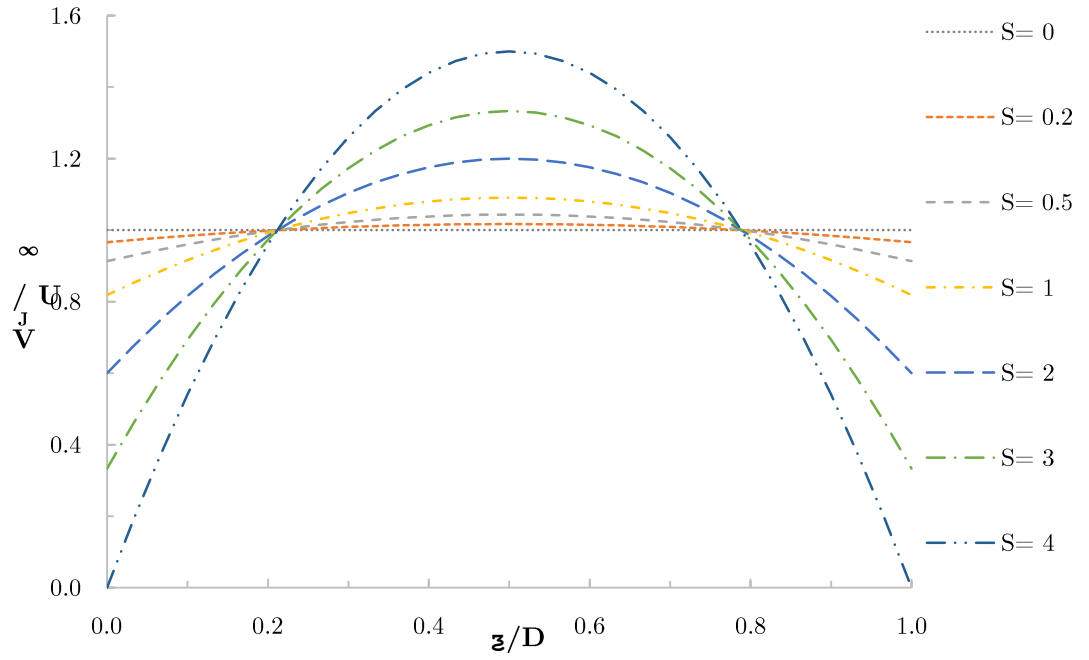


Figure 5-3 Velocity profile at the jet exit for the seven cases analysed

As can be seen in Table 10, the impact of different velocity profiles at the jet exit on the lift and drag coefficients is minimal and less than one percent for most cases, taking the top-hat distribution as a reference (case 1). In the present study, it was decided to use the distribution with $S=2$, as it simulates the closest with the velocity profile would be expected from a flow inside a pipe.

Table 10 Percentage variations in C_L/C_D due to different jet-exit velocity profiles.

Case	S	$V_{J,max}/U_\infty$	$(C_L/C_D - C_L/C_{Dcase1}) / (C_L/C_{DCase1})$			
			AOA			
			15°	17°	19°	21°
1	0.0	1.00				
2	0.2	1.02	0.01%	0.02%	0.02%	0.02%
3	0.5	1.04	0.02%	0.06%	0.08%	0.04%
4	1.0	1.09	0.05%	0.15%	0.22%	0.13%
5	2.0	1.20	0.16%	-0.08%	0.74%	-0.13%
6	3.0	1.33	0.32%	0.38%	1.75%	0.00%
7	4.0	1.50	0.56%	0.87%	3.46%	0.96%

5.2.4 Simulation procedure

Three main steps were followed for each simulation. Firstly, a steady state calculation is performed for 15,000 iterations or until the continuity residual reaches 1×10^{-6} . Secondly, the simulation is continued as a transient calculation for at least two seconds of simulated time (corresponding to 65 flow through times based on the airfoil chord) with 20 iterations per time step to get statistically converged solutions. Here the total number of iterations varies since the time step size depends on the jet frequency as noted earlier in Section 5.2.2. Finally, the simulation is run for another 10,000 time steps (corresponding to about 27 jet oscillation cycles) to obtain the time-averaged solutions.

Each set of simulations is performed for 23 different angles of attack from 0° to 22° , taking approximately 72 hours with 32 parallel Intel E5-2660 (Sandy Bridge) CPUs (2x16 cores at 2.2 GHz) using the S-A model.

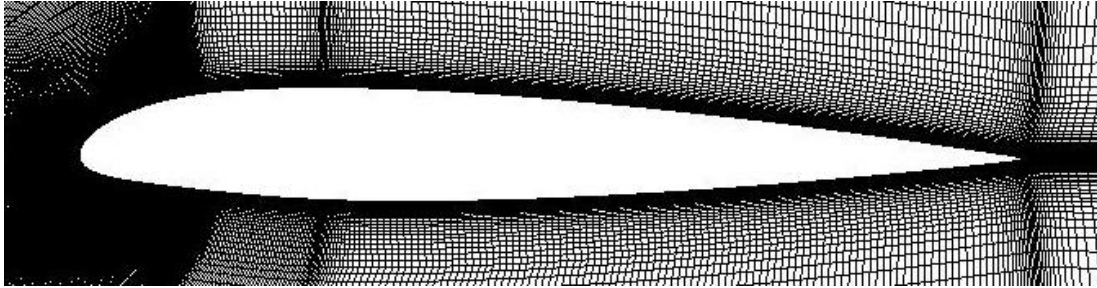
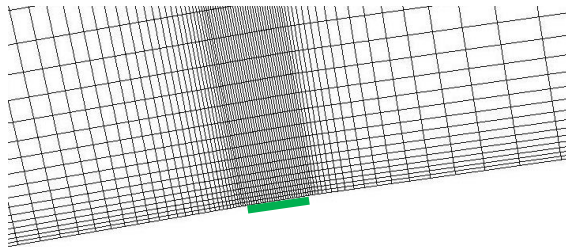
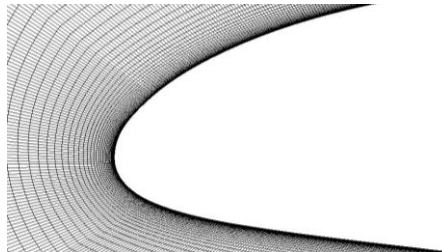


Figure 5-4 Computational mesh around the profile



b) Elements close to the Jet exit (47k mesh)



c) Elements close to the leading edge

Figure 5-5 Computational Grid, NACA-23012

5.2.5 Grid convergence

To examine the grid sensitivity, six different mesh resolutions were assessed for the case without flow control. The mesh was generated using a C-Grid topology with a layer close to the wall made of 10 elements (for the coarsest grid consisting of 13 thousand elements in total) to 60 elements (for the finest grid consisting of 140,000 elements) at a growth ratio from 2 to 1.1. The mesh refinements were performed in both wall-normal and chord-wise directions near the wall, while the far field was not modified through the refinements. The average y^+ value along the chord at all AOA is less than 1 for the finest grid with 140,000 elements. Also, the finest grid was modified with redistribution of elements around the aerofoil to improve the quality as well as reduce the size of the final grid, which has 66,000 elements and is hereafter referred to as “Redistributed 66k” grid (shown in Fig. 5-4). As can be seen from the lift coefficient results in Figure 5-6 and Table 11, the final “Redistributed 66k” grid shows little difference from both experiments¹⁴⁹ and the 140,000-element grid; therefore, this grid is used for the rest of the analyses in this chapter.

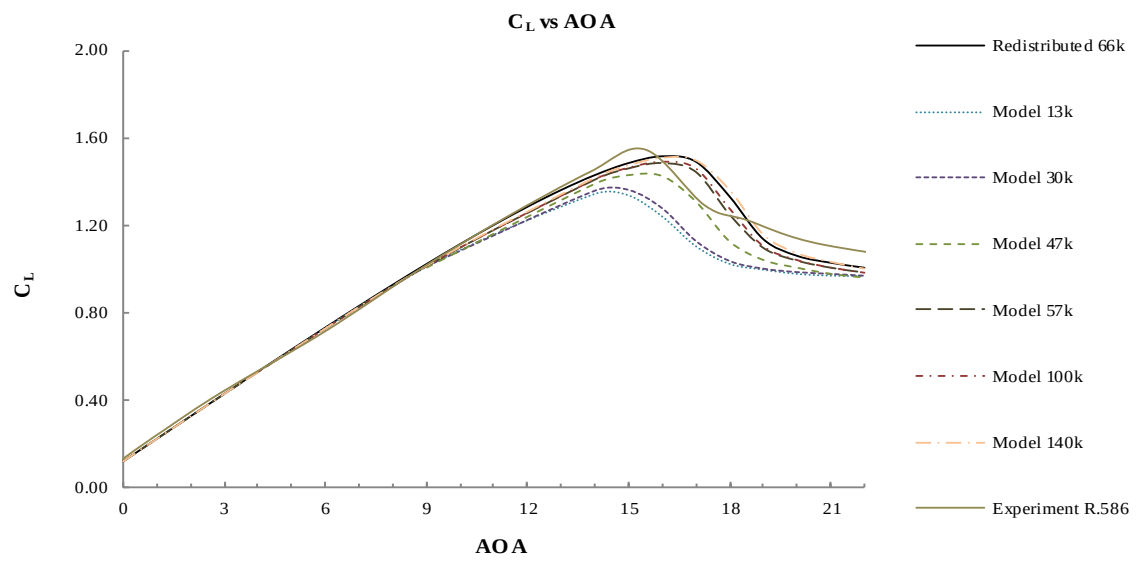


Figure 5-6 Lift coefficient comparison.

Table 11 shows the values of each grid and a column at the right with the relative difference between the lift coefficient of each grid and the experiment, showing a clear converging trend.

Table 11 **Average differences in lift coefficient for different grids**

	C_L Experiment R.586	Model 13k	Model 30k	Model 47k	Model 57k	Model 100k	Model 140k	Redistributed 66k
AOA	$(C_{L@case} - C_{L@Exp}) / C_{L@Exp}$							
9	1.017	-1%	-1%	-1%	-1%	-1%	0%	0%
14	1.454	-8%	-6%	-5%	-3%	-3%	-3%	-2%
15	1.517	-12%	-10%	-6%	-4%	-4%	-3%	-2%
16	1.471	-16%	-13%	-3%	1%	1%	3%	3%
17	1.326	-17%	-15%	-2%	9%	10%	13%	12%
18	1.250	-18%	-17%	-10%	0%	2%	9%	6%
19	1.198	-17%	-17%	-13%	-9%	-8%	-4%	-5%
20	1.148	-15%	-14%	-13%	-10%	-9%	-7%	-8%
21	1.105	-13%	-12%	-12%	-9%	-9%	-7%	-7%
22	1.082	-11%	-11%	-11%	-9%	-9%	-8%	-7%
Average difference		-13%	-12%	-8%	-4%	-3%	-1%	-1%

5.3 RESULTS AND DISCUSSION

5.3.1 Effect of direction

Firstly, a comparative analysis of six configurations of BDSJ is performed to study the influence of jet angles on the aerofoil lift. The jet angles are modified gradually from the classical synthetic jet configuration (090-090, where the first and second values indicate the inflow and outflow jet angles in degrees, respectively) to the “most efficient” configuration achieved for the NASA 2D hump case in the previous chapter (165-015). For this comparative analysis, the maximum jet amplitude and frequency are fixed at $V_{j,max}/U_\infty = 3$ and at 37Hz, respectively. Figure 5-7 shows the lift coefficient for all six configurations. It can be seen from the figure how the most inclined jet configuration (165-015) tends to improve the lift coefficient for most AOA. In the following, the “165-015” configuration is employed.

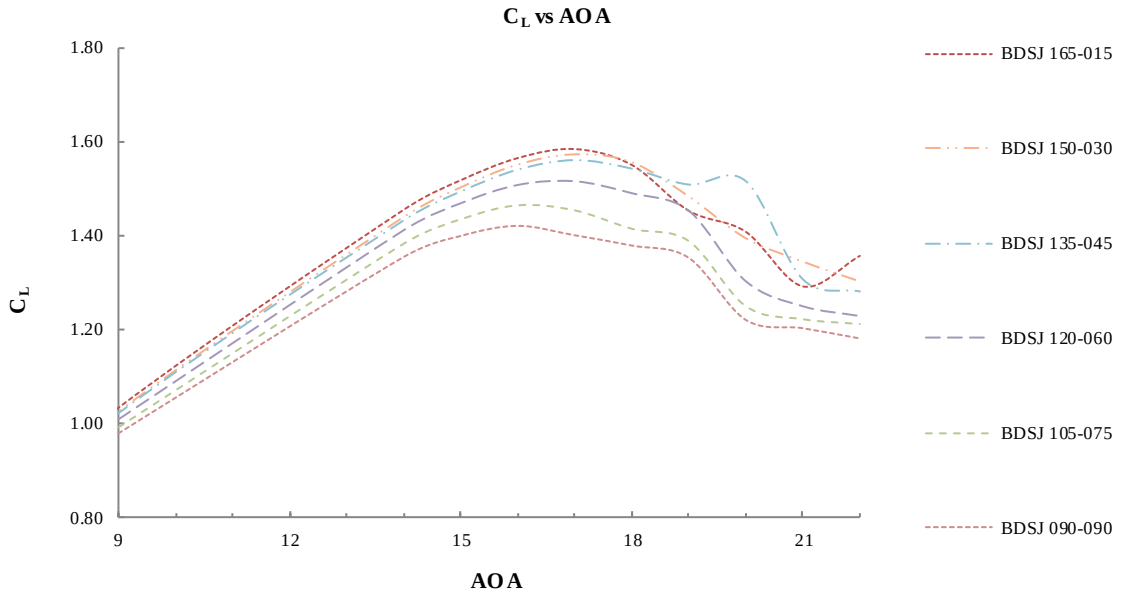


Figure 5-7 C_L vs AOA comparison for different BDSJ configurations

5.3.2 Effect of the Jet Amplitude and Frequency

The velocity at the jet exit (located at $x/c=0.12$) fluctuates in time as $V_{jet}(\xi, t) = V_j(\xi) \cdot \sin \omega t$, where $V_j(\xi)$ is the velocity defined in section 5.2.3. Here three different values of the maximum jet velocity $V_{j,max}$ are tested: 32.64 m/s, 97.92 m/s, and 163.2 m/s for cases “1A”, “3A” and “5A”, respectively (where “XA” means that the maximum jet velocity is X times higher than the freestream velocity). Also, three different values of the jet frequency ω are tested: 5.90 Hz, 37.09Hz and 120Hz. The horizontal and vertical components of the jet velocity, considering the prescribed jet direction (165-015), are specified through a User Defined Function (UDF) in FLUENT, which can be found in Appendix A.1, A.2 and A.3.

In Figure 5-8, the lift coefficient values for the set of nine different analyses as well as the case without flow control are presented, showing the influence of both jet amplitude and frequency for a wide range of angles of attack. As can be seen from this figure, at low angles of attack, there is little impact of the amplitude and frequency. It can also be observed that, at high angles of attack, the lift coefficient values depend largely on the jet amplitude. It can be concluded that an increment in the jet amplitude results in a significant increase in lift coefficient at high angles of attack, whereas the jet frequency has limited influence.

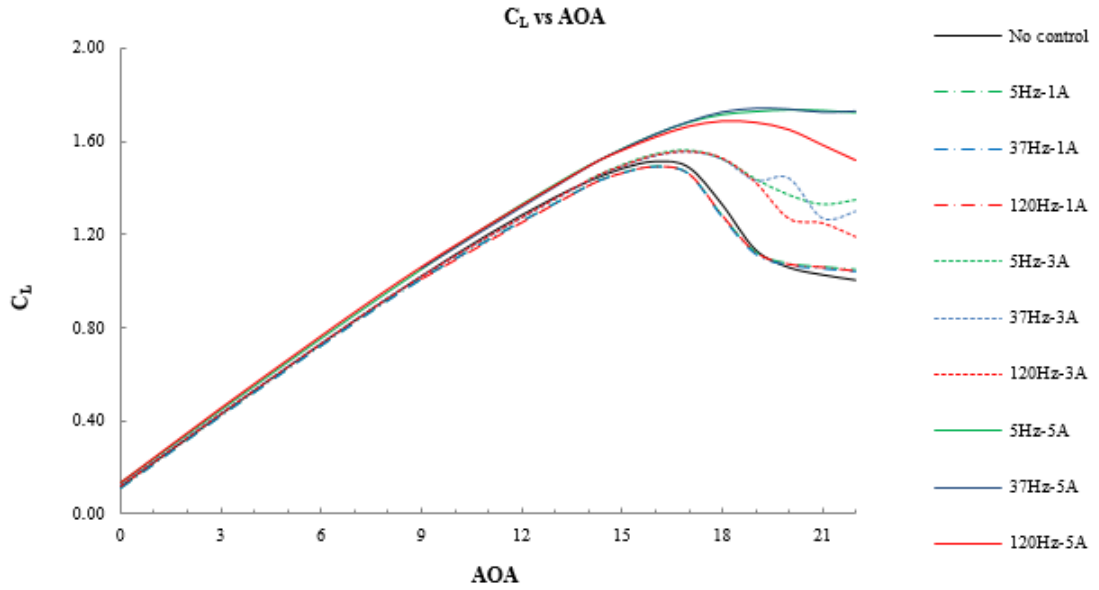


Figure 5-8 C_L vs AOA comparison for a BDSJ

Table 12 shows the differences in percentage for the nine analyses against the baseline case with no flow control. On this table, the small variations in lift coefficient due to different jet frequencies can be appreciated.

Table 12 **Percentile variations of lift coefficient against the baseline case for all frequencies and amplitudes**

AOA	$(C_L - C_{L, \text{Baseline}}) / C_{L, \text{Baseline}}$								
	1A			3A			5A		
	5Hz	37Hz	120Hz	5Hz	37Hz	120Hz	5Hz	37Hz	120Hz
15	0.7%	0.2%	-0.2%	3%	3%	2%	6%	7%	7%
16	0.8%	0.2%	0.1%	4%	4%	4%	8%	9%	9%
17	1.1%	0.6%	0.2%	8%	7%	7%	14%	15%	14%
18	2.9%	3.5%	2.1%	18%	17%	18%	31%	31%	31%
19	7%	4%	3%	32%	28%	32%	55%	54%	55%
20	12%	3%	3%	37%	33%	28%	67%	64%	64%
21	14%	4%	4%	39%	26%	27%	72%	68%	63%
22	8%	5%	5%	42%	35%	27%	75%	75%	59%
Average	6%	2%	2%	23%	19%	18%	41%	40%	38%

As observed in the experiments of Nishizawa¹⁵⁰ with a synthetic jet produced using MEMS at the leading edge of a NACA-0015 aerofoil, higher jet velocities tend to improve the lift but the jet frequency does not tend to affect it significantly.

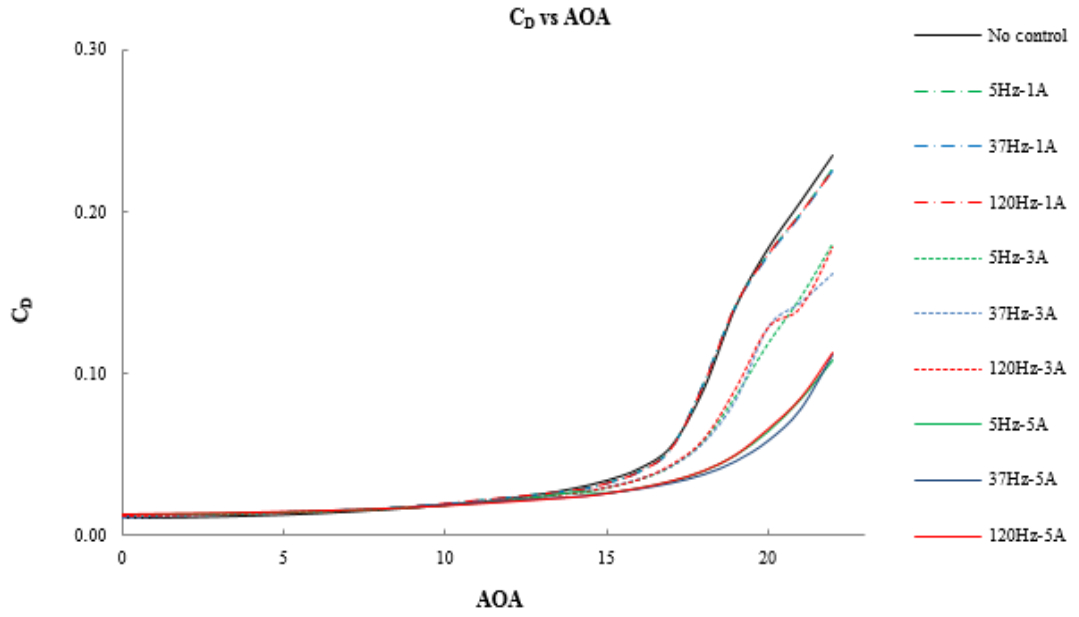


Figure 5-9 C_D vs AOA comparison for a BDSJ

The impact of the jet amplitude and frequency on the drag coefficient can be identified from Figure 5-9. It can be noticed that the jet frequency does not have a clear influence on the drag coefficient. A closer look to the drag distribution is shown in Figure 5-10, in which the contribution of the viscous drag as a percentage of the total drag for all cases with high angles of attack is shown. Here it is clear that the contribution of the viscous drag decreases as the AOA increases. It is worth noting that the viscous contribution to the total drag appears to be less dependent of the amplitude of the jet as the AOA increases.

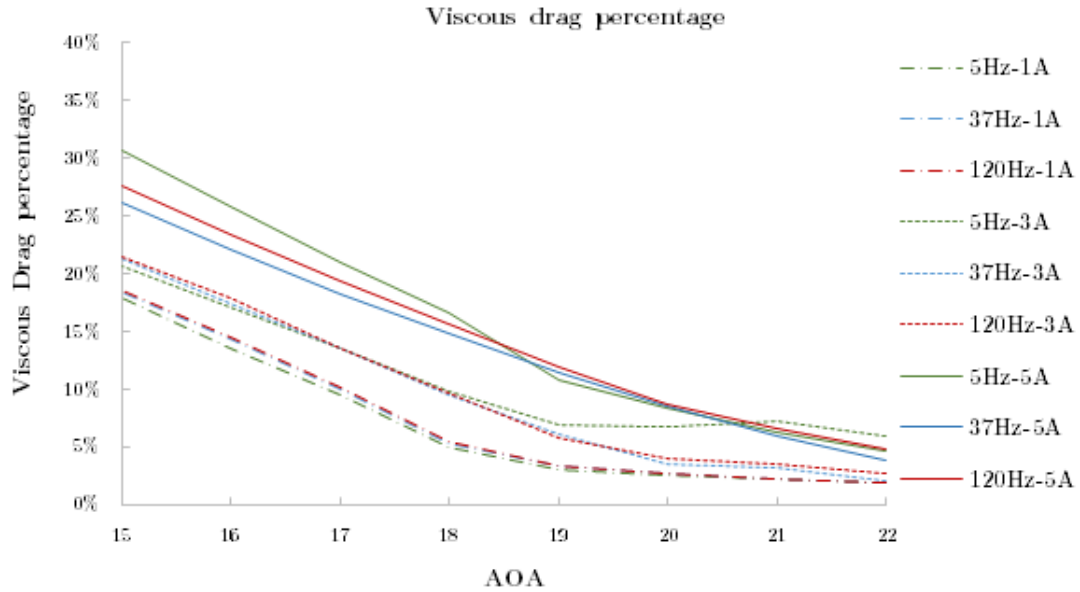


Figure 5-10 Contribution of the viscous drag to the total drag.

To have a better understanding of the influence of the jet amplitude, Figure 5-11, shows the contours of mean pressure coefficient around the aerofoil, at 16° , 18° , 20° and 22° AOA, with the three different jet amplitudes. Here the jet frequency is fixed at 37Hz. As could be expected^{tt}, the suction near the leading edge increases with the amplitude of the jet, and decreases with the AOA in this high AOA range.

^{tt} A numerical analysis was made by Azim¹⁵⁷, who investigated the effect of suction to delay boundary layer separation for a NACA4412 profile, in which they concluded that at low angles of attack, the lift to drag ratio is increased by 2.24 times by the effect of a 65kPa suction.

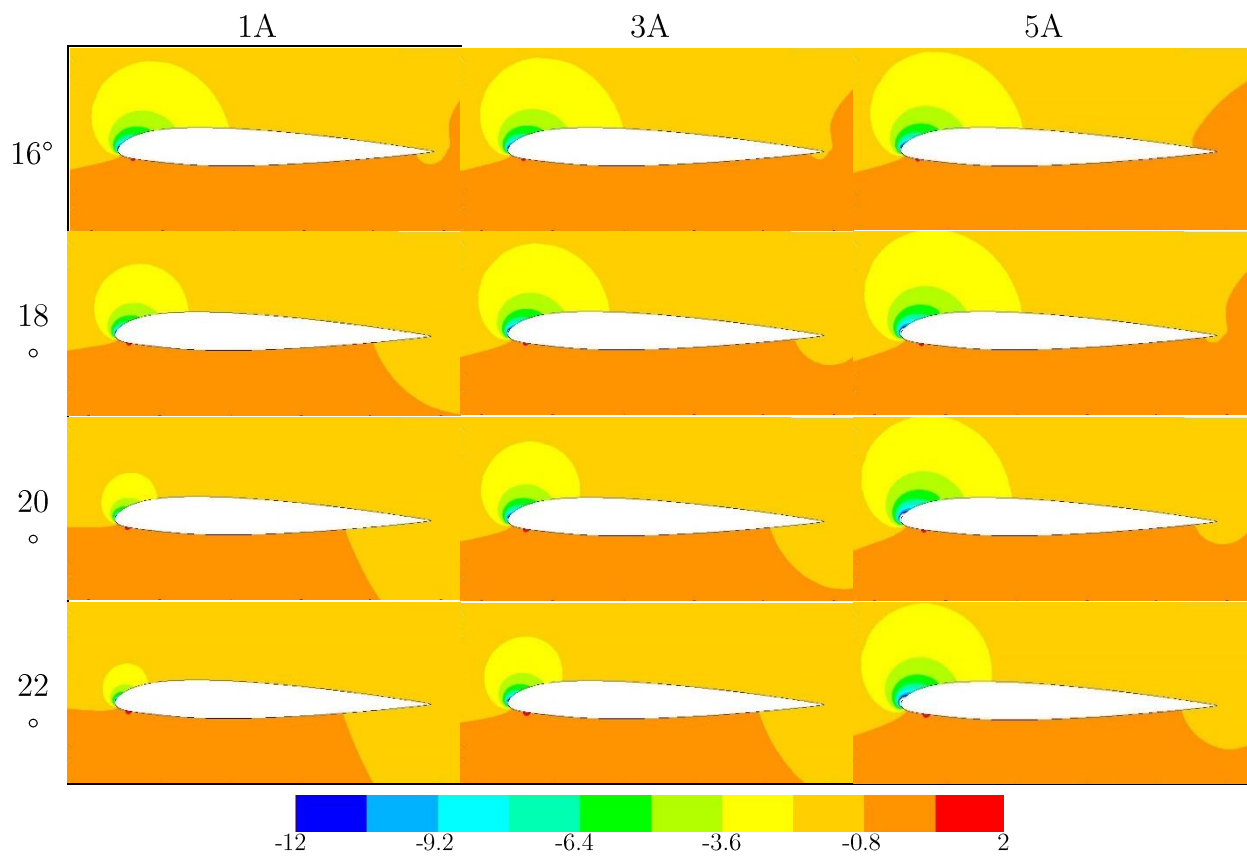


Figure 5-11 Contours of Mean Cp for 37Hz

Table 13 Percentile variations of drag coefficient against the baseline case for all frequencies and amplitudes.

AOA	$(C_{D@case} - C_{D@Baseline})/C_{D@Baseline}$								
	1A			3A			5A		
	5Hz	37Hz	120Hz	5Hz	37Hz	120Hz	5Hz	37Hz	120Hz
15	6%	0%	1%	3%	-7%	-6%	-10%	-16%	-16%
16	5%	-1%	0%	0%	-11%	-10%	-15%	-23%	-23%
17	2%	-1%	0%	-8%	-18%	-17%	-24%	-34%	-32%
18	-4%	-3%	-4%	-25%	-33%	-34%	-43%	-52%	-50%
19	-6%	-4%	-5%	-29%	-39%	-41%	-53%	-63%	-62%
20	0%	-6%	-5%	-17%	-31%	-35%	-48%	-62%	-63%
21	6%	-7%	-6%	-2%	-29%	-32%	-36%	-56%	-59%
22	2%	-8%	-7%	11%	-22%	-31%	-24%	-44%	-55%
Average	1%	-4%	-3%	-8%	-24%	-26%	-32%	-44%	-45%

Figure 5-12 shows a comparison of both C_L and C_D for all 10 cases. As can be seen from the figure, the higher the jet amplitude becomes, the higher the maximum ratio C_L/C_D , which happens at a higher AOA.

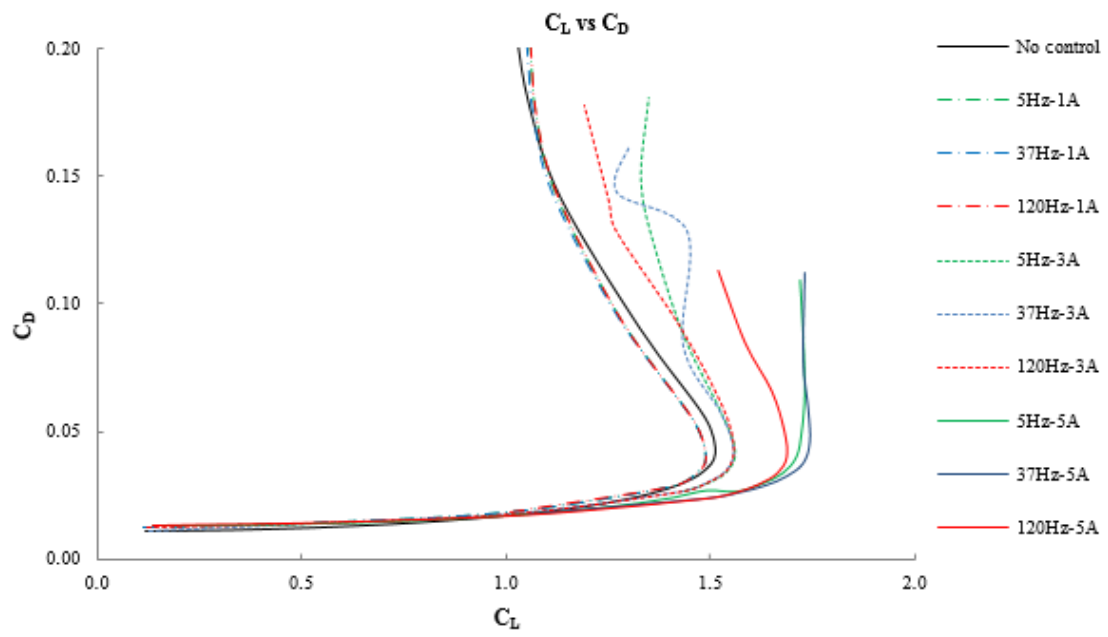


Figure 5-12 C_L vs C_D comparison

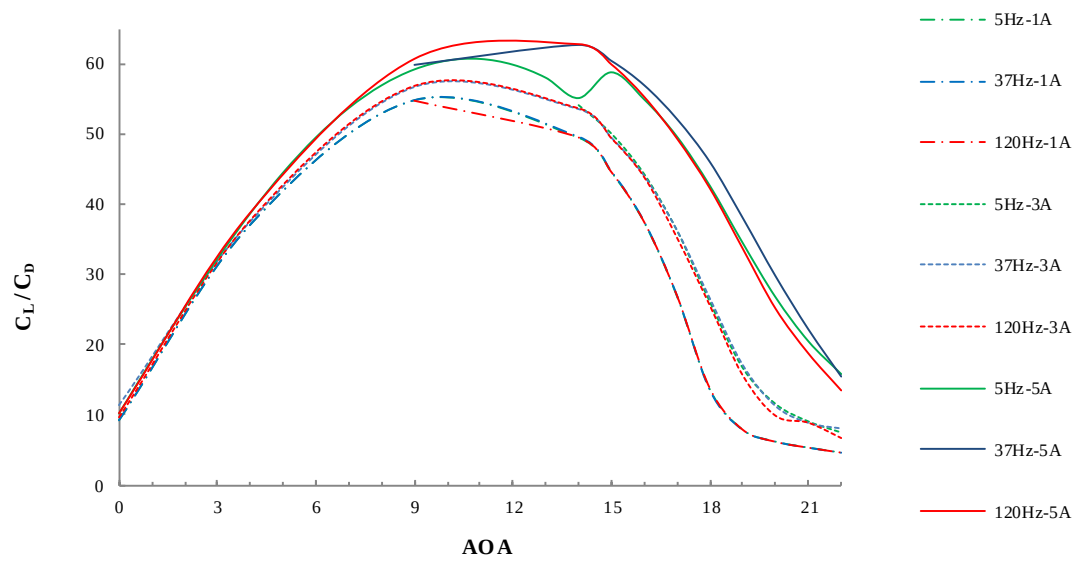


Figure 5-13 C_L / C_D vs AOA

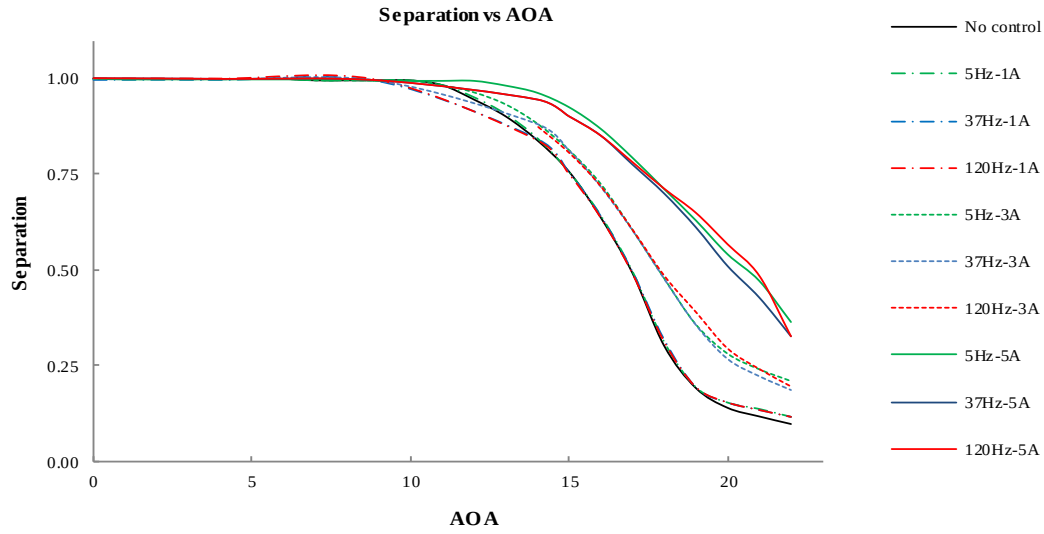


Figure 5-14 Flow separation point

Figure 5-14 shows the streamwise position at which the boundary-layer separates from the upper surface of the airfoil for each set of jet amplitudes and frequencies. The separation points were estimated by calculating the minimal mean skin-friction coefficient over the profile. The results shown in this figure are consistent with the results presented earlier, as the delay in the separation of the boundary-layer leads to the lift increase. The impact of different jet amplitudes on the separation point is significant, while the effect of the jet frequency is limited.

5.3.3 Effect of the Phase of Jet Oscillations

The difference in the phase at which the inflow and outflow of the BDSJ take place was parametrized to understand its influence on the aerodynamic coefficients of the airfoil^{um}. Here the jet amplitude and frequency are fixed at 1A and 37Hz, respectively. Figure 5-15 shows the inflow and outflow jet velocity

^{um} Orkwis and Filz⁹¹ followed a similar comparison in this study.

conditions for five different phase differences tested in this study; “Phase 0” means that the outflow takes place immediately after the inflow and vice versa, while “Phase 180” means that both inflow and outflow take place simultaneously.

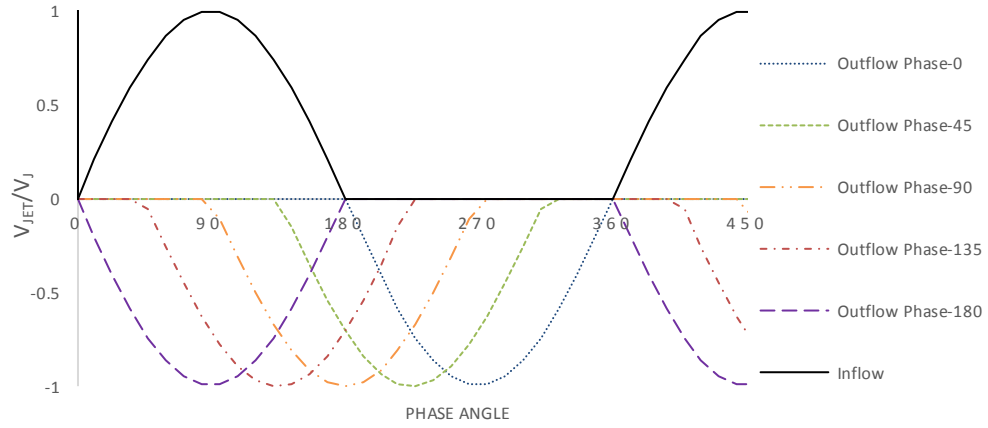


Figure 5-15 Phase differences between the inflow and outflow of BDSJ

In Figure 5-16, the lift and drag coefficients are plotted for the five different phase differences. As can be seen from the figure, the phase difference has no apparent effect on the aerofoil performance at angles of attack below 16° , while at higher angles of attack a slight increase in lift is achieved by changing the phase difference.

Table 14 shows the comparative differences in C_L , C_D and C_L/C_D ratio from the baseline case (Phase 0°). The results indicate a slight increase in C_L for the cases of Phase 135° and Phase 180° . Although these results suggest that the effect of phase differences tend to be relatively minor, this may potentially be an important design parameter for real BDSJ devices to be developed in the future since the BDSJ, by its nature, has two orifices and an internal part connecting them (i.e. the phase difference would depend on the size and design of the internal part).

In the experiment of Holfman⁸¹, the relative phase of the two synthetic jets used for his study over a six-inches chord NACA0025 profile with $Re_c=10^5$, did not show significant influence in the separation control either.

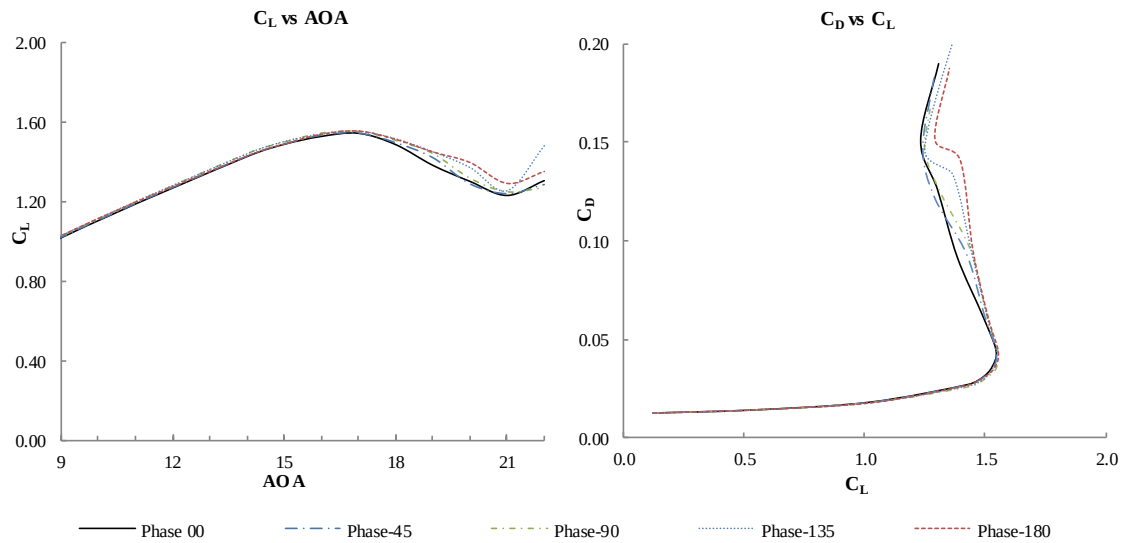


Figure 5-16 Phase sensitivity analysis on C_L and C_D

Table 14 Differences in lift and drag due to the BDSJ phase differences
(against the baseline case “Phase 0”).

AOA	C_L				C_D				C_L / C_D			
	Phase-45	Phase-90	Phase-135	Phase-180	Phase-45	Phase-90	Phase-135	Phase-180	Phase-45	Phase-90	Phase-135	Phase-180
15	0.1%	0.6%	0.6%	0.0%	-0.9%	-2.7%	-2.8%	-1.4%	1.0%	3.4%	3.6%	1.4%
16	0.4%	0.9%	0.4%	0.6%	-1.6%	-3.6%	-1.4%	-2.2%	2.0%	4.7%	1.9%	2.8%
17	0.1%	0.5%	0.4%	0.6%	-0.1%	-1.3%	-0.8%	-1.6%	0.1%	1.8%	1.2%	2.3%
18	0.7%	1.4%	1.7%	1.7%	-1.0%	-2.0%	-2.4%	-2.7%	1.7%	3.4%	4.2%	4.5%
19	2.8%	4.2%	4.6%	4.8%	1.5%	2.8%	2.1%	3.0%	1.3%	1.4%	2.4%	1.7%
20	-0.9%	1.1%	5.1%	7.2%	-3.4%	-1.9%	6.1%	11.8%	2.5%	3.1%	-1.0%	-4.1%
21	0.8%	1.4%	1.6%	4.9%	-2.8%	-3.8%	-2.3%	0.2%	3.6%	5.4%	4.0%	4.7%
22	-1.4%	-2.8%	13.1%	3.5%	-4.2%	-12.4%	28.4%	-1.5%	2.9%	11.1%	-11.9%	5.1%
Average	0.3%	0.9%	3.4%	2.9%	-1.6%	-3.1%	3.4%	0.7%	1.9%	4.3%	0.5%	2.3%

5.3.4 Effect of the gap between two jet orifices

To understand better the effect of the position of the BDSJ slot over the 2-dimensional profile, the exit was first discretized in such a way that the boundary condition of the jet could be moved forward and backward by up to ten slot diameters. The simulations are performed with these variations and the effect of this movement is plotted in the following graphs.

The discretized domain in the jet exit consists of 24 ports of 15 elements each and two ports are used to simulate any exit as is shown in the magnification on Figure 5-17. In the Gap 0D case, port K-L and M-N are activated, with the velocity profiles shown earlier; K-L as an inlet and M-N as an outlet. To get a one diameter gap between inflow and outflow, the port J and O were activated to

replace L and M which were set to a wall boundary. Similar procedures were followed for the subsequent configurations.

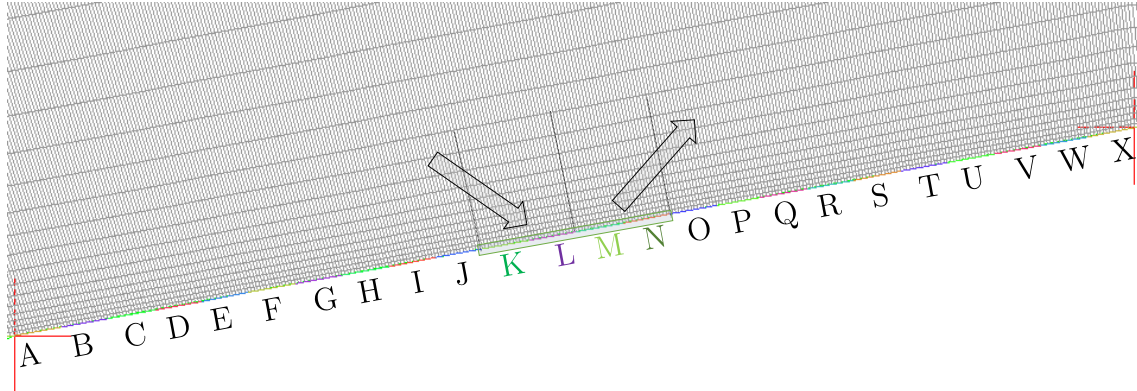


Figure 5-17 Close-up of the mesh elements around the jet exit

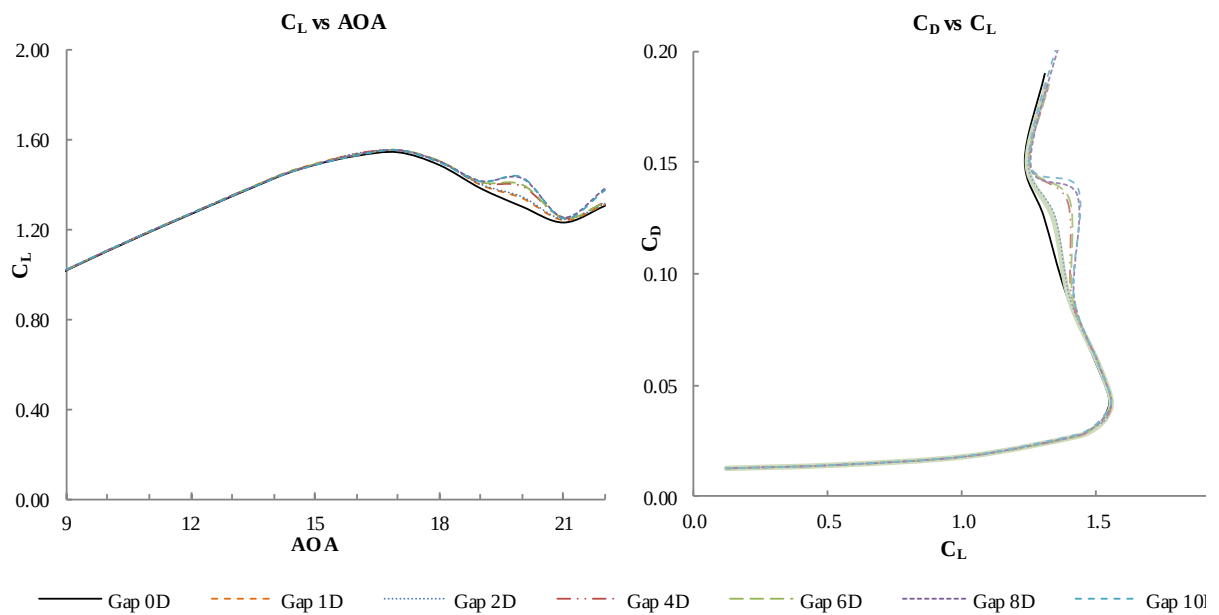


Figure 5-18 C_L vs AOA and C_D vs C_L for different gap cases

In Figure 5-18 it can be appreciated that the effect the gap has on the lift coefficient is very limited in almost all cases, with the exception of 20°. This is because at this angle of attack, the separation takes place exactly at the location

of the slot, which is when the effect of the control becomes more relevant. Figure 5-19 shows the comparison of the coefficients for the 20° case. As can be seen from this figure and Table 15, the lift and drag seem to increase together. The results show, however, that a small gap (for example, one or two diameter gap) seems to improve the C_L/C_D ratio.

Table 15 Differences in lift and drag due to the gap between two jet orifices (against the baseline case “Gap 0D”)

AOA	C_L					C_D					C_L/C_D				
	1D	2D	4D	8D	10D	1D	2D	4D	8D	10D	1D	2D	4D	8D	10D
15	0.3%	-0.1%	-0.1%	-0.1%	-0.2%	-1.0%	0.3%	0.7%	1.1%	1.9%	1.3%	-0.3%	-0.8%	-1.2%	-2.1%
16	0.5%	0.6%	0.5%	0.2%	0.0%	-1.6%	-2.0%	-1.6%	-0.3%	1.1%	2.2%	2.6%	2.1%	0.5%	-1.1%
17	0.6%	0.4%	0.5%	0.5%	0.5%	-2.2%	-1.3%	-1.5%	-1.4%	-0.9%	2.9%	1.7%	2.0%	2.0%	1.5%
18	1.1%	0.7%	1.0%	1.0%	0.7%	-3.5%	-2.3%	-3.1%	-2.8%	-1.4%	4.8%	3.1%	4.2%	3.9%	2.1%
19	1.1%	1.2%	1.6%	2.3%	2.1%	-3.3%	-3.5%	-2.9%	-2.3%	-0.5%	4.5%	4.8%	4.7%	4.7%	2.6%
20	2.9%	3.2%	6.8%	9.6%	9.9%	-1.0%	-0.1%	5.0%	7.7%	9.7%	3.9%	3.3%	1.8%	1.7%	0.2%
21	0.9%	1.0%	1.5%	1.8%	1.4%	-1.3%	-1.6%	-1.5%	-1.2%	-0.2%	2.2%	2.7%	3.1%	3.1%	1.6%
22	0.4%	0.6%	1.1%	5.8%	5.2%	-2.7%	-2.7%	-2.8%	12.2%	12.2%	3.3%	3.4%	4.0%	-5.7%	-6.2%
Average	1.0%	0.9%	1.6%	2.6%	2.4%	-2.1%	-1.7%	-1.0%	1.6%	2.7%	3.1%	2.7%	2.6%	1.1%	-0.2%

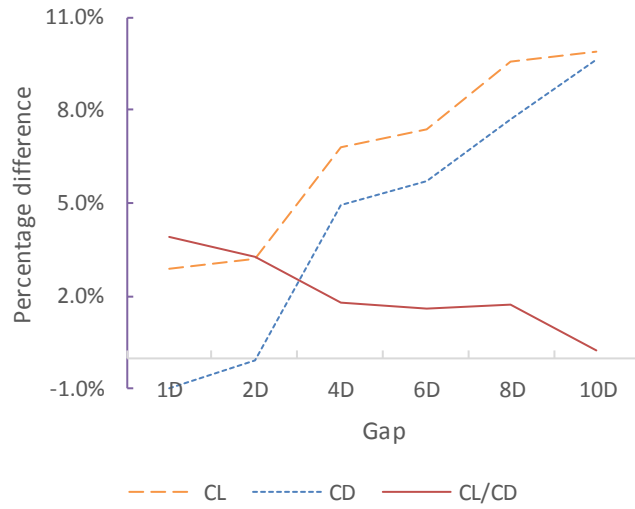


Figure 5-19 Effect of the jet gap on drag and lift coefficients 20° AOA

Two conclusions can be raised from Figure 5-19. Firstly, the effectiveness of the gap in the BDSJ would only be relevant when the position of the slot coincides with the separation point. Secondly, if this condition is met, a small gap between the inflow and outflow would delay separation of the boundary-layer, increasing the C_L/C_D ratio.

Figure 5-20 shows the mean streamwise velocity contours for some of the configurations analysed in this study. There is an obvious delay in the separation region from the baseline case to the BDSJ 90-90 and the BDSJ 165-015.

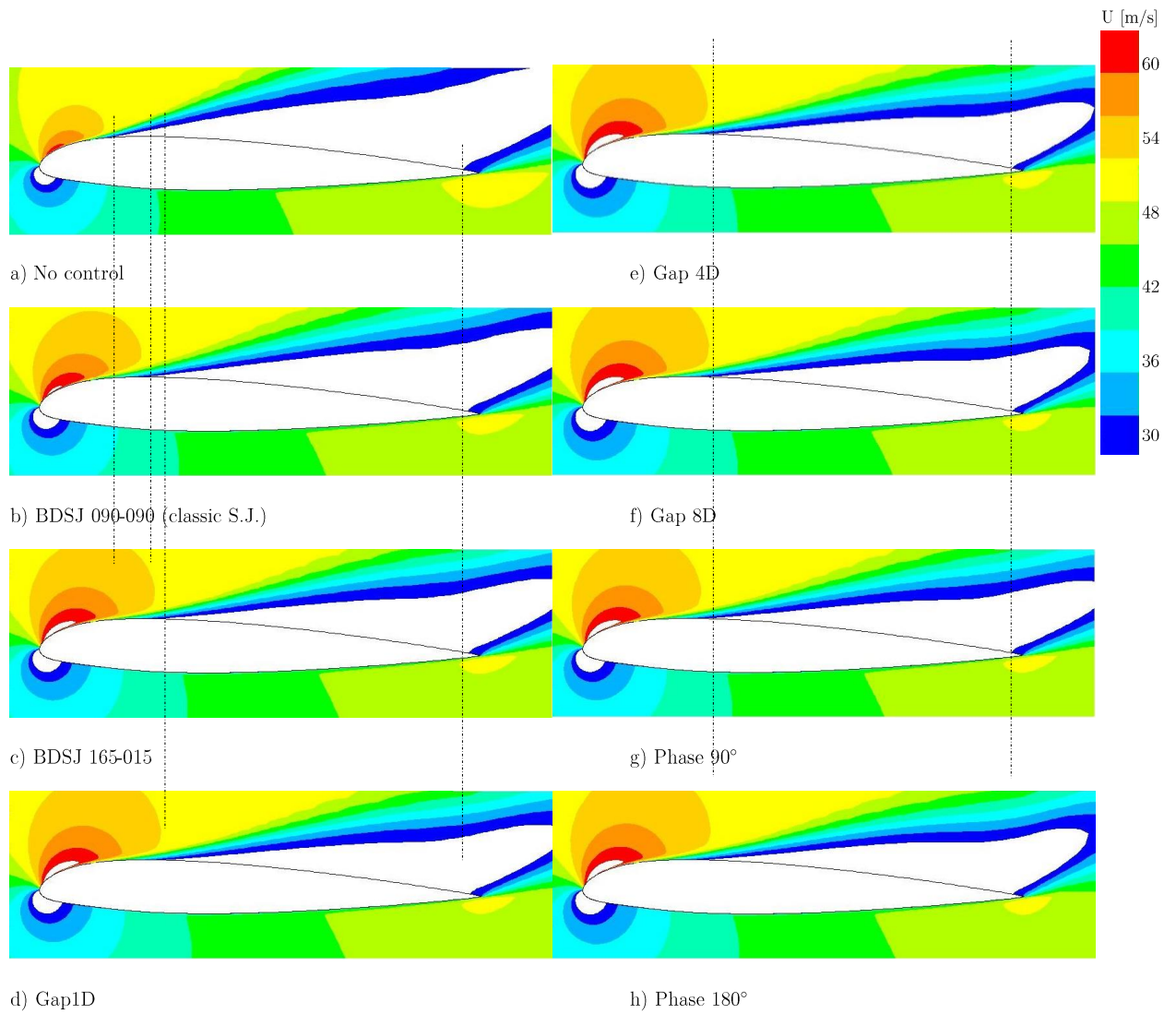


Figure 5-20 Comparison of mean steam-wise velocity contours at 20°

5.3.5 Jet exit configuration

A common question to answer in the flow control community is related with the study of multiple SJA, and previous¹⁵¹ attempts have been made for the classical SJ. To understand better, the influence of the number of slots and their position, three analyses were performed with multiple jet exits with a gap of two jet exit diameters between them. This proposed configuration is based on the results of Watson⁹⁴, whose results suggested that the use of two jet orifices may produce two vortex rings with a total circulation greater than a ring produced by a single orifice of equivalent area. Figure 5-21 and Table 16 show this comparison, where the jet labelled as “2Jets 3D Forward” consist of two jet slots, whose inflow corresponds to ports AB and KL (c.f. Figure 5-17) and out-take CD and MN, while for “2Jets 3D backward” the inlet is KL and UV and out-take MN and WX. Finally, on the 3 Jet configurations, all simulated boundaries are separated by 2 diameters.

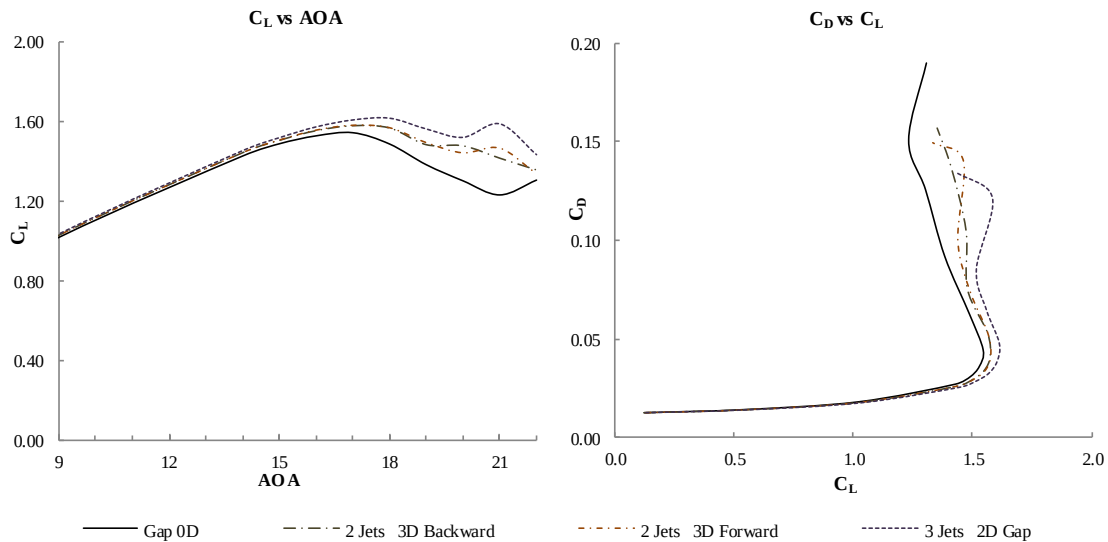


Figure 5-21 Sensitiveness of position on the slot on C_L and C_D

As can be seen from the Figure 5-21 and Table 16, little difference in the lift curves is appreciated between the 2 jets configurations at both positions.. On the other hand, an increase in the lift coefficient at angles of attack above 15° for the two and three jet configurations can be appreciated. This was expected as the amount of flow entering the domain is higher for the two and three jet configurations. It should be noted that the difference between the two and three configurations is not as large as between the single and double configuration. The numerical result suggests that two jet configurations separated by a distance may be the most efficient configuration as a lift enhancer, as it appears to be an apparent threshold, after which increasing the amount of flow, the lift does not increase as much.

Table 16 Drag and Lift differences in percentage for the three jet configurations

AOA	C_L			C_D		
	2 Jets 3D Backward	2 Jets 3D Forward	3 Jets 2D Gap	2 Jets 3D Backward	2 Jets 3D Forward	3 Jets 2D Gap
15	0.9%	1.1%	1.8%	-3.9%	-4.1%	-6.4%
16	1.5%	1.7%	2.6%	-5.7%	-6.2%	-9.3%
17	2.0%	2.3%	3.8%	-7.3%	-7.6%	-12.8%
18	5.2%	5.3%	8.4%	-15.0%	-15.1%	-23.0%
19	7.0%	7.9%	12.7%	-18.1%	-19.1%	-29.4%
20	13.3%	10.6%	16.5%	-16.8%	-20.5%	-32.1%
21	14.7%	18.9%	28.6%	-9.6%	-7.1%	-19.6%
22	3.6%	2.2%	9.5%	-17.7%	-21.3%	-29.2%
Average	6.0%	6.3%	10.5%	-11.7%	-12.6%	-20.2%

5.4 KEY FINDINGS SUMMARY

The key findings of this chapter could be summarise in the following points:

- The computational results show that the higher the skewed angle between the inlet and outlet ejection of flow, the higher the lift coefficient for a particular angle of attack.
- The amplitude at the jet exit have a much higher impact in the lift coefficient than the effect of different frequencies (which for this particular set, results in being non conclusive or without any impact)
- The study on the phase gap between inlet-outflow shows that a sinusoidal or classical pattern of injection-ejection of fluid flow yield to a higher lift coefficient for the majority of AOA
- A gap of one jet exit diameter length between inlet and outlet is particularly important when the jet exit coincides with the separation point. However, there is no particular effect for AOA which don't yield to this condition.

6 DESIGN OF THE BIDIRECTIONAL SYNTHETIC JET

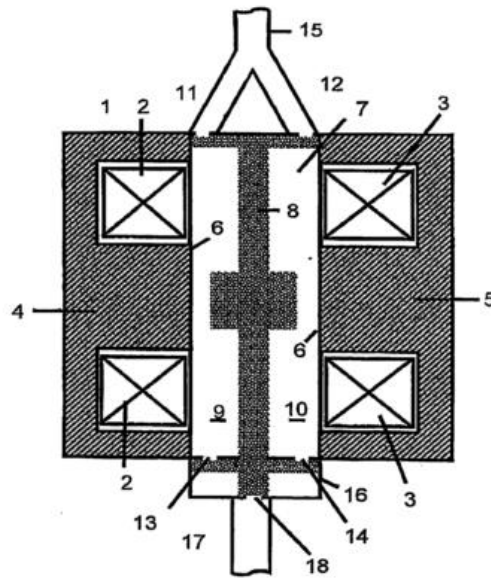
Developing the design from concept to manufacture of a bidirectional synthetic jet with the required performance characteristics is a challenging task; however, for flow control purposes, with the current materials and technology in the market its creation seems feasible in the foreseeable future.



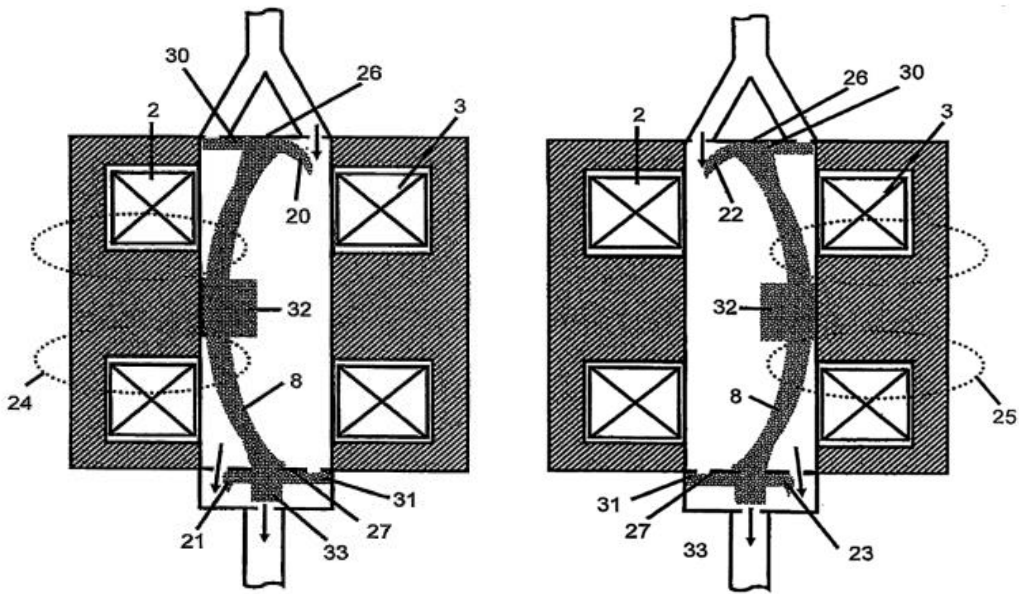
6.1 CONCEPTUALIZATION

Due to the numerical nature of the analyses presented in this work, a conclusion about the effective improvements of a Bi-directional jet over a classical jet is pending and needs to be confirmed by experimental results. It is true that the easiest way to change the direction of the flow over an aerodynamic surface is by the action of a pump or a rotodynamic device under the surface, which will subtract air by the movement of rotating blades and eject it in a different direction by the same principle. However, this device would include rotating elements in the system, which will add complexity in the operation and maintenance. Moreover, its fair comparison with the classical synthetic jet, and its advantages for simplicity of operation would be nullified.

To create an effective system which can keep all the advantages of a zero-net-mass-flow jet, including a change in direction without adding multifaceted mechanism, a polymeric film with strategically placed magnetic strips may be used, which with the help of a solenoid would create oscillatory movements to subtract and eject the air in different directions, substituting in this way the parallel membrane of the classical synthetic jet.



a)



b)

c)

Figure 6-1 Sketch of a diaphragm pump.

The patent from Heinrich¹⁵² 2008, shows a diaphragm pump mechanism actuated by the movement of a membrane, which is made from foamed plastic with soft magnetic particles inside, and form up to 70% of the total weight. This membrane, numbered 32 in Figure 6-1, oscillates from left to right due to a

magnetic field created by the coils 2 & 3, that are attached to the tubular cavity. At the top and bottom of the membrane, two flat plate shaped ends allow and restrict the flow passing through the valves from both chambers besides the membrane. The rims of the plates are pressed inwardly and outwardly due to the positive and negative pressure produced within the chambers, creating in this way a continuous subtraction of fluid downwards.

Some of the features depicted within this patent may be used by the Bidirectional Synthetic Jet, borrowing the principle of oscillation due to an external magnetic field, squeezing the fluid from inlet to outlet, and therefore, changing efficiently the direction of the flow without mass injection or rotating elements include..

6.2 DESIGN

The design of zero net mass flow devices remains still a challenge for the active flow control community even though the usage of Synthetic jets is widespread. It is an exhausting and time-consuming task to design properly a zero-net mass flow actuator for a precise application. Seifert affirms there are no clear choices or figures that are universal for all applications; however, some guidelines are provided, which allowed a new configuration to be presented for discussion.

Some of the requirements for the effective performance of a Synthetic Jet design include electric efficiency, power consumption, bandwidth⁴⁸, weight, size,

⁴⁸ Here bandwidth is defined as the difference between the lowest and highest frequencies the actuator can support.

robustness, geometry of the driver, orifice/slot, material, cavity shape, and of course manufacture, operation and maintenance cost.

As there are many parameters influencing the BDSJ performance, and trial-and-error approaches are expensive and time consuming, the influence of each parameter on the global properties for generic applications is still uncertain.

Generally speaking, the basic elements of a classical synthetic jet actuator could be pointed out to be the cavity, the slot exit, a vibrating membrane and, for the design of a Bidirectional synthetic jet, a controlled magnetic frame which would be required to be included. All these parameters are coupled in the formation, development and effectiveness of the vortices generated in the flow, and the influence on the flow for individual components is still unclear, even for the classical synthetic jet. However, the guidelines learnt from the present work and classical configuration can be used as a preliminary approximation to excel control effectiveness in the bidirectional synthetic jet.

6.2.1 Cavity

The cavity in the BDSJ is actually a storage chamber, which keeps air inside it without any disturbance from the outer flow for some time, energizing it with the oscillation of the membrane, and ejecting it through the slot exit. Several attempts to understand the influence of the cavity on the development and effectivity of the vortices generated by the classical synthetic jet in the flow field have been carried out and the knowledge brought by these investigations could be used to set up starting dimensions for the design of the bidirectional synthetic jet cavity.

For instance, in the experiments of Chaudhari^{93, 153}, changes in the size of the orifice diameters turned out to be more influential than changes in the depth of the cavity. Therefore, it is suspected that for the bidirectional synthetic jet cavity, the length of the inner tubular cavity that stores partially the fluid may not influence the flow as significantly as the shape of the inlet and outlet jet exits.

The results of Ukutaar¹³⁵ conclude that there is no significant influence of cavity size on the performance of the jet. However, when comparing different cavity shapes, like in the most recent study by Ferro⁹⁹, it is suggested a slight influence on the overall flow with increased efficiency in complex cavities, but no detailed description of the extent of the influence of each parameter is mentioned. therefore some freedom can be expected for not only the length but also the design and shape of the inner tubular cavity.

According to previous investigations about the cavity effect on the main flow, it can be concluded that the main influence of the cavity on the vortex quality depends more on the configuration and shape close to the jet exit than on the general cavity silhouette. For this reason, a tubular cavity with an elliptical cross section and a funnel and conical shaped jet exit is proposed, from which the cavity of the bidirectional synthetic jet could subtract air in one direction and eject it into another.

The following picture shows a proposed dual helicoidal cavity for the BDSJ. The slightly pitched phase shown in Figure 6-3, helps the inlet-outflow exit funnels join smoothly at the end of the chamber.

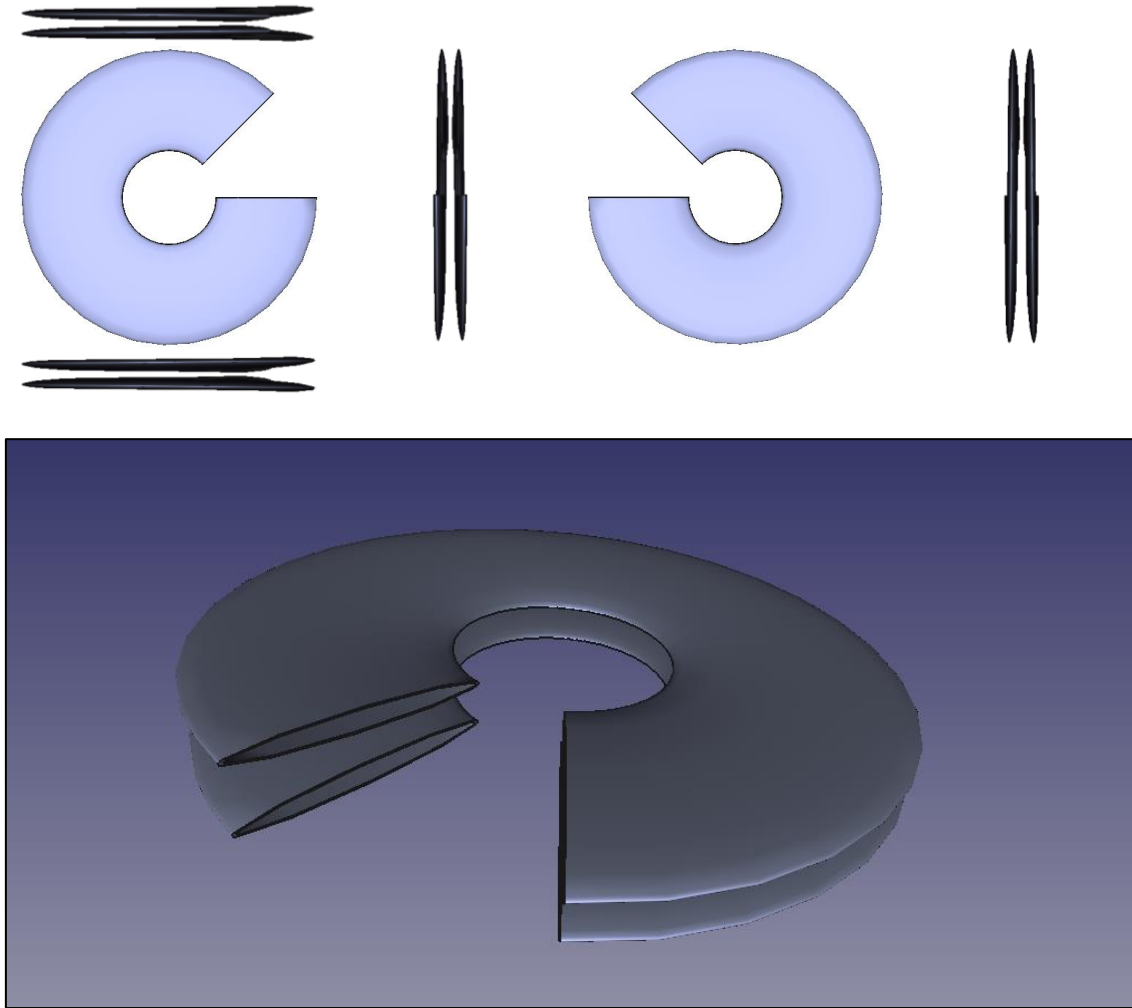


Figure 6-2 Proposed BDSJ elliptical-circular cavity

Regarding the dimensions of the chamber, the elliptical cross section has a long axis of 60 mm and a short axis of 5mm. The chamber has a circular shape with a radius of 60 mm and a 45° cut, at which both ends the inlet and outlet funnels would be attached. This chamber could store a total volume of about 77,723 mm³, being this a similar value to that of a classical synthetic jet with 100 mm diameter and 10 mm width, which could store 78,539 mm³. With the proposed configuration, two cavities could be attached to take advantage of a single solenoid arrangement, and improve energy efficiency.

The elliptical tubular shape is selected because in these way the elliptical cavity would store and squeeze the highest amount of air towards the exit while the flat membrane oscillate.

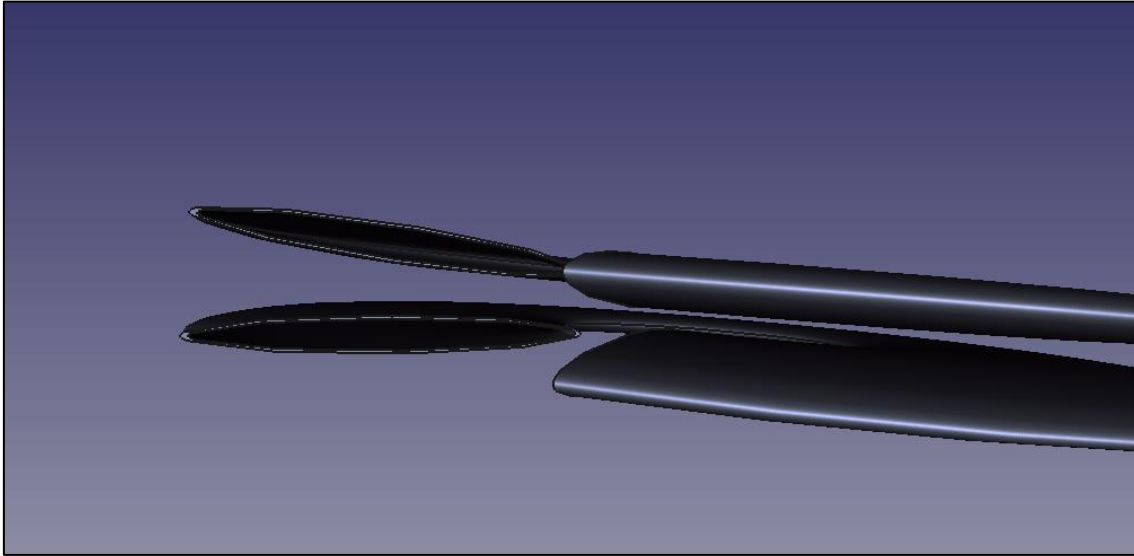


Figure 6-3 Dual Cavity configuration

6.2.2 Slot exit

The work of Kim¹⁵⁴, Lee¹⁴³, Watson⁹⁴ and Milanomic²⁷ to name a few, represent the efforts that have been made to understand the influence that the SJ exit arrangement, configuration or shape have in the flow field. For the Bidirectional Synthetic jet, beyond the rectangular or circular profile of the orifice, it is required for it to modify its form dynamically in phase with the frequency due to the required change in directions. Another option would be a defined separation for the intake and outlet orifices where the chamber could connect both exits to change the flow direction and a tubular cavity seems logical to face such a challenge.

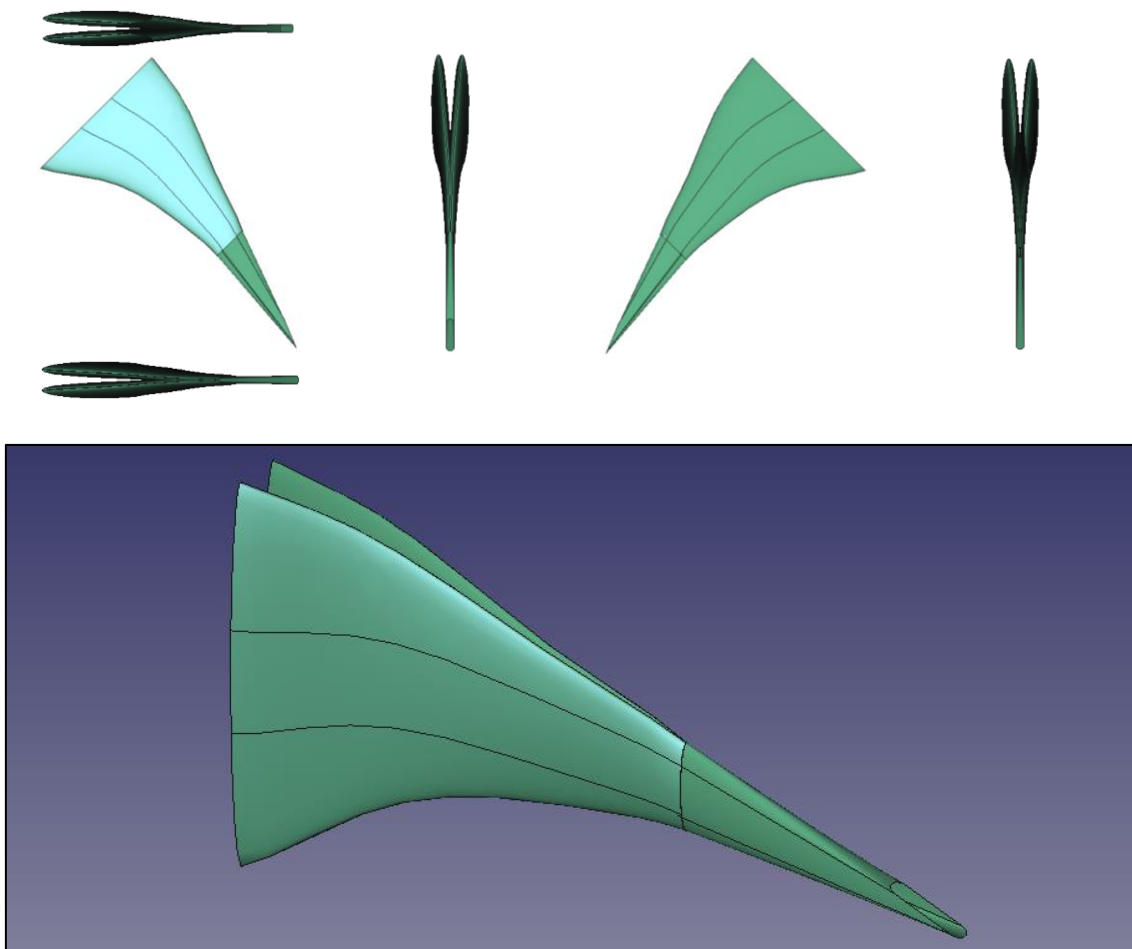


Figure 6-4 Proposed BDSJ funnel-jet exit

A perfect and flawless cavity jet exit design would only be achieved after a few physical try and error experimentation. Within the mindset of conceptualization, Figure 6-4 shows the last of several attempts to design the ideal and most likely cavity jet exit that could eject the highest amount of air and momentum at the outlet, with the smother transition, which, as it was studied on chapter 3, happens at 15° with respect to the horizontal plane.

The area of the jet exit in Figure 6-4 is 37 mm^2 ; however, as the tubular exhaust is inclined by 15° , the transversal area becomes smaller (10 mm^2)

Note that the jet outlet possess plays a crucial role in the BDSJ performance, as is responsible to accomplish a two important objectives, like help discharge the flow in the required direction. and add velocity to the flow. Reason by

6.2.3 Slot inlet

Due to the nature of the BDSJ, a slot jet inlet is required separately from the outlet, being the mass flow rate at the inlet more important than the size of the inlet port. Figure 6-5 shows a proposed dual inlet harbor, whose transversal area is bigger than its counterpart for the outlet. This decreasing change in transversal area allows a higher amount of fluid to pass through the inlet bay that the one through the outlet, increasing the velocity at the exit, aiming to inject more momentum into the system.

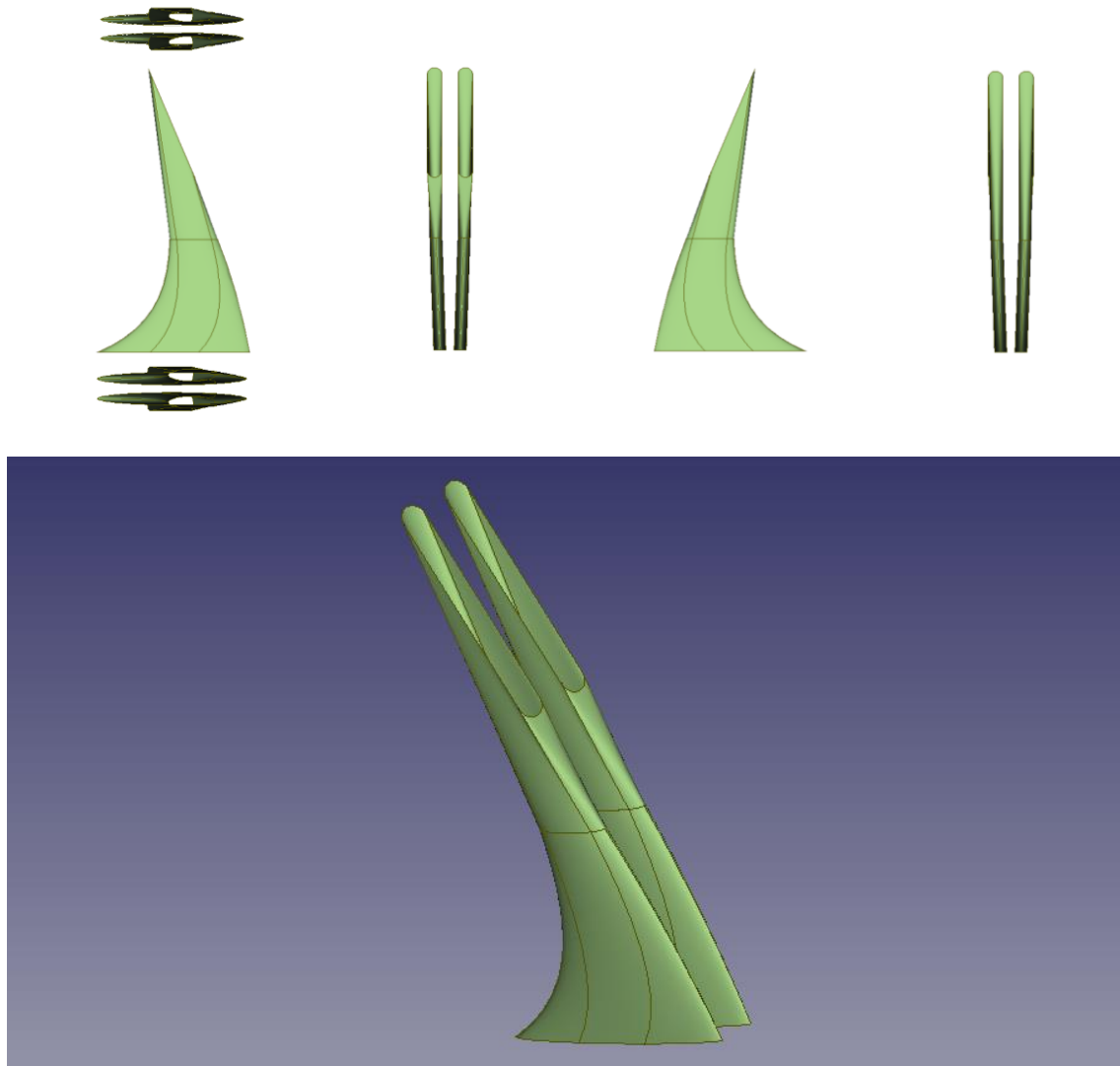


Figure 6-5 Proposed BDSJ elliptical-circular cavity

As seen in section 5.3.4, the gap between inlet and outlet of the bidirectional synthetic jet is particularly important for lift improvement when the position of the BDSJ matches the separation point. In the case of an aerofoil, for instance, not significant effect is seen by the gap at any AOA except for the one at which the position of the BDSJ equals the separation region, as seen in Figure 5-18.

In the present case, the gap between inlet and outlet is notorious, however, only try an error experimentation would confirm or define the ideal geometry, as it is obvious that the proposed inlet configuration could be subjected to plenty of improvements for particular applications; the size, orientation, location and geometry of the slot is still a mystery on the global effect for active flow control.

6.2.4 Oscillating membrane

A flat circular polymeric membrane with slim ferric threads in its interior would be held between the cavity walls, the ferric fibres inside the membrane would be subjected to a magnetic field created by the solenoidal cores. This would move and squeeze the air from one side to another without the usage of fans, keeping the intrinsic advantage and simplicity of a synthetic jet.

A proposed configuration for the membrane is shown in Figure 6-6, which depicts three sets of strips that would be aligned with the solenoids shown in Section 6.2.5.

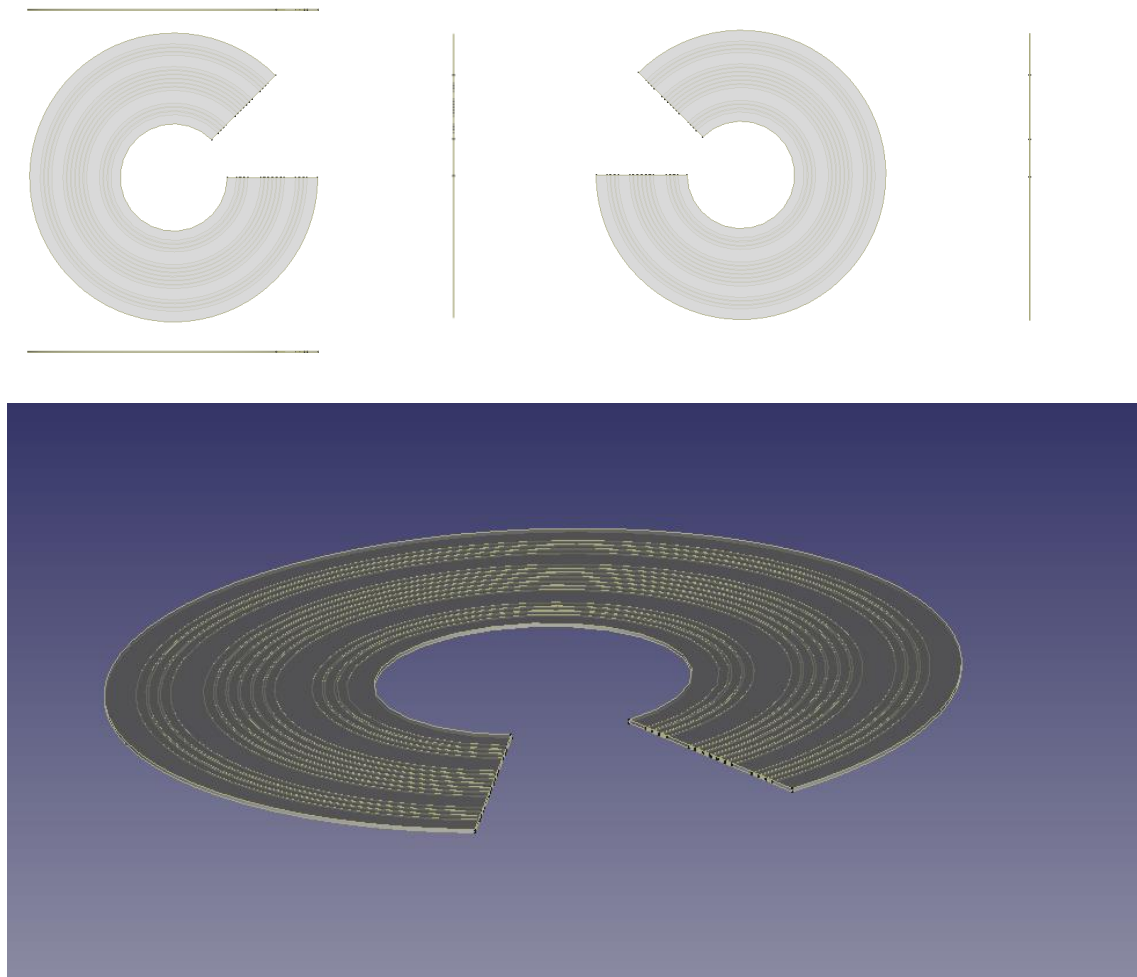


Figure 6-6 Proposed BDSJ membrane

As the membrane moves away from the walls, the pressure within the tubular cavity is decreased resulting in an entrainment of low momentum fluid. Slowly, after the first bulk of fluid is caught into the cavity, the membrane would close the inlet due to the action of the magnetic switch, and the ejection stroke would begin. The oscillatory magnetic field displace the bulked fluid towards the orifice, resulting in the expulsion of fluid through the orifice changing direction of the flow.

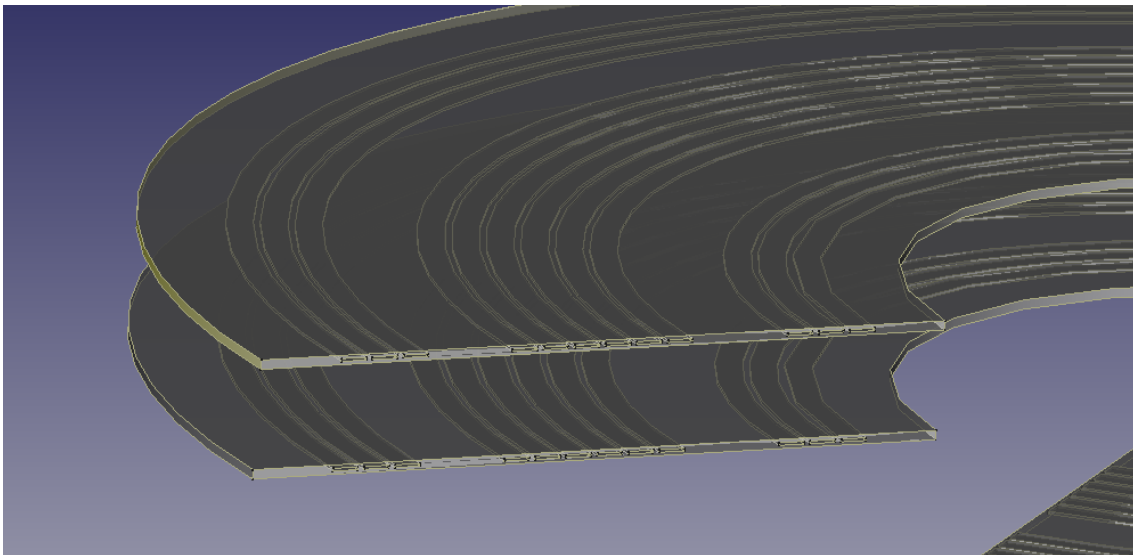


Figure 6-7 Close up dual membrane configuration

6.2.5 Magnetic frame

In order to move the flow from inlet to outlet in a synchronized manner, a static magnetic frame would be required. The tubular cavity needs to be in contact with the individual solenoids of the magnetic frame configuration, which by an integrated circuit, would alternatively moves the magnetic field from one solenoid to the other.

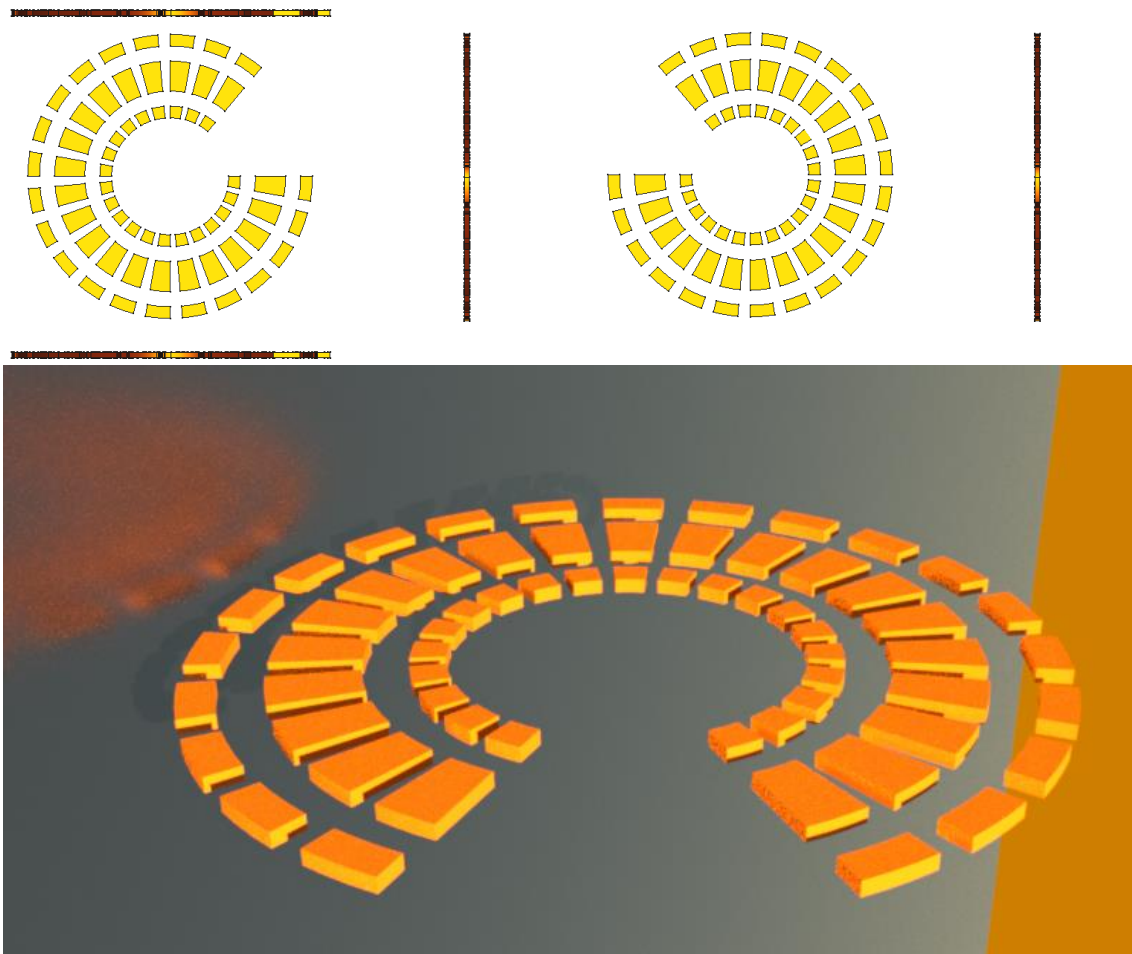


Figure 6-8 Proposed BDSJ solenoid configuration

The position and configuration of the integrated circuit that controls the magnetic field on all solenoids is trivial for the purposes of this work, and is not

showed in Figure 6-8. However, the space in the centre of the BDSJ, could be used to store any electrical component required to accomplish the task of control.

6.2.6 Functional ensemble

A sketch of the conceptual BDSJ with the previous individual parts can be seen in Figure 6-9, where the frontal and lateral views, as well as the isometric perspective of the ensemble are shown.

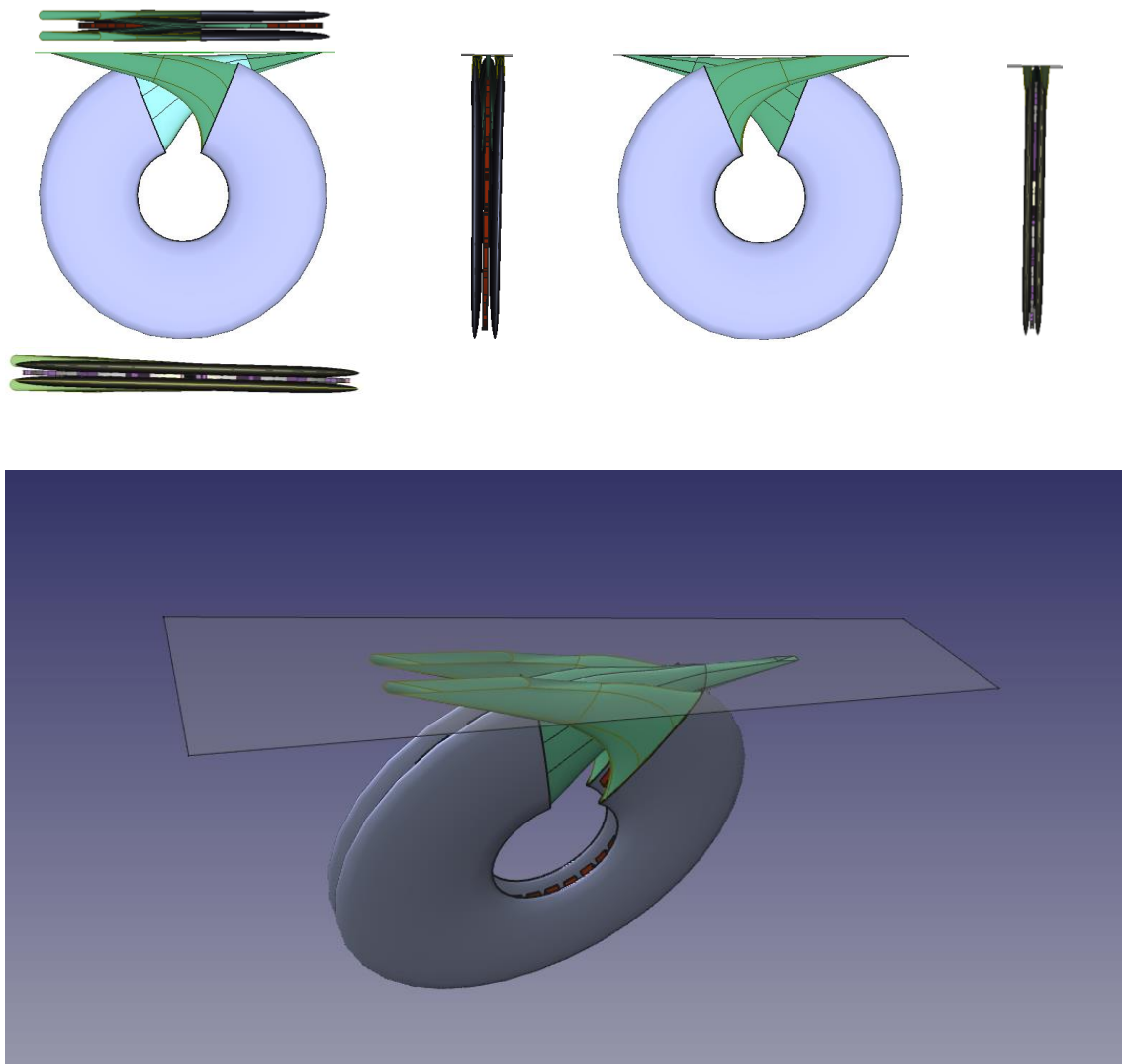


Figure 6-9 Proposed BDSJ solenoid configuration

A transversal cut of a section of the BDSJ can be observed in Figure 6-10. As it can be seen from this picture, the ferric filaments of the membrane are seated parallel with the solenoid configuration, so when the charge in the solenoids changes the filaments would move and squeeze the air, in order to create the required flow.

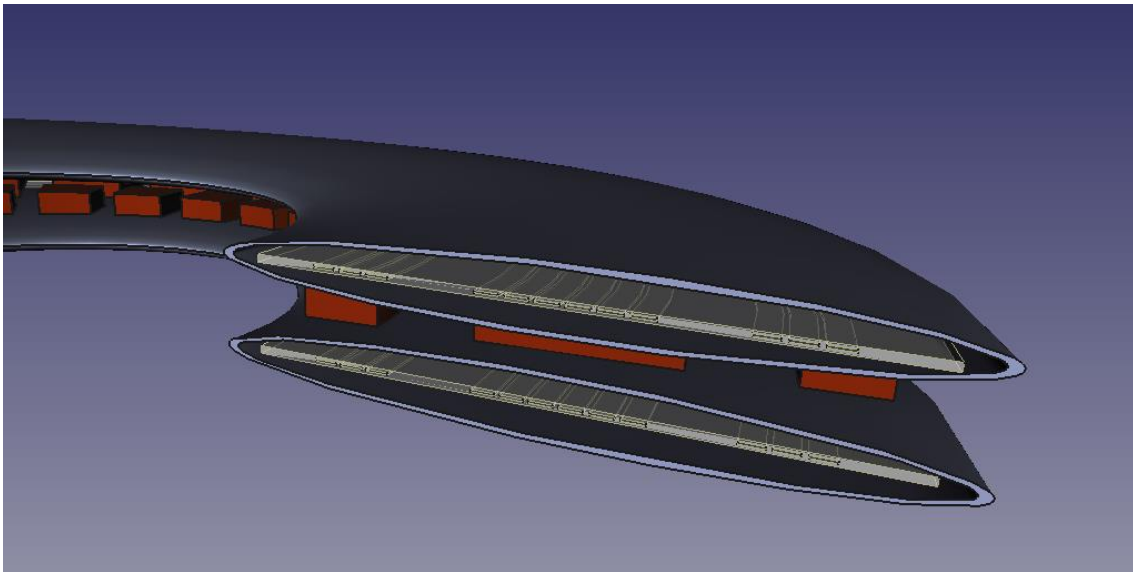


Figure 6-10 BDSJ transversal cut

Figure 6-11-a) illustrates the movement of the membrane due to the magnetic field emitted by the solenoid. The first set of solenoids close to the inlet, are activated with a positive charge, so they attract the negative charge of the ferric filaments inside the membrane, moving it at the top and bottom walls of the lower and upper cavities respectively, creating a vacuum, which would subtracts air from the surroundings.

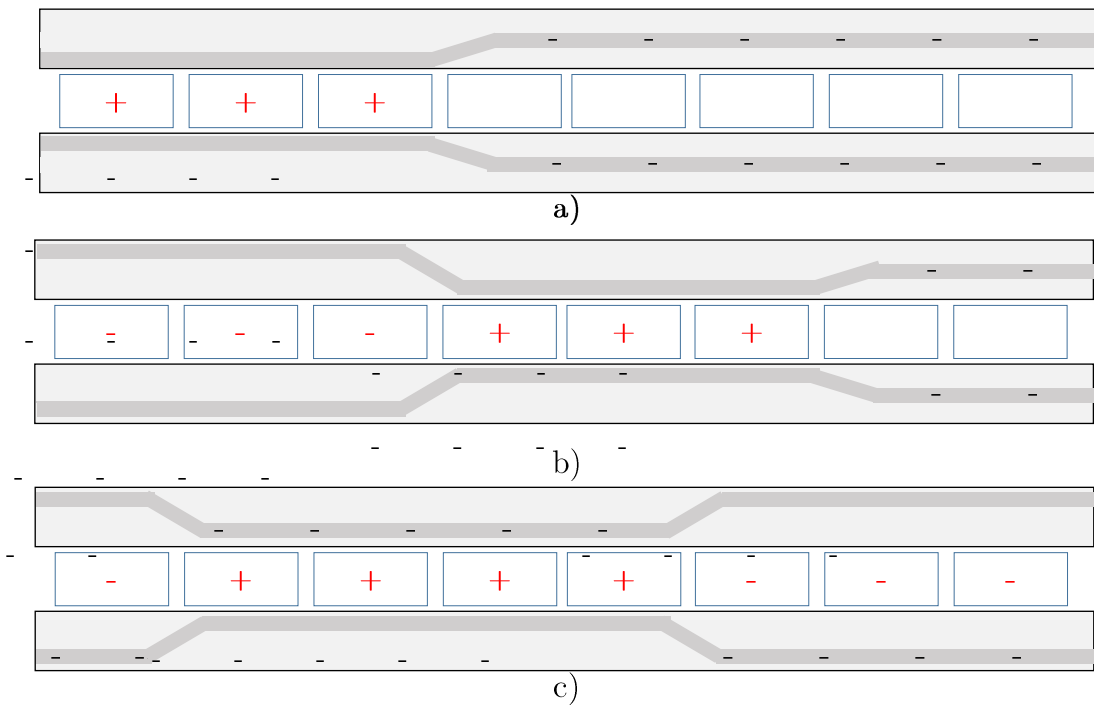


Figure 6-11 Double configuration cavity cut

In Figure 6-11-b) and c), the following individual solenoids change the direction of the current. This change in polarity repels the filaments of the membrane at a particular section of the chamber and transport the trapped air inside the cavity towards the other end.

7 CONCLUSIONS



The present work is a concept exploration study which focused its efforts on understanding numerically the implications of changing the inflow and outflow directions of a synthetic jet with the objective of improving flow control capabilities.

This new conceptual device, named Bi-directional Synthetic Jet, is first tested with a well-known and widely validated "NASA 2D Hump" configuration to assess its performance compared to the classical synthetic jet and Directional Synthetic Jet. Three different turbulence models were employed to obtain general trends of the performance of the BDSJ. Also, three different approaches regarding the jet exit conditions, namely the "Adaptive mass flow rate", "Adaptive velocity" and "Adaptive jet area" cases were considered, to make a fair comparison with the classical synthetic jet.

One of the key findings from this study is that the performance of a synthetic jet could improve significantly if the inflow (suction) and outflow (ejection) directions of the jet are optimized separately. The simulation results suggest that the performance of the BDSJ increases as the difference between the inflow and outflow angles of the jet is increased. The results also show that there is little influence of the different parabolic velocity profiles specified at the jet exit on the separation bubble size. All three turbulence models employed (S-A, SST and realizable k- ϵ) predict similar general trends. However, the results depend on the type of jet exit conditions employed; the jet performance improves most significantly if the mass flow rate and the jet slot width are fixed while changing the inflow/outflow directions of the jet, which is in other words, when the velocity at the jet exit increases, the separation bubble size decreases.

Another important point clarified in this research is that when comparing the flow control properties of the SJ, BDSJ and DSJ, the lowest mean average separation region is obtained by the BDSJ configuration with inflow-outflow direction 165° - 15° . This configuration is the one used as a reference for numerical applications and design.

Once the basic concept of the BSJD is validated, a set of 2-D unsteady RANS simulations is performed to examine the potential benefits of the Bidirectional Synthetic Jet as a flow control device to improve the lift coefficient of an aerofoil or maximize the C_L/C_D ratio. The simulations are carried out for the NACA23012 profile, first validated with experimental values and then evaluated on the frequency, amplitude, phase, and jet position. It was numerically demonstrated that the BDSJ effectively delays separation on the aerofoil.

It should be noted that this is the first study to investigate the effect of frequency and amplitude in a Bidirectional Synthetic Jet and the first one to investigate the effect of overlapped phases and gap on inflow/outflow for any type of synthetic jet.

As expected, it was found that the jet amplitude plays a significant role in the effectiveness of the BDSJ, improving the lift coefficient at high angles of attack in a more obvious way than the jet frequency. The higher the jet amplitude is at the exit, the higher the C_L/C_D ratio becomes, while the frequency has limited influence for the conditions here studied.

Regarding the phase at which the jets actuate, the results suggest that instantaneity in the momentum injection after subtraction of the flow could delay the separation of the boundary layer, increasing the lift of the aerofoil. However, as the increment of this amount is relatively small, in the order of 3%, and

because the turbulence model plays a significant role once the separation takes place, a higher resolution model, such as LES, would be required to provide a solution and confirm this hypothesis.

Another key feature raised by this study is the effect of the gap between the inflow and outflow of the jet. It was found that the effect of the gap in the BDSJ would only be relevant when the position of the gap coincides with the separation point. If this condition is met, a gap between the inflow and outflow would delay the separation of the boundary layer, increasing the C_L/C_D ratio.

It should also be noted that the optimal jet configuration may depend on the shape of the aerodynamic body and other flow parameters, such as the Reynolds number, freestream turbulence and the frequency and magnitude of the jet excitation.

Finally, the conceptualization of the BDSJ shown in the last chapter of this thesis, gives a first step in the process of design and manufacture of the BDSJ.

Key Findings in quantitative form:

- When the inflow (suction) and outflow (ejection) directions of the jet are separated by 150° (BDSJ 165-15) the separation bubble length could be reduced to up to 6%.
- The difference in the parabolic profile at the jet exit represents less than 1% impact in the separation region. Implying a degree of freedom in the shape of the jet exit.

- A gap of 1 jet exit diameter length between inlet and outlet would maximize the effectiveness of the BDSJ when the separation coincides with the position of the device.

7.1 RESEARCH LIMITATIONS AND FUTURE WORK

1. One limitation of the present study, is the relatively low speed at which it is tested. Higher freestream velocities need to be investigated to know the impact of the BDSJ at Mach numbers higher than 0.3. This could yield to application in full size airplanes, racing cars, helicopters, and turbine blades to mention a few.
2. Another limitation of the present work, is the fidelity of the turbulence models used. Higher order RANS, DES or LES models could improve the insights about the reattachment regions and open new questions in the designed model.
3. The development of a new product is always a challenge. The model shown in this work represents the first effort to land the conceptual design of a BDSJ, and undoubtedly there are many areas to improve. Better materials, or efficient configuration. Moreover, a study about the power consumption and its comparison with the classical synthetic jet should be done to confirm an economic advantage as a flow control device.
4. An experimental evaluation must be carried out to confirm the computational results. The proposed configuration presented here, may be built and subjected to a cross flow over a hump or a profile to measure separation downstream and either confirm or reject its advantages in flow control over the classical and directed synthetic jet.

REFERENCES

- ¹ Hepp, H. J., Box, E. O., and Ray, G. C., “Utilization of Kerosene Stocks for Jet Fuels by Treatment with Urea,” *Industrial and Engineering Chemistry*, 1953.
- ² Davidson, B. J., and Riley, N., “Jets induced by oscillatory motion,” *J. Fluid Mechanics*, vol. 53, 1972, pp. 287–303.
- ³ Glezer, A., “The formation of vortex rings,” *Physics of Fluids*, vol. 31, 1988, p. 3532.
- ⁴ Smith, B. L., and Glezer, A., “The formation and evolution of synthetic jets,” *Physics of Fluids*, vol. 10, 1998, p. 2281.
- ⁵ Mallinson, S. G., Hong, G., and Reizes, J. A., “Some Characteristics of Synthetic Jets,” *AIAA*, vol. 99–3651, 1999, p. 11.
- ⁶ Glezer, A., and Amitay, M., “Synthetic Jets,” *Annual Rev. Fluid Mechanics*, vol. 34, 2002, p. 33.
- ⁷ Zhong, S., Millet, F., and Wood, N. J., “The behaviour of circular synthetic jets in a laminar boundary layer,” *The Aeronautical Journal*, 2005, pp. 461–470.
- ⁸ Hong, G., “Effectiveness of micro synthetic jet actuator enhanced by flow instability in controlling laminar separation caused by adverse pressure gradient,” *Sensors and Actuators A: Physical*, vol. 132, Nov. 2006, pp. 607–615.
- ⁹ Mohseni, K., and Mittal, R., *Synthetic Jets: Fundamentals and Applications*, CRC Press, Taylor & Francis Group, 2015.
- ¹⁰ Ranga Dinesh, K. K. J., Savill, A. M., Jenkins, K. W., and Kirkpatrick, M. P., “LES of intermittency in a turbulent round jet with different inlet conditions,” *Computers & Fluids*, vol. 39, Oct. 2010, pp. 1685–1695.
- ¹¹ Nishino, T., Hahn, S., and Shariff, K., “Large-eddy simulations of a turbulent Coanda jet on a circulation control airfoil,” *Physics of Fluids*, vol. 22, 2010.
- ¹² Choi, K.-S., Paul E. Roach, DeBisschop, J.-R., and Clayton, B. R., “Turbulent Boundary-layer Control by means of Spanwise-wall Oscillation,” *AIAA*, 1997, p. 9.

- ¹³ Seifert, A., Bachar, T., Koss, D., Shepshelovich, M., and Wygnanski, I., "Oscillatory Blowing: A Tool to Delay Boundary Layer Separation," *AIAA Journal*, vol. 31, 1993, pp. 2052–2060.
- ¹⁴ Chang, R. C., Hsiao, F. B., and Shyu, R. N., "Forcing Level Effects of Internal Acoustic Excitation on the Improvement of Airfoil Performance," *Journal of Aircraft*, vol. 29, 1992, pp. 823–833.
- ¹⁵ Rumsey, C., and Gatski, T., "Summary of the 2004 CFD Validation Workshop on Synthetic Jets and Turbulent Separation Control," *2nd AIAA Flow Control conference* *AIAA Flow Control conference, AIAA-2004-2217*, 2004.
- ¹⁶ Rumsey, C. L., "Successes and Challenges for Flow Control Simulations," *AIAA 2008-4311*, 2008, pp. 1–26.
- ¹⁷ Kamnis, S., and Kontis, K., "Numerical studies on the application of synthetic jets for the active control of subsonic flow configurations," *AIAA Conference*, vol. 2004–2609, 2004, pp. 1–13.
- ¹⁸ Seifert, A., and Pack, L. G., "Active flow separation control on wall-mounted hump at high Reynolds numbers," *AIAA Journal*, vol. 40, 2002, pp. 1363–1372.
- ¹⁹ Seifert, A., and Pack, L. G., "Compressibility and Excitation Location Effects on High Reynolds Numbers Active Separation Control," *Journal of Aircraft*, vol. 40, 2003, pp. 120–126.
- ²⁰ Seifert, A., and Pack, L. G., "Effect of sweep on active separation control at high Reynolds numbers," *Journal of Aircraft*, vol. 40, 2003, pp. 120–126.
- ²¹ Gan, S., Ghia, U., and Ghia, K., "Active flow control of separated turbulent flow over a hump using RANS, DES and LES," *FEDSM2008*, ASME, ed., Jacksonville, Florida: ASME, 2008, p. 8.
- ²² Greenblatt, D., Paschal, K. B., Yao, C.-S., Harris, J., Schaeffler, N. W., and Washburn, A. E., "A Separation Control CFD Validation Test Case Part 2 - Zero Efflux Oscillatory Blowing," *AIAA 2005-0485*, 2005, pp. 1–24.
- ²³ Rumsey, C. L., "Reynolds-Averaged Navier-Stokes Analysis of Zero Efflux Flow Control over a Hump Model," *Journal of Aircraft*, vol. 44, 2006, pp. 444–452.
- ²⁴ Arwatz, G., Fono, I., and Seifert, A., "Suction and oscillatory blowing actuator Modeling and Validation," *AIAA Journal*, vol. 46, 2008, pp. 1107–1117.
- ²⁵ Childs, R. E., Stremel, P. M., Garcia, J. A., Heineck, J. T., Kushner, L. K.,

- and Storms, B. L., "Simulation of Sweep-Jet Flow Control , Single Jet and Full Vertical Tail," *54th AIAA Aerospace Sciences Meeting*, 2016, pp. 1–28.
- ²⁶ Krishnan, G., and Mohseni, K., "An Experimental and Analytical Investigation of Rectangular Synthetic Jets," *Journal of Fluids Engineering*, vol. 131, 2009, p. 121101.
- ²⁷ Milanovic, I. M., and Zaman, K. B. M. Q., "An experimental study of synthetic jets from rectangular orifices," *ASME HTFED04*, vol. HT-FED2004, 2004, p. 11.
- ²⁸ Lam, J. H., Timchenko, V., and Reizes, J., "Numerical and experimental studies of a channel flow with multiple circular synthetic jets," vol. 10, 2012, p. 8.
- ²⁹ Lee, B., Kim, M., Choi, B., Kim, C., and Kim, H. J., "Closed-loop Active Flow Control of Stall Separation using Synthetic Jets," *AIAA Applied Aerodynamics Conference*, 2013, pp. 1–23.
- ³⁰ Lee, D., Orkwistt, P. D., Turner, M. G., Pes, M., Filzt, C., and Caldwell, N., "Numerical analysis of two dimensional directed synthetic jets in cross flow at low match number," *AIAA*, vol. 2003–637, 2003, p. 15.
- ³¹ Florea, R., and Wake, B. E., "Parametric Analysis of Directed - Synthetic Jets for Improved Dynamic-Stall Performance," *AIAA*, 2003, pp. 1–11.
- ³² Cannata, M., and Iuso, G., "Spanwise directed synthetic jets for wall turbulence control," *AIAA*, vol. 2008–4205, 2008, pp. 1–18.
- ³³ McCormick, D. C., "Boundary Layer separation Control with directed synthetic jets," *AIAA*, 2000, pp. 1–11.
- ³⁴ Shevell, R., *Fundamentals of Flight*, 1998.
- ³⁵ McLean, D., *Understanding Aerodynamics*, John Wiley & Sons Ltd, 2013.
- ³⁶ Hoffman, J., and Johnson, C., "Resolution of d ' Alembert ' s Paradox," 2008, pp. 1–14.
- ³⁷ Anderson, D. F., and Eberhardt, S., *Understanding Flight*, McGraw-Hill, 2001.
- ³⁸ Geropp, D., and Odenthal, H.-J. H.-J., "Drag reduction of motor vehicles by active flow control using the Coanda effect," *Experiments in fluids*, vol. 28, 2000, pp. 74–85.
- ³⁹ John D Anderson, "Ludwig Prandtl ' s Boundary Layer," *Physics Today*, 2005, p. 7.
- ⁴⁰ Pope, S. B., "Turbulent Flows," *Journal of Turbulence*, vol. 1, 2000, p. 771.

- 41 Hazen, D. C., *Boundary Layer Control*, USA: Princeton University, 1965.
- 42 Schlichting, H., *Boundary Layer Theory*, 1979.
- 43 Kluwick, A., *Recent advances in boundary layer theory*, 1998.
- 44 Kral, L. D., Donovan, J. F., Cain, A. B., and Cary, A. W., “Numerical Simulation of Synthetic Jet Actuators,” *AIAA 97-1824*, 1997, p. 14.
- 45 Amitay, M., Smith, D. R., Kibens, V., Parekh, D. E., and Glezer, A., “Aerodynamic Flow control over an Unconventional Airfoil Using Synthetic Jet Actuators,” *AIAA Journal*, vol. 39, 2001, pp. 361–371.
- 46 Koumoutsakos, P., and Mezic, I., *Control of Fluid Flow*, Berlin Heidelberg 2006: Springer-Verlag, 2006.
- 47 Lee, C., Hong, G., Ha, Q. P., and Mallinson, S. G., “A piezoelectrically actuated micro synthetic jet for active flow control,” *Sensors and Actuators A: Physical*, vol. 108, Nov. 2003, pp. 168–174.
- 48 Smith, B. L., and Glezer, A., “Jet vectoring using synthetic jets,” *Journal of Fluid Mechanics*, vol. 458, May 2002, pp. 1–34.
- 49 Guo, D., and Gary, A. W., “Vectoring Control of a Primary Jet with Synthetic Jets,” *AIAA*, vol. 2001–738, 2001, p. 11.
- 50 Smith, B. L., and Glezer, A., “Vectoring of Adjacent Synthetic Jets,” *AIAA Journal*, vol. 43, Oct. 2005, pp. 2117–2124.
- 51 Chen, Y., Liang, S., Anug, K., Glezer, A. and Jagoda, J., “Enhanced Mixing in a Simulated Combustor Using Synthetic Jet Actuators,” *37th AIAA Aerospace Sciences Meeting and Exhibit*, 1999, p. AIAA-99-0449.
- 52 Chiekh, M. Ben, Bera, J.-C., and Sunyach, M., “Synthetic jet control for flows in a diffuser: vectoring, spreading and mixing enhancement,” *Journal of Turbulence*, vol. 4, 2003, pp. 37–41.
- 53 Mahalingam, R., and Glezer, A., “Design and Thermal Characteristics of a Synthetic Jet Ejector Heat Sink,” *Journal of Electronic Packaging*, vol. 127, 2005, p. 6.
- 54 Pavlova, A., and Amitay, M., “Electronic Cooling Using Synthetic Jet Impingement,” *Journal of Heat Transfer*, vol. 128, 2006, p. 897.
- 55 Vukasinovic, B., Brzozowski, D., and Glezer, A., “Fluidic Control of Separation Over a Hemispherical Turret,” *AIAA Journal*, vol. 47, 2009, pp. 2212–2222.
- 56 Vukasinovic, B., Glezer, A., Gordeyev, S., Jumper, E., and Kibens, V., “Fluidic Control of a Turret Wake, Part I: Aerodynamic Effects,” *AIAA*

2009-816, vol. 48-8, 2009.

- ⁵⁷ III, L. N. C., and Sheplak, M., “Actuators for Active Flow Control,” *Annual Review of Fluid Mechanics*, vol. 43, 2011, pp. 247–272.
- ⁵⁸ Maldonado, V., Farnsworth, J., Gressick, W., and Amitay, M., “Active control of flow separation and structural vibrations of wind turbine blades,” *Wind Energy*, 2009, pp. 221–237.
- ⁵⁹ Esmaeili Monir, H., Tadjfar, M., and Bakhtian, a., “Tangential synthetic jets for separation control,” *Journal of Fluids and Structures*, vol. 45, Feb. 2014, pp. 50–65.
- ⁶⁰ Tang, H., Salunkhe, P., Zheng, Y., Du, J., and Wu, Y., “On the use of synthetic jet actuator arrays for active flow separation control,” *Experimental Thermal and Fluid Science*, vol. 57, 2014, pp. 1–10.
- ⁶¹ Nae, C., “Unsteady Flow control using synthetic jet actuators,” *AIAA*, 2000, p. 12.
- ⁶² Zhang, S., and Zhong, S., “Turbulent Flow Separation Control over a Two-Dimensional Ramp Using Synthetic Jets,” *AIAA Journal*, vol. 49, Dec. 2011, pp. 2637–2649.
- ⁶³ Montoya, L. D., Jackson, J. L., and Amitay, M., “Control of aerosol dispersion and removal in a room using synthetic jet actuators,” *Building and Environment*, vol. 45, Jan. 2010, pp. 165–175.
- ⁶⁴ Baeder, K. D. and J. D., “Active Flow control for rotor Airfoils using synthetic jets,” *AIAA*, vol. 2002–2835, 2002.
- ⁶⁵ Smith, B. L., and Swift, G. W., “A comparison between synthetic jets and continuous jets,” *Experiments in Fluids*, vol. 34, Feb. 2003, pp. 467–472.
- ⁶⁶ Tang, H., and Zhong, S., “2D numerical study of circular synthetic jets in quiescent flows,” *The Aeronautical Journal*, 2005, pp. 89–97.
- ⁶⁷ Lee, C. Y., and David B. Goldstein, “Two Dimensional Synthetic Jet Simulation,” *AIAA*, vol. 40, 2002, pp. 510–516.
- ⁶⁸ Lee, C. Y., and Goldstein, D. B., “Two-Dimensional Synthetic Jet Simulation,” *AIAA Journal*, vol. 40, 2002, pp. 510–516.
- ⁶⁹ Jabbal, M., Wu, J., and Zhong, S., “The performance of round synthetic jets in quiescent flow,” *The Aeronautical Journal*, 2006, pp. 385–393.
- ⁷⁰ Utturkar, Y., Holman, R., Mittal, R., Carroll, B., Sheplak, M., and Cattafesta, L., “A Jet Formation Criterion for Synthetic Jet Actuators,” *AIAA Conference*, vol. 2003–636, 2003, pp. 1–9.

- 71 Holman, R., Utturkar, Y., Mittal, R., Smith, B. L., and Cattafesta, L., "Formation Criterion for Synthetic Jets," *AIAA*, vol. 43, 2005, p. 8.
- 72 Zhong, S., Jabbal, M., Tang, H., Garcillan, L., Guo, F., Wood, N., and Warsop, C., "Towards the design of synthetic-jet actuators for full-scale flight conditions: Part 1: The fluid mechanics of synthetic-jet actuators," *Flow, Turbulence and Combustion*, vol. 78, 2007, pp. 283–307.
- 73 New, D. T. H., and Yu, S. C. M., *Vortex Rings and Jets: Recent Developments in Near-Field Dynamics*, 2015.
- 74 Zhang, S., and Zhong, S., "An experimental investigation of turbulent flow separation control by an array of synthetic jets," *AIAA Conference*, 2010, pp. 1–16.
- 75 Mossi, K., Castro, N. D., Bryant, R., Mane, P., Mossi, K., Castro, N. D., Bryant, R., and Mane, P., "Boundary Condition Effects on Piezo-Synthetic Jets," *Integrated Ferroelectrics*, vol. 71:1, 2006, pp. 257–266.
- 76 Alstrom, R. B., "Aerodynamic flow control of a high lift system with dual synthetic jet arrays," Clarkson, 2013.
- 77 Feero, M. a., Goodfellow, S. D., Lavoie, P., and Sullivan, P. E., "Flow Reattachment Using Synthetic Jet Actuation on a Low-Reynolds-Number Airfoil," *AIAA Journal*, vol. 53, 2015, pp. 1–10.
- 78 Greenblatt, D., and Wygnanski, I. J., "The control of flow separation by periodic excitation," *Progress in Aerospace Sciences*, vol. 36, 2000.
- 79 Hong, G., "Effectiveness of micro synthetic jet actuator enhanced by flow instability in controlling laminar separation caused by adverse pressure gradient," *Sensors and Actuators A: Physical*, vol. 132, Nov. 2006, pp. 607–615.
- 80 Hong, G., Lee, C., Ha, Q. P., Mack, A. N. F., and Mallinson, S. G., "Effectiveness of synthetic jets enhanced by tollmien-schlichting waves," *AIAA*, 2002, pp. 1–9.
- 81 Holman, R., Gallas, Q., Carroll, B., and Cattafesta, L., "Interaction of Adjacent Synthetic Jets in an Airfoil Separation Control Application," *AIAA Conference*, 2003, pp. 1–11.
- 82 Goodfellow, S. D., Yarusevych, S., and Sullivan, P. E., "Momentum Coefficient as a Parameter for Aerodynamic Flow Control with Synthetic Jets," *AIAA Journal*, vol. 51, Mar. 2013, pp. 623–631.
- 83 Smith, D., Amitay, M., Kibens, V., Parekh, D., and Glezer, A., "Modification of lifting body aerodynamics using synthetic jet actuators,"

36th AIAA Aerospace Sciences Meeting and Exhibit, 1998.

- ⁸⁴ Kim, W., Kim, C., and Jung, K. J., “Separation Control Characteristics of Synthetic Jets Depending on Exit Configuration,” *Aiaa*, vol. 50, 2012, pp. 1–25.
- ⁸⁵ Kim, S. B., Bong, W., and Kim, C., “Separation Control Mechanism of Airfoil Using Synthetic Jet,” *Journal of Mechanical Science and Technology*, vol. 21, 2007, pp. 1367–1375.
- ⁸⁶ Kim, S. H., and Kim, C., “Separation control on NACA23012 using synthetic jet,” *Aerospace Science and Technology*, vol. 13, Jun. 2009, pp. 172–182.
- ⁸⁷ Margaris, P., and Gursul, I., “Wing tip vortex control using synthetic jets,” *The Aeronautical Journal*, 2006, pp. 673–681.
- ⁸⁸ Kim, W., Kim, S., Choi, K., and Kim, C., “Experimental and Computational study on flow characteristics by Synthetic Jets Configuration,” *AIAA Conference*, vol. 2010–864, 2010, pp. 1–12.
- ⁸⁹ Smith, D. R., “Interaction of a Synthetic Jet with a Cross ow Boundary Layer,” *AIAA*, vol. 40, 2002, p. 12.
- ⁹⁰ Aram, S., and Mittal, R., “Computational Study of the Effect of Slot Orientation on Synthetic Jet-Based Separation Control,” *International Journal of Flow Control*, 2011, pp. 1–23.
- ⁹¹ Orkwis, P. D., and Filz, C., “Characterization of dual two dimensional synthetic jets in cross flow at low mach number,” *AIAA Conference*, 2003, pp. 1–14.
- ⁹² Xiong, D., Zhixun, X., Zhenbing, L., and Lin, W., “A novel optimal design for an application-oriented synthetic jet actuator,” *Chinese Journal of Aeronautics*, Apr. 2014, p. 9.
- ⁹³ Chaudhari, M., Verma, G., Puranik, B., and Agrawal, A., “Frequency response of a synthetic jet cavity,” *Experimental Thermal and Fluid Science*, vol. 33, Mar. 2009, pp. 439–448.
- ⁹⁴ Watson, M., Jaworski, A. J., and Wood, N. J., “A study of synthetic jets from rectangular and dual-circular orifices,” *The Aeronautical Journal*, 2003, p. 11.
- ⁹⁵ Albright, S. O., and Solovitz, S. A., “Examination of a Variable- Diameter Synthetic Jet,” vol. 138, 2016, pp. 1–9.
- ⁹⁶ Van Buren, T., Leong, C. M., Whalen, E., and Amitay, M., “Impact of

- orifice orientation on a finite-span synthetic jet interaction with a crossflow,” *Physics of Fluids*, vol. 28, 2016.
- ⁹⁷ Montazer, E., Mirzaei, M., Salami, E., Ward, T. A., Romli, F. I., and Kazi, S. N., “Optimization of a synthetic jet actuator for flow control around an airfoil,” *IOP Conference Series: Materials Science and Engineering*, vol. 152, 2016, p. 12023.
- ⁹⁸ Kosasih, E. A., Harinaldi, H., and Trisno, R., “Characteristics of Vortex Ring Formation by Synthetic Jet Actuators in Different Cavities,” *International Journal of Technology*, vol. 7, 2016, p. 296.
- ⁹⁹ Feero, M. A., Lavoie, P., and Sullivan, P. E., “Influence of cavity shape on synthetic jet performance,” *Sensors and Actuators, A: Physical*, vol. 223, 2015, pp. 1–10.
- ¹⁰⁰ Ma, L., and Feng, L., “Experimental investigation on control of vortex shedding mode of a circular cylinder using synthetic jets placed at stagnation points,” *Science China Technological Sciences*, vol. 56, Nov. 2012, pp. 158–170.
- ¹⁰¹ Versteeg, H. K., and Malalasekera, W., *An Introduction to Computational Fluid Dynamics*, 2007.
- ¹⁰² Ferziger, J. ., and Peric, M., *Computational Methods for Fluid Dynamics*, Germany: Springer, 2002.
- ¹⁰³ David C. Wilcox, *Turbulence-Modeling-for-CFD*, 1993.
- ¹⁰⁴ Lesieur, M., Metais, O., and Comte, P., *Large-Eddy Simulations of Turbulence*, Cambridge, 2005.
- ¹⁰⁵ Frank M. White, *Fluid Mechanics*, McGraw-Hill, 2011.
- ¹⁰⁶ Spalart, P. R., “Strategies for turbulence modelling and simulations,” *International Journal of Heat and Fluid Flow*, vol. 21, Jun. 2000, pp. 252–263.
- ¹⁰⁷ Coleman, G. N., and Sandberg, R. D., “A primer on direct numerical simulation of turbulence - methods, procedures and guidelines,” *Technical Report*, 2010, pp. 1–21.
- ¹⁰⁸ Davidson, P. A., Kaneda, Y., Moffatt, K., and Sreenivasan, K. R., *A Voyage Through Turbulence*, 2014.
- ¹⁰⁹ Schmitt, F. G., “About Boussinesq’s turbulent viscosity hypothesis: historical remarks and a direct evaluation of its validity,” *Comptes Rendus Mécanique*, vol. 335, Sep. 2007, pp. 617–627.

- 110 You, D., and Moin, P., "Active control of flow separation over an airfoil using synthetic jets," *Journal of Fluids and Structures*, vol. 24, Nov. 2008, pp. 1349–1357.
- 111 Baldwin, B. S., and Barth, T. J., "A One-Equation Turbulence Transport Model for High Reynolds Number Wall-Bounded Flows," *Nasa Technical Memorandum*, 1990.
- 112 Spalart, P. R., and Allmaras, S. R., "A one-equation turbulence model for aerodynamic flows," *La Recherche Aerospatiale*, vol. 1, 1994, pp. 5–21.
- 113 Launder, B. E., and Sharma, B. I., "Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc," *Letters in Heat and Mass Transfer*, vol. 1, 1974, pp. 131–137.
- 114 Chou, P. Y., "On velocity correlations and the solution of the equations of turbulent fluctuations," *National Tsing Hua Univerity*, vol. 3, 1945, pp. 38–54.
- 115 Harlow, F. H., and Nakayama, P. I., "Transport of Turbulence Energy Decay Rate," *Los Alamos Sci Lab. University of California Report*, vol. LA-3854, 1968, p. 7.
- 116 Jones, W. ., and Launder, B. ., "The prediction of laminarization with a two-equation model of turbulence," *International Journal of Heat and Mass Transfer*, vol. 15, Feb. 1972, pp. 301–314.
- 117 Shih, T. H., Liou, W. W., Shabbir, A., Yang, Z., and Zhu, J., "A New k-epsilon Eddy Viscosity Model for High Reynolds Number Turbulent Flows: Model Development and Validation," *Computer \& Fluids*, vol. 24, 1995, pp. 227–238.
- 118 Menter, F. R., "Two-equation eddy-viscosity turbulence model for engineering applications," *AIAA journal*, vol. 32, 1994, pp. 1598–1605.
- 119 Menter, F. R., Kuntz, M., and Langtry, R., "Ten Years of Industrial Experience with the SST Turbulence Model," *Turbulence Heat and Mass Transfer 4*, vol. 4, 2003, pp. 625–632.
- 120 Catalano, P., and Amato, M., "An evaluation of RANS turbulence modelling for aerodynamic applications," *Aerospace Science and Technology*, vol. 7, Oct. 2003, pp. 493–509.
- 121 Barton, I. E., "Comparison of SIMPLE- and PISO-type algorithms for transient flows," *International Journal for Numerical Methods in Fluids*, vol. 26, 1998, pp. 459–483.
- 122 Bettini, C., and Cravero, C., "Computational Analysis of Flow Separation

- Control for the Flow Over a Wall-Mounted Hump Using a Synthetic Jet,” *AIAA 2007-516*, 2007, pp. 1–7.
- ¹²³ He, C., Corke, T. C., and Dame, N., “Numerical and Experimental Analysis of Plasma Flow Control Over a Hump Model,” *AIAA paper*, vol. 2007–935, 2007, pp. 1–16.
- ¹²⁴ Greenblatt, D., Paschal, K. B., Yao, C. S., Harris, J., Schaeffler, N. W., and Washburn, A. E., “Experimental Investigation of Separation Control Part 1: Baseline and Steady Suction,” *AIAA Journal*, vol. 44, 2006, pp. 2820–2830.
- ¹²⁵ Gatski, T., and Rumsey, C., “Langley Research Center Workshop,” *web site* Available: <http://cfdval2004.larc.nasa.gov/index.html>.
- ¹²⁶ Rumsey, C. L., “Proceedings of the 2004 Workshop on CFD Validation of Synthetic Jets and Turbulent Separation Control,” *NASA/CP-2007-214874*, 2007.
- ¹²⁷ Morgan, P. E., Rizzetta, D. P., and Visbal, M. R., “Large-Eddy Simulation of Separation Control for Flow over a Wall-Mounted Hump,” *AIAA Journal*, vol. 45, 2007.
- ¹²⁸ Larsson, J., and Kawai, S., “Wall-modeling in Large Eddy Simulation: Length scales , grid resolution , and accuracy,” *Center for Turbulence Research, Annual Research Briefs 2010*, vol. 15105, 2010, pp. 39–46.
- ¹²⁹ Rumsey, C. L., “Reynolds-Averaged Navier-Stokes Analysis of Zero Efflux Flow Control over a Hump Model,” *AIAA Journal*, vol. 44, 2007, pp. 444–452.
- ¹³⁰ Viken, S. A., Vatsa, V. N., Rumsey, L., and Carpenter, M. H., “Flow Control Analysis on the Hump Model with RANS Tools,” *AIAA Journal*, vol. 2006–218, 2003, p. 26.
- ¹³¹ Laouedj, S., Azzi, A., and Benazza, A., “New Analysis in 3D-Synthetic Jets Within Numerical Investigation Using CFD Validation Code: Moving Boundary Techniques,” *Energy Procedia*, vol. 19, Jan. 2012, pp. 226–238.
- ¹³² Lyons, D. C., Zajackowski, F. J., Peltier, L. J., and Paterson, E. G., “Assessment of des models for separated flow from a hump in a turbulent boundary layer,” *Proceedings of the 5th ASME/JSME*, 2007, pp. 1–10.
- ¹³³ *ANSYS FLUENT 14 Theory Guide*, 2011.
- ¹³⁴ Findanis, N., and Ahmed, N. A., “A CFD analysis of the flow over a side-supported sphere with localised synthetic jet,” *AIAA Conference*, 2013, pp. 1–20.

- 135 Utturkar, Y., Mittal, R., Rampunggoon, P., and Cattafesta, L., "Sensitivity of synthetic jets to the design of the jet cavity," *AIAA 2002-0124*, Jan. 2002, pp. 1–8.
- 136 Inc., A., *ANSYS FLUENT 14 Users Guide*, ANSYS, 2011.
- 137 Naughton, J. W., "Wall Shear Stress Measurements on the NASA Hump Model for CFD Validation," *AIAA 2004-2607*, 2004, pp. 1–13.
- 138 Greenblatt, D., Paschal, K. B., Yao, C.-S., Harris, J., Schaeffler, N. W., and Washburn, A. E., "A Separation Control CFD Validation Test Case Part 1: Baseline & Steady Suction," *AIAA 2004-2220*, 2004, pp. 1–17.
- 139 Greenblatt, D., Paschal, K. B., Yao, C. S., and Harris, J., "Experimental Investigation of separation Control Part2: Zero Mass-Flux Oscillatory Blowing," *AIAA Journal*, vol. 44, 2006, pp. 2831–2845.
- 140 Šarić, S., Jakirlić, S., Djugum, a., and Tropea, C., "Computational analysis of locally forced flow over a wall-mounted hump at high-Re number," *International Journal of Heat and Fluid Flow*, vol. 27, 2006, pp. 707–720.
- 141 Rumsey, C. L., Gatski, T. B., Sellers III, W. L., Vasta, V. N., and Viken, S. a., "Summary of the 2004 Computational Fluid Dynamics Validation Workshop on Synthetic Jets," *AIAA Journal*, vol. 44, 2006, pp. 194–207.
- 142 Kim, S. H., and Kim, C., "Separation control on NACA23012 using synthetic jet," *Aerospace Science and Technology*, vol. 13, 2009, pp. 172–182.
- 143 Lee, B., Kim, M., Lee, J., Kim, C., and Jung, K. J., "Separation Control Characteristics of Synthetic Jets with Circular Exit Array," *AIAA Flow Control Conference*, 2012, pp. 1–25.
- 144 Nae, C., "Synthetic jets influence on a naca 0012 airfoil at high angles of attack," *AIAA Journal*, vol. 98, 1998, pp. 854–859.
- 145 Ishibashi, Y., and Miyaji, K., "Detached Eddy Simulations of a Synthetic Jet for a High-angle-of-attack Airfoil -Influence of the Jet Angle-," *AIAA SciTech*, vol. 2014–768, 2014, pp. 1–19.
- 146 Alexandru Catalin, M., and Florin, F., "Numerical simulations of synthetic jets in aerodynamic applications," *Incas Bulletin*, vol. 6, Apr. 2014, pp. 81–93.
- 147 Duvigneau, R., and Visonneau, M., "Optimization of a synthetic jet actuator for aerodynamic stall control," *Computers & Fluids*, vol. 35, Jul. 2005, pp. 624–638.

- 148 Kotapati, R. B., Mittal, R., Marxen, O., Ham, F., You, D., and Cattafesta, L. N., “Nonlinear dynamics and synthetic-jet-based control of a canonical separated flow,” *Journal of Fluid Mechanics*, vol. 654, 2010, pp. 65–97.
- 149 Jacobs, E., and Clay, W., “Report No.530 -Characteristics of the NACA 23012 airfoil from tests in the full-scale and variable-density tunnels,” 1936, pp. 435–444.
- 150 Nishizawa, A., Takagi, S., Abe, H., Segawa, T., and Yoshida, H., “Toward smart control of separation around a wing -Active Separation Control System part 2-,” 2004.
- 151 Hasnain, Z., Flatau, A. B., Jr., J. E. H., and Mulinti, R., “Numerical Study and Experimental Validation of the Interaction of Multiple Synthetic Jet Actuators with Cross Flow,” *AIAA-2012-1247*, 2012.
- 152 Heinrich, R., Meyer, D., and Schaubert, T., “US 2008/0240942 A1,” US 2008/0240942 A1, 2008.
- 153 Chaudhari, M., Verma, G., Baramade, A., Puranik, B., and Agrawal, A., “Near-field measurements and cavity design for synthetic jet,” *AIAA 2008-4068*, 2008, pp. 1–12.
- 154 Kim, W., Kim, C., and Jung, K. J., “Separation Control Characteristics of Synthetic Jets Depending on Exit Configuration,” *AIAA Journal*, vol. 50, Mar. 2012, pp. 559–570.
- 155 Patankar, S., “Numerical heat transfer and fluid flow,” *Series in computational methods in mechanics and thermal sciences*, 1980, pp. 1–197.
- 156 Fisher, R., Nishino, T., and Savill, A. M., “Numerical Analysis of a Bi - Directional Synthetic Jet for Active Flow Control,” *AIAA Journal*, 2016, pp. 1–13.
- 157 Azim, R., Hasan, M. M., and Ali, M., “Numerical investigation on the delay of boundary layer separation by suction for NACA 4412,” *6th BSME International Conference on Thermal Engineering, ICTE 2014*, vol. 105, 2015, pp. 329–334.

APPENDICES

Appendix A

UDF for the Jet velocity profiles

The following UDF was created in Fluent with the objective to model the velocity profile at the jet exit. This code shows the basic form of all different types of UDF for the jet exit velocity profiles created in this project. The values of the variables A, B, D, shape, as well as the values of the function F_profile after the conditional decision is made, change accordingly to the requirements needed. For instance, for a pulsed jet suction, the condition in the if sentence would change from >0 to <0 . All configurations of jet are based on this UDF template.

A.1 Control of the velocity Jet exit (Pulsed Jet Blow)

```
#include "udf.h"
#define pi 3.1415926535897
DEFINE_PROFILE(jet, thread, position)
{
    real xx[ND_ND];    /*this holds the position vector*/
    real yy;
    face_t f;
    real A = 32.64*1;    /* Amplitud 1A*/
    real B = 0.0683018    /* B for jet R-Q*/
    real w = 2 * pi * 37.09; /* 37.09Hz or 100Hz, or 5.90Hz or 120Hz */
    real D=0*pi/180;    /* Displacement= 0, 45, 90, 135, 180*/
    real shape = 1800;    /* 800,1300,1800,2300,2800,3322*/

    begin_f_loop(f,thread)
    {
        F_CENTROID(xx,f,thread);
        yy = xx[1];

        if( sin(w * CURRENT_TIME)>0 )
            F_PROFILE(f, thread, position) = A * (1 - (yy - B) * (yy - B) *
            shape * shape) * sin(w * CURRENT_TIME + D);
        else
            F_PROFILE(f, thread, position) = 0;
    }
}
```

```

    end_f_loop(f, thread)
}

```

A.2 “Horizontal” X, Direction of the Velocity.

```

#include "udf.h"
#define pi 3.141592635897
DEFINE_PROFILE(x, thread, position)
{
    face_t f;
    real w = 2 * pi * 100;
    begin_f_loop(f, thread)
    {
        if( sin(w * CURRENT_TIME)>0)
            F_PROFILE(f, thread, position) = 1;
        else
        {
            F_PROFILE(f, thread, position) = 1;
        }
    }
    end_f_loop(f, thread)
}

```

A.3 “Vertical” Y, Direction of the Velocity .

```

#include "udf.h"
#define pi 3.1415926535897
DEFINE_PROFILE(y, thread, position)
{
    face_t f;
    real w = 2 * pi * 100;
    begin_f_loop(f, thread)
    {
        If (A * sin(w * CURRENT_TIME)>0)
            F_PROFILE(f, thread, position) = 1.52220;
        else
            F_PROFILE(f, thread, position) = 3.02326;
    }
    end_f_loop(f, thread)
}

```

Appendix B

Scripts used for Fluent

B.1 Journal File for the set of BDSJ analyses

The following text is an example script with comments to run some of the hump simulations. This complete file can run with almost no alterations, with the exception on the places where a decision should be made on the selection of turbulence model. For reasons of space and illustration, the symbol ¶ was included, which represents the number of enters that the user must apply for the file to work.

```
; Read case and data files from the file directory
file/read-case-data name_of_file.cas.gz
file/auto-save/root-name name_of_file.dat.gz

; Reorder domainww
mesh/reorder/reorder-domain
mesh/reorder/reorder-domain¶

; Define the Turbulence model for Spalart Almaras
define/models/viscous/spalart-allmaras
Y
; or for Realizable k-epsilon
define/models/viscous/ke-realizable
Y
; or for k-w SST
define/models/viscous/kw-sst
Y

; Stablish the convergence criteria for continuity, u and v momentum and
turbulence k or k and epsilon, or k and omega.
solve/monitors/residual/convergence-criteria
0.000001      ; continuity
0.00001       ; x-velocity
0.00001       ; y-velocity
0.00001       ; nut, kappa or kappa
```

^{ww} Reorder the nodes, faces, and cells along zones and in memory to increase memory access efficiency. This instruction is used twice because normally two iteration are enough to order all the domain.

```

0.00001          ; epsilon or omega for k-e and k-w respectively

; Define the analysis on steady state
define/models/steady y

; Initialize the solution from the inlet (stablished on file)
solve init init

; Run for 10, 100 or 1000 iterations for steady simulations
it 10¶¶¶¶¶¶¶¶¶¶xx

; Save case and data file with the same name
file w-c-d¶
y

; Stablish transient analysis
define/models/unsteady-2nd-order y

; Do not collect data sampling for time statistics
solve/set/d-s n¶

; Stablish first order discretization scheme for S-A
solve/set/discretization-scheme/mom 0
solve/set/discretization-scheme/nut 0
; or for For k- epsilon
solve/set/discretization-scheme/k 0
solve/set/discretization-scheme/epsilon 0
; or for For k- omega
solve/set/discretization-scheme/k 0
solve/set/discretization-scheme/omega 0

; Compile the UDF for the velocity x.c, y.c and amplitude jet.c
define/user-defined/compiled-functions compile¶¶
x.c
y.c
jet.c¶¶
define/user-defined/compiled-functions load¶¶

; Save case and data file with the same name
file w-c-d¶
y

; or Run for 10,100 or 1000 time-steps and 20 iteration per time step
solve/dual-time-iterate¶ 100 20¶¶¶¶¶¶¶¶¶¶

; Save file with same name
file w-c-d¶
y

; Stablish second order discretization scheme for S-A
solve/set/discretization-scheme/mom 1
solve/set/discretization-scheme/nut 1
; or for For k- epsilon
solve/set/discretization-scheme/k 1
solve/set/discretization-scheme/epsilon 1
; or for For k- omega

```

^{xx} After the iteration instruction, the software request whether we want to overwrite over the old monitors or continue at the bottom of the existing file. This set of 10 continuous marks ¶ represent each monitor of the variables used for an analysis, and an agreement to keep writing at the end of the existing files.

```

solve/set/discretization-scheme/k 1
solve/set/discretization-scheme/omega 1

; Run for 10000 time-steps, 20 iteration per time step
solve/dual-time/10000 20

; Collect data sampling for time statistics
solve/set/d-s y¶¶¶

; Run for 6000 time-steps and 20 iteration per time step
solve/dual-time/6000 20

; Save file with same name
file w-c-d¶
y

; Get the Mean Skin Friction coefficient at the lower wall and write it into a
file with the name meanSkinFricCoef.txt
plot/plot¶
meanSkinFricCoef.txt¶¶¶
mean-skin-friction-coef
y¶¶
wall¶¶¶

; Get the pressure coefficient at the lower wall with the name cp.txt
plot/plot¶
cp.txt¶¶¶
pressure-coefficient
y¶¶
wall¶¶¶

; Exit the application
Exit

```

B.2 Journal File for the set of Aerofoil Studies

As in the previous Appendix, the following text is an example script with comments to run some of the aerofoil simulations. The values on this file should be adapted for each angle of attack, as here only $\text{AOA}=22^\circ$ is presented. In this file, it is assumed to use the Spalart-Allmaras turbulent model, however, the changes to a different turbulent model follow the same structure as in the previous appendix. For reasons of space and illustration, the symbol ¶ was included, which represents the number of enters that the user should apply for the file to work.

```
; Read case and data files from the file directory
file/read-case-data name_of_file.cas.gz
file/auto-save/root-name name_of_file.dat.gz

; Reorder domainyy
mesh/reorder/reorder-domain
mesh/reorder/reorder-domain¶

; Defining the velocity inlet boundary condition at 32.64 m/s, with direction
x=0.927 and y=0.374, turbulence intensity of 5% and 0.001 turbulent length
scale
define/b-c/v-i
inlet¶¶¶
32.64¶¶¶
0.927¶
0.374¶¶
5
0.001¶

; Defining the pressure at the outlet with direction x=0.927 and y=0.374,
turbulence intensity of 10% and 0.001 turbulent length scale
define/b-c/p-o
outlet¶
0¶¶
0.927¶
0.374¶¶
10
0.001¶¶

; Create monitors with the name CD_022_History.txt for the Drag coefficient in
upper and lower walls
solve/monitors/force/set-d-m
cd
Y
```

^{yy} Reorder the nodes, faces, and cells along zones and in memory to increase memory access efficiency.

```

wall_down
wall_up
y
y
CD_022_History
0.927
0.374

; Create monitors with the name CL_022_History.txt for the Lift coefficient in
upper and lower walls
solve/monitors/force/set-l-m
cl
y
wall_down
wall_up
y
y
CL_022_History
-0.374
0.9271

; Create monitors for the Average Area-weighted of the Skin Friction
solve/monitors/surface/set-monitor
avrg_skinfrictcoeff
"Area-Weighted Average"
skin-friction-coef
wall_up
no
yes
yes
Avrg_SkinFrictCoeff_22_History.txt
1
yes
time-step

; Save case and data file with the name Naca23012_aoa-22.cas.gz
file w-c-d
Naca23012_aoa-22.cas.gz /

; Initialize the solution from the inlet (stablished on file)
solve init init
ok

; Stablish the convergence criteria for continuity, u and v momentum and
turbulence k or k and epsilon, or k and omega.
solve/monitors/residual/convergence-criteria
1.00E-08
0.0000001
0.0000001
0.0000001

; Define the analysis on steady state and iterate
define/models/steady y
it 3000

; Stablish new convergence criteria
solve/monitors/residual/convergence-criteria
1.00E-06
0.0001
0.0001
0.0001

; Define transient analysis with no sampling
define/models/unsteady-2nd-order y

```

```

solve/set/d-s n

; Define second order discretization scheme for S-A
solve/set/discretization-scheme/mom 1
solve/set/discretization-scheme/nut 1

; Run for 30000 time-steps, 20 iteration per time step
solve/dual-time/30000 20

; Collect data sampling for time statistics
solve/set/d-s y
1
yes
yes

; Run for 4000 time-steps, 20 iteration per time step
solve/dual-time/4000 20

; Save case and data
file w-c-d
Naca23012_aoa-22.cas.gz
y

; Get the Mean Skin Friction coefficient at the upper wall and write it into a
file with the name meanSkinFricCoef.txt
plot/plot
Mean_Skin_Fric_Coef_aoa-22.txt
mean-skin-friction-coef
y
wall_up

; Get the Total Pressure at both walls and write it into a file with the name
Total_Pressure_aoa-22.txt
plot/plot
Total_Pressure_aoa-22.txt
total-pressure
y
wall_down
wall_up

; Get the pressure coefficient at both walls and write it into a file with the
name Cp_aoa-22.txt
plot/plot
Cp_aoa-22.txt
pressure-coefficient
y
wall_down
wall_up

; Exit the software
Exit

```


Appendix C

Useful Equations

Generic Scalar Transport Equation	$\underbrace{\frac{\partial \rho \phi}{\partial t}}_{\text{Transient term}} + \underbrace{\nabla \cdot (\rho \vec{u} \phi)}_{\text{Convection term}} = \underbrace{\nabla \cdot (\Gamma \nabla \phi)}_{\text{Diffusion}} + \underbrace{S_\phi}_{\text{Source term}}$
--------------------------------------	--

	$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \quad \text{Continuity}$
Navier-Stokes	$\frac{\partial u}{\partial t} + (u \cdot \nabla)u = -\frac{1}{\rho} \nabla p + F + \frac{\mu}{\rho} \nabla^2 u \quad \text{Motion}$
	$\rho \left(\frac{\partial \varepsilon}{\partial t} + u \cdot \nabla \varepsilon \right) - \nabla \cdot (K_H \nabla T) + p \nabla \cdot u = 0 \quad \text{Energy}$

RANS ^{zz}	$\rho \frac{\partial U_i}{\partial t} + \rho U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} (2\mu S_{ji} - \overline{\rho u'_i u'_j})$
--------------------	--

S _{ij}	Shearing linear deformation component ^{aaa} $S_{ij} = \frac{1}{2} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)$
-----------------	--

Law of Viscosity	$\tau_{xy} = \mu \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)$ $-\rho \bar{u} \bar{v} = \mu_t \left(\frac{\partial \bar{u}}{\partial y} + \frac{\partial \bar{v}}{\partial x} \right)$
------------------	---

μ	Turbulent viscosity ^{bbb} $\mu_t = \rho C_\mu \frac{\kappa^2}{\epsilon}$
-------	---

Boussinesq eddy viscosity ^{ccc,ddd}	$\tau_{ij} = 2\mu_t S_{ij}^* - \frac{2}{3} \rho \kappa \delta_{ij}$
---	---

^{zz} Similar Equation can be found on Pag. 64 of Ref. ¹⁰¹

^{aaa} Taken from Pag. 21 of Ref. ¹⁰¹

^{bbb} Taken from Pag. 75 of Ref. ¹⁰¹

^{ccc} The Boussinesq eddy viscosity hypothesis is not valid according to ¹⁰⁹

^{ddd} Equation obtained from Pag. 76 of Ref. ¹⁰¹

$$y^+ = \frac{U^* \cdot y}{\nu} \quad \text{dimensionless wall distance}$$

$$y^+ \quad U^* = \sqrt{\frac{\tau_{ij}}{\rho}}; \quad \text{friction velocity}$$

$$\tau_w = \mu \left(\frac{\partial u}{\partial y} \right)_{y=0} \quad \text{Wall shear stress}$$

$$\text{Law of the wall} \quad \text{if } y^+ > 30; \quad U^+ = \frac{1}{\kappa} \log y^+ + C; \quad \kappa \approx 0.41; \quad C \approx 5.2$$

$$\text{Re} \quad Re = \frac{\rho V L}{\mu}$$

$$C_f \quad C_f = \frac{\tau_w}{\frac{1}{2} \rho u^2} \quad \text{Friction coefficient}$$

$$\begin{aligned} & \frac{\partial(\rho\kappa)}{\partial t} + \nabla \cdot (\rho\kappa U) = \nabla \cdot \left(\frac{\mu_t}{\sigma_\kappa} \nabla \kappa \right) + 2\mu_t S_{ij} \cdot S_{ij} - \rho\varepsilon \\ \mathbf{k}\text{-}\varepsilon^{\text{eee}} & \quad \underbrace{\frac{\partial(\rho\varepsilon)}{\partial t}}_{\text{Rate of change of } \kappa \text{ or } \varepsilon} + \underbrace{\nabla \cdot (\rho\varepsilon U)}_{\text{Transport of } \kappa \text{ or } \varepsilon \text{ by convection}} = \underbrace{\nabla \cdot \left(\frac{\mu_t}{\sigma_\varepsilon} \nabla \varepsilon \right)}_{\text{Transport of } \kappa \text{ or } \varepsilon \text{ by diffusion}} + \underbrace{C_{1\varepsilon} \frac{\varepsilon}{\kappa} 2\mu_t S_{ij} \cdot S_{ij}}_{\text{Rate of production of } \kappa \text{ or } \varepsilon} - \underbrace{C_{2\varepsilon} \rho \frac{\varepsilon^2}{\kappa}}_{\text{Rate of destruction of } \kappa \text{ or } \varepsilon} \end{aligned}$$

^{eee} Equation obtained from Pag. 75 of Ref. ¹⁰¹