Theses and Dissertations

2006-09-21

Numerical Simulation of Plasma-Based Actuator Vortex Control of a Turbulent Cylinder Wake

Nathan Keith McMullin
Brigham Young University - Provo

Follow this and additional works at: <https://scholarsarchive.byu.edu/etd>

Part of the [Mechanical Engineering Commons](#)

BYU ScholarsArchive Citation

McMullin, Nathan Keith, "Numerical Simulation of Plasma-Based Actuator Vortex Control of a Turbulent Cylinder Wake" (2006). *Theses and Dissertations*. 783.
<https://scholarsarchive.byu.edu/etd/783>

This Thesis is brought to you for free and open access by BYU ScholarsArchive. It has been accepted for inclusion in Theses and Dissertations by an authorized administrator of BYU ScholarsArchive. For more information, please contact scholarsarchive@byu.edu, ellen_amatangelo@byu.edu.

NUMERICAL SIMULATION OF PLASMA-BASED ACTUATOR
VORTEX CONTROL OF A TURBULENT CYLINDER WAKE

by

Nathan K. McMullin

A thesis submitted to the faculty of

Brigham Young University

in partial fulfillment of the requirements for the degree of

Master of Science

Department of Mechanical Engineering

Brigham Young University

December 2006

Copyright © 2006 Nathan K. McMullin

All Rights Reserved

BRIGHAM YOUNG UNIVERSITY

GRADUATE COMMITTEE APPROVAL

of a thesis submitted by

Nathan K. McMullin

This thesis has been read by each member of the following graduate committee and by majority vote has been found to be satisfactory.

Date

Deryl O. Snyder, Chair

Date

Timothy W. McLain

Date

Scott L. Thomson

BRIGHAM YOUNG UNIVERSITY

As chair of the candidate's graduate committee, I have read the thesis of Nathan K. McMullin in its final form and have found that (1) its format, citations, and bibliographical style are consistent and acceptable and fulfill university and department style requirements; (2) its illustrative materials including figures, tables, and charts are in place; and (3) the final manuscript is satisfactory to the graduate committee and is ready for submission to the university library.

Date

Deryl O. Snyder
Chair, Graduate Committee

Accepted for the Department

Matthew R. Jones
Graduate Coordinator

Accepted for the College

Alan R. Parkinson
Dean, Ira A. Fulton College of Engineering
and Technology

ABSTRACT

NUMERICAL SIMULATION OF PLASMA-BASED ACTUATOR VORTEX CONTROL OF A TURBULENT CYLINDER WAKE

Nathan K. McMullin

Mechanical Engineering

Master of Science

A numerical study has been performed to investigate the mechanics of the turbulent wake of a circular cylinder that is controlled by a plasma actuator. The numerical investigation implements a straightforward moving wall boundary condition to model the actuator's effects on the flow. Validations of the moving wall for this simulation are set forth with the understanding that the moving wall can model the plasma actuator bulk flow effects at a distance downstream and not in a region near or on the plasma actuator. The moving wall boundary condition is then applied to a circular cylinder at a Reynolds number of 8,000. At this unsteady transitional flow regime, a large eddy simulation solver is utilized to resolve flow features. The moving walls are placed at the top and bottom ninety degree points of the cylinder and alternately activated at a frequency to produce lock-in behavior. Investigation into the flowfield mechanics reveals that a harmonic frequency of the forced frequency occurs from the creation of sub-vortices from the instantaneous starting and stopping of the moving-wall actuators. With the forcing frequency close to the natural shedding

frequency it is found that the aerodynamic drag increases due to the moving wall creating an average low pressure region on to the downstream side of the cylinder. It is also found that drag can be reduced when the forcing frequency is closer to half the natural shedding frequency. This happens because of a decrease in the average pressure on the downstream side of the cylinder.

ACKNOWLEDGMENTS

This project has been a great experience for me. It has been such because of the many people who have given me support of one kind or another. First, thanks go to my advisor Dr. Deryl O. Snyder. Through this research his guidance has been invaluable. His insights, and his ability to see and explain details has been an inspiration to me. I would also like to thank my committee members Dr. Timothy W. McLain and Dr. Scott L. Thomson. Thanks are also due to the members of the lab, specifically Douglas Hunsaker and Gregory Larson for their willingness to edit and consult. Brigham Young University must also be thanked for the use of the supercomputing facility and staff that were critical for this research. Finally, I would like to acknowledge the continual support and patience of my beautiful wife Kristi. She has truly been a champion throughout this project.

Table of Contents

Acknowledgements	xi
List of Tables	xv
List of Figures	xx
1 Introduction	1
1.1 Overview	1
1.2 Organization	3
1.3 Literature Review	4
1.3.1 Cylinder and Other Bluff Body Flow Control	4
1.3.2 Plasma Actuator	4
1.3.3 Plasma Actuator Flow Control	5
1.4 Contribution	8
2 Flow Solver	11
2.1 Algorithm Summary	11
2.2 Validation	13
2.3 Moving Wall Boundary Condition	16
2.4 Flow Field Point Monitoring	19
2.5 Mesh Interpolating Utilities	20
2.6 Conclusion	21

3	Moving Wall Approximation For Plasma Actuators	23
3.1	Validation of Moving Wall Boundary Condition	23
3.2	Determination of Appropriate Wall Velocity	26
3.2.1	Reverse Blasius Solution	28
3.3	Application of Method	34
3.4	Conclusion	37
4	Circular Cylinder Flow Control at $Re = 8,000$	39
4.1	Introduction	39
4.2	Computational Domain	42
4.3	Results And Discussion For No Actuator	46
4.4	Moving Wall Actuator	51
4.5	Lock-In Case	53
4.6	Lock-In With Harmonics	62
5	Conclusion	71
	Bibliography	76

List of Tables

2.1	Results of SFELES compared with various experimental results . . .	15
4.1	Strouhal numbers and drag coefficients at $Re = 8,000$ for both coarse and fine mesh results from SFELES as compared to published results.	51
4.2	Effects of the moving wall lock-in case on the average aerodynamic properties of the cylinder versus the unforced case.	58
4.3	Effects of the moving wall lock-in with harmonics case on the average aerodynamic properties of the cylinder versus the unforced case. . . .	65

List of Figures

1.1	Schematic of the Dielectric barrier discharge (DBD) plasma actuator.	5
2.1	Sample two dimensional geometry with in-plane triangular mesh. . .	12
2.2	Mesh and computational domain for SFELES $Re = 3,900$ validation case.	14
2.3	Instantaneous plot of iso-surfaces of vorticity magnitude, $ \vec{\omega} D/U_\infty$, in the region $-0.5D \leq x \leq 10D$ for flow around a cylinder at $Re = 3,900$.	15
2.4	Mean stream-wise velocity (a), mean lateral velocity (b), and time-averaged Reynolds shear stress (c) at different stream-wise locations in the wake at $Re = 3,900$	17
2.5	Schematic of the two parts of the moving wall cycle a) First part, b) Second part.	18
2.6	Three different possible areas that are made by the vertices of the element with the point p , when p lies in the element	19
2.7	How the elements in SFELES appear in the transverse direction with the monitoring point z_p lying between the z_a element plane and the z_b element plane.	20
2.8	Tecplot add-on graphical user interface for creating restart files for SFELES	21
3.1	Roth's results of a plasma actuator induced velocity profile plotted against a solution to the Glauert wall jet problem.	24
3.2	Schematic of the plasma actuator.	24
3.3	Schematic of a moving wall applied to a flat plate.	25
3.4	Computational domain used for simulation of Corke's experimental work, a) Mesh used in simulation (37901 nodes) , b) Schematic of simulation key dimensions.	27

3.5	Velocity profiles from a CFD calculation compared with Corke's experimental results of a plasma actuator induced velocity profile. . . .	28
3.6	Schematic of Blasius solutions to the boundary layer equations with modified boundary conditions.	29
3.7	Solution to the Blasius solution with reversed boundary conditions. .	30
3.8	u velocity profiles from the Blasius solution with reversed boundary conditions.	31
3.9	Mesh used for CFD validation of the Blasius solution with reversed boundary conditions.	31
3.10	CFD validation versus the analytical solution of the Blasius solution with reversed boundary conditions.	32
3.11	Schematic of the wall jet.	33
3.12	Computational domain used for simulation of Roth's experimental work, a) Mesh used in simulation (90681 nodes) , b) Schematic of simulation key dimensions.	36
3.13	Moving wall boundary conditions simulation results compared to Roth's data.	36
4.1	Experimental data for the lock-in case at $Re = 7,400$ a) Downstream u velocity time trace , b) Lock-in PSD.	40
4.2	Experimental data for the lock-in with harmonics case at $Re = 7,400$ a) Downstream u velocity time trace , b) Lock-in with harmonics PSD. .	41
4.3	Smoke visualization of the shedding off of a cylinder in cross flow for the lock-in case through the period of the forcing cycle of the plasma actuator.	41
4.4	Smoke visualization of the shedding off of a cylinder in cross flow for the lock-in with harmonics case through the period of the forcing cycle of the plasma actuator.	42
4.5	Perspective view of the mesh used for the $Re = 8,000$ simulation with the spatial dimensions of the domain labeled.	43
4.6	Boundary layer mesh around the cylinder used for simulation of $Re = 8,000$ case.	44
4.7	Refined boundary layer mesh around the cylinder for simulation of $Re = 8,000$ case.	45

4.8	Sample partitioning of the mesh for parallel computations.	45
4.9	Schematic of the placement of the probes for monitoring pressure and velocity components a) Side view , b) Top view.	46
4.10	Iso-surface plots of vorticity magnitude colored with pressure for the $Re = 8,000$ case, $ \vec{\omega} /D=5$ a) The von Karman Street , b) Zoomed in view of the recirculation region.	47
4.11	Vorticity contours of the recirculation region at $Re = 8,000$ for both experiment and simulation.	48
4.12	Top view of iso-surface plots of vorticity magnitude for the $Re = 8,000$ case. Iso-surfaces are plotted at a single value of $ \vec{\omega} D/U_\infty = 5$ and are colored with pressure.	49
4.13	u velocity contours at 15 levels between values of -1.4 and 1.4 in the mid-plane of the cylinder ($y = 0$).	49
4.14	v velocity contours at 15 levels between -1.4 and 1.4 in the mid-plane of the cylinder ($y = 0$).	49
4.15	w velocity contours at 15 levels between -1.4 and 1.4 in the mid-plane of the cylinder ($y = 0$).	50
4.16	u velocity components measure in the spanwise direction at $0.5D, 1D, 1.5D, 2D$, and $2.5D$ as a function of time for $Re = 8,000$	50
4.17	Schematic of the moving wall process a) Top moving wall activated , b) Bottom moving wall activated.	51
4.18	Geometry of the moving wall located on the cylinder.	52
4.19	Lock-in case u velocity time trace for a) Experiment, b) Simulation with moving wall boundary condition.	54
4.20	Statistical correlation of simulation results for both the unforced case and the lock-in case.	55
4.21	Power spectral density calculation of the u velocity time trace data for the lock-in case ($f/f_o \approx 1.1$).	55
4.22	Cycle of lock-in simulation at certain percentages of the period of the forcing frequency. For 10%-40% the top is on and for 60%-90% the bottom is on.	56
4.23	Aerodynamic coefficients through time for both the unforced and the lock-in cases a) C_l , b) C_d	58

4.24	Power spectral density plot of the aerodynamic coefficient time trace a) Unforced C_l PSD , b) Lock-in C_l PSD.	59
4.25	Average velocity profiles on the top of a cylinder with a moving wall actuator. The left profile is before the moving wall, the middle profile is in the middle of the moving wall, and the furthest right profile is just after the moving wall actuator.	60
4.26	Contours of average pressure with values from -1.2 to 0.5 incremented by 0.1 a) Unforced case b) Lock-in case.	61
4.27	C_p plotted from $x/D = 0.5$ to 3.5 for both the unforced case and the lock-in case.	61
4.28	Lock-in with harmonics case u velocity time trace for a) Experiment, b) Simulation with moving wall boundary condition.	63
4.29	Statistical correlation of both the unforced case and the lock-in with harmonics case using the moving wall approach.	63
4.30	Power spectral density calculation of the u velocity time trace data for the lock-in with harmonics case ($f/f_o \approx 0.6$) with moving walls. . . .	64
4.31	Aerodynamic coefficients through time for both the unforced and the lock-in with harmonics case a) C_l , b) C_d	64
4.32	Power spectral density plot of the aerodynamic coefficient time trace a) Unforced C_l PSD , b) Lock-in with harmonics C_l PSD.	65
4.33	C_p plotted from $x/D = 0.5$ to 3.5 for the unforced, the lock-in, and the lock-in with harmonics cases.	66
4.34	Contours of average pressure for the unforced and lock-in with har- monics cases with contour values from -1.2 to 0.5 incremented by 0.1 a) Unforced b) Lock-in with harmonics case.	67
4.35	Forcing cycle of the simulation for the lock-in with harmonics at certain percentages of the period of the forcing cycle. For 10%-40% the bottom is on and for 60%-90% the top is on.	69

Chapter 1

Introduction

1.1 Overview

The investigation of circular cylinder vortex dynamics has been the subject of many studies over the years because of its simple geometry, complex flow, and many engineering applications [1]. The control of the wake that forms behind a cylinder in certain flow regimes is of particular interest due to the large pressure drag and the oscillating von Karman Street that develops down stream. Flow regimes with Reynolds numbers from 1,000 to 200,000 are known as Shear-Layer Transition Regimes in which there is a laminar region of the flow that separates off the cylinder and transitions to turbulence in the shear layer [1]. The increased drag and oscillating von Karman Street that exist for these Reynolds numbers could potentially have adverse effects in certain applications. Therefore, it has been of interest to control the wake of a cylinder.

Various methods to control the turbulent wake of a cylinder have been implemented and proven effective. One particular method is the use of a dielectric barrier discharge (DBD) plasma-based actuator. These actuators are made up of two offset electrodes separated by a dielectric. As an A.C. voltage is applied to the electrodes, a plasma forms. As the plasma is formed, ions move from the exposed electrode to the surface of the dielectric directly over the second electrode. As the ions are transported, collisions between these ions and the surrounding gas particles occur, inducing a velocity on the external gas.

The act of placing DBD plasma actuators in special locations on a cylinder has proven to be an effective flow control device in recent experimental studies [2, 3, 4]. The actuators can be tuned to alter the shedding frequency and create more coherent

behavior in the span-wise flow structure downstream of the cylinder. In addition to being an effective flow control device, a noted advantage of the plasma actuator is its quick response time.

Because of the effectiveness of these actuators, numerical simulations have been performed to better understand them. In most cases these simulations have treated the actuators as a body force on the flow. Alternative approaches to the body force method have also proven successful. One possible alternative is to replace the actuators by a moving wall boundary condition. The fact that the effects induced by the plasma actuator occur so close to the wall allow it to be numerically simulated with a moving wall boundary condition. The simplicity of the moving wall approximation makes it computationally efficient and easy to implement, and therefore a desirable approach.

Validation of the moving wall boundary condition was achieved by applying it to a flat plate and comparing resulting velocity profiles to experimental results of plasma actuator induced flow. A method for determining an approximate wall velocity is derived from the laminar boundary layer equations. This method was formulated through a similar approach to that taken by Blasius for laminar flow over a flat plate. It is presented as a practical tool for using the moving wall boundary condition for plasma actuator simulations.

With the moving wall boundary condition validated, it is then applied to a circular cylinder in cross-flow at $Re = 8,000$. Using an in-house large eddy simulation (LES) code as the solver, the effects of the moving wall boundary condition on the cylinder flow field are determined. Because the Reynolds number for this flow regime is 8,000, LES is an ideal choice for two main reasons. LES is an unsteady solution technique, so it is able to capture time-accurate results for this unsteady phenomenon. The second reason is the transitional nature of this Reynolds number. On the front of the cylinder the flow is laminar. Near the top and bottom of the cylinder the flow separates and proceeds to transition to turbulence in the shear layers.

With the moving wall boundary condition applied to a cylinder, distinct flow control behavior is seen. This includes the ability to shift the shedding frequency

to the forcing frequency along with increasing the span-wise coherence of the wake. A discussion of why these phenomena exist is offered, including an explanation of the physics behind the forcing frequency harmonics seen in experiments at certain conditions. These insights provided by this simulation give new understanding of the plasma actuator effects on the cylinder wake.

In summary, many researchers have investigated circular cylinder wake dynamics. Through these studies, numerous tactics to control the turbulent wake of a cylinder have been successful. Plasma actuator flow control is one of these strategies. To gain greater insight into the plasma actuator, numerical simulations have been performed, treating the actuator as a body force. A novel alternative for simulating plasma actuator effects is the use of a moving wall boundary condition. The moving wall method has been validated with experimental data. By applying the moving wall procedure to a circular cylinder in cross-flow ($Re = 8,000$), wake control has been achieved. The numerical nature has allowed insight into the physical mechanisms at work.

1.2 Organization

Now that an overview of this research has been presented, the rest of this document will proceed as follows. This chapter will continue by reviewing the current literature available on plasma actuators and cylinder wake flow control. The final section of this chapter will present a succinct discussion of the contributions made by this research. Chapter 2 will cover the large eddy simulation solver that was utilized during this project. This will consist of summarizing the algorithm and reviewing validation cases. Chapter 3 presents the validation of the moving wall approximation for the plasma actuator. Chapter 4 gives results and discussion for applying the moving wall to a cylinder at $Re = 8,000$. In this chapter, two types of forcing regimes seen in experiments (lock-in and lock-in with harmonics) are investigated. Discussion of the flow fields and aerodynamic properties of these regimes are given. Chapter 5 summarizes this research project and states possible future work.

1.3 Literature Review

1.3.1 Cylinder and Other Bluff Body Flow Control

Controlling the wake of a circular cylinder has piqued the interest of many researchers over the years. This interest has produced several methods for achieving turbulent wake flow control. Methods such as oscillating the cylinder, acoustic techniques and even suction techniques have all been employed [5, 6, 7]. Optimal and feedback control techniques have also been investigated [8, 9, 5]. Griffin and Hall [10] present a detailed review of vortex shedding and flow control of the wake behind a bluff body. They define the term “lock-on,” referring to the resonance between the bluff body, or some perturbation, and the vortex shedding frequency. Others have referred to this phenomenon as “lock-in” [2]. Griffin and Hall review various flow control techniques; specifically, the use of oscillating the bluff body or the use of sound to induce lock-in are discussed. Another flow control technique is moving surface boundary layer control (MSBC) [11]. The basic idea of MSBC is to make certain portions of the aerodynamic surface move at some desired velocity, which imparts momentum into the flow. The moving surfaces are created by applying bearing-mounted, motor-driven, hollow cylinders to the body. When applied to various bluff bodies, MSBC causes a delay in separation and significant reduction in pressure drag. Employing a plasma actuator has also proven to be effective in controlling the vortex shedding behind bluff bodies [2, 4, 12, 3]. The use of plasma in fluid flow is documented in a short review article by Fomin [13].

1.3.2 Plasma Actuator

The dielectric barrier discharge (DBD) plasma actuator is made up of two offset electrodes separated by a dielectric. The two electrodes are made of conductive materials, such as copper, with the dielectric separating them. Figure 1.1 illustrates a typical DBD plasma actuator. This schematic elucidates how the dielectric separates the two electrodes, with one electrode being buried under the dielectric and the other

Figure 1.1: Schematic of the Dielectric barrier discharge (DBD) plasma actuator.

offset and exposed to the ambient gas. This configuration is then placed on an aerodynamic surface.

As shown in Fig. 1.1, an A.C. voltage is applied to the electrodes. This generates a plasma on the exposed surface of the dielectric directly above the buried electrode. The formation of plasma causes an induced velocity on the external gas through the collisions of accelerated ions with the external gas particles [14, 15]. For the generation of plasma, appropriate voltage (5-10 kV), and frequency ranges (2-20 kHz) as well as gap distances (on the order of millimeters) must be maintained [16, 15].

1.3.3 Plasma Actuator Flow Control

The plasma actuator has been studied in several different aerodynamic applications. Corke [17] applied the plasma actuator to the upper surface of an airfoil. He recorded an increase in lift through a range of angles of attack up to stall. He also documented an increased drag coefficient. Ramakumar [18] found the actuators had the ability to cause separation reduction on low pressure turbine blades as well as finding lift enhancement on a 3D airfoil. Post [19] also demonstrated separation control on low-pressure turbine blades and airfoils. Roth [14] found that when applied

to a flat plate, the actuators could be configured to produce either significant thrust or drag on the plate.

Another application of the plasma actuator has been to control a circular cylinder turbulent wake [2, 4, 12, 3]. Unique aspects of this application are its ability to function at relatively high Reynolds numbers. Asghar [3] placed two cylinders perpendicular to the flow at the same downstream location with a pitch to diameter ratio of 4, where the pitch is the vertical distance between cylinder centers. That is, when the pitch to diameter ratio is 1 the cylinders are touching each other. He studied this configuration at a Reynolds number range of 8,000 to 41,000. In this study Asghar showed that, when the plasma actuators were placed near $\pm 90^\circ$ from the upstream stagnation point, the vortex shedding off the two cylinders was significantly synchronized. Asghar also reported that the in-phase synchronization and the actuator power input was highly dependent on the Reynolds number. McLaughlin [2] noted that the unforced vortex shedding frequency could be driven to the plasma actuator forcing frequency. This created lock-in behavior similar to results from other control methods. Munzka [4] has shown that the plasma actuator has the ability to produce lock-in up to a Reynolds number as high as 88.0×10^3 . He also reported that the actuators are able to increase the two-dimensionality of the wake across a 30 cm span. Both McLaughlin [2] and Munzka [4] agree that one major strength the plasma actuator has over many other techniques is its capacity to rapidly alter the forcing frequency.

A further noteworthy result of the cylinder application is the ability to control the separation point. In various configurations the plasma actuators are able to delay separation close to, or beyond, the downstream stagnation point. These results give additional evidence to the ability of the actuator to be an effective flow control device [2].

The literature does not contain as many numerical studies as experimental studies of this phenomenon. However, these studies are becoming more common in recent years. In the vast majority of numerical studies available, the plasma actuator is modeled as a body force acting on the fluid. The body force is derived from the

electric properties of the plasma [20, 21, 22, 23]. One of the early body force equations used can be found in Roth [15], and the one dimensional form is,

$$F_b = \frac{d}{dx} \left(\frac{1}{2} \varepsilon_o E^2 \right) \quad (1.1)$$

where E is the electric field, ε_o is the permittivity of free space, and F_b is the body force.

More recently, an alternate body force equation has been presented by Enloe et. al [24]

$$\vec{f}_b^* = - (\varepsilon_o / \lambda_D^2) \varphi \vec{E}. \quad (1.2)$$

In this equation, ε_o is the permittivity of free space, λ_D is the Debye length, φ is the electric potential, $\vec{E}$ is the electric field, and $\vec{f}_b^*$ is the induced electrostatic body force. This formulation has been used by Orlov and Corke [25] in a full numerical simulation that is designed for DNS simulations of flows with plasma actuators and allows for variations in both time and space. It was shown that this body-force approach agreed well with experimental results showing that the thrust resulting from the actuator was proportional to the applied voltage raised to the 7/2 power. Orlov and Corke conclude that this model can be used as an effective tool to optimize actuator designs [25].

The modeling of the plasma as a body-force involves either providing some analytical solution for the plasma induced body force or coupling the governing equations of the flow directly with the electrodynamic equations for the plasma actuator. As an example, Suzen et. al [20] created a numerical model by solving the Navier-Stokes equations with the plasma actuator effects added through a body-force vector from Eq. (1.2). The body force was computed by solving two additional equations; the first results from solving for the electric field that occurs from the A. C. voltage, and the second is for the charge density in place of the ionized air.

One challenge of coupling the governing equations of the fluid with that of the plasma actuator is the computational cost. This coupling is expensive because the characteristic time scales of the plasma are significantly smaller than those of

the surrounding gas [26]. To deal with this type of problem, researchers have taken more approximate approaches. Shyy et. al [22] approached this problem by averaging the effects of the plasma on the flow to create a body force equation that could be coupled with the Navier-Stokes and energy equations. Equation (1.3) shows the x and y components of the body force parameters

$$\begin{aligned} F_{tave x} &= \frac{f_{eff x} \Delta t}{T_t} \\ F_{tave y} &= \frac{f_{eff y} \Delta t}{T_t}. \end{aligned} \tag{1.3}$$

Here, the $f_{eff x}$ and $f_{eff y}$ terms are the effective forces on neutral particles, Δt is the time in which plasma discharge takes place (in Shyy's study this value is $67 \mu s$), and T_t is the time period of the cycle which corresponds the applied voltage period. According to Shyy et. al., this procedure was successful in simulating plasma actuator effects.

1.4 Contribution

This study provides a significant contribution to the general field of fluid mechanics in three main areas. First, the bulk flow produced by a DBD plasma actuator can be simulated by specifying a moving wall boundary condition. This is shown by comparing experimental results of a plasma actuator on a flat plate to simulated results using a moving wall. For this case it is shown that the velocity profile down stream of a plasma actuator can be accurately captured by the moving wall approximation, provided the moving wall velocity injects sufficient momentum. An approximate method for determining the appropriate moving wall velocity was determined. This method was derived from the flat plate boundary layer equations and requires experimental measurements to be used.

The second contribution comes in the application of the moving wall boundary condition to a cylinder in cross-flow ($Re = 8,000$). For this case it is shown that the moving wall approximation alters the turbulent wake of a circular cylinder in cross-flow in a manner similar to plasma actuators. Specifically, the vortex shedding

frequency of the cylinder can be altered, provided that the ratio of the forcing frequency of the moving wall to the unforced vortex shedding frequency of the cylinder is in an appropriate range [2]. Not only does this study simulate the flow control behavior of the plasma actuator, but the numerical nature allows improved understanding of the underlying physics of this phenomenon. In particular, the physical mechanisms behind the harmonics seen in the frequency spectra of the wake are identified, and the effect on lift and drag is investigated

The third area of contribution comes in the further development of an in-house LES code. An additional boundary condition was implemented that enables a moving wall on the curved surface of a circular cylinder to be specified as a boundary condition. This provides the ability to simulate DBD plasma actuators with an accurate LES solver. The code was further developed by advancing point monitoring capability from two-dimensions to three-dimensions. Other contributions to the code were mesh interpolation tools to generate restart files for the solver.

Chapter 2

Flow Solver

In this chapter the turbulent flow solver used in this study will be discussed. First, an overview of the algorithm is presented along with consideration of the solver validation. For a more thorough examination on the details of the solver the reader is referred to Snyder [27, 28]. The second part of this chapter focuses on contributions to the solver as a result of this effort. These include an extra boundary condition to implement a moving wall on a cylinder, the extension of the point monitoring capabilities to three-dimensions and the development of mesh interpolation tools.

2.1 Algorithm Summary

The solver used in this study is a hybrid spectral/finite-element LES flow solver known as SFELES. Figure 2.1 is an example of the type of problem that SFELES can be used to solve. In general SFELES can solve for turbulent flows over two-dimensional or axi-symmetric geometries. It does this by treating the in-plane mesh (as shown in Figure 2.1) by the finite-element method, while using a truncated Fourier series in the transverse direction. This allows the 3D problem to be converted into a number of loosely coupled 2D problems.

The non-dimensional form of the Navier Stokes equations are given by

$$R_{Cont}^{n+1} = \nabla \cdot (\vec{u}^{n+1}) = 0 \quad (2.1)$$

$$\begin{aligned} \vec{R}_{Mom}^{n+1} = & \quad \frac{1}{\Delta t}(\vec{u}^{n+1} - \vec{u}^n) + \nabla p^{n+\frac{1}{2}} - \frac{1}{2Re} \nabla^2 (\vec{u}^{n+1} + \vec{u}^n) \\ & + \frac{3}{2}[\vec{u} \cdot \nabla \vec{u} + \vec{u} \nabla \cdot \vec{u}]^n - \frac{1}{2}[\vec{u} \cdot \nabla \vec{u} + \vec{u} \nabla \cdot \vec{u}]^{n-1} = 0, \end{aligned} \quad (2.2)$$

Figure 2.1: Sample two dimensional geometry with in-plane triangular mesh.

with the superscript n and $n + 1$ being the values at time t and $t + \Delta t$ respectively. In Eqs. 2.1 and 2.2, $\vec{u} = (u, v, w)$ is the velocity vector and p is the pressure. The algorithm requires that the residuals are zero at each time-step. For advancement in time, the solver uses an implicit Crank-Nicholson method for the viscous and pressure terms while an Adam-Bashforth scheme is used for the convective terms.

The finite-element, or in-plane, discretization of the flow quantities at time t can generally be represented by Eq. (2.3). In this equation N_{Nod} is the number of nodes, j indicates a particular node and ϕ_j is the basis function that is associated with node j .

$$q^n(x, y, z) = \sum_{j=1}^{N_{Nod}} q_j^n(z, t) \phi_j(x, y) \quad (2.3)$$

The full discretization of the in-plane domain is accomplished by integrating over the entire domain:

$$\int \phi_i R_{\text{Cont}}^{n+1} d\Omega + \text{PSPG} = 0 \quad (2.4)$$

$$\int \phi_i \vec{R}_{\text{Mom}}^{n+1} d\Omega + \text{SUPG} = 0, \quad (2.5)$$

where PSPG and SUPG represent stabilization terms. These terms are designed to address compatibility and convective instabilities. With the full discretization

performed the terms can be gathered into a matrix equation form

$$\left[M_1 + M_2 \frac{\partial}{\partial z} + M_3 \frac{\partial^2}{\partial z^2} \right] \begin{bmatrix} \Delta p(z) \\ \Delta \vec{u}(z) \end{bmatrix} = \begin{bmatrix} R_p^n(z) \\ \vec{R}_u^n(z) \end{bmatrix}, \quad (2.6)$$

where $R_p^n(z)$ and $\vec{R}_u^n(z)$ represent the residuals for the pressure and velocity terms respectively.

To discretize in the z direction, a truncated Fourier series is employed with periodicity assumed. By symmetry of the Fourier modes, only $N/2$ complex modes are solved. The Fourier transform of a general flow quantity at a given node is given by

$$q_j^n(z) = \frac{1}{N} \sum_{k=-N/2+1}^{N/2-1} \hat{q}_k^n \exp(2\pi I k z / L_z), \quad (2.7)$$

with N being the number of nodes in the z direction and L_z being the periodic length in the z direction.

With the Fourier transform being taken at each node, a matrix equation can again be formulated similar to Eq. 2.6

$$A_k \begin{bmatrix} \Delta \hat{p}_k \\ \Delta \hat{\vec{u}}_k \end{bmatrix} = \begin{bmatrix} \hat{R}_{p,k}^n \\ \hat{\vec{R}}_{u,k}^n \end{bmatrix}, \quad (2.8)$$

with $A_k = [M_1 + M_2(2\pi I k / L) - M_3(2\pi k / L)^2]$.

The 3D problem can then be thought of as a series of loosely coupled 2D problems. The nonlinear $\hat{R}_{p,k}^n$ and $\hat{\vec{R}}_{u,k}^n$ terms are computed in physical space and then transformed to Fourier space, which leads to a pseudo-spectral procedure. The parallelization process partitions the problem in physical space and solves in Fourier space, which results in nearly linear scaling.

2.2 Validation

SFELES has been validated on several classic CFD cases as documented in Ref. [27, 28, 29]. The validation of flow over a cylinder is of particular interest for

this study. A thorough validation of the $Re = 3,900$ case is presented in Snyder et. al [28]. In this article the solver is validated at Reynolds numbers of 190, 300 and 3,900. Although all the details of this validation won't be discussed, a summary of the results for the $Re = 3,900$ case will be presented. An overview of the grid and problem set up used for this validation is shown in Fig. 2.2. The time step for this simulation was fixed at $\Delta t U_\infty / D = 0.001$. Both a coarse and fine grid were used to ensure adequate spatial resolution of the grid. The coarse grid was made of 23,500 in-plane triangle elements, equating to approximately 12,000 nodes. The fine grid consisted of 43,000 triangular elements with 22,000 nodes. For the transverse direction, 32 modes were used; doubling the value suggested from Williamson's work [30]. This results in a total of 380,000 nodes for the coarse mesh and 700,000 nodes for the fine mesh.

Figure 2.2: Mesh and computational domain for SFELES $Re = 3,900$ validation case [28].

Results from this simulation present a strong case for the ability of the solver to simulate flows with sub-critical Reynolds numbers. The first of the results is seen in Fig. 2.3. One major feature in Fig. 2.3 is that the recirculation region has a length of $1.4D$. This corresponds well with experimental results at this Reynolds number. In addition to the recirculation length, one can also see that the von Karman Street has developed downstream of the cylinder.

Figure 2.3: Instantaneous plot of iso-surfaces of vorticity magnitude, $|\vec{\omega}|D/U_\infty$, in the region $-0.5D \leq x \leq 10D$ for flow around a cylinder at $Re = 3,900$ [28].

Additional quantitative results of certain flow quantities were measured and compared to experimental results. These can be seen in Table 2.1. Note that each of the simulation values falls within the measurement error of the experimental values except the drag coefficient.

Table 2.1: Results of SFELES compared with various experimental results. (see Ref. [28])

	$\bar{C}_{D_P}$	$\bar{U}_{min}$	L_f/D	θ_{sep}	St
Snyder and Degrez	1.09	-0.29	1.30	88.0°	0.2179
Experiment	0.99±0.05	-0.24 ±0.1	1.4±0.1	86° ± 2°	0.215±0.005

Another quantitative validation of SFELES is seen from computing mean stream-wise velocity, mean lateral velocity and time-averaged Reynolds shear stress values at various positions in the wake. These results are shown in Fig. 2.4 with the fine grid (solid line) and coarse grid (dashed line) plotted against various experimental and benchmark CFD results.

The Reynolds number 3,900 validation case is of particular interest because this study considers flow around a cylinder at $Re = 8,000$. As Williamson discusses [1], cylinder flows with a Reynolds number ranging between 1,000 and 200,000 have similar flow properties. The flow in this Reynolds number range has been referred

to as the Shear-Layer Transition Regime. In this range the base suction and 2D Reynolds stress levels increase, while the Strouhal number and the formation length of the mean recirculation region decrease with Reynolds number. It can be seen in both the $Re = 3,900$ and the $Re = 8,000$ case that the flow undergoes a laminar to turbulent transition. The flow on the upstream half of the cylinder remains laminar, and separates at approximately 90° from the front stagnation point. Once the flow separates it transitions to turbulence in the shear layers and the ensuing wake is turbulent. This behavior can be seen in Fig. 2.3.

2.3 Moving Wall Boundary Condition

The moving wall boundary condition is designed for a cylinder centered at the origin. The boundary condition takes two parameters, one being the magnitude of the velocity (V) of the moving wall and the other is the frequency (f in Hz) at which the moving wall will be pulsed. In SFELES an edge is specified as a moving wall boundary type. At the first part of the cycle the portion of the edge with positive y values is active and the wall is given a specified velocity, while the edge with negative y values get set to zero velocity. When the second portion of the cycle begins, the part of the edge with negative y values is given a velocity while the portion of the edge with positive y values is given a zero wall velocity. The velocity of the wall is such that it always runs tangent to the cylinder. For the edge portion with positive y values the velocity of the wall runs clockwise, while for edge portions with negative y values the wall velocity runs counter-clockwise. Figure 2.5 shows a schematic of the operation of the moving wall boundary condition.

Figure 2.4: Mean stream-wise velocity (a), mean lateral velocity (b), and time-averaged Reynolds shear stress (c) at different stream-wise locations in the wake at $Re = 3,900$. Results of the coarse mesh are represented by a dashed line and the fine grid is the solid line. The rest are either experimental results or benchmark CFD results (see Ref. [28]).

Figure 2.5: Schematic of the two parts of the moving wall cycle a) First part, b) Second part.

The boundary condition, as mentioned previously, is fixed for a cylinder that is centered at the origin. With this being the case, we can approximate the velocity that is tangent to the cylinder. This is done by determining which node is under consideration and then getting the x and y coordinates (x_j and y_j). The frequency (f), which is read as an input parameter, is converted to an angular frequency (ω) through the simple relation $\omega = 2\pi f$. From x_j and y_j the radius (R) of the cylinder can be computed. This is then used to find the angular velocity by $\alpha = V/R$. To keep track of which side of the cylinder is being actuated, a flag value ($flag$) is continuously calculated by the equation $flag = \sin(\omega t)$. If $flag > 0$ all the locations of the moving wall with $y_j > 0$ get set to the appropriate values using the relations $u = y_j\alpha$, $v = -x_j\alpha$, and $w = 0.0$ (with the velocity vector at the node location being equal to $\vec{u}_j = (u_j, v_j, w_j)$). Also at $flag > 0$, all the velocity components for locations with $y_j < 0$ get set to zero. Now, if $flag < 0$, moving wall edges with $y_j < 0$ get a velocity following $u = y_j(-\alpha)$, $v = -x_j(-\alpha)$, and $w_j = 0.0$. And following a similar procedure as before, when $flag < 0$ all velocity components with $y_j > 0$ get set to zero.

2.4 Flow Field Point Monitoring

SFELES has the capability of monitoring flow quantities through time at any point in the domain. During this project the point monitoring capability was extended from a two-dimensional monitoring tool to a three-dimensional monitoring tool. The algorithm to monitor flow quantities follows a procedure that starts by reading the desired coordinate points into SFELES from an external file. Once the coordinates are read, a procedure of searching and interpolation is followed to determine what the flow quantities are at the desired monitoring point. A summary of the algorithm is presented here.

As an example, suppose SFELES was given the point to monitor (x_p, y_p, z_p) . The point monitor algorithm would first determine which 2-D element contains this point by only looking at x_p and y_p . To determine which in-plane element the point lies in the algorithm uses cross products to calculate the area of triangles made with point p and the vertices of an element (see Fig. 2.6). If point p lies in an element, then the cross product area of all three triangles made from two of the elements vertices and p will have the same sign (see Fig. 2.6). If p does not lie in an element, the areas of triangles made by vertices with p will not have the same sign. By this distinction one can determine if the appropriate element has been found. It is also worth noting that this search occurs for each monitoring point on each of the processors since it is not known a-priori which processor contains the desired element.

Figure 2.6: Three different possible areas that are made by the vertices of the element with the point p , when p lies in the element

Suppose the x and y coordinates for the three nodes on the element surrounding point p were (x_0, y_0) , (x_1, y_1) and (x_2, y_2) . Once the 2-D element is determined

then the two z locations that z_p lies between must be determined. This is a simple search to determine what z_a and z_b satisfy the relationship $z_a \leq z_p \leq z_b$. Through linear interpolation between each node point of the element on plane a with plane b , flow quantities of artificial node points surrounding point p at z_p can be determined. These are seen in Fig. 2.7 as the nodes 0p, 1p, and 2p. Now that the flow quantities for the artificial node points around point p are known, the flow quantities at point p can be computed using

$$q^n(x, y, z) = \sum_{j=1}^{N_{Nod}} q_j^n(z, t) \phi_j(x, y) \quad (2.9)$$

where q_j is the flow quantity at each node of the element (0p, 1p, 2p) and $\phi_j(x, y)$ is the finite element shape function. This function is linear because SFELES utilizes P1/P1 elements for its in-plane finite element computations.

Figure 2.7: How the elements in SFELES appear in the transverse direction with the monitoring point z_p lying between the z_a element plane and the z_b element plane.

2.5 Mesh Interpolating Utilities

To aid in the speed of the calculations, various tools were developed to provide a method for SFELES to restart from an existing solution of either a previous SFELES run on a different grid, or some other solver. The tool is an add-on utility to the post-processing software Tecplot (see Fig. 2.8 for a picture of an add on GUI). Hence, any CFD solution that can be loaded into Tecplot can be used to create a restart file for

SFELES. Because of the amount of time these LES simulations take, to be able to start from an existing solution is very advantageous.

Figure 2.8: Tecplot add-on graphical user interface for creating restart files for SFELES

2.6 Conclusion

In this chapter the turbulent flow solver SFELES was discussed. An overview of the algorithm was first presented along with consideration of the solver validation. Following this, a discussion of the contributions to the solver as a result of this study were considered. These include an extra boundary condition for a moving wall on a cylinder, the extension of the point monitoring capabilities to three-dimensions, and the development of mesh interpolation tools.

Chapter 3

Moving Wall Approximation For Plasma Actuators

In recent years various methods have been used to simulate the DBD plasma actuators. As previously mentioned, the numerical simulations of these actuators have traditionally treated the actuator as a body-force in CFD solvers. Various studies have shown that other modeling techniques for the actuators have promise. In this chapter we will look at a moving wall boundary condition as a way to simulate the bulk effects of a plasma actuator on a flow. We do this by first looking at the results of the moving wall boundary condition applied to a flat plate. To validate the moving wall approximation, these results will be compared to experimental results of a plasma actuator applied to a flat plate. Next, this chapter presents an approach for determining the appropriate moving wall velocity to approximate the plasma actuator. The final part of this chapter presents an example of this approach.

3.1 Validation of Moving Wall Boundary Condition

Roth [31] showed that when a plasma actuator is applied to a flat plate in a quiescent environment, the velocity profile induced downstream can be approximated by a Glauert wall jet [32]. This may be interpreted to mean that as long as the appropriate injection of momentum is maintained by the wall jet, the wall jet can be used to simulate the bulk flow effects of the plasma actuator. Figure 3.1 shows Roth's experimental results plotted against a Glauert wall jet solution.

This example allows one to accept the possibility of alternative modeling techniques for plasma actuators. To understand these possibilities, it is helpful to be aware of some of the plasma actuator details. As mentioned earlier, when an A. C. voltage is applied to the electrodes (see Fig. 3.2) a plasma is formed on the surface

Figure 3.1: Roth’s results of a plasma actuator induced velocity profile plotted against a solution to the Glauert wall jet problem [31].

of the dielectric which, through accelerated ion collisions with gas particles, induces a flow [14, 15].

Figure 3.2: Schematic of the plasma actuator.

With this being understood, one might reasonably ask: What is the spatial dimension of the region directly affected by plasma actuator’s electrostatic force? The characteristic electrostatic shielding length of the plasma is known as the Debye length. In one successful DNS simulation, a Debye length of 0.17 mm was appropriate [25]. This suggests that the plasma actuator’s effect on the fluid happens close to the wall. Roth’s experiment also suggests that the effect on the flow occurs very close to the wall [31]. In Roth’s experiment the velocity profile was measured 15 mm downstream of a 1 mm long actuator. At this 15 mm location, the ensuing height of the jet maximum velocity is still only about 1.5 mm (see Fig. 3.1) from the wall. With this plasma actuator behavior, it is easy to surmise that a “moving wall” boundary condition might suffice for modeling bulk flow effects. The boundary condition of a moving wall is simply a Dirichlet boundary condition. All that is required is to specify a wall velocity as shown in Fig. 3.3.

Figure 3.3: Schematic of a moving wall applied to a flat plate.

To test this theory, experimental results of Corke [17] were compared with simulation results when a moving wall is applied. The setup is a flat plate with a zero freestream velocity ($U_\infty = 0$). Velocity profile measurements downstream of the moving wall were taken of the x velocity component. The CFD simulation uses the grid seen in Fig. 3.4a. This particular grid consists of 37,901 nodes. With the moving

wall length L , the grid extended $3L$ upstream of the moving wall while extending $9L$ downstream. The top boundary is located $8L$ above the moving wall. The solution was computed using a two-dimensional laminar model. Figure 3.4b gives a summary of the boundary conditions used in this simulation. Before and after the moving wall, the velocity (both x and y components) is set to zero at the wall. The top, left, and right portions of the boundary are specified as outlet conditions where the pressure is fixed and the velocities are extrapolated. At the moving wall, an x velocity of 1 is specified. With a moving wall length L equal to 1, the velocity profile measurements were taken at a distance of $0.05L$ after the trailing edge of the moving wall. For validation, non-dimensionalized velocity profiles were compared with those published by Corke. To non-dimensionalize the values, the x velocity component was normalized using the maximum velocity seen in the flow at the location measured. The y distance off the wall was normalized by a “jet thickness” δ . The jet thickness is defined as the y location such that the x velocity component is equal to 5% of the maximum velocity (i.e. $\delta = y$ if $u(y) = 0.05U_{\max}$).

The CFD results presented in Fig. 3.5 match up very well with Corke’s experimental data. This shows that if the moving wall is configured so that the appropriate amount of momentum is injected into the flow, it has the ability to simulate the bulk flow effects of the plasma actuator.

3.2 Determination of Appropriate Wall Velocity

Studies suggest that the plasma actuator has the ability to induce a flow between 3 and 5 m/s [2, 14, 15]. Depending on the configuration of the actuator, velocities induced by the plasma will vary and there is still some uncertainty as to what those values are. For this study there is no claim made that the moving wall can simulate or is even a good approximation of the flow *at the actuator*, only downstream. The validation results of the moving wall show that it requires a velocity significantly higher than the 3 to 5 m/s range seen for actual plasma actuators. This section outlines a method for determining an appropriate wall velocity. The method is summarized in the following outline:

- Compute the mass flow rate from an experimentally measured u velocity profile.
- Use this mass flow rate to determine appropriate constants for the solution of the wall jet.
- With the wall jet solution, compute the momentum flux injected close to the wall jet inlet.
- Solve the laminar boundary layer equations for a moving flat plate in a quiescent environment (i.e. a moving wall boundary).
- Set the momentum flux from the wall jet equal to that added by the moving flat plate.
- Compute the appropriate velocity for the moving flat plate.

Figure 3.4: Computational domain used for simulation of Corke's [17] experimental work, a) Mesh used in simulation (37901 nodes) , b) Schematic of simulation key dimensions.

Figure 3.5: Velocity profiles from a CFD calculation compared with Corke's [17] experimental results of a plasma actuator induced velocity profile.

3.2.1 Reverse Blasius Solution

The well known solution by Blasius [33] solves for laminar uniform flow over a flat plate. If we modify the boundary conditions of this solution, a different flow scenario occurs as seen in Fig. 3.6.

The laminar boundary layer momentum and continuity equations are given by

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (3.1)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = U \frac{dU}{dx} + \nu \frac{\partial^2 u}{\partial y^2}. \quad (3.2)$$

These equations are subject to the following boundary conditions:

$$u(x, 0) = U(x) = U_w$$

$$v(x, 0) = 0$$

$$u(x, \infty) = U_\infty$$

Figure 3.6: Schematic of Blasius solutions to the boundary layer equations with modified boundary conditions.

By imposing the assumptions that $Re \gg 1$ and $U(x) = \text{constant} = U_w$, Eqs. 3.1 and 3.2 adequately model this flow problem. With these boundary conditions and assumptions, we can approach this problem similar to the way Blasius approached laminar boundary layer flow on a flat plate. By letting $\eta = y\sqrt{\frac{U_w}{2\nu x}}$ and taking $\psi = \sqrt{2\nu U_w x}f(\eta)$ we can solve for u and v by

$$u = \frac{\partial \psi}{\partial y} = U_w f'(\eta) \quad (3.3)$$

$$v = -\frac{\partial \psi}{\partial x} = \sqrt{\frac{\nu U_w}{2x}}(\eta f' - f). \quad (3.4)$$

After substituting u and v from Eq. (3.3) and Eq. (3.4) into the governing laminar boundary layer equations and performing some manipulation we arrive at the familiar ordinary differential equation

$$f''' + ff'' = 0. \quad (3.5)$$

The boundary conditions on Eqs. 3.1 and 3.2 can then be translated into conditions on Eq. (3.5). Because as $y \rightarrow 0$, $\eta \rightarrow 0$ and as $y \rightarrow \infty$, $\eta \rightarrow \infty$, the equivalent

boundary conditions are as follows:

$$\begin{aligned} u(x, 0) = U_w &\longrightarrow f'(0) = 1 \\ v(x, 0) = 0 &\longrightarrow f(0) = 0 \\ u(x, \infty) = U_\infty &\longrightarrow f'(\infty) = \frac{U_\infty}{U_w} \end{aligned}$$

With the boundary conditions known and letting $U_\infty = 0$, Eq. (3.5) can be solved numerically and solutions for f , f' , and f'' are shown in Fig. 3.7. To get the f' boundary condition at infinity, an “artificial” boundary condition of $f''(0) = -0.62756257$ is imposed. This particular solution is a similarity solution and the actual u velocity profile at various locations of the moving wall are displayed in Fig. 3.8.

Figure 3.7: Solution to the Blasius solution with reversed boundary conditions.

To support this solution, a computational experiment was performed with the grid presented in Fig. 3.9. This particular mesh is made up of 25351 nodes. The left, right and top sides of this computational domain were given a pressure outlet boundary condition. The bottom boundary was split and given two different

Figure 3.8: u velocity profiles from the Blasius solution with reversed boundary conditions.

boundary conditions: the left side was given a x direction slip wall condition and the right side was given a constant x velocity moving wall condition of 1 m/s.

Figure 3.9: Mesh used for CFD validation of the Blasius solution with reversed boundary conditions.

This particular CFD solution was computed with a two-dimensional laminar steady-state solver. Results from this are compared to the analytical solution by

plotting the velocity profiles at different x locations on the moving wall. These results can be seen in Fig. 3.10. Some variation can be seen due to the assumptions in boundary conditions required by the CFD solution. In the analytical solution the moving wall is considered infinitely long ($U_w = \text{constant}$) where as in the CFD solution the moving wall is finite. Nevertheless, the results compare very well.

Figure 3.10: CFD validation versus the analytical solution of the Blasius solution with reversed boundary conditions.

These results allow us to compute how much x momentum flux is added into the flow at a certain x location of the moving wall. To compute the x momentum flux we can use

$$J = \rho \int_0^\infty u^2 dy. \quad (3.6)$$

Since $u = U_w f'(\eta)$, $\eta = y \sqrt{\frac{U_w}{2\nu x}}$, $\frac{d\eta}{dy} = \sqrt{\frac{U_w}{2\nu x}}$, as $y \rightarrow 0$ $\eta \rightarrow 0$, and as $y \rightarrow \infty$ $\eta \rightarrow \infty$ the integral in Eq. (3.6) can be rewritten as

$$J = \rho \int_0^\infty (U_w f'(\eta))^2 \sqrt{\frac{2\nu x}{U_w}} d\eta. \quad (3.7)$$

This can be simplified to

$$J = \rho U_w^2 \sqrt{\frac{2\nu x}{U_w}} \int_0^\infty (f'(\eta))^2 d\eta. \quad (3.8)$$

By numerical integration, the integral in Eq. (3.8) approximately equals 0.6272. This leads to an equation for the x momentum flux as a function of the moving wall length and velocity.

$$J = 0.6272 U_w \rho \sqrt{U_w 2\nu x} \quad (3.9)$$

In theory this result can be used to match x momentum fluxes of experimental data for plasma actuator velocity profiles (in a quiescent environment) to get an approximate velocity for the moving wall. Because the moving wall is not a good approximation for the plasma actuator in the very near field, a method for patching this momentum flux to the momentum flux at a location downstream is necessary. As mentioned previously, the velocity profile downstream can be modeled by the well known Glauert wall jet [32]. Glauert found a solution to the laminar boundary layer equations when they are applied to the wall jet scenario as pictured in Fig. 3.11. Here we present the basics of the wall jet analysis. For a more thorough treatment of this analysis the reader is referred to Ref. [34, 32].

Figure 3.11: Schematic of the wall jet.

The laminar boundary layer equations with a zero pressure gradient are

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \nu \frac{\partial^2 u}{\partial y^2},$$

with the boundary conditions of $u = v = 0$ at $y = 0$ and $u \rightarrow 0$ as $y \rightarrow \infty$. If we let the streamfunction $\psi = 4x^{1/4}f_\infty f(\eta)$ and $\eta = f_\infty y / \nu x^{3/4}$ (with f_∞ being the asymptote of $f(\eta)$ as $\eta \rightarrow \infty$) then we can substitute the streamfunction into the boundary layer equations through the relationships $u = \partial\psi/\partial y$ and $v = -\partial\psi/\partial x$. Through some mathematical simplifications Eq. (3.10) can be achieved with the boundary conditions $f(0) = f'(0) = f'(\infty) = 0$.

$$f''' + ff'' + 2f'^2 = 0 \quad (3.10)$$

The solution to this equation, $f(\eta)$, is a similarity solution and is valid downstream of the wall jet inlet where similarity applies. The solution to the wall jet essentially gives us the velocity profile as a function of x . Once we have the solution to the ordinary differential equation the mass flow rate can be computed

$$\dot{m} = \rho \int_0^\infty u dy = \rho \psi(\infty) = 4x^{1/4}f_\infty^2. \quad (3.11)$$

With this, the mass flow rate from an experimentally measured velocity profile can be used to compute f_∞ . This then allows the computation of the momentum flux close to the inlet of the wall jet. Assuming that this x location is close enough that losses due to drag are negligible, this momentum value J can be substituted into Eq. (3.9) to compute the approximate moving wall velocity boundary condition.

3.3 Application of Method

This method can now be applied to the experiment by Roth et. al [31]. Roth studied the effects of multiple actuators placed in series on a flat plate in a quiescent

environment. In his study Roth includes data on the effects of a single actuator. Roth's data regarding a single plasma actuator was set up such that the plasma actuator had a length of $L = 1$ mm and the velocity profile measurements were taken 15 mm downstream of the actuator.

To determine what the moving wall velocity should be for a simulation of Roth's experiment, we apply the method just proposed. Because Roth's data does not extend sufficiently high in the y direction, we compute the mass flow rate using the approximate wall jet profile presented in the experimental work. From this an $f_\infty = 0.0353$ is computed from (3.11). With f_∞ known, a value for $f''(0)$ was determined to be $9.85e - 6$. With these key parameters, the velocity profile at a location of $0.1L$ was chosen as sufficiently close to the wall jet inlet to compute the velocity profile. The u velocity profile and the y distance off the wall were computed using

$$u = 12.649 \frac{f_\infty^2 f'}{\nu L^{1/2}} \quad (3.12)$$

and

$$y = 0.1778 \frac{\nu L^{3/4} \eta}{f_\infty}. \quad (3.13)$$

The momentum computed from this profile can be substituted into Eq. (3.9) to give a moving wall velocity of 12.5 m/s.

Now that an approximate moving wall velocity has been determined it can be applied to a CFD simulation. For this case a steady laminar model was used. Figure 3.12a shows the grid used for this study. This grid is made up of 90681 nodes with similar boundary conditions to the simulation used for the Corke study. It extends $5L$ upstream of the moving wall and $40L$ downstream. The vertical distance was set at $10L$ above the moving wall boundary. Figure 3.12b is the schematic illustrating the boundary condition setup at the wall for this simulation. The moving wall length was set to 1 mm with the velocity profile measurement location being 15 mm downstream of the trailing edge of the moving wall.

The velocity profile results are pictured in Figure 3.13. These results show that in order for the moving wall to inject sufficient momentum to accurately approximate

Figure 3.12: Computational domain used for simulation of Roth's [31] experimental work, a) Mesh used in simulation (90681 nodes) , b) Schematic of simulation key dimensions.

the plasma actuator's effects on the flow, a moving wall velocity of 12-13 m/s must be maintained. This confirms both the usefulness of the method to determine the required moving wall velocity as well as again showing the accuracy of the moving wall approximation to simulate the bulk flow effects downstream of the plasma actuator.

Figure 3.13: Moving wall boundary conditions simulation results compared to Roth's [31] data.

3.4 Conclusion

This chapter has explored the idea of simulating the effects of a plasma actuator on a flow field by replacing the actuator with a moving wall boundary condition. The length scales of the plasma give qualitative support to the moving wall method. Validation of the moving wall method was performed by comparing moving wall results against experimental results for two different cases. An approach was developed and presented to determine an approximate wall velocity to sufficiently simulate the effects of the plasma actuator on the flow. These findings show that the moving wall boundary condition has the ability to produce the same bulk flow effects as plasma actuators as long as sufficient momentum is imparted to the flow from the moving wall.

Chapter 4

Circular Cylinder Flow Control at $Re = 8,000$

4.1 Introduction

This analysis references experimental work performed by McLaughlin et. al [2], where plasma actuators were applied to a circular cylinder in cross flow at Reynolds number 7,400. Various configurations of the plasma actuators were studied to analyze their capability to control the vortex shedding of the cylinder.

When discussing flow control of a cylinder wake it is useful to understand something about vortex shedding frequencies. For a cylinder at a Reynolds number of 8,000, the Strouhal number (St) is approximately 0.2. The Strouhal number is a non-dimensional frequency and is defined as

$$St = \frac{f_{\circ} D}{U}, \quad (4.1)$$

where D is a characteristic length, U is a characteristic velocity and $f_{\circ}$ is the frequency at which the vortices are shed off the cylinder body. Hence, in any given flow over a cylinder, the shedding frequency can be determined through

$$f_{\circ} = \frac{StU}{D}. \quad (4.2)$$

Experimental studies have shown that plasma actuators have the ability to control the turbulent wake of a cylinder. This has been shown to be directly related to the ratio $f/f_{\circ}$, where f is the frequency at which the actuator is being forced.

For $f/f_{\circ} \approx 1.1$ the plasma actuators have the ability to shift the shedding frequency from $f_{\circ}$ to f . In addition, the u velocity time trace downstream of the

cylinder at various spanwise locations becomes more coherent, making the flow field more two-dimensional. This particular case has been referred to as lock-in. A u velocity time trace of McLaughlin's data can be seen in Fig. 4.1a. For this particular case the unforced shedding frequency is just above 10 Hz and the forcing frequency is at about 12 Hz. Figure 4.1b shows a power spectral density (PSD) calculation of the time trace revealing that the actuators have shifted the vortex shedding to the forcing frequency.

Figure 4.1: Experimental data for the lock-in case at $Re = 7,400$ a) Downstream u velocity time trace , b) Lock-in PSD [2].

The second type of vortex shedding control observed in the experiments occurred when $f/f_o \approx 0.6$. This case also forced the shedding frequency to the forcing frequency, but with additional harmonics seen in the wake. These types of results have been referred to as lock-in with harmonics. Similar to the lock-in case, more coherent u velocity time traces, as well as power spectral density calculations, can be seen in Fig. 4.2. Notice that the vortex shedding has been shifted to the forcing frequency (6 Hz) and the 2nd and 3rd harmonics are also apparent.

Another significant result from the experimental work is the ability of the plasma actuators to delay separation on the cylinder. In a series of photos from experiments one can see the effects on separation that the lock-in and the lock-in

Figure 4.2: Experimental data for the lock-in with harmonics case at $Re = 7,400$ a) Downstream u velocity time trace , b) Lock-in with harmonics PSD [2].

with harmonics cases have through time. Figure 4.3 shows results of the lock-in case and Fig. 4.4 shows separation control results for the lock-in with harmonics case.

Figure 4.3: Smoke visualization of the shedding off of a cylinder in cross flow for the lock-in case through the period of the forcing cycle of the plasma actuator a) $t=0$, b) $t=0.024$ at 0.24 of the period, c) $t=0.048$ at 0.48 of the period, d) $t=0.072$ at 0.72 of the period [2].

Figure 4.4: Smoke visualization of the shedding off of a cylinder in cross flow for the lock-in with harmonics case through the period of the forcing cycle of the plasma actuator a) $t=0$, b) $t=0.027$ at 0.25 of the period, c) $t=0.054$ at 0.5 of the period, d) $t=0.081$ at 0.75 of the period [2].

These experimental results will be used as a benchmark for the moving wall boundary condition. The nature of the numerical simulation will allow us to explore the physics of the flow providing insight into the mechanisms by which the actuators affect the wake.

4.2 Computational Domain

The computational mesh used for this simulation is pictured in Fig. 4.5. The mesh consists of 32,383 nodes in the 2-D plane. To determine the spanwise mesh resolution required, we look to the results of Mansy et. al [35] and Williamson [30], who both agree that the spanwise structures in the wake of a cylinder are proportional to the Re :

$$\lambda_z/D = \alpha Re^{-1/2} \quad (4.3)$$

where D is the cylinder diameter, λ_z is the structure wavelength, and α is a constant which is approximately between 20-25. At $Re = 8,000$ the spanwise structures will have a wavelength of $\lambda_z/D \approx 0.22$. Since the spanwise direction is treated by a

spectral method at least two grid points per wavelength are required. With $L_z/D = \pi$, where L_z is the spanwise length, the number of spanwise grid points required is 29. For this study 32 grid points were used to insure spanwise spatial resolution. The full three-dimensional mesh consisted of about 990,000 nodes. The cylinder is located $5D$ downstream of the inlet and $20D$ upstream of the outlet. The domain stretches $10D$ above and below the cylinder, with the spanwise dimension set to πD . The inlet condition is given a Dirichlet velocity condition. The outlet consists of a fixed pressure with a Neumann condition for the velocity. The top and bottom of the domain are given slip wall conditions, and a periodic condition is used in the spanwise direction.

Figure 4.5: Perspective view of the mesh used for the $Re = 8,000$ simulation with the spatial dimensions of the domain labeled.

In order to resolve gradients at critical locations in the flow, grid point clustering was used in appropriate locations. The clustering used in the wake region of the cylinder fans out to follow the diffusing behavior of the von Karman Street as

it moves downstream. Clustering was also utilized on the boundary of the cylinder through a boundary layer mesh, as shown in Figure 4.6. The boundary layer mesh on the cylinder started with the first node off the wall being at $0.001D$. The boundary layer mesh growth factor was set to 1.25 with 12 rows for the complete boundary layer mesh. This resulted in a boundary layer mesh depth of 0.0542 and a y^+ value of 1.3 at $\theta \approx 45^\circ$ (with $\theta = 0$ at the upstream stagnation point).

Figure 4.6: Boundary layer mesh around the cylinder used for simulation of $Re = 8,000$ case.

To ensure spatial resolution, a second, more refined, mesh was used. This mesh had 56,187 nodes in the 2-D plane. The spanwise direction was resolved with 64 modes, resulting in a total mesh size of 3.6 million nodes. The boundary layer was also refined and can be seen in Fig. 4.7. The first node off the wall was located at $0.0005D$, with a growth factor of 1.25. By extending the number of rows to 16, the total boundary layer mesh depth was 0.0691. This gave the finer mesh a y^+ value of 0.6924, also measured at $\theta \approx 45^\circ$.

For both the coarse and fine case, the Smagorinsky [36] subgrid-scale model was used and the time step ($\Delta t U_\infty / D$) was set to 0.0005. The mesh was partitioned onto 32 processors, and required about two weeks to compute 400,000 time steps. The

Figure 4.7: Refined boundary layer mesh around the cylinder for simulation of $Re = 8,000$ case.

partitioning of the grid used in the simulation can be seen in Fig. 4.8. The solution proceeded by initializing all the flow quantities with random noise. With this initial flow field, the solver was then allowed to run until the flow field developed. Once this had occurred, data was gathered for analysis.

Figure 4.8: Sample partitioning of the mesh for parallel computations.

To monitor points in the wake of the cylinder in the experiment, 4 probes were placed $3D$ downstream and $1D$ above the cylinder, and were equally spaced in the spanwise direction. In the simulation, monitor points were also located $3D$ downstream and $1D$ off axis of the cylinder. Figure 4.9 shows a schematic of the monitor points for the simulation. One major difference between the simulation and the experiment is that the simulation monitored more points. The simulation monitored approximately 30 points in the spanwise direction, with each point being spaced $0.1D$ apart.

Figure 4.9: Schematic of the placement of the probes for monitoring pressure and velocity components a) Side view , b) Top view.

4.3 Results And Discussion For No Actuator

Results for a cylinder at $Re = 8,000$ can be seen in Fig. 4.10. This figure shows iso-surface plots of vorticity magnitude. Figure 4.10a shows the familiar von Karman Street that has developed downstream of the cylinder. Figure 4.10b gives a closer view of the recirculation region. As can be seen, the recirculation region extends about $0.8D$ from the trailing edge of the cylinder, which is consistent with the experimental observation [1, 37].

Figure 4.10: Iso-surface plots of vorticity magnitude colored with pressure for the $Re = 8,000$ case, $|\vec{\omega}|/D=5$ a) The von Karman Street , b) Zoomed in view of the recirculation region.

Another comparison may be made with the experimental results of Chyu and Rockwell [37], seen in Fig. 4.11. In this figure, very similar behavior in the shear layers is seen between the experiment and the simulation. In both cases one can see the instabilities in the shear layers. This instability is a characteristic of flows at this Reynolds number. Worth noting is that the instabilities in the shear layers are both occurring at similar relative distances downstream of the cylinder. In addition, the size of the recirculation region is validated.

Figure 4.12 shows the top view of the cylinder flow. This is an image of iso-surfaces of vorticity magnitude colored with pressure. The structures in this image are seen to have a spanwise wavelength of approximately $0.2D$ down stream of the cylinder. These results are in compliance with observations of both Williamson [30] and Mansy et. al [35].

Figure 4.13 displays contours of the u velocity component taken at the mid-plane ($y = 0$). The expanse of the recirculation region is displayed very clearly here due to the dark line representing the zero contour. The fluid in the recirculation region does not have high velocities, but small turbulent structures are present. One can also see the extent of the small scale turbulent structures downstream of the recirculation region.

Figure 4.14 shows the contours of the v velocity component taken at the mid-plane. One noticeable feature of this image is the alternating signs of this velocity

Figure 4.11: Vorticity contours of the recirculation region at $Re = 8,000$ for both experiment and simulation a) Chyu and Rockwell experimental results with $|\omega_{min}| = 5s^{-1}$ with contour increments at $\Delta\omega = 5s^{-1}$. Thin lines represent positive contours and thick lines represent negative contours, b) SFELES simulation of z vorticity contours. Vorticity range was $5 \leq |\omega_z| \leq 210$. The contours are incremented at $\Delta\omega_z = 5s^{-1}$. Dotted lines represent negative vorticity and solid lines represent positive vorticity

component. This oscillating positive/negative behavior is caused by the von Karman street that has developed in the turbulent wake. Figure 4.15 shows contours of the w velocity component, again at the mid-plane.

Figure 4.16 shows the u velocity time trace measured at the monitor points discussed earlier. It is appropriate to note the out-of-phase behavior at the different probe points. From this data we can determine the frequency at which the vortices are shedding. This is done through a spectral analysis of the u velocity time trace data. These results were compared to published values and are presented in Table (4.1). This table also presents the SFELES results for the time-averaged drag coefficient on the cylinder at $Re = 8,000$. These results are compared to results published by White [33]. As can be seen in the table both the coarse and the fine mesh values are very close to published values. Also, since the fine mesh has over three and a half times the number of nodes as the coarse mesh and the relative difference in results are small, it is apparent that there is sufficient spatial resolution.

Figure 4.12: Top view of iso-surface plots of vorticity magnitude for the $Re = 8,000$ case. Iso-surfaces are plotted at a single value of $|\vec{\omega}|D/U_\infty = 5$ and are colored with pressure.

Figure 4.13: u velocity contours at 15 levels between values of -1.4 and 1.4 in the mid-plane of the cylinder ($y = 0$).

Figure 4.14: v velocity contours at 15 levels between -1.4 and 1.4 in the mid-plane of the cylinder ($y = 0$).

Figure 4.15: w velocity contours at 15 levels between -1.4 and 1.4 in the mid-plane of the cylinder ($y = 0$).

Figure 4.16: u velocity components measure in the spanwise direction at $0.5D$, $1D$, $1.5D$, $2D$, and $2.5D$ as a function of time for $Re = 8,000$.

Table 4.1: Strouhal numbers and drag coefficients at $Re = 8,000$ for both coarse and fine mesh results from SFELES as compared to published results from White [33].

Case	St	C_d
Coarse	0.214	1.197
Fine	0.212	1.285
White [33]	0.21	1.2

4.4 Moving Wall Actuator

In this simulation, modeling of the plasma actuator effects is accomplished through the use of applying a moving wall boundary condition where the plasma actuator is normally located. For a brief review of the process of how the moving wall boundary condition works, Fig. 4.17 is a reprint from a Chapter 2. As this figure shows, for the first part of the cycle the moving wall on the top portion of the cylinder is active, while the bottom portion is not. Then, on the second half of the cycle, the bottom portion of the cylinder is active, while the top portion is not. Key to the

Figure 4.17: Schematic of the moving wall process a) Top moving wall activated , b) Bottom moving wall activated.

success of this approach is the location of the moving wall. It has been noted by Ashgar and Jumper [3] that plasma actuators need to be located at $\pm 90^\circ$ from the upstream stagnation point for them to be most effective. This placement is close to where separation occurs, and the addition of momentum into the flow is intended to energize the boundary layer. With these guidelines, this study does the same but with a moving wall in place of a plasma actuator. Figure 4.18 gives the geometric setup of the cylinder with moving walls. For this study, the length of each of the moving walls (L_p) is approximately 2% of the circumference of the cylinder.

Figure 4.18: Geometry of the moving wall located on the cylinder.

4.5 Lock-In Case

The lock-in case is expected to occur when $f/f_o = 1.1$. As seen in experiments, two aspects of the flow need to be present for the flow to be considered “locked-in”. The first is that the shedding frequency must be shifted to the forcing frequency. Second, the spanwise time trace of the u velocity components in the wake must be more coherent. If we look at these two aspects of the flow when a moving wall is applied, we can see this behavior occurring. Results from the time trace of the u velocity component and a spectral analysis are presented. In addition, one sees the ability of the flow to stay attached longer, similar to what is seen in the experiments.

By looking at a time trace next to experimental results, a more coherent behavior is apparent. This comparison can be seen in Fig. 4.19. Although it is visually apparent that the simulation produces a more coherent behavior, this is only a qualitative approach. For a more quantitative approach, we take a statistical correlation of all the data at the $z = 0$ point with each spanwise monitor location by using [38]

$$c_j = \frac{cov(u_o, u_j)}{\sqrt{var(u_o)var(u_j)}}. \quad (4.4)$$

In this equation c_j is the statistical correlation value at the spanwise location j , with u_o being the u velocity trace data at the first point in the z direction ($z = 0$) and u_j being the j th point in the spanwise direction. The functions cov and var are the commonly used covariance and variance functions in statistical analysis. A plot of the correlation results of the lock-in case compared to the unforced case can be seen in Fig. 4.20. This figure shows that the moving wall actuator satisfies the criteria of increasing the spanwise coherence of the wake. By taking the PSD of the u velocity time trace data, we are able to see that the shedding has shifted to the forcing frequency. This alteration can best be seen in Fig. 4.21. One notes in this image that the primary shedding frequency is adjusted to the forcing frequency ($f = 0.23$ Hz), but in addition there is another peak. This second peak occurs at a frequency of 0.46 Hz, or one harmonic of the forcing frequency. At first glance one does not see this extra harmonic in the experimental results. It can be argued that

it does exist, however it is just less apparent. There are two reasons for this. First, experiments are subject to more noise than a computer model, thus hiding this extra harmonic. Secondly, spectral plots for the experimental results were computed using about 4,000 data samples, which the simulation spectral plots were computed using about 400,000 data points. Thus, the experimental spectral calculations were not as resolved as the computational results. These two reasons, coupled with the fact that the moving wall produced flow control results to satisfy the lock-in criteria, strongly support the conclusion that the extra harmonic does exist, but was hidden in the “noise” of the experimental results.

Figure 4.19: Lock-in case u velocity time trace for a) Experiment [2] , b) Simulation with moving wall boundary condition.

Figure 4.20: Statistical correlation of simulation results for both the unforced case and the lock-in case.

Figure 4.21: Power spectral density calculation of the u velocity time trace data for the lock-in case ($f/f_o \approx 1.1$).

Figure 4.22: Cycle of lock-in simulation at certain percentages of the period of the forcing frequency. For 10%-40% the top is on and for 60%-90% the bottom is on. a) 10% , b) 20%, c) 30%, d) 40%, e) 60%, f) 70%, g) 80%, h) 90%.

One way to investigate the cause of the extra harmonic is to visualize the shedding of the vortices through time. This also illustrates the moving wall's ability to force the flow to stay attached longer. The results can be seen in the series of images in Fig. 4.22. In this figure the black dot represents the downstream location that was monitored in the flow. One sees a significant difference from the unforced case. When the moving wall is applied at this lock-in frequency, a collision of the shear layers occurs downstream of the cylinder. As the flow is forced to stay attached, it collides into the shear layer on the opposite side (best seen in frame b). The primary shedding frequency observed is obviously shifted to the forcing frequency because the primary flow feature that passes through the control point is the major vortex formed each time the moving wall is activated. However, the instantaneous starting and stopping of the moving wall causes smaller vortices to form, as seen in frame h. This effectively causes a higher frequency to show up because it is seeing two features every shedding cycle.

Finally, we can look at how the moving wall affects the aerodynamic characteristics. Figure 4.23 shows a history of the drag and lift coefficients for both the unforced and the lock-in case. Figure 4.23a shows how the C_l for the lock-in case oscillates with higher amplitudes and has approximately the same mean value as the unforced case. In Fig. 4.23b one immediately sees that the oscillations for the lock-in case are again of higher amplitude and that the average drag coefficient value is higher than the no actuator case.

The PSD calculations of this time trace data give insight into the frequencies at which the aerodynamic variables are oscillating. Figure 4.24a and b show the PSD results for the C_l data for both the unforced and the lock-in case, respectively. These figures give values indicating that the lift coefficient is oscillating at the primary vortex shedding frequency, 0.21 Hz for the no actuator case and at the forcing frequency 0.23 Hz for the lock-in case. This, of course, is in direct correlation with the frequency of vortex shedding. Note that no other harmonics show up for the lock-in PSD of the lift coefficient. This indicates that the 0.46 Hz frequency that shows up downstream in the PSD plots of the u velocity time trace data for the lock-in case is due to

some downstream wake phenomenon and not to the primary vortex shedding of the cylinder.

Figure 4.23: Aerodynamic coefficients through time for both the unforced and the lock-in cases a) C_l , b) C_d .

Statistical averages of aerodynamic data are summarized in Table (4.2). As can be seen in this table, the lock-in case causes a significant increase in the mean drag coefficient. It is also apparent that all drag components (both from friction, C_{xf} , and pressure, C_{xp}) are responsible for the increase.

Table 4.2: Effects of the moving wall lock-in case on the average aerodynamic properties of the cylinder versus the unforced case.

Case	C_{xp}	C_{xf}	C_d	C_{yp}	C_{yf}	C_l
No Actuator	1.16	0.034	1.20	0.005	-0.00055	0.0044
Lock-in	1.35	0.088	1.44	0.015	-0.0018	0.013

These results raise the question: Why are lift and drag coefficients magnitudes and amplitudes of oscillation larger for the lock-in case? We can break this discussion

Figure 4.24: Power spectral density plot of the aerodynamic coefficient time trace a) Unforced C_l PSD , b) Lock-in C_l PSD.

up into two parts: First, the friction drag. For both the lock-in and unforced case the front half of the cylinder will have similar contributions to the drag due to friction. It is when the flow begins approaching the $\pm 90^\circ$ point that differences occur. At this point, the lock-in case has a moving wall at the 90° mark whereas the unforced case has a stationary wall. So, for the no actuator case, the boundary layer grows until separation (close to 90°) at which point the shear stress then decreases rapidly. In contrast, for the lock-in case the moving wall delays separation by an instantaneous step change in velocity (from zero velocity at the stationary wall to the moving wall velocity). Now the boundary layer is energized and the fluid stays attached longer. The moving wall is a very small portion of the cylinder, but will cause some thrust that will effectively cause a reduction on the drag. However, beyond the end of the moving wall, the fluid at the wall returns to zero velocity, producing a significant velocity gradient that results in a higher shear stress. Because of the increased shear stress we find higher values for the C_{xf} and C_{yf} portions of the lift and drag coefficients. Fig. 4.25 shows time averaged velocity profiles on the top portion of the cylinder. This figure has three profiles, one before the moving wall, one in the middle of the

moving wall, and one just after the moving wall. As discussed earlier, notice the different velocity gradients for each of these profiles.

Figure 4.25: Average velocity profiles on the top of a cylinder with a moving wall actuator. The left profile is before the moving wall, the middle profile is in the middle of the moving wall, and the furthest right profile is just after the moving wall actuator.

We can understand the increase in pressure drag by referring to the contour plots of average pressure through time. Figure 4.26 shows both the unforced case and the lock-in case. Note the similar pressure behavior on the front of the cylinder. As we move to the downstream side of the cylinder, one notes that the low pressure zone is located closer to the cylinder. This occurs because the moving wall actuators cause the developing low pressure vortex to stay close to the body of the cylinder. This behavior creates a larger pressure difference between the upstream portion of the cylinder and the downstream portion, which naturally leads to increased drag.

Another way to see this behavior is to view the pressure downstream of the cylinder trailing edge. Figure 4.27 shows the coefficient of pressure (C_p) taken from $x/D = 0.5$ to $3D$ downstream. Here C_p was computed as

$$C_p = \frac{P - P_\infty}{\frac{1}{2}\rho U_\infty^2},$$

where P is the pressure, P_∞ is the reference pressure, ρ is the fluid density, and U_∞ is the freestream velocity. Note the large difference in values in Fig. 4.27 at $x/D = 0.5$. This x location corresponds to the downstream stagnation point of

Figure 4.26: Contours of average pressure with values from -1.2 to 0.5 incremented by 0.1 a) Unforced case b) Lock-in case.

the cylinder. As can be seen, the lock-in case creates a much lower pressure at the cylinder's downstream side, again causing an increase in drag.

Figure 4.27: C_p plotted from $x/D = 0.5$ to 3.5 for both the unforced case and the lock-in case.

4.6 Lock-In With Harmonics

As discussed earlier, for $f/f_o \approx 0.6$ additional harmonics were seen in the experiment [2]. Just as for the lock-in case, the moving wall can be actuated in such a way to produce the lock-in with harmonics effects. In this section we present the results for the lock-in with harmonics case when using a moving wall actuator. Analysis and discussion of the flow field for the lock-in with harmonics case is presented here.

By looking at a time trace next to experimental results, very similar in-phase behavior can be seen. This comparison is shown in Fig. 4.28. Although it is visually apparent that the simulation produces more coherent behavior for the u velocity component, a more quantitative measure can again be attained through a statistical correlation. The procedure is the same as that discussed in the previous section and a plot of the results can be seen in Fig. 4.29. This plot shows the correlation for the lock-in with harmonics, lock-in, and the unforced cases.

Again, by taking the PSD of the time trace data we are able to see that the shedding has been shifted to the forcing frequency ($f = 0.1167$ Hz), as seen in Fig. 4.30. One notes that even through the primary shedding frequency is adjusted to the forcing frequency, there are also other less prominent peaks at frequencies just higher than the primary. It is apparent though, that the large harmonic peaks seen in the experiments are absent. This will be discussed shortly.

Now we will look at how the moving wall for the lock-in with harmonics case affects the aerodynamic characteristics. The time trace of the aerodynamic coefficients is seen in Fig. 4.31 and shows a history of the drag and lift coefficients for both the unforced and the lock-in with harmonics cases. Figure 4.31a shows how the C_l for the lock-in with harmonics case oscillates with higher amplitudes and has about the same mean value as the unforced case. In Fig. 4.23b one immediately sees that the oscillations for the lock-in with harmonics case are also of higher amplitude, but that the average drag coefficient is lower than for the unforced case.

The PSD calculations of this aerodynamic data are displayed in Fig. 4.32. Figure 4.32a and b show the PSD results for the C_l data for both the unforced

Figure 4.28: Lock-in with harmonics case u velocity time trace for a) Experiment [2], b) Simulation with moving wall boundary condition.

Figure 4.29: Statistical correlation of both the unforced case and the lock-in with harmonics case using the moving wall approach.

and the lock-in case respectively. These figures give values that indicate that the lift coefficient is oscillating at about 0.118 Hz for the lock-in with harmonics case. The time history of C_l is of course in direct correlation with the frequency of vortex shedding. Note that on Fig. 4.32b we see another, much smaller but nevertheless distinct, peak. This peak occurs at a frequency of 0.217 Hz, or about at the second harmonic of the forcing frequency. If we refer back to the u trace for the lock-in

Figure 4.30: Power spectral density calculation of the u velocity time trace data for the lock-in with harmonics case ($f/f_0 \approx 0.6$) with moving walls.

Figure 4.31: Aerodynamic coefficients through time for both the unforced and the lock-in with harmonics case a) C_l , b) C_d .

with harmonics case there were some oscillations around this higher frequency value. However, this spectral analysis of C_l clearly demonstrates that at this ratio a harmonic does appear.

Figure 4.32: Power spectral density plot of the aerodynamic coefficient time trace a) Unforced C_l PSD , b) Lock-in with harmonics C_l PSD.

The aerodynamic results are summarized in Table (4.3). As can be seen in this table, the lock-in with harmonics case causes a decrease in the mean drag coefficient.

Table 4.3: Effects of the moving wall lock-in with harmonics case on the average aerodynamic properties of the cylinder versus the unforced case.

Case	C_{xp}	C_{xf}	C_d	C_{yp}	C_{yf}	C_l
No Actuator	1.16	0.034	1.20	0.005	-0.00055	0.0044
Lock-in with harmonics	1.00	0.076	1.08	-0.026	-0.0022	-0.028

As with the lock-in case, the friction part of the drag, C_{xf} , is larger than the unforced case. This, of coarse, is due to the same behavior seen in the lock-in case discussed in the previous section.

The component due to pressure is smaller than the unforced case. Figure 4.33 reproduces Fig. 4.27 but includes data for the lock-in with harmonics case. Note how the C_p for the lock-in with harmonics case is higher than the unforced case right at

the cylinder's downstream side ($x/D = 0.5$). Figure 4.34 shows the pressure contours of the lock-in with harmonics case compared with the unforced case.

Figure 4.33: C_p plotted from $x/D = 0.5$ to 3.5 for the unforced, the lock-in, and the lock-in with harmonics cases.

We now investigate the cause of the harmonic seen in the spectral analysis of the C_l data. This can be done by viewing the series of images in Fig. 4.35. This figure shows the shedding through time using iso-surface plots of vorticity magnitude colored with pressure. One immediately sees significant differences from both the unforced and lock-in with harmonics case. When the moving wall is applied at this lock-in with harmonics frequency, the wake behaves in a very chaotic manner. As one sees in Fig. 4.35a we first see two major vortices shed off both the bottom and top of the cylinder. Note that this is while the bottom actuator is active. After these two vortices have moved downstream, the next major vortex rolls up off the top of the cylinder while there is another smaller vortex that is shed off the bottom the cylinder. Again note that the top actuator is active during this frame. Each time the actuator is on, it creates a large vortex to roll up fairly close to the cylinder. Yet at this same time another vortex is shed off the opposite side. This smaller vortex seems to occur due to the sudden stopping of the moving wall and the shear layer attempting to

Figure 4.34: Contours of average pressure for the unforced and lock-in with harmonics cases with contour values from -1.2 to 0.5 incremented by 0.1 a) Unforced b) Lock-in with harmonics case.

return to its “natural” separated state. Usually these “sub” vortices are smaller than the larger primary vortex shed due to the actuator being active. It should also be noted that each time a moving wall is activated, a smaller vortex is created separate from the primary vortex. In summary, a major vortex develops due to the activated actuator. It develops on the same side as the activated actuator. However, when the actuator switches, smaller vortices are formed in the shear layers on both sides of the cylinder: one from the instantaneous stopping of one actuator and the other from the instantaneous starting of the other. These secondary vortices are small, but are shed twice as often. These secondary vortices therefore appear to be responsible for the harmonic seen in the spectral analysis of the C_l data.

It is not completely clear why the additional harmonics seen in the experiment were not seen in the simulation. However, an explanation is offered. When the moving wall approximation of the plasma actuator was validated, we found that in order to simulate the plasma actuator’s effects, the moving wall needed to have a significantly higher velocity value than what is generally noted for the actual plasma actuators. With this being understood, and since our moving wall for the cylinder case had

a velocity of $4U_\infty$, then it is reasonable to suggest that the moving wall velocity on the cylinder is simply not large enough. It is moving fast enough to create a harmonic, as was seen in the lift data, but the sub-vortices that cause the harmonic are not substantial enough to persist downstream to create a distinct harmonic at the monitor point. Another part of the problem is that because $f/f_o \approx 0.6$ we are much further away from the natural shedding frequency, unlike the lock-in case where $f/f_o \approx 1.1$. This adds more reason why having the additional moving wall velocity is critical to overcome the natural shedding tendency. Higher velocities were attempted in this study. Unfortunately, due to limitations of the solver in its current state, it was impossible to obtain a stable solution with a moving wall velocity higher than $4U_\infty$. The reason for this instability is the use of explicit Adam-Bashforth time stepping for the convective terms. In doing so, a stability limit is placed on the time step size. By increasing the wall velocity to, say, $8U_\infty$, the time step would need to be halved. Considering that each simulation required approximately 3 weeks with 32 processors, it was not feasible to double (or triple) the computing resources. Future efforts on the solver development will introduce more sophisticated time-stepping algorithms to relax the stability limit, but such efforts are beyond the scope of this work.

Even with this limitation, the configuration used in this study was able to impart sufficient momentum to excite a harmonic and produce lock-in. Therefore it was valuable and justifiable to study this case, and it produced a better understanding into the mechanics of why harmonics are occurring.

Figure 4.35: Forcing cycle of the simulation for the lock-in with harmonics at certain percentages of the period of the forcing cycle. For 10%-40% the bottom is on and for 60%-90% the top is on. a) 10% , b) 20%, c) 30%, d) 40%, e) 60%, f) 70%, g) 80%, h) 90%.

Chapter 5

Conclusion

In conclusion, many researchers have investigated circular cylinder wake dynamics. Through these studies, numerous tactics to control the turbulent wake of a cylinder have been successful. Plasma actuator flow control is one of these strategies. Because of the effectiveness of plasma actuators, numerical simulations have been performed to better understand them. In most cases, these simulations have treated the actuators as a body force on the flow. Alternate approaches to the body force method have also proven successful. One possible alternative is the one presented in this study, which is to replace the actuators by a moving wall boundary condition. The fact that the plasma actuator's effects occur close to the wall justify numerical simulation this way. The simplicity of the moving wall approximation makes it computationally efficient and easy to implement and therefore a desirable approach.

This novel alternative has been validated as an accurate approach sufficiently far downstream of the actuator. Validation of the moving wall boundary condition was achieved by applying it to a flat plate and comparing velocity profiles to experimental results of plasma actuator induced flow. A method for determining an approximate wall velocity was derived from the laminar boundary layer equations. It is presented as a practical tool for using the moving wall boundary condition for plasma actuator simulation.

With the moving wall boundary condition validated, it was then applied to a circular cylinder in cross-flow at $Re = 8,000$. Using an in-house large eddy simulation (LES) code for the solver, the effects of the moving wall boundary condition on the cylinder flow field were modeled. Because the Reynolds number for this flow regime is 8,000, LES was a good choice because it captures time-accurate results for this

unsteady phenomenon. In addition, the transitional nature of this Reynolds number makes LES desirable.

An additional boundary condition was implemented into the solver that allows simulation of a moving wall to be applied to the curved surface of a circular cylinder as a boundary condition. This provides a tool for the solver to simulate plasma actuators with an accurate LES solver. The code was further developed by advancing point monitoring capability from two-dimensions to three-dimensions, in addition to mesh interpolation tools to generate more useful restart files for the solver.

The moving wall boundary condition on a cylinder resulted in distinct flow control features. This includes the ability to induce lock-in. That is, to shift the shedding frequency to the forcing frequency and to increasing the spanwise coherence of the wake. It was concluded that for the lock-in with harmonics case, a harmonic was seen because the primary vortices induced sub-vortices at a rate that was twice as fast as the primary shedding. It was also noted that the drag coefficient for the lock-in case went up, while for the lock-in with harmonics case it went down when compared to the unforced case. This was shown to be due to the proximity of the average low pressure region of the primary vortex to the downstream side of the cylinder. It was also noted that the amplitude of fluctuation for both the drag and lift coefficients was higher for all cases when compared to the unforced case. These insights provide new understanding into the plasma actuator effects on the cylinder wake.

This study has also been an excellent beginning for possible future work. The first obvious piece of possible future work would be to increase the stability of the solver to allow for moving walls of higher velocity on the cylinder. It seems fair to hypothesize that if the moving wall was to have a higher velocity one would get stronger secondary vortices. These secondary vortices could potentially be strong enough to induce another subset of vortices and hence excite more harmonics, as were observed in experiments. Another possible future work would be to determine a direct relationship between the plasma actuator parameters and the moving wall boundary condition parameters.

Bibliography

- [1] Williamson, C. H. K., "Vortex Dynamics In The Cylinder Wake," *Annual Reviews of Fluid Mechanics*, Vol. 28, 1996, pp. 477–539. 1, 15, 46
- [2] McLaughlin, T. E., Munska, M. D., Vaeth, J. P., Dauwalter, T. E., Goode, J. R., and Siegel, S. G., "Plasma-Based Actuators for Cylinder Wake Vortex Control," *Proceedings of 2nd AIAA Flow Control Conference*, Portland, Oregon, June 2004, AIAA 2004-2129. 1, 4, 6, 9, 26, 39, 40, 41, 42, 54, 62, 63
- [3] Asghar, A. and Jumper, E. J., "Phase Synchronization Of Vortex Shedding From Multiple Cylinders Using Plasma Actuators," *Proceedings of the 41st AIAA Aerospace Sciences*, American Institute of Aeronautics and Astronautics, Reno, NV, 2003, AIAA-2003-1028. 1, 4, 6, 52
- [4] Munska, M. D. and McLaughlin, T. E., "Circular Cylinder Flow Control Using Plasma Actuators," *Proceedings of the 43rd AIAA Aerospace Science Meeting*, American Institute of Aeronautics and Astronautics, Reno, NV, 2005, AIAA 2005-141. 1, 4, 6
- [5] Cohen, K., Siegel, S., and McLaughlin, T., "Feedback Control of a Cylinder Wake Low-Dimensional Model," *AIAA Journal*, Vol. 41, No. 7, 2003, pp. 1389–1391. 4
- [6] Blevins, R., *Flow Induced Vibrations*, Van Nostrand Reinhold, New York, 2nd ed., 1990. 4
- [7] Ohle, F. and Eckelmann, H., "Modeling of a von Karman vortex street at low Reynolds numbers," *Phys. Fluids A*, Vol. 4, No. 8, 1992, pp. 1707–1714. 4
- [8] Homescu, C., Navon, I. M., and Li, Z., "Suppression of vortex shedding for flow around a circular cylinder using optimal control," *International Journal For Numerical Methods In Fluids*, Vol. 38, 2002, pp. 43–69. 4
- [9] Gillies, E. A., "Low-dimensional control of the circular cylinder wake," *J. Fluid Mech.*, Vol. 371, 1998, pp. 157–178. 4
- [10] Griffin, O. M. and Hall, M. S., "Review–Vortex Shedding Lock-on and Flow Control in Bluff Body Wakes," *Journal of Fluids Engineering*, Vol. 113, 1991, pp. 526–537. 4
- [11] Modi, V. J., "Moving Surface Boundary-Layer Control: A Review," *Journal of Fluids and Structures*, Vol. 11, 1997, pp. 627–663. 4

- [12] McLaughlin, T. E., Felker, B., and Enloe, C. L., "Further Experiments in Cylinder Wake Modification with Dielectric Barrier Discharge Forcing," *Proceedings of the AIAA 44th Aerospace Sciences Meeting and Exhibit*, Reno, Nevada, January 2006, AIAA 2006-1409. 4, 6
- [13] Fomin, V. M., Tretyakov, P. K., and Taran, J.-P., "Flow control using various plasma and aerodynamic approaches (Short review)," *Aerospace science and technology*, Vol. 8, No. 5, 2004, pp. 411–421". 4
- [14] Roth, J. R., Sherman, D. M., and Wilkinson, S. P., "Boundary Layer Flow Control With A One Atmosphere Uniform Glow Discharge Surface Plasma," *Proceedings of the 36th Aerospace Sciences Meeting and Exhibit*, Reno, Nevada, Jan. 1998, AIAA 98-0328. 5, 24, 26
- [15] Roth, J. R., Sherman, D. M., and Wilkinson, S. P., "Electrohydrodynamic Flow Control with a Glow-Discharge Surface Plasma," *AIAA Journal*, Vol. 38, No. 7, 2000, pp. 1166–1172. 5, 7, 24, 26
- [16] Post, M. L., *Plasma Actuators For Separation Control On Stationary And Oscillating Airfoils*, Ph.D. thesis, University of Notre Dame, 2004. 5
- [17] Corke, T. C., Jumper, E. J., Post, M. L., Orlov, D., and McLaughlin, T. E., "Application of Weakly-Ionized Plasmas As Wing Flow-Control Devices," *Proceedings of the 40th AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, 2002, AIAA-2002-350. 5, 25, 27, 28
- [18] Ramakumar, K. and Jacob, J. D., "Flow Control And Lift Enhancement Using Plasma Actuators," *Proceedings of the 35th Fluid Dynamics Conference and Exhibit*, Toronto, Ontario, 2005, AIAA-4635. 5
- [19] Post, M. L. and Corke, T. C., "Flow Control with Single Dielectric Barrier Plasma Actuators," *Proceedings of the 35th Fluid Dynamics Conference and Exhibit*, Toronto, Ontario, June 2005, AIAA 2005-4630. 5
- [20] Suzen, Y. B., Huang, P. G., and Jacob, J. D., "Numerical Simulations of Plasma Based Flow Control Applications," *Proceedings of the 35th Fluid Dynamics Conference and Exhibit*, Toronto, Ontario, 2005, AIAA 2005-4633. 7
- [21] Roy, S. and Gaitonde, D. V., "Multidimensional Collisional Dielectric Barrier Discharge for Flow Separation Control at Atmospheric Pressures," *Proceedings of the 35th AIAA Fluid Dynamics Conference and 36th AIAA Plasma Dynamics and Lasers Conference*, 2005, AIAA 2005-4631. 7
- [22] Shyy, W., Jayaran, B., and Andersson, A., "Modeling of glow discharge-induced fluid dynamics," *Journal Of Applied Physics*, Vol. 92, No. 11, 2002, pp. 6434–6443. 7, 8

- [23] Jacob, J. D., Ramakumar, K., Anthony, R., and Rivir, R. B., “Control Of Laminar And Turbulent Shear Flows Using Plasma Actuators,” *Proceedings of the 4th International Symposium on Turbulence and Shear Flow Phenomena*, Williamsburg, VA, June 2005. 7
- [24] Enloe, C., McLaughlin, T. E., VanDyken, R. D., Kachner, K., Jumper, E. J., and Corke, T. C., “Mechanisms and Responses of a Single Dielectric Barrier Plasma Actuator: Plasma Morphology,” *AIAA Journal*, Vol. 42, No. 3, 2004, pp. 589 – 594. 7
- [25] Orlov, D. and Corke, T. C., “Numerical Simulation of Aerodynamic Plasma Actuator Effects,” *Proceedings of the AIAA 43rd Aerospace Sciences Meeting*, Reno, Nevada, January 2005, AIAA 2005-1083. 7, 25
- [26] Enloe, C., McLaughlin, T. E., VanDyken, R. D., Kachner, K., Jumper, E. J., Corke, T. C., Post, M., and Haddad, O., “Mechanisms and Responses of a Single Dielectric Barrier Plasma Actuator: Geometric Effects,” *AIAA Journal*, Vol. 42, No. 3, 2004, pp. 595 – 604. 8
- [27] Snyder, D. O., *A Parallel Finite-Element/Spectral LES Algorithm for Complex Two-Dimensional Geometries*, Ph.D. thesis, Utah State University and Von Karman Institute For Fluid Dynamics, January 2002. 11, 13
- [28] Snyder, D. O. and Degrez, G., “Large-eddy simulation with complex 2-D geometries using a parallel finite-element/spectral algorithm,” *International Journal For Numerical Methods In Fluids*, Vol. 41, 2002, pp. 1119–1135. 11, 13, 14, 15, 17
- [29] Nierhaus, T., Thomas, J.-F., Detandt, Y., and Abeele, D. V., “Direct numerical simulation of bubbly Taylor-Couette flow,” *Proceedings of the 4th International Conference on Computational Fluid Dynamics*, Gent, Belgium, July 2006. 13
- [30] Williamson, C. H. K., Wu, J., and Sheridan, J., “Scaling of streamwise vortices in wakes,” *Phys. Fluid*, Vol. 7, No. 10, 1995, pp. 2307–2309. 14, 42, 47
- [31] Roth, J. R., Modhan, R. C. M., Ydav, M., and Rahel, J., “Flow Field Measurements Of Paraelectric, Peristaltic, and Combined Plasma Actuators Based On The One Atmosphere Uniform Glow Discharge Plasma (OAUGDP),” *Proceedings of the 42nd AIAA Aerospace Sciences Meeting and Exhibit*, Vol. AIAA-2004-845, Reno, Nevada, 2004. 23, 24, 25, 34, 36
- [32] Glauert, M. B., “The wall jet,” *Journal of Fluid Mechanics*, Vol. 1, 1956, pp. 625–643. 23, 33
- [33] White, F. M., *Viscous Fluid Flow*, The McGraw-Hill Companies, 3rd ed., 2006. 28, 48, 51
- [34] Kundu, P. K. and Cohen, I. M., *Fluid Mechanics*, Academic Press, 2nd ed., 2002. 33

- [35] Mansy, H., Yang, P. M., and Williams, D. R., “Quantitative measurements of three-dimensional structures in the wake of a circular cylinder,” *Journal of Fluid Mechanics*, Vol. 270, 1994, pp. 227–296. 42, 47
- [36] Smagorinski, J., “General circulation experiments with the primitive equations part I: the basic experiment,” *Monthly Weather Review*, Vol. 91, 1963, pp. 99–164. 44
- [37] Chyu, C. and Rockwell, D., “Near-wake structure of an oscillating cylinder: effect of controlled shear-layer vortices,” *Journal of Fluid Mechanics*, Vol. 322, 1996, pp. 21–49. 46, 47
- [38] Weisstein, E. W., “Statistical Correlation,” MathWorld—A Wolfram Web Resource, <http://mathworld.wolfram.com/StatisticalCorrelation.html>. 53