A THESIS

Entitled

Flow Control Around Circular Cylinder: Ventilation holes Method

By

Abdulaziz Mohammed AlRefaie

Submitted as partial fulfillment of the requirements for

the Master of Science Degree in

Mechanical Engineering

Advisor: Dr. Terry Ng

College of Graduate Studies

The University of Toledo

December 2009
The University of Toledo

COLLEGE OF ENGINEERING

I HEREBY RECOMMEND THAT THE THESIS PREPARED UNDER MY SUPERVISION BY Abdulaziz Mohammed AlReafie

ENTITLED Flow Control Around Circular Cylinder: Ventilation holes Method

BE ACCEPTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF Master of Science in Mechanical Engineering

Thesis Advisor: Dr. Terry Ng

Recommendation concurred by

Dr. Douglas Oliver

Dr. Sorin Cioc

Committee On Examination

Dean, College of Engineering
An Abstract of

Flow Control Around Circular Cylinder: Ventilation holes Method

Abdulaziz Mohammed AlRefaie

Submitted as partial fulfillment of the requirements for

the Master of Science Degree in

Mechanical Engineering

The University of Toledo

December 2009

This thesis presents experimental and numerical investigations of flows around a circular cylinder with different ventilation hole shapes and ventilation holes arrangement. Ten models with different ventilation configurations are tested at Re in 55000, and data were acquired for each case at flow angles ranging from 0° to 180° at increment of 2.5°. It has been found that the best results in terms of drag force reduction are models where the holes are located on the sides (shoulders). With the best hole-configuration, the mean drag coefficient is reduced by 33.5% compared to the smooth surface case. Numerical investigations were
conducted using FLUENT on a smooth cylinder and the cylinder with the experimentally-determined best hole-configuration. The CFD results show that the flow through the holes fixes the local separation locations on the cylinder, thereby reduce the flow induced force oscillation. The Strouhal number associated with vortex shedding is reduced from the smooth cylinder.
Acknowledgements

I would like to express my deep thanks and appreciation to my advisor Dr. Terry Ng. His acceptance, advising, patience, and supervision of my work through my long academic journey of pursuing my Master’s degree was a major factor in my success, and the support and knowledge I have gained from him is priceless.

To my colleagues: Yang, Vijaya, Peter, Marco, Chris and Katie. I really enjoy working with you, and your cooperation in using the wind tunnel facility and our discussion and your comments were a great benefit to me.

I thank Dr. Douglas Oliver and Dr. Sorin Cioc for agreeing to be on the thesis committee. I also thank the MIME faculty for their contribution in conveying knowledge to me.
Contents

Abstract iv
Acknowledgements vi
Contents vii
List of Figures ix
List of Tables xi
Nomenclature xii

SECTION ONE - Introduction

1.0 Basic Cylinder Flow and Controls 1
1.1 Regions of Flow Around a Bluff Body 3
1.2 Transition in Unstable Region 4
1.3 Common Sources of Perturbations in Experiments 5
1.4 Current Flow Regime 6
1.5 Transitional Flow 7
1.6 Transition in Shear Layers 8
1.7 Flow Control 8
1.8 Main Objective of the Present Study 9

SECTION TWO - Experimental Models and Wind Tunnel Setup

2.0 Cylinder Models 10
2.1 Model One (M1) 11
2.2 Model Two (M2) 12
2.3 Ventilation Holes Patterns 13
2.4 Experiment Setup in Wind Tunnel 16
2.5 Experiment Condition 18
2.6 Data Acquisition 19
## SECTION THREE - Experimental Observations

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.0</td>
<td>Introduction</td>
<td>21</td>
</tr>
<tr>
<td>3.1</td>
<td>Model One (M1) Observation</td>
<td>21</td>
</tr>
<tr>
<td>3.2</td>
<td>Model Two (M2) Observation</td>
<td>24</td>
</tr>
<tr>
<td>3.3</td>
<td>Models One and Two Comparison</td>
<td>26</td>
</tr>
</tbody>
</table>

## SECTION FOUR - Unsteady Numerical Simulation

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.0</td>
<td>Introduction</td>
<td>28</td>
</tr>
<tr>
<td>4.1</td>
<td>3-D Mesh</td>
<td>29</td>
</tr>
<tr>
<td>4.2</td>
<td>Boundary Conditions</td>
<td>30</td>
</tr>
<tr>
<td>4.3</td>
<td>Modeling Turbulent Flow</td>
<td>31</td>
</tr>
<tr>
<td>4.4</td>
<td>Simulation Results and Discussions</td>
<td>35</td>
</tr>
<tr>
<td>(a)</td>
<td>General wake behaviors</td>
<td>35</td>
</tr>
<tr>
<td>(b)</td>
<td>Drag/Lift Coefficient</td>
<td>40</td>
</tr>
<tr>
<td>(c)</td>
<td>Force Coefficients and Power Spectral Density (PSD)</td>
<td>44</td>
</tr>
</tbody>
</table>

## SECTION FIVE

- Conclusion: 49

References: 51
List of Figures

1.1 Regions of unstable flow 3
1.2 Transitions in unstable regions 4
1.3 Typical unstabilities 7
1.4 Flow control classification and some examples 9
2.1-a Baseline (Top view) 10
2.1-b Baseline (Front view) 10
2.2-a Model 1 (Top view) 11
2.2-b Model 1 (Front view) 11
2.3-a Model 2 (Top view) 12
2.3-b Model 2 (Front view) 12
2.4 Tested cases 15
2.5 Part of the wind tunnel where the cubic test section 16
2.6-a Cylinder assembly scheme 17
2.6-b Plate assembly 17
2.7 Experiment arrangement in test section 18
2.8 Illustration of where models of single sided hole starts and ends (M1&M2) 19
2.9 Illustration of where models of double sided holes starts and ends (M1&M2) 20
3.1 Drag coefficient of Model 1 21
3.2 Lift coefficient of Model 1 23
3.3 Drag coefficient of Model 2 24
3.4 Lift coefficient of Model 2 25
3.5 Drag coefficient of best three cases 26
3.6 Lift coefficient of best three cases 27
4.1-a Top view of the simulated model 28
4.1-b Side view of the simulated model 28
4.2 Geometry and domain mesh 30
4.3 Computational domain and boundary conditions 31
4.4-a Baseline vortex shedding, colored by velocity 36
4.4-b M2P2D vortex shedding, colored by velocity 36
4.5 ZY planes of velocity contours for the wake shapes
   at X =0.2D, X=0.4D, and X=1.0D 37
4.6 Comparison of velocity contours at different flow time
   between base line (left column) and M2P2D (right column) 38
4.7: Flow visualizations showing the wake of the baseline (top)
   and ventilated (bottom) cylinder. 39
4.8-a Baseline path lines colored by velocity 40
4.8-b M2P2D path lines colored by velocity 40
4.9 M2P2D velocity vectors-far filed 41
4.10 M2P2D velocity vectors exiting-near filed 41
4.11 M2P2D at different flow times 43
4.12 Baseline drag coefficient 46
4.13 M2P2D drag coefficient 46
4.14 Baseline lift coefficient 47
4.15 M2P2D lift coefficient 47
4.16 Baseline frequency 48
4.17 M2P2D frequency 48
List of Tables

1.1 Stable flow regimes 7
2.1 Models tested 14
2.2 Models Tested-continued 15
4.1 Summery of FLUENT simulation setting 34
4.2 Simulation results 44
List of abbreviation

M1 ..... Model one
M2 ..... Model two
V ..... Local velocity
V₀ ..... Free stream velocity
β ..... Roll angle
C_D ..... Drag coefficient
C_L ..... Lift coefficient
St ..... Strouhal number
T ..... Period
R_D ..... Reynolds number based on circular cylinder
l ..... Turbulence length
C_μ ..... Empirical constant
ω ..... Turbulence dissipation rate
I ..... Turbulence intensity
k ..... Turbulence energy
SECTION ONE

Introduction

1.0 Basic Cylinder Flow and Controls

The circular cylinder is a classical example of bluff body flow that has been study extensively over the years. A large amount of research has been conducted experimentally and numerically to give better understanding of the basic flow behaviors as well as the control of the flow. The use of passive porosity as a flow control method was developed in the early 1980s, and the control has since been successfully implemented for various applications [1].

Ling and Fang [2] studied the flow around a circular cylinder with surface suction or blowing. They found that if suction was located at the shoulder and the blowing was located at the rear of the cylinder, the asymmetry of the wake vortices could be suppressed. The control also lowered the unsteady lift force. Locating suction at the rear and blowing at the shoulder, however, did not result in the same behaviors.

Takayama and Ako [3] studied the flow round a circular cylinder with different grooves on the surface at $Re_D$ of $1.2 \times 10^5$. Depending on the groove depth, the presence of grooves can affect the flow by transforming the
laminar boundary layer to a turbulent boundary layer and thus delayed flow separation, or promote early separation.

Ozturk, Akkoca, and Sahin [4] studied the flow past a circular cylinder confined by two end plates for Re range of $1,500 \leq \text{Re}_D \leq 6,150$. They found that the flow structure is highly 3-D and very complex, and as the main flow reaches the cylinder a half-saddle of attachment occurs on the surface of the cylinder and horseshoe vortices form.

Nakamura [5] conducted experiments on the vortex shedding from bluff bodies. He found that the Strouhal number is not affected by the details of afterbody shape but by the aspect ratios L/D, with the greater the ratio the smaller the Strouhal number.

Recent research conducted by Patil [6] studied the effects of steady suction on the surface of a circular cylinder with different porosity pattern. The flow wake was studied experimentally using flow visualization and hot-wire measurements, and numerically by solving the Navier-Stokes equations. His study found that the discontinuous perturbations induce vortices in stream direction and suppress vortices spanwise, which result in the reduction or even elimination of vortex-induced vibrations [6].
1.1 **Regions of Flow Around a Bluff Body**

When flow passes a standstill object or an object is moving in a flow at rest, regions of disturbed flow formed around the object as governed by parameters including speed, geometry, and the size of the object [7].

A “bluff body” can have sharp edges on its circumference, like a flat plate normal to the flow, or rounded edges like circular cylinders. Regardless of the shape, the flows around bluff bodies share certain common features. The flow around a circular cylinder is characterized by variation of local velocity in terms of magnitude and direction in time compared to the free stream velocity [7]. Figure 1.1, shows the flow regions around a circular cylinder:

![Figure 1.1: Regions of unstable flow. Cited from Ref.[7]](image)

- One narrow region of retarded flow.
- Two boundary layers attached to the surface of the cylinder.
iii. Two sidewise regions of displaced and accelerating flow.

iv. One wide region downstream of the separated flow called the wake.

1.2 Transition in Unstable Regions

The transition of the flow from laminar to turbulent was first discovered by Reynolds (1883). The transition phenomena that occur in several unstable regimes were first noted by Dryden (1941) and later studied in detail by Roshko and Fizdon (1969)[7]. Figure 1.2, shows the development of transition in several unstable regions at different Re numbers: (a) transition in wakes (TrW); (b) and (c) transition in shear layers (TrSL); and (d) transition in boundary layers (TrBL).

Figure 1.2: Transitions in unstable regions: (a) TrW (b) TrSL (c),(d) TrBL
BL: boundary layer, L=laminar, T: Turbulent,
Tr: transition, S: separation. Cited from Ref.[7]
1.3 Common Sources of Perturbations in Experiments

In practical applications the flow around a circular cylinder is subjected to many types of perturbations. Some classic examples of perturbations are shown in Fig. 1.3. A common one is free stream turbulence (Figure 1.3-a). The surface roughness shape and its depth can also be governing factors (Figure 1.3-b). Wall blockage can restrict the stable flow and increase the adverse pressure gradient (Figure 1.3-c). Blocking the cylinder from one side (asymmetric blockage) could become the dominant influence (Figure 1.3-d). Attaching both ends of the cylinder to the wall will penetrate the boundary layer of the wall and the flow along the span of the cylinder will be affected (Figure 1.3-e). Similar to the single side blockage mentioned earlier, penetrating of the cylinder on the wall boundary layer on one end and leaving the other end free can be an important factor (Figure 1.3-f). Lastly, oscillation in any direction could dominate all other instability sources (Figure 1.3- g and h).
1.4 Current Flow Regime

In the absence of the above perturbations, the transition of the boundary layer is dependent mostly on the Reynolds number and body geometry. Based on the accumulated experimental data around a circular cylinder, the flow regimes can be fully laminar or any of the three transitions or fully turbulent states. The range of $Re_D$ in this research falls in the transition regime (further detail are available in [7]). The table below classifies the flow regimes based on $Re_D$ and the characteristics of drag coefficient, length of near-wake and length of eddy formation.
Table 1.1: Stable flow regimes

<table>
<thead>
<tr>
<th>STATE</th>
<th>REGIME</th>
<th>Re RANGES</th>
<th>$L_w/L_f$</th>
<th>$C_D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>L</td>
<td>LAMINAR</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>NO-SEPARATION</td>
<td>0 to 4-5</td>
<td>NONE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>CLOSED WAKE</td>
<td>4-5 to 30-48</td>
<td>\           \</td>
<td></td>
</tr>
<tr>
<td></td>
<td>PERIODIC WAKE</td>
<td>30-48 to 180-200</td>
<td>\           \</td>
<td></td>
</tr>
<tr>
<td>Tr W</td>
<td>TRANSITION IN WAKE</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>FAR-WAKE</td>
<td>180-200 to 220-250</td>
<td>\           \</td>
<td></td>
</tr>
<tr>
<td></td>
<td>NEAR-WAKE</td>
<td>220-250 to 350-400</td>
<td>\           \</td>
<td></td>
</tr>
<tr>
<td>Tr SL</td>
<td>TRANSITION IN SHEAR LAYERS</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>LOWER</td>
<td>350-400 to 1k-2k</td>
<td>\           \</td>
<td></td>
</tr>
<tr>
<td></td>
<td>INTERMEDIATE</td>
<td>1k-2k to 20k-40k</td>
<td>\           \</td>
<td></td>
</tr>
<tr>
<td></td>
<td>UPPER</td>
<td>20k-40k to 100k-200k</td>
<td>SAME       SAME</td>
<td></td>
</tr>
<tr>
<td>Tr BL</td>
<td>TRANSITION IN BOUNDARY LAYERS</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PRECRITICAL</td>
<td>100k-200k to 300k-340k</td>
<td>\           \</td>
<td></td>
</tr>
<tr>
<td></td>
<td>SINGLE BUBBLE</td>
<td>300k-340k to 380k-400k</td>
<td>(?) \</td>
<td></td>
</tr>
<tr>
<td></td>
<td>TWO-BUBBLE</td>
<td>380k-400k to 500k-1M</td>
<td>(?) \</td>
<td></td>
</tr>
<tr>
<td></td>
<td>SUPERCRITICAL</td>
<td>500k-1M to 3.5M-6M</td>
<td>NONE \</td>
<td></td>
</tr>
<tr>
<td></td>
<td>POSTCRITICAL</td>
<td>3.5M-6M to (?)</td>
<td>(?) \</td>
<td></td>
</tr>
<tr>
<td>T</td>
<td>FULLY TURBULENT</td>
<td>(?) to $\infty$</td>
<td>(?) \</td>
<td></td>
</tr>
<tr>
<td></td>
<td>IN_VARIABLE</td>
<td>(?) \</td>
<td>SAME</td>
<td></td>
</tr>
<tr>
<td></td>
<td>ULTIMATE</td>
<td>(?) \</td>
<td>(?) \</td>
<td></td>
</tr>
</tbody>
</table>

Accumulated experimental data revealed there are three subdivisions or three $Re_D$ ranges for the transitional flow: (1) transition in the wake, (2) transition in shear layer, and (3) transition in the boundary layer. A brief description of transitional flow in shear layer of current flow regime will be presented in the following section, and for the remaining types of flow the reader can refer to Refs. [1] and [7].
1.6 Transition in Shear Layers

When transition happens in the shear layer with the boundary layer remains fully laminar, $C_D$ and $C_L$ remain mostly constant as $Re_D$ increases. Additionally, there is a gradual decrease in the Strouhal number. These characteristics are a result of the developing instability of the separating shear layers from the sides of the circular cylinder [7].

1.7 Flow Control

Flow control refers to any modification on the body or object of interest in an active or passive way that will lead to favorable improvement in its characteristics [1]. In particular, the modifications are intended to change the behaviors of the boundary layer or the free-shear layers, delay or advance separation, reduce drag, enhance lift enhancement, and reduce noise. More details on flow control are presented in Refs. [1], [6], and [9]. A brief classification and some examples of flow controls are illustrated in Figure 1.4.
1.8 Main Objective of the Present Study

The present study will focus on using passive ventilation through discrete holes as a means of controlling the cylinder flow. Different hole geometries and distributions will be investigated using a combination of experiments and computational fluid dynamics (CFD).

Figure 1.4: flow control classification and some examples
SECTION TWO

Experimental Models and Wind Tunnel Setup

2.0 Cylinder Models

Three models are used in this study: (1) Baseline (BL), (2) Model one (M1) and (3) Model two (M2). All models are made of PVC pipes. All models are 23.6 in. (0.6 m) long with 2.4 in. (0.06 m) outer diameter and 2.02 in. (0.051 m) inner diameter. The ventilation holes are 0.25 in. (0.00635 m) in diameter.

Figure 2.1-a: Baseline (Top view)

Figure 2.1-b: Baseline (Front view)
2.1 Model One (M1)

Model 1 is sketched in Fig. 2.2 a and b, the main characteristic of model 1 is that all of the hole axes intersect the axis of the cylinder. The model originally consisted of 264 holes, with 12 lines of ventilation holes evenly distributed (30° apart) around the model circumference and 1.0 in. between every two holes along the length of the model.

Figure 2.2-a: Model 1 (Top view)

Figure 2.2-b: Model 1 (Front view)
2.2 Model Two (M2)

Model 2 is sketched in Fig. 2.3. The model is characterized by the axis of each hole not intersecting with the axis of the model. Twenty two pairs of holes were drilled along the model length as shown in Fig. 2.3. The axis of each hole is drilled 1.0 \textit{in.} from the center of the model.
2.3 Ventilation Holes Patterns

The following notations were used to denote experimental cases. Each notation consists of a combination of five letters and alphabets. The first pair of alpha and letter represents the model number. The second pair represents the hole pattern (from 1 to 4). The last alphabet is: A - all holes on one side open, B - holes open on every other row, or D - all holes on both sides open. Details of the configurations will be discussed in later sections.

![Diagram showing the notation system]
Table 2.1 shows all patterns tested. Different holes patterns were generated on the circular cylinder by covering selected holes using very thin tape and leaving other holes uncovered.

Table 2.1: Models Tested

<table>
<thead>
<tr>
<th>Model</th>
<th>Side</th>
<th>Top β = -90˚</th>
</tr>
</thead>
<tbody>
<tr>
<td>M1P1A</td>
<td><img src="image1.png" alt="Image" /></td>
<td><img src="image2.png" alt="Image" /></td>
</tr>
<tr>
<td>M1P1D</td>
<td><img src="image3.png" alt="Image" /></td>
<td><img src="image4.png" alt="Image" /></td>
</tr>
<tr>
<td>M1P2A</td>
<td><img src="image5.png" alt="Image" /></td>
<td><img src="image6.png" alt="Image" /></td>
</tr>
<tr>
<td>M1P2B</td>
<td><img src="image7.png" alt="Image" /></td>
<td><img src="image8.png" alt="Image" /></td>
</tr>
<tr>
<td>M1P3A</td>
<td><img src="image9.png" alt="Image" /></td>
<td><img src="image10.png" alt="Image" /></td>
</tr>
<tr>
<td>M1P3B</td>
<td><img src="image11.png" alt="Image" /></td>
<td><img src="image12.png" alt="Image" /></td>
</tr>
<tr>
<td>M1P4A</td>
<td><img src="image13.png" alt="Image" /></td>
<td><img src="image14.png" alt="Image" /></td>
</tr>
</tbody>
</table>
Table 2.2: Models Tested-continued

<table>
<thead>
<tr>
<th>Model</th>
<th>pattern</th>
<th>Side</th>
<th>Top $\beta = -90^\circ$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Model 2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>M2P1A</td>
<td></td>
<td><img src="image" alt="M2P1A pattern" /></td>
<td><img src="image" alt="M2P1A Top" /></td>
</tr>
<tr>
<td>M2P1D</td>
<td></td>
<td><img src="image" alt="M2P1D pattern" /></td>
<td><img src="image" alt="M2P1D Top" /></td>
</tr>
<tr>
<td>M2P2D</td>
<td></td>
<td><img src="image" alt="M2P2D pattern" /></td>
<td><img src="image" alt="M2P2D Top" /></td>
</tr>
</tbody>
</table>

Figure 2.4: Tested cases, (sim: means that this case is also simulated)
2.4 Experiment Setup in Wind Tunnel

The University-of-Toledo low-speed tunnel is a closed-loop design that is driven by a 14-blade, variable-pitch fan coupled to a 150 hp electric motor. The test section is 3' x 3' in cross-section. Two tempered-glass side walls and a large Plexiglas window on the ceiling provide access for flow visualizations. Tunnel dynamic pressure and temperature are monitored continuously by a computer during testing. The flow in the test-section is very uniform, with a turbulence level of about 0.5% outside of the wall boundary layers.

As sketched in Figure 2.5, the model is attached vertically to an external platform balance that measures drag and lift. The balance was designed and built during other studies at the University of Toledo [6 and 10].

Figure 2.5: Part of the wind tunnel where the cubic test section
The external balance is attached to a Newport model 495 Power Rotary Stage (turntable) with precision of ±0.001°. The turntable was mounted on the test section ceiling and controlled by a 855 Newport Controller (see Figure 2.6-b).

For this experiment two rectangular end plates were used on the model to reduce effects of the wind tunnel boundary layers. Dimensions of each end plate are the following: thickness of 0.25 in., span of 33 in., and chord of 23.75 in. The model was mounted between the plates as sketched in Figure 2.7. Notice there are small gaps between the model and the top and bottom plates to allow the model to freely respond to the flow.

Figure 2.6-a: Cylinder assembly scheme

Figure 2.6-b: Plate assembly
2.5 Experimental Condition

All experiments were conducted at Reynolds number (Re_D) of 55000 based on the cylinder diameter and free stream velocity of 12.50 m/s. The model was rotated from angle -90° to 90° with increments of 2.5° to 5°. Figures 2.8 and 2.9 show three positions of the cylinders with single-sided and double-sided pattern respectively.
2.6 Data Acquisition

The balance output was acquired using LabVIEW 8.2 software with a sampling rate of 50 Hz and 1000 samples (i.e. the data was averaged over 20 seconds). To confirm the repeatability of the data each case was repeated two to three times.

![Diagram](image)

Figure 2.8: Illustration of where models of single sided hole starts and ends (M1&M2)
Figure 2.9: Illustration of where models of double sided holes starts and ends (M1&M2)
SECTION THREE

Experimental Observations

3.0 Introduction

Results of the experiments will be presented. In particular, mean values of the drag coefficient and the lift coefficient versus angles will be discussed.

3.1 Model one (M1) Observations

Figure 3.1: Drag coefficient of Model 1
Figure 3.1 shows drag coefficient results for all versions of Model One (M1). The lowest drag coefficient among all versions of model 1 is that of M1P1D where the ventilation holes are located at the “shoulders”. The coefficient of the drag decreases to $CD = 0.79$ as compared to the baseline drag value of $C_D = 1.1$ (Baseline) at roll angle ($\beta$) of 0°. Increasing the ventilation holes number on the cylinder surface has no positive effect on the value of the drag (e.g. model M1P2B has $1\frac{1}{2}$ times the ventilation holes of M1P1A and, even though the ventilation patterns are similar otherwise, the drag is higher by $\approx 10\%$.). When the ventilation holes are located farther downstream from the shoulders the positive effect on the drag at $\beta = 0$ reduces, and the drag becomes relatively insensitive to the roll angle.
Figure 3.2 shows the lift coefficients of various versions of model 1 as a function of $\beta$. The highest lift coefficient was produced by M1P2B at $\beta = -65^\circ$ where three lines of holes are located at (60°, 90° and 120°). The lift coefficient increases gradually from $\beta = 0^\circ$ to its peak value at -65° and then decreases until the holes are located at the rear of the model.

For the case of M1P1D that gives lowest drag coefficient at $\beta = 0^\circ$, the maximum lift coefficient of 0.093 occurs at $\beta = -60^\circ$. At this $\beta$, one line of ventilation hole is located at the front stagnation point while the other is located at low pressure area at the shoulder.
3.2 Model Two (M2) Observations

Figure 3.3: Drag coefficient of Model 2

Figure 3.3 shows drag coefficient results for all versions of Model 2 (M2). In comparison to model 1, the results show that the drag of model 2 is more sensitive to the roll angle. For M2P1A the lowest drag occurs at $0^\circ$. The drag coefficient gradually increases to local peaks of approximately 0.9 at $\beta = +35$ and $-35$ degrees, and then decreases at higher magnitudes of $\beta$. For M2P1D the lowest drag also occurs at $0^\circ$ with local drag peaks at approximately $+/50^\circ$ and $+/12.5^\circ$. For M2P2D the drag reaches its peak at $\beta = -35^\circ$ where $C_D = 0.96$. 
Figure 3.4: Lift coefficient of Model 2

Figure 3.4 shows the lift coefficients of various versions of model 2 as a function of $\beta$. The highest lift coefficient was produced by M2P1A at $\beta = -27.5^\circ$. For M2P2D there is a maximum lift of $C_L = -0.16$ at angle $0^\circ$ where the lift coefficient is 0.067 and drag coefficient is 0.73 (see Figure 3.3).
3.3 Models One and Two Comparison

A comparison between the three lowest drag cases: M1P1D, M2P1D, and M2P2D are shown in Figure 3.5 for angles ranging from -30˚ to +30˚. Each of the three configurations produce its lowest drag at $\beta = 0$. In comparison to the other two models, M2P2D produces the lowest drag that increases more gradually away from $\beta = 0$ compared with the other two. The configuration is therefore the most effectiveness in that a significant reduction in drag is maintained over a wide range of flow angles.
Figure 3.6 compares the lift coefficients of the same three cases for the same range of $\beta$ from $-30^\circ$ to $+30^\circ$. The results show that the cases of M2P1D and M2P2D, which can produce significant drag benefits, can produce noticeable lift as the roll angle changes.
SECTION FOUR

Unsteady Numerical Simulation

4.0 Introduction

This chapter presents 3-D numerical solutions of unsteady incompressible turbulent flow around two cases: Baseline and M2P2D using the FLUENT simulation package to solve the unsteady Navier-Stokes equations. A periodic section of the experimental model with a length of 3” and the same outer and inner diameters was used. For the M2P2D configuration, the periodic section as such contains three sets of porous holes as shown in Figures 4.1 a and b. The same Reynolds number of 55000 as the experiment was used, and the simulation were conducted at $\beta = 0^\circ$.

Figure 4.1-a: Top view of the simulated model

Figure 4.1-b: side view of the simulated model
4.1 3-D Mesh

The flow around a circular cylinder has been studied extensively using different 3-D mesh schemes [13]. In this study hybrid mesh of quad map and quad pave schemes have been successfully constructed for the baseline, and hybrid of hex and tetrahedral mesh schemes have been constructed for the M2P2D. The grid used for the M2P2D is shown in Figure 4.2. On the circumference of the baseline cylinder eight rows were placed within the boundary layer, with a first row height of 0.0007525 m and growth factor not exceeding 1.050. There were 400 nodes on the circumference of the cylinder, 36 nodes on the x-coordinate, 24 nodes on Y-coordinate, and 10-nodes on Z-coordinate. For the M2P2D eight rows were for boundary layers, 400 nodes on the circumference of the cylinder, 26 nodes on the circumference of each ventilation hole, 36 nodes on the X-coordinate, 24 nodes on Y-coordinate, and 10 nodes on Z-coordinate.
4.2 Boundary Conditions

The downwind domain extended $20d$ behind the model to capture vortex shedding in the wake area, and for upwind and crossflow the boundaries are located at $10d$. The boundaries of the computational domain were set at sufficiently remote distance from the model to ensure satisfaction of inlet and outlet conditions. The inlet boundary condition on the computational domain is Velocity-Inlet, and in all cases the speed is set at $12.50 \text{ m/s}$. The outlet boundary condition is Pressure-Outlet, and in all cases this is set at a pressure of $101325 \text{ kPa}$. The top and bottom faces of the computational domain are set to be symmetric, since the simulated model is symmetric. Lastly, the side faces are set to periodic boundary conditions.
4.3 Modeling Turbulent Flow

After meshing the geometry in GAMBIT the meshed file was exported to FLUENT and ready to be simulated. While the experimental Reynolds number corresponds to a transition flow, FLUENT™ is implicitly simulating the phenomenon by giving an option for transitional flows correcting when the SST- $k-w$ model is used as a turbulent flow model. Recent research on circular cylinder simulation showed different approaches and different turbulence models [13-24].

In this study the Unsteady Reynolds Averaged Navier-Stokes (URANS) was chosen as the approach since it takes less time and computing resources compared to other approaches [13][22][24].
The *Shear-Stress Transport Model* (SST $k$-$w$) was selected as the turbulence model. SST $k$-$w$ model is a two-equation model that solves the transport equations for $k$ and $w$, where $k$ is the turbulence kinetic energy and $w$ is the turbulence dissipation rate. The SST $k$-$w$ effectively blends the robust and accurate formulation of the $k$-$w$ model in the near-wall region with the free-stream independence of the $k$-$\varepsilon$ model in the far field [13][14]. Unlike other two-equation model SST $k$-$w$ does not involve the damping function, even it is as accurate as any other model in predicting the mean flow profile [24].

In the SST $k$-$w$ model, the Turbulence Kinetic Energy $k$ is given by:

$$ k = \frac{3}{2} (V_{ave} I)^2 $$

Where $I$ is the turbulence intensity $I = 0.16(R_{e_D})^{-\frac{1}{6}}$

And, $R_{e_D} = \frac{DV}{v} = 55000$

The Specific Dissipation rate is given by:

$$ \omega = \frac{k^{\frac{3}{2}}}{c_\mu^{\frac{1}{2}} I} $$

*Empirical constant* $c_\mu = 0.09$

*Turbulence length* $l = 0.07 D$
Strouhal number \( St = \frac{f D}{v} \)

Assumed \( St = 0.19 \) from (Re vs St Chart) based on Re 55000

Thus, the frequency \( f = 39 \) Hz

And, \( t = 0.0255 \) s

The time step \((\Delta t)\) was chosen to be small enough to account for the smallest cell in the computational domain [27]:

\[
(\Delta t) = \frac{\text{Typical cell size}}{\text{Free stream velocity}} \approx 0.00006 \text{ s}
\]

A summary of the simulation settings is given in Table 4.1.
Table 4.1:
A summary of FLUENT simulation setting

<table>
<thead>
<tr>
<th></th>
<th>3-D double precision</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Solver</strong></td>
<td>Pressure based, Unsteady</td>
</tr>
<tr>
<td><strong>Boundary conditions</strong></td>
<td>Inlet $\rightarrow$ velocity</td>
</tr>
<tr>
<td></td>
<td>Outlet $\rightarrow$ pressure outlet</td>
</tr>
<tr>
<td></td>
<td>Sagittal sides $\rightarrow$ periodic</td>
</tr>
<tr>
<td></td>
<td>Transverse sides $\rightarrow$ symmetry</td>
</tr>
<tr>
<td><strong>Viscous scheme</strong></td>
<td>SST-Kw</td>
</tr>
<tr>
<td></td>
<td>*with transitional flows correction option</td>
</tr>
<tr>
<td><strong>Pressure-velocity coupling</strong></td>
<td>PISO</td>
</tr>
<tr>
<td><strong>Time(t)</strong>*</td>
<td>0.0255s</td>
</tr>
<tr>
<td></td>
<td>*Calculated from assumed Strouhal Number of 0.19</td>
</tr>
<tr>
<td><strong>Turbulence kinetic energy (k)</strong></td>
<td>269.398</td>
</tr>
<tr>
<td><strong>Specific Dissipation rate (ω)</strong></td>
<td>0.398</td>
</tr>
</tbody>
</table>
4.4 Simulation Results and Discussions

(a) General wake behaviors

Figures 4.4-a and b shows the vortex structures as revealed by the velocity. For the baseline case the vortex shedding is typical of that behind a smooth cylinder. For the M2P2D case the flows through the ventilation holes appear as striations on the separated shear layers. The shear layers then roll into vortices that are more tilted and farther separated relative to each other compared to the baseline. The vortices also start to become 3-dimensional sooner that the baseline case.

Figure 4.5 shows the wake cross section along the ZY plane at different streamwise locations. At these locations the baseline shear layers remain 2-dimensional and appear as thin sheets. The wake of the M2P2D is 3-D due to the flows through the holes.

Figure 4.6 shows comparisons of the flows at different relative times: 0.0452s, 1.0542s, and 1.6598s. The results show a wider wake for the ventilated case compared to the baseline. It also can be observed that the vortices downstream spaced increasing farther apart compared to the baseline where the converting vortices maintain their alignments with the freestream. Additionally, the entire wake oscillates up and down at a low frequency. The experimental flow visualizations in Fig. 4.7 generally agree with the global flow features reveal by CFD, i.e. as the vortices went by it has been observed they spread out wider comparing to baseline case.
Figure 4.4a: Baseline vortex shedding, colored by velocity

Figure 4.4b: M2P2D vortex shedding, colored by velocity
Figure 4.5: ZY planes of velocity contours for the wake shapes at $X = 0.2D$, $X = 0.4D$, and $X = 1.0D$
Figure 4.6: Comparison of Velocity Contours at different flow time between base line (left column) and M2P2D (right column)
Baseline

M2P2D

Figure 4.7: Flow visualizations showing the wake of the baseline (top) and ventilated (bottom) cylinder.
(b) Detailed flow field

Figure 4.8-a: Baseline path lines colored by velocity

Figure 4.8-b: M2P2D path lines colored by velocity
Figure 4.9: M2P2D velocity vectors-far filed

Figure 4.10: M2P2D velocity vectors exiting-near filed

41
Figures 4.8a and 4.8b show respectively the baseline and M2P2D flows. The flow through the ventilation holes is entrained into the vortices in the wake. The near and far wakes of M2P2D are then modified from the baseline. Figure 10 shows close-ups of the flow around a hole. The orientation of the hole allows a smooth passage of flow without any internal separation, which is possible one of the reasons that model 2 configuration is more effective than model 1 in affecting control.

Figure 4.11 shows a sequence of the near wake flow of M2P2D at a cross-section through the holes. While the wake flow is obviously oscillatory, the flows through the holes apparently fix the separation locations. This behavior is quite different from a smooth cylinder where feedbacks from the downstream vortices cause the separation locations to oscillate.
Figure 4.11: M2P2D at different flow time
(c) **Force Coefficients and Power Spectral Density (PSD)**

Throughout the numerical simulation different parameters of interest were monitored and recorded, and the data are tabulated in Table 4.2: The simulation shows that the drag is reduced, which is in agreement with the experimental results. Additionally, the vortex shedding frequency of the M2P2D case is reduced from the baseline.

<table>
<thead>
<tr>
<th></th>
<th>Baseline</th>
<th>M2P2D</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Experiment</td>
<td>Simulation</td>
</tr>
<tr>
<td>Drag coefficient (mean)</td>
<td>1.1</td>
<td>0.96</td>
</tr>
<tr>
<td>Vortex-frequency</td>
<td>39°</td>
<td>38</td>
</tr>
<tr>
<td>*based on calculation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Strouhal Number</td>
<td>0.19°</td>
<td>0.183</td>
</tr>
<tr>
<td>*based on assumtion</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 4.2 Drag coefficients and frequencies
Figures 12 and 13 show respectively the drag and lift as a function of time. The results show that, additional to reductions of the mean values, fluctuations in both the drag and lift are significantly reduced. The reduction in force fluctuation is a result of the elimination of oscillation of the separation locations as seen in Fig. 4.10. This behavior is further illustrated by the spectra in Figs. 13a and b. In the baseline case there is only one dominant frequency in the frequency spectrum, whereas in the M2P2D case there were three frequency peaks. For the latter case the mid-frequency peak at 30 Hz is identified (through flow visualizations) as one associated with vortex shedding, and the low-frequency peak is associated with the previously observed slow oscillation of the wake. The high-frequency peak is relatively low in magnitude and is likely due to the initial 2-dimensional vortex roll up. The magnitude of the single dominant peak of the baseline case is 1.00e-01, while magnitudes of all the frequency peaks of M2P2D are significantly lower with the maximum being 2.8e-02 at the lowest frequency.
Figure 4.12: Baseline Drag Coefficient

Figure 4.13: M2P2D Drag Coefficient
Figure 4.14: Baseline Lift Coefficient

Figure 4.15: M2P2D Lift Coefficient
Figure 4.16: Baseline frequency

Figure 4.17: M2P2D frequency
SECTION FIVE

Conclusions

The use of passive ventilation through discrete holes as a means of controlling the cylinder flow was studied using a combination of experiments and computational fluid dynamics (CFD). The following can be concluded from the results.

- The flow around the circular cylinder with different ventilation holes configuration was investigated. The experimental results are in good agreement with numerical solution results.
- With the best hole configuration tested, the mean drag coefficient is reduced by 33.5% compared to the smooth surface case.
- The best location of ventilation holes appears to be on the cylinder sides (shoulders).
- The hole-shape, location and the angle of the flow are all factors leading to the most effective result.
- Flows through the holes fix the local separation locations on the cylinder, thereby reduce the flow induced force oscillation.
• For the M2P2D case the Strouhal number associated with vortex shedding is reduced from the smooth cylinder. Additionally, a low-frequency oscillation of the wake is induced by the control.
References


5. **Y. Nakamura** “Vortex Shedding From Bluff Bodies And a Universal Strouhal Number” J of Fluids and Structure, 10, 159-171, 1996.


