Cylinder drag reduction by geometric modification in 3-Diameter confined space

Towards speed skiing application

Jaydeep Koradiya
Cylinder drag reduction by geometric modification in 3-Diameter confined space

Towards speed skiing application

by

Jaydeep Koradiya

to obtain the degree of Master of Science
at the Delft University of Technology,
to be defended publicly on Wednesday April 25, 2018 at 2:00 PM.

Student number: 4516125
Project duration: April 1, 2017 – April 25, 2018
Thesis committee: Dr. ir. B. W. van Oudheusden, TU Delft, Chair committee
Dr. A. Sciacchitano, TU Delft, Supervisor
Ir. WA. Timmer, TU Delft, Wind energy
Ir. W. Terra, TU Delft, Ph.D candidate

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

Cylinder flow has been extensively studied for decades due to its numerous direct and indirect relevance to the real-life applications. Being a bluff body, the cylinder experiences flow separation with large wake leading to significant drag force thus the drag reduction on the cylinder is sought. In the long-running history of cylinder flow, drag reduction is achieved by geometric modification, active/passive flow control techniques.

The aim of this research is to investigate the drag reduction on the cylinder with the help of geometric modification and passive flow control devices. The geometric modifications are restricted to be made only in the downstream direction from the cylinder leading point within the constrained space of 3 diameters. The constraints are inspired from speed skiing fairing design and other applications where a complete aerofoil cannot be fitted due to limited available space. The passive flow control devices like zigzag strip and vortex generator are implemented on the modified geometry to seek further drag reduction.

According to O’Connor et al. [27], enveloping the cylinder by an aerofoil is the most effective geometric modification for drag reduction. In the scope of this work, two semi-circular nose shaped aerofoils are designed to inscribe the cylinder at their maximum thickness. The aerofoils are designed using analytical solution from XFOIL. The models are designed such that the length-to-thickness ratio is equal to 3. Models which include wings, a cylinder-splitter plate assembly and the reference cylinder are tested for drag in the wind tunnel at three different velocities and various angles of attack. Traditionally, the drag quantification on an object is performed via balance measurement which yields the force acting on the object without revealing the causes behind drag generation. Another means of drag quantification is the wake survey using techniques like Hot-wire anemometry and Pitot-static tube measurements. In this research, Particle Image Velocimetry (PIV) technique is used for flow measurement which reveals entire flow field with different flow structures. Pressure from the PIV data is computed by solving Poisson’s equation. Thereafter, wake analysis study including momentum and pressure deficit is performed for drag quantification.

Zigzag strips are effective in tripping the boundary layer and delaying/avoiding flow separation, thus reducing total drag acting on the geometry. The application of zigzag strips on the cylinder achieves 35% drag reduction. Whereas the cylinder-splitter plate assembly with zigzag strips experiences 50% less drag compared to the smooth cylinder. The highest drag reduction of 92% is achieved by envelopment of an aerofoil around the cylinder. Two aerofoils designed in this project have different trailing edge thickness which leads to a difference in the drag force experienced by them and their operational angles of attack range without flow separation. Vortex generators along with the zigzag strips are used to extend this operational range.

These aerofoil designs are useful for drag reduction in a various practical application with high Reynolds number flow; especially in a situation where geometric modification is restricted in an upstream direction. To name few applications, it can be used to make an aerodynamic fairing around the legs of a skier, around a wind turbine tower etc. Upstream wind turbines have limited space upstream the tower due to the presence of the rotor. Implementation of aerodynamic fairing not only reduces the drag on the tower but also limits the wake size of the tower. Thus, installation of wind turbines with smaller intermediate distance can be made possible. Also due to smaller wake size and reduced wake flow fluctuations, downstream wind turbines can be encouraged.
Acknowledgement

It was 20th August 2015, when I landed in the Netherlands for the first time. After two and half swift years, here I stand on the verge of graduation. This master thesis concludes my education in Aerospace engineering at TU Delft. Before I set out on a new journey, I would like to take this moment to thank all the people for investing their time and energy in making this happen.

I would like to thank my supervisor Dr Andrea Sciacchitano for the proposal of this interesting project. His engaging lectures on the ‘Flow Measurement Technique’ are the major reason behind choosing a thesis involving experimental aerodynamics. His patience and advice are deeply appreciated. A special thanks to Dr Daniele Ragni for all the guidance in pressure reconstruction. Furthermore, I would like to thank the technical staff of Aerodynamics Lab, Nico van Beek, Frits Donker-Duyvis, Peter Duyndam and Stefan Bernardy for the support during the experimental campaign.

The studies would have been exhaustive without friends. A special thanks to all the fellow students from the basement for engaging discussion and refreshing coffee breaks. I thank Chinmay, Greeshma, Ram, Hugo, Niko, Bart, Salil, Abhimanyu, Abdullah, Sumanth, Shashwat, Johnathan, Tomi, Felipe and the whole Schiedam gang for keeping my spirit up.

I thank all my favourite people back home in India. Most importantly, none of these would have been possible without the constant support of my parents. I could never thank you enough but this is my humble attempt. I hope I have made you proud.

Jaydeep Koradiya
Delft, April 2018
# Contents

## List of Figures

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ix</td>
</tr>
</tbody>
</table>

## 1 Introduction

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
</tr>
</tbody>
</table>

## 2 Research background

<table>
<thead>
<tr>
<th>Subsection</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1 Cylinder flow</td>
<td>5</td>
</tr>
<tr>
<td>2.2 Cylinder with a splitter plate</td>
<td>8</td>
</tr>
<tr>
<td>2.3 Drag reduction techniques used in the past</td>
<td>12</td>
</tr>
<tr>
<td>2.4 Passive flow control techniques</td>
<td>15</td>
</tr>
<tr>
<td>2.4.1 Zigzag strip</td>
<td>15</td>
</tr>
<tr>
<td>2.4.2 Vortex generators</td>
<td>15</td>
</tr>
<tr>
<td>2.5 Research objective and questions</td>
<td>16</td>
</tr>
<tr>
<td>2.6 Particle Image Velocimetry (PIV)</td>
<td>17</td>
</tr>
<tr>
<td>2.6.1 Tracing particles</td>
<td>18</td>
</tr>
<tr>
<td>2.6.2 Illumination</td>
<td>19</td>
</tr>
<tr>
<td>2.6.3 Imaging</td>
<td>20</td>
</tr>
<tr>
<td>2.6.4 Evaluation of the particle motion</td>
<td>21</td>
</tr>
<tr>
<td>2.7 Load determination from PIV and Control surface approach</td>
<td>22</td>
</tr>
<tr>
<td>2.8 Pressure reconstruction using velocity field</td>
<td>24</td>
</tr>
</tbody>
</table>

## 3 Model design & Experimental setup

<table>
<thead>
<tr>
<th>Subsection</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1 Model design</td>
<td>29</td>
</tr>
<tr>
<td>3.2 Experimental setup</td>
<td>33</td>
</tr>
<tr>
<td>3.2.1 Wind tunnel, Test section and Model mounting</td>
<td>33</td>
</tr>
<tr>
<td>3.2.2 Seeding particles and Seeder</td>
<td>33</td>
</tr>
<tr>
<td>3.2.3 Laser</td>
<td>34</td>
</tr>
<tr>
<td>3.2.4 Field of view and Recording device</td>
<td>34</td>
</tr>
<tr>
<td>3.2.5 Pressure measurement setup</td>
<td>36</td>
</tr>
<tr>
<td>3.3 PIV Data acquisition</td>
<td>36</td>
</tr>
<tr>
<td>3.4 PIV data processing</td>
<td>37</td>
</tr>
<tr>
<td>3.5 Flow control devices used</td>
<td>39</td>
</tr>
<tr>
<td>3.5.1 zigzag strips</td>
<td>39</td>
</tr>
<tr>
<td>3.5.2 Vortex generators (VG)</td>
<td>39</td>
</tr>
</tbody>
</table>

## 4 Data reduction techniques

<table>
<thead>
<tr>
<th>Subsection</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1 Pressure reconstruction and the boundary conditions</td>
<td>41</td>
</tr>
<tr>
<td>4.2 Drag computation</td>
<td>44</td>
</tr>
<tr>
<td>4.3 Drag uncertainty analysis</td>
<td>45</td>
</tr>
<tr>
<td>4.3.1 Uncertainty of the mean velocity ($U_{mean}$)</td>
<td>45</td>
</tr>
<tr>
<td>4.3.2 Uncertainty of the Reynolds normal stress ($U_{\overline{u'w'}}$)</td>
<td>47</td>
</tr>
<tr>
<td>4.3.3 Uncertainty in time-averaged pressure computation ($U_{\overline{p}}$)</td>
<td>47</td>
</tr>
<tr>
<td>4.4 Grid independence study</td>
<td>48</td>
</tr>
<tr>
<td>4.5 Free stream velocity correction</td>
<td>49</td>
</tr>
</tbody>
</table>
5 Results and discussion

5.1 Drag quantification on the reference geometry ........................................ 51
  5.1.1 Trend analysis of drag contributing terms for cylinder: ....................... 51
5.2 Effect of surface roughness on cylinder drag ........................................... 53
  5.2.1 Flow field comparison for smooth and rough cylinder for $u_\infty = 27.5 \text{m/s}$ . . 55
5.3 Effect of splitter plate on cylinder drag ................................................... 58
  5.3.1 Comparison between cylinder A and cylinder B at $u_\infty = 27.5 \text{m/s}$ ........ 58
  5.3.2 Effect of roughness on cylinder-splitter plate assembly ...................... 59
5.4 Wing A .............................................................................................. 61
  5.4.1 Wing A at non-zero angles of attack ................................................. 63
5.5 Wing B .............................................................................................. 66
  5.5.1 Wing B at zero angle of attack with three different free stream velocities .... 66
  5.5.2 Drag determination on wing B at non-zero angles of attack ............... 69
5.6 Minimum drag comparison for the geometries ....................................... 72

6 Conclusions and Recommendations ......................................................... 75
  6.1 Conclusions ......................................................................................... 75
  6.2 Recommendations and future work ...................................................... 77

Bibliography ......................................................................................... 79
List of Figures

1.1 Skier Joost Vandendries (Belgium) in action. Reproduced from: www.skiinfo.nl .......................... 1
1.2 Free body diagram of a skier. Reproduced from: Janura et al. [18]. ........................................... 2
1.3 Qualitative illustration of pressure and skin friction drag contribution in case of bluff and streamline body. Reproduced from: John D. Anderson [19]. ........................................... 2
1.4 The source regions of pressure and friction drag. Largest source region is indicated by colour Red and the smallest source region is by colour Blue. Reproduced from: Airshaper [3]. ........................................... 3
1.5 The flow speed is 40 m/s from left to right. Reproduced from: Asai et al. [6]. ............................. 3
1.6 Design layout of fairing. Reproduced from: The International Ski competition rules (ICR) [44] .... 4
2.1 Chapter road map. .................................................. 5
2.2 Turbulent wake behind a cylinder at $Re = 10,000$. Large separated wake (Red arrow) and free shear layer (Green arrows) are highlighted (reproduced from Van Dyke [48]). 6
2.3 Plot showing dependence of $C_d$ on $Re$. Data from Wieselsberger, Delany and Sorenson, Finn, Tritton, and Roshko. Data compilation by (Panton [29]). Reproduced from: (Panton [29]) .................................................. 6
2.4 Schematics of the different flow regime around a circular cylinder. Reproduced from: (Houghton and Carpenter [16]) .................................................. 7
2.5 Base pressure coefficient($C_{pb}$) vs splitter-plate length $(l/h)$. $\times$, $Re = 1.4 \times 10^5$ and $\circ$, $Re = 2.45 \times 10^5$. Image taken from (Bearman [7]) .................................................. 8
2.6 Percentage velocity fluctuation($\theta$) traverses along y-axis (normal to the wake) at different downstream locations at $Re = 1.45 \times 10^5$. Image taken from (Bearman [7]) .................................................. 9
2.7 Percentage velocity fluctuation($\theta$) traverses downstream the blunt trailing edge at $y/h = 0.25$. The experiment is carried out at $Re = 1.4 \times 10^5$ using different splitter-plate length. Image taken from (Bearman [7]) .................................................. 10
2.8 Percentage velocity fluctuation($\theta$) obtained from traverses along y-axis (normal to the wake) at different downstream locations at $Re = 1.45 \times 10^5$ for plate length $(l/h)$ ranging from 0 to 4. $\times$, position of maximum $\theta$. Image taken from (Bearman [7]) .................................................. 10
2.9 Wake centreline pressure with and without splitter plate at $Re = 14,500$. Reproduced from: (Roshko [34]) .................................................. 11
2.10 $C_d$ vs $Re$ for varying splitter plate length. Reproduced from: (Apelt et al. [5]) ......................... 11
2.11 (a) The flow visualisation around a cylinder in presence of the upstream control rod. (b) The pressure coefficient on the cylinder surface in terms of angles measured from the upstream stagnation point; in absence of the control rod. Other symbols: different upstream distance of control rod. Reproduced from Lee et al. [23] .................................................. 12
2.12 The time-averaged streamlines for the unsteady flow at $Re = 100$: (a) $G/d = 2.0$, (a) $G/d = 2.6$ and (a) $G/d = 2.7$. $G$ is the distance between the cylinder base point and leading edge of downstream plate, $d$ is the diameter of the cylinder and length of the plates. Reproduced from Hwang and Yang [17] .................................................. 13
2.13 Illustrative shroud design around a cylinder using NACA0033. Reproduced from: O’Connor et al. [27] .................................................. 13
2.14 Drag coefficient at $\alpha = 0^\circ$ for a cylinder, ellipse, and symmetric NACA aerofoils ranging from NACA0012 to NACA0050, where D is the maximum circle diameter than can be inscribed in a given shape and c is the streamwise length. (ellipse data from Blevins [8], and reduced drag data from Lee et al. [23], Hwang and Yang [17] and Triyogi et al. [46]). Reproduced from: O’Connor et al. [27].

2.15 (a) Baseline geometry NACA0033 with trimmed front (b) Baseline geometry added with a linear cuff decreasing from a certain length at y=0 to zero at y=maximum thickness. Reproduced from: O’Connor et al. [27].

2.16 (a) Sketch of a zigzag strip, (b) Mean velocity flow field downstream the zigzag strip. The blue coloured contour indicates the back flow and green contour represents the vortical motion as detected by Q-criterion. Reproduced from: Elsinga and Westerweel [12].

2.17 Sketch of a counter-rotating vane-type vortex generator. Reproduced from: Lin [24].

2.18 Schematic diagram showing PIV set-up. (Reproduced from: Raffel et al. [31])

2.19 Scattering light intensity as a function of the scattering angle for a 1 $\mu$m oil droplet in air. (Reproduced from: Raffel et al. [31]).

2.20 Schematic diagram showing synchronisation between the laser pulses and camera acquisition.

2.21 Schematic diagram showing PIV imaging.

2.22 Particle displacement determination using cross-correlation. Reproduced from Sciacchitano [37].

2.23 (a) PIV setup schematic diagram (b) Vector field in the wake of mannequin. Reproduced from: Terra et al. [42].

2.24 Schematics of control surface approach.

2.25 PIV integrated pressure (Left) and RANS computed pressure (Right). Reproduced from: Ragni et al. [32].

2.26 Variation of lift, drag and moment with respect to incidence angle; comparison of results from PIV and pressure measurements at $Re = 3 \times 10^5$. Reproduced from Van Oudheusden et al. [50].

2.27 Finite difference stencil for the five-point method (reproduced from Hoffman and Frankel [15]).

3.1 The models tested in the wind tunnel experiment. All dimensions are in cm.

3.2 Wing A design illustration.

3.3 Drag coefficient as a function of chord-to-thickness ratio for NACA aerofoils at $Re_D = 84,500$.

3.4 Setup of the PIV experiment. The position of the camera and the laser sheet wrt to the model is highlighted.

3.5 Schematic diagram showing the laser setup.

3.6 Camera position wrt the laser sheet. The field of view (FOV) is also highlighted here.

3.7 The experimental setup for pressure measurements.

3.8 example of intensity normalisation filter. Reproduced from (LaVision [22]).

3.9 The comparison between the raw and processed PIV sample.

3.10 The vector fields resulted from cross-correlation analysis.

3.11 Zigzag strip applied on the reference cylinder.

3.12 Vortex generator strip.

4.1 Mean velocity field on which pressure computation is carried out with pressure boundary conditions.
4.2 Comparison between the PIV-pressure and Pitot-static pressure values in the wake centerline of cylinder and wing A. ................................................................. 43
4.3 The x-velocity standard deviation in the flow field. ........................................... 46
4.4 Mean x-velocity fields around a cylinder obtained using two different window sizes. . 48
4.5 Mean x-velocity fields around a cylinder obtained using two different window sizes. . 49
4.6 Velocity variation on the top boundary. ................................................................. 50

5.1 Mean fields and drag terms plot for reference cylinder for free stream velocity of 25 m/s. 52
5.2 Centreline velocity and pressure behind a rough cylinder for 3 free stream velocities. . 54
5.3 Streamline highlighting the flow separation line for smooth and rough cylinder mean fields. ........................................................................................................ 55
5.4 Contour lines highlighting the difference in recirculation region length. .................... 55
5.5 Streamline highlighting the difference in recirculation region for a smooth and a rough cylinder mean fields. ................................................................. 56
5.6 Pressure and velocity profiles in the wake at $\frac{x}{D} = 1$. ........................................... 56
5.7 Plots highlighting the difference in pressure deficit term for a smooth and a rough cylinder. ........................................................................................................ 57
5.8 Centreline Pressure and velocity for a smooth and a rough cylinder at free stream velocity of 27.5 m/s (Re = $9.85 \times 10^4$). ................................................................. 57
5.9 Mean x-velocity fields for cylinder A and cylinder B. ................................................. 59
5.10 Wake profile comparison for cylinder A and cylinder B depicting smaller wake width behind the cylinder B. The profiles are taken at $\frac{x}{D} = 3$ as shown with green lines in Figure 5.9. ................................................................. 59
5.11 Pressure field around cylinder A and cylinder B indicating the difference in the base pressures. ........................................................................................................ 60
5.12 Mean x-velocity fields for smooth and rough cylinder B. ......................................... 60
5.13 Velocity profiles for smooth and rough cylinder B. ................................................... 61
5.14 Mean fields around wing A for free stream velocity of 25.8 m/s. ................................. 62
5.15 Drag contributing terms plot for three different free stream velocities. The 'red'-momentum deficit term, 'blue'- pressure deficit term, 'green'- Reynolds stress term and 'black'- total coefficient of drag. 'solid lines' represents the $C_d$ for 20.8 m/s, 'dashed lines' represents the $C_d$ for 25.9 m/s and 'dotted lines' represents the $C_d$ for 30.9 m/s. ............... 62
5.16 The wake velocity profiles for wing A at three different free stream velocities. The profile is taken at 2 cylinder diameter-distance downstream the trailing edge. ............... 63
5.17 Mean fields around wing A at $\alpha = 3^\circ$ with and without boundary layer tripping for free stream velocity of 25.8 m/s. ..................................................... 64
5.18 The velocity profiles taken at 0.5 chord-distance in the wake of wing A. ....................... 65
5.19 (a) and (b): Mean x-velocity fields around wing A with zigzag tripping and with/without vortex generators, at $\alpha = 8^\circ$, (c) and (d): Mean x-velocity field close to the maximum thickness location in presence and absence of VGs. The free stream velocity of 25.8 m/s. .................. 65
5.20 Mean x-velocity and pressure field around wing B $\alpha = 0^\circ$ and $u_\infty = 25$. ............... 67
5.21 Mean x-velocity around wing B in absence and presence of trailing edge vortex generators. ........................................................................................................ 68
5.22 Contours showing the effect of trailing edge vortex generators. ................................. 68
5.23 Wake profiles at $\frac{x}{D} = 1$ for wing B with and without the trailing edge VGs. ................. 68
5.24 Plot showing the drag terms for wing B at $\alpha = 0^\circ$ and 3$^\circ$. ................................ 70
5.25 Mean x-velocity and pressure field around wing B $\alpha = 3^\circ$ and $u_\infty = 25.45$. .......... 70
5.26 The mean x-velocity plots with streamlines highlighting the avoided flow separation due to boundary layer tripping. $\alpha = 5^\circ$. ................................. 71
5.27 The mean x-velocity plots with streamlines highlighting the avoided flow separation due to vortex generators. $\alpha = 8^\circ$. ................................................................. 71
5.28 The drag plot to highlight the spatial dependency of drag coefficient. ....................... 72
5.29 Wing B $C_d$ at different angles of attack. ................................................................. 72
5.30 The chart displaying lowest drag coefficient on different geometries. ....................... 73
Introduction

A split second delay can decide the placing in professional sports events. The final outcome of any sports event is a complex amalgamation of multiple parameters like physiology, psychology, technical advancement in the equipment etc. Aerodynamics play a decisive role in almost every sport ranging from ball games like football, cricket, tennis, golf etc. to athletics like skiing, speed skating, bob-sleigh, cycling. Thus, the aerodynamic loads analysis is usually included in the overall performance analysis. In recent history, sports aerodynamics has become one of the leading topics of interest and the flow around athletes/sportsmen and equipment is widely studied via both computational and experimental analysis.

Speed skiing is one such sport where the aerodynamic drag on skier is the dominant factor contributing to more than 80% of total resistance when a skier passes through timing gate (Thompson et al. [45]). Speed skiing race involves the downhill skiing on a straight path aiming to achieve maximum possible speed while passing through the speed gate located at the end of the track.

An average speed skier achieves the maximum speed ranging between 150-180kmph depending upon the track. Men’s world record is clocked at the speed of 254.9 kmph. Besides being fast, speed skiing is an extremely competitive sport too. In Super G races at the 2006 Torino winter Olympics, the difference between a third-place podium finish and fourth place was 0.10 seconds (0.11%) in the Men’s event and 0.03 seconds (0.03%) in the Women’s race (Brownlie et al. [9]). At such high speed, the aerodynamic drag on skier is the dominant resisting force. Effective drag reduction is a key to success in such a highly competitive sport. To achieve drag reduction, participants optimise the design of their skiing accessories like helmet, boot-fairings, ski poles, skiing apparel and also focus on the aerodynamically optimised skiing posture.

Figure 1.2 shows the forces acting on a skier in generic skiing posture. The resisting forces on a skier comprise of aerodynamic drag and kinetic friction force on the skies. As mentioned the aerodynamic drag is the dominant resisting force, performance enhancement of a skier largely depends on aerodynamic drag reduction.
The drag force acting on a skier consists of pressure drag and skin friction drag. The skin friction drag is caused by the shearing action of the air moving over the skier. Whereas the pressure drag is a result of the low-pressure region caused by flow separation from the body. Figure 1.3, shows the contribution of these two drags in case of a streamlined and a bluff body. The helmet, ski poles and boot-fairings are streamline designed to avoid separated flow and significantly large pressure drag. The roughness of the apparel fabric is optimised for drag reduction. Depending upon the Reynolds number of the flow certain roughness of the fabric can trigger boundary layer transition from laminar to turbulent and delay or avoid flow separation reducing pressure drag (Terra [41]).

Drag acting on a body immersed in the fluid is given as:

\[ \text{Drag} = \frac{1}{2} \rho V^2 C_d A \]  

Here \( \rho \) is the density of the fluid, \( V \) is the relative velocity between the fluid and the body, \( C_d \) is the coefficient of drag and \( A \) is the frontal area exposed to the flow. As it can be inferred from the equation 1.1, the drag force is proportional to the velocity squared. Hence higher the velocity, larger would be the drag resistance. Optimising the skiing posture can reduce the frontal area and considerably reduce the drag force. Studying the airflow around a skier in different posture also helps to build the understanding of the regions of drag production. According to a computational study by Asai et al. [6], main sources of drag in the full tuck posture are the head, upper arms, upper legs, and thighs (including buttocks). The company Airshaper [3] conducted numerical simulation on the 3D scanned model of a skier (Joost Vandendries) and the sources of pressure and skin friction drag are visualised in Figure 1.4. The result shown in Figure 1.4 agrees with the findings of Asai et al. [6].
(a) Source regions of pressure drag.  
(b) Source regions of friction drag.

Figure 1.4: The source regions of pressure and friction drag. Largest source region is indicated by colour Red and the smallest source region is by colour Blue. Reproduced from: Airshaper [3].

(a) The velocity magnitude field on cut-plane bisecting the skier model in half. The position of the plane is demonstrated in the top-left corner.

(b) The velocity magnitude field on cut-plane bisecting a leg in half. The position of the plane is demonstrated in the top-left corner.

Figure 1.5: The flow speed is 40 m/s from left to right. Reproduced from: Asai et al. [6].
Figure 1.5 shows the velocity magnitude fields around a skier in the mid-plane and the plane cutting vertically through the leg. As inferred from Figure 1.5(a), the flow separation region is right behind the buttocks of a skier which can be altered by optimising the body posture. Whereas in Figure 1.5(b), the separated flow region is present behind the lower legs. This is because the lower legs act as bluff bodies in the flow and are responsible for considerable pressure drag. According to an article on the University of Cambridge website [1], the implementation of aerodynamically designed fairings around the legs of a skier can reduce the drag as much as 38% in the wind tunnel.

This thesis focuses on the design of a streamlined fairing to reduce the drag on skier's legs. The fairings are worn on the skiing boots which have fairly cylindrical shin part. Thus, for simplification, the lower leg is assumed as a cylinder. The flow around a circular cylinder is experimentally studied and the fairing design to envelope the cylinder is developed in this thesis.

The design regulations of ski fairings are established by the International Ski Federation. According to the regulations, the skiers may wear rear fairings under the ski suit while meeting the following requirements ([44]):

1. Each fairing may not exceed 1 kg in weight, must be constructed from a pliable material, must not cover or inhibit the working of the ski bindings, and must be non-wounding when braking.

2. The maximum depth of the fairings, measured perpendicular to the leg, must not exceed 30 cm.

3. Front fairings must be rounded and follow the standard shape of the boot.

The first design constraint has little to do with the aerodynamic design of the fairing while the other two constraints are to be strictly met. Figure 1.6 shows the design layout of a generic fairing.

Figure 1.6: Design layout of fairing. Reproduced from: The international Ski competition rules (ICR) [44].

Thesis outline

As explained earlier in this chapter, the skier's leg is assumed as a circular cylinder. The flow around a cylinder and the drag reduction techniques are discussed in the chapter 2 and the research question of the thesis is established. The process involved in designing of the model is explained in the first half of the chapter 3. The second half of the chapter explains the experimental setup used and data acquisition performed during the drag measurements in the wind tunnel. Thereafter the processing of the data to obtain mean velocity fields around the models is explained in the chapter 4. The extraction of pressure field from velocity information and the process of drag force measurement is also discussed in the same chapter. The findings of this thesis are presented and discussed in the chapter 5. Finally, the conclusions are drawn and the recommendation for the future work are provided in the chapter 6.
Part I: Fluid dynamics

2.1. Cylinder flow

The flow around a circular cylinder has been extensively studied for decades due to its numerous real-life applications. Besides its direct application, the flow around a cylinder serves as a simple model for relevant engineering flows especially for the study of blunt bodies immersed in the free stream (Hwang and Yang [17]). Flow around an industrial chimney, wind turbine tower, skier's legs are few examples of the real-life cylinder flow.

The cylinder acts as a bluff body in the flow and exhibits flow separation from the surface. This separated flow results in larger wake size and hence larger pressure drag. Figure 2.2 shows a flow around a cylinder with larger separated wake. The separated wake and the free shear layers are highlighted in Figure 2.2 using red and green arrows respectively. The drag force acting on a body consists of skin-friction drag and pressure drag. In case of streamlined bodies like airfoils, the skin-friction drag is dominant whereas, for bluff bodies like a cylinder, pressure drag is the major source of total drag experienced by the body.
The free shear layers on top and bottom of the cylinder roll to form vortices which are shed alternatively (Bearman [7]). The wake in case of bluff bodies are complex mainly due to interaction between multiple shear layers namely boundary layer, separated shear layer and the wake (Tropea and Yarin [47]). The drag force acting on a cylinder placed in flow is given by:

\[
\text{Drag} = \frac{1}{2} \rho C_d SV^2
\]

where, \(\rho\) is the density of the fluid, \(C_d\) is the drag coefficient of the cylinder and \(S\) is the frontal area. \(V\) is free stream velocity of the flow.

The drag force on a cylinder would always increase proportionally to the square of flow velocity if \(C_d\) is a constant. However, in reality, \(C_d\) is a function of Reynolds number. The relation between coefficient of drag (\(c_d\)) and Reynolds number (\(Re\)) is shown in Figure 2.3.

As stated earlier, the cylinder wake is complex due to an interaction between multiple shear layers. It exhibits different flow features at different Reynolds number range. Depending upon the Reynolds number, the flow around cylinder can be classified into distinguished flow regimes. Flows belonging to a particular regime would demonstrate similar characteristics. Since the flow regimes are the consequence of varying Reynolds number, they exhibit different location of laminar to turbulent transition or may not exhibit the transition at all if Reynolds number is lower than the critical value.

The schematic of different flow regimes is shown in Figure 2.4. The flow regimes are briefly discussed as follows:
At very low Reynolds number \((Re = 0 - 5)\), the flow is highly viscous and stays attached all around the cylinder. Such flow is called *creeping flow* or *Stoke's flow*. At Reynolds number between 5-40, the flow separates at the rear half of the cylinder forming a symmetrical recirculating wake as seen in Figure 2.4b. As the Reynolds number increases beyond 40, the flow becomes unsteady with the periodic shedding of counter-rotating vortices famously known as von Karman vortex street. At very low Reynolds number the flow stays entirely laminar and with the increase in Reynolds number the turbulence gradually comes into the picture. When the Reynolds number exceeds 400, the far wake starts becoming turbulent while the boundary layers and shear layers close to the cylinder stay laminar. Turbulence starts to appear more upstream as the Reynolds number increases. Eventually, the boundary layers transit to turbulent and show turbulent separation.

The sub-critical and critical regime are important for engineering applications, they are briefly discussed here:

**Sub-critical regime** \((Re = 400 - 2 \times 10^5)\)

With further increase in Reynolds number, von Karman vortex street becomes turbulent while the boundary layer on the cylinder surface still remains laminar. The flow separation is laminar and the separation point are located approximately at 80° from the stagnation point (John D. Anderson [19]). The value of \(C_d\) remains relatively constant and close to unity (see Figure 2.3). The schematic of this flow regime is shown in Figure 2.3(d) which also highlights wide turbulent wake behind the cylinder.
**Critical regime** \((Re = 2 \times 10^5 - 5 \times 10^5)\)

This flow regime is characterised by the formation of laminar separation bubble. Like the previous regime, the laminar separation of boundary layer occurs in the front half of the cylinder (John D. Anderson [19]). Thus forms a free shear layer over the top and bottom point of the cylinder. Immediately after the separation, this newly formed shear layer undergoes a transition to turbulent flow which has enhanced mixing characteristics. The energised shear layer attaches back to the surface as a turbulent boundary layer and hence forms a laminar separation bubble (Anthoine et al. [4]). This reattachment point lies at the back of the cylinder. Being turbulent, the freshly reattached boundary layer has higher ability to withstand adverse pressure gradient. The flow stays attached to the surface till around \(120^\circ\) measured along circumference from stagnation point. The formation of a laminar separation bubble helps to delay the flow separation resulting in a narrower wake. The smaller width of the wake reduces pressure drag acting on the cylinder which is advocated by the dramatic drop in \(C_d\) value from \(~1.2\) to \(0.3\). The schematic of this regime highlighting the narrower wake is shown in Figure 2.4(e).

With further increase in the Reynolds number, flow regime changes to super and trans-critical in succession which is characterised by increasing \(C_d\) value.

### 2.2. Cylinder with a splitter plate

Splitter plate is a simple flat plate attached to the cylinder base. The splitter plate increases the base pressure coefficient and hence reduces the drag acting on the cylinder. In some cases, the splitter plate is also found to suppress the vortex shedding.

Bearman [7] investigated the flow around a 2D model with blunt trailing edge fitted with splitter plates of different lengths. Total boundary layer thickness to base height ratio was selected as 0.5 and experiment was performed at chord-based Reynolds number of \(2.56 \times 10^5\). Splitter plate lengths examined were in the range of 0 to \(4h\), where \(h\) is the base height. Hot-wire anemometry technique is the techniques used for velocity quantification by Bearman.

![Figure 2.5: Base pressure coefficient((\(C_p\))\(_b\)) vs splitter-plate length \((l/h)\). \(\times\), \(Re = 1.4 \times 10^5\) and \(\circ\), \(Re = 2.45 \times 10^5\). Image taken from (Bearman [7])](image)
2.2. Cylinder with a splitter plate

Figure 2.5 shows the effect of splitter plate length on base pressure coefficient at two different Reynolds number. It is interesting to note that \((C_p)_b\) is Reynolds number depended only when the splitter-plate length \((l/h)\) ranges from 0 to 2.5. The coefficient of base pressure exhibits the increasing trend with the increase in plate length except for the plate-length range of 1.0 to 1.5 where \((C_p)_b\) behaves otherwise.

In order to inspect the changes in velocity field around a bluff body due to the presence of splitter plate, Bearman [7] carried out two traverses of hot-wire measurements. The first traverse was carried out in a direction normal to the wake of a basic geometry, at different downstream locations. The findings are presented in Figure 2.6. While the second traverse was carried out in a streamwise direction (parallel to splitter plate) and its findings are shown in Figure 2.7.

According to Bearman [7], the percentage velocity fluctuation \((\theta)\) is defined as \(((u_{rms}/U_0) \times 100)\) where \(u_{rms}\) is streamwise velocity fluctuation and \(U_0\) is free stream velocity. Inferring from Figure 2.6, the velocity fluctuations closest to trailing edge \((x/h=0.25)\) show a sharp peak at \(y/h = 0.5\) in the traverse direction. This is explained by the presence of turbulent shear layer springing from the bluff trailing edge (Bearman [7]). The peak attains it apex for \(x/h = 1\), i.e. one base-height distance downstream the trailing edge. This location of peak apex is in good agreement with findings presented in Figure 2.7(a), where measurements were taken streamwise at \(y/h = 0.25\). While moving further downstream the location of the peak moves away from the centre of wake and weakens in intensity. The location of peak signify the presence of vortex core (Bearman [7]). Bearman [7]'s observation shows agreement with (Kovasznay [20]), (Roshko [33]) and (Schaefer and Eskinazi [36]).

![Figure 2.6](image-url)  
Figure 2.6: Percentage velocity fluctuation \((\theta)\) traverses along y-axis (normal to the wake) at different downstream locations at Re =1.45 × 10^5. Image taken from (Bearman [7])

Figure 2.7 shows the percentage velocity fluctuation measured streamwise with different splitter-plate length. For purpose of comparison of Figure 2.7(a) with rest of the subfigures, it is advised to compare values of percentage velocity fluctuations \(\theta\) at \(y/h = 0.25\) since the presence of splitter plate would not permit measurements at \(y/h = 0\). It can be inferred from Figure 2.7 that the peak of velocity fluctuation moves downstream with the increasing plate length which suggests the vortex formation is taking place further downstream from the model base. Fluctuations attain lower peak value in the vicinity of the model base when splitter plate is present. For shorter plate length i.e. \(l/h\) ranging from 0.5 to 1, the large increases in the \(\theta\) occurs beyond end of the plate(Figure 2.7(b),(c)). Whereas for \(l/h > 1\), the increase in \(\theta\) begins before the plate ends which suggests the beginning of vortex formation in shear layer present on either side of the plate(Figure 2.7(d),(e)). When \(l/h > 3\), the peak in \(\theta\) is found to be present before the plate ends (Figure 2.7(f),(g)). According to Bearman, the \(\theta\) peak for \(l/h > 3\) occurs above the position of flow reattachment determined using oil-film technique and it is in agreement with (Tani et al. [40]).

A clear representation of maximum \(\theta\) location obtained from hot-wire traverse in the direction normal to the wake, for different plate length is presented in Figure 2.8. The velocity fluctuations con-
2. Research background

Figure 2.7: Percentage velocity fluctuation($\theta$) traverses downstream the blunt trailing edge at $y/h=0.25$. The experiment is carried out at $Re=1.4 \times 10^5$ using different splitter-plate length. Image taken from (Bearman [7])

Figure 2.8: Percentage velocity fluctuation($\theta$) obtained from traverses along y-axis (normal to the wake) at different downstream locations at $Re=1.45 \times 10^5$ for plate length($l/h$) ranging from 0 to 4. $x$, position of maximum $\theta$. Image taken from (Bearman [7])
2.2. Cylinder with a splitter plate

verge towards the centre of the wake and start to diverge again at a different downstream location depending upon the plate length. This necking effect was observed by Schaefer and Eskinazi [36].

Roshko [34] determined pressure coefficient in the wake of cylinder in presence and in absence of a splitter plate at \( Re = 14,500 \). The results are presented in Figure 2.9. Though Roshko [34] carried out the experiment at lower Reynolds number compared to Bearman [7], the results indicated in Figure 2.9 and Figure 2.5 the similar trend of increased base pressure in presence of splitter plate.

![Figure 2.9: Wake centreline pressure with and without splitter plate at \( Re = 14,500 \). Reproduced from: (Roshko [34])](image)

Apelt et al. [5] plotted the relation between \( C_d \) and Reynolds number for different splitter plate length varying from 0 to 2 times the diameter. The trend is shown in Figure 2.10. It can be seen that drag reduction can be achieved due to the presence of splitter plate.

![Figure 2.10: \( C_d \) vs \( Re \) for varying splitter plate length. Reproduced from:(Apelt et al. [5])](image)

In summary, these results show that the base pressure is inversely proportional to the location of velocity fluctuation peak from the model base. This peak represents the formation of a fully formed vortex. The presence of splitter plate forces vortex formation downstream and hence increasing the base pressure.
2.3. Drag reduction techniques used in the past

Lee et al. [23] performed an experimental study on the drag reduction of a cylinder by installation of an upstream control rod. The maximum drag reduction on the main cylinder was found to be 29%. The main cylinder lies inside the vortex formation region behind the control rod which leads to substantial decrease in the pressure on the windward side on main cylinder (Lee et al. [23]). The flow visualisation around the cylinder and pressure coefficient distribution on cylinder surface is shown in Figure 2.11.

Triyogi et al. [46] performed experimental study similar to Lee et al. [23] but used I-type bluff body instead of a circular rod, upstream the main cylinder. I-shape bluff body is a small circular cylinder with cuts at both the sides parallel to the main cylinder. The idea behind I-shape bluff body is to enhance a large wake emanating from it and hence the stronger shear layer affects the main cylinder present downstream. The flow separation process on main cylinder is altered so that the aerodynamic forces are reduced (Triyogi et al. [46]). The drag reduction achieved with the presence of upstream I-shape body is nearly 50%.

Hwang and Yang [17] carried out an numerical study on the cylinder drag in presence of dual splitter plates placed along the horizontal centreline, one upstream and another downstream the cylinder. The findings reported the maximum drag reduction of 38.6%. The upstream plate significantly reduces the pressure in the vicinity of front stagnation point with secondary effect of increased base pressure. While the downstream splitter plate also increases the base pressure on the cylinder along with effective vortex shedding suppression. The flow streamlines in presence of the dual splitter plates are shown in Figure 2.12.

According to O’Connor et al. [27], the drag reduction of more than 90% can be achieved by incorporating aerodynamically faired shroud around the cylinder. An illustration of an aerodynamic shroud geometry around the cylinder is shown in Figure 2.13. According to O’Connor et al. [27], symmetric NACA aerofoils outperformed elliptical aerofoils even at lower Reynolds number, which is significant since drag coefficient typically higher at lower Reynolds number. The plot in Figure 2.14 shows the comparative assessment of $C_d$ for NACA aerofoils, elliptical aerofoils and other previously discussed drag reduction techniques.

It can be inferred from Figure 2.14 that the aerodynamic shroud implementation is more effective than any other drag reduction method. From the application point of view, the implementation of such shroud on a wind turbine tower can not only reduce the drag force acting on the tower but
2.3. Drag reduction techniques used in the past

Figure 2.12: The time-averaged streamlines for the unsteady flow at Re=100: (a) $\frac{G}{d} = 2.0$, (a) $\frac{G}{d} = 2.6$ and (a) $\frac{G}{d} = 2.7$. G is the distance between the cylinder base point and leading edge of downstream plate, d is the diameter of the cylinder and length of the plates. Reproduced from Hwang and Yang [17].

Figure 2.13: Illustrative shroud design around a cylinder using NACA0033. Reproduced from: O’Connor et al. [27].

Figure 2.14: Drag coefficient at $\alpha = 0^\circ$ for a cylinder, ellipse, and symmetric NACA aerofoils ranging from NACA0012 to NACA0050, where D is the maximum circle diameter than can be inscribed in a given shape and c is the streamwise length. (ellipse data from Blevins [8], and reduced drag data from Lee et al. [23], Hwang and Yang [17] and Triyogi et al. [46]). Reproduced from: O’Connor et al. [27].

it also reduces the unsteadiness in the wake. The downwind rotor design can be encouraged if the unsteadiness in the wake of a tower is reduced. The extreme-scale turbine is more feasible with downwind turbines due to relaxed stiffness constraint compared to conventional upwind turbines.
O’Connor et al. [27] designed shroud geometries based on XFOIL analysis to envelope wind turbine tower. The calculation was carried out at tower diameter-based Reynolds number of $8.3 \times 10^6$. The shroud was needed to meet the following requirements:

1. Low drag i.e., minimum diameter based drag coefficient ($C_{d,D}$).

2. self-aligning characteristics about the cylinder centre i.e., $C_{mz,D} < 0$ about $x_D$ (negative gradient of moment coefficient calculated around the centre of the inscribed cylinder.)

3. avoid strong adverse pressure gradient to avoid flow separation.

4. short trailing edge i.e. minimum $x_{TE} - x_D$ (distance from the inscribed cylinder centre to the trailing edge point).

O’Connor et al. [27] carried out a batch XFOIL analysis of symmetrical NACA aerofoils to discover the aerofoil chord-to-thickness ratio which leads to minimal drag. The aerofoil corresponding to the minimal drag is selected as base geometry. Furthermore, the frontal part upstream the maximum thickness of the base geometry is trimmed and a linear cuff is added. The addition of the frontal cuff maintains the chord-to-thickness ratio responsible for minimal drag. According to O’Connor et al. [27], the semi-circular headed aerofoil will have negative $C_{mz}$ around the centre of the tower because the maximum thickness of the aerofoil lies further close to the leading edge.

The geometry with trimmed upstream front and the geometry with the frontal cuff is shown in Figure 2.15. Furthermore, the inverse design method in PROFOIL is used to design the aerofoil profile with smoother pressure gradient to avoid flow separation.

![Figure 2.15: (a) Baseline geometry NACA0033 with trimmed front (b) Baseline geometry added with a linear cuff decreasing from a certain length at y=0 to zero at y=maximum thickness. Reproduced from: O’Connor et al. [27].](image)

Extending the work of fairing design, O’Connor et al. [28] conducted an experimental study on the developed design by O’Connor et al. [27]. The experiments were carried out at Reynolds number which is two order of magnitude lower than the Reynolds number considered during the design process. O’Connor et al. [28] conducted the PIV experiments in a water tunnel at diameter-based Reynolds number ($Re_D$) of $3.35 \times 10^4$ and studied the wake deficit for two different aerofoils and a cylinder. The designed aerofoils produced lower wake deficit compared to the cylinder. The aerofoils exhibited flow separation at $10^\circ$ angle of attack and the wake deficit were comparable to that of the cylinder.

The aerofoils designed by O’Connor et al. [27] have a cuff attached to the cylinder front which makes the aerofoil head non-circular. While such a frontal cuff design is not permitted in this project due to the design constraints (refer chapter 1). Hence, the circular-headed self-designed aerofoils are made and tested for drag reduction on a cylinder. The experiments carried out in this project are at higher Reynolds number compared to the experiment of O’Connor et al. [28] and detailed drag quantification is conducted.
2.4. Passive flow control techniques

The separated flow results in larger pressure drag on the body. To avoid/delay the boundary layer separation, flow control techniques have been proven substantially helpful. Depending upon the energy expenses made, the flow control techniques can be classified into active and passive techniques. Latter one requires no external energy source for operation and usually rely upon geometrical changes in the object to control the flow. Roughness element and vortex generator are examples of passive flow control technique.

2.4.1. Zigzag strip

It is a glue-tape with characteristic zigzag shape and is available in different thickness. Figure 2.16(a) shows a sketch of the zigzag strip. They are used to trip the boundary layer which means forcing the boundary layer transition from laminar to turbulent. According to Van Rooij and Timmer [51], zigzag strips are highly effective for boundary layer tripping since they can initiate the transition at lower thickness-based Reynolds number i.e. 200.

Elsinga and Westerweel [12] conducted tomographic-PIV measurement of the flow around zigzag trips to measure the forced boundary layer transition. They observed shear layer separating from the strip with undulating spanwise vortices (as seen in Figure 2.16(b)). Underneath the separated shear layer, backflow with streak structures is observed. The spanwise vortices break into smaller arches close to the reattachment point and subsequently forms hairpin structures. Further downstream, the smaller vortical structures close to the wall are produced which are characteristics of a turbulent boundary layer.

2.4.2. Vortex generators

Vortex generators are passive flow control devices. They usually consist of series of small plates or aerofoils protruding normally to the surface and at a non-zero incidence angle to the flow to produce streamwise trailing vortices (Lin [24]). These vortices entrap higher momentum fluid from
the outer flow into the boundary layer hence energising the boundary layer. The energised boundary layer can withstand higher adverse pressure gradient before separating from the surface. Thus, vortex generators help to delay or avoiding boundary layer separation.

Figure 2.17: Sketch of a counter-rotating vane-type vortex generator. Reproduced from: Lin [24].

2.5. Research objective and questions

It is clearly evident from the literature study about drag reduction on the cylinder that the streamlined fairing design is the most effective passive drag reduction technique. Streamlined fairing design would reduce the size of the cylinder wake and suppress the pressure drag component at an expense of a relatively small increase in skin friction drag.

“The research objective of this thesis to design streamlined fairing to reduce drag acting on a cylinder and quantify the reduced drag.”

In order to meet the research objective mentioned above, the self-designed models are developed and drag quantification is carried out in the wind tunnel. The following research questions regarding these models are to be answered using the data acquired from the experiment:

1. Prior to the streamlined fairing, how much drag reduction can be achieved by applying roughness on a cylinder to trip the boundary layer?

2. Quantify the drag reduction achieved by attaching a splitter plate of 2 diameter length to the base of the cylinder and examine its effectiveness compared to the other geometries.

3. Drag quantification on the designed aerofoil geometries at different flow velocities and angles of attack. Also, examining their sensitivity towards the flow separation for non-zero angles of attack.

4. What is the further drag reduction that can be achieved by implementing passive flow control techniques (roughness strips and vortex generators) on aerofoil geometries?

5. Which of the above model has least calculated drag value and wider operational range in terms of angles of attack with significantly lower drag?

The computed drag values for wings is compared to its counterpart obtained from XFOIL to examine the level of agreement.
Part II - Experimental techniques

Thus far, the flow features around the cylinder and drag reduction techniques used till date are discussed. In the scope of this project, the wind tunnel testing on self-developed models using Particle Image Velocimetry (PIV) technique is performed to quantify the drag force. This part includes the brief discussion of the experimental techniques for data acquisition and introduces control surface approach which is the theoretical principle behind drag calculation.

2.6. Particle Image Velocimetry (PIV)

In recent decades, PIV has emerged as the most advanced technique for flow measurement. As the name suggests, it is a Velocimetry technique thus provides instantaneous velocity flow field. PIV offers not only the quantitative velocity field but also the qualitative flow visualisation unravelling different flow structures in the domain.

The working principle of PIV is based on the measurement of particle displacement in a known time period which yields velocity field. PIV involves insertion of tracing particles into the flow to visualise the fluid motion. These particles should be sufficiently small to follow the course of the flow as accurately as possible. The tracing particles are illuminated by a light source making it visible to the recording device. Typically, the pulsed light source is used and the light beam is transformed into a sheet. Only the particles inside the light sheet are illuminated. CCD camera is usually chosen as a recording device. The light scattered by the tracing particles is recorded onto this camera in two subsequent frames forming an image pair. The cross-correlation analysis is performed on an image pair which gives instantaneous velocity field. Multiple image pairs can be acquired and each produces an individual instantaneous velocity field. The mean flow field can be obtained by taking a statistical average of all the instantaneous velocity fields. For the records, the author is interested in the study of mean flow. The schematic diagram of a typical PIV set-up is shown in Figure 2.18.

Traditionally, PIV was used to obtain the velocity field in a planar section of the flow but it is
recently extended to a volume. The spatial region where the flow is being evaluated is known as ‘Field of view’. In this chapter, the elements of PIV are discussed from the planar PIV perspective. For a detailed description about stereoscopic and tomographic PIV reader is advised to refer (Raffel et al. [31]).

2.6.1. Tracing particles

Making a right choice of tracing particles for PIV experiment is essential since they are representatives of the fluid elements. During PIV, the motion of particles is evaluated and particle velocity at a particular spatial location is appointed as local fluid velocity. Thus it is crucial that particles follow the flow faithfully. Besides following the flow accurately, the particles also have to scatter enough light in order appear distinguished from the background to the camera. Hence an ideal tracing particle needs to have excellent mechanical properties (to follow the fluid flow) and light scattering property (to be visible to the camera).

Mechanical properties

The density difference between a particle and the fluid is one of the parameters which decides the ability of a particle to follow the flow precisely. If $\rho_p - \rho_f / \rho_f << 1$ then particles can be assumed to be buoyancy neutral. This buoyancy neutral condition is easier to achieve in the case of liquid flows since the density difference between particle and liquid is smaller than that compared to gas flows. This allows larger tracing particles in liquid flows compared to gas flows.

Relaxation time depicts the spontaneity with which the particle reacts to sudden velocity jumps. Lower the relaxation time faster would be the reaction of a particle. The relaxation time of a particle ($\tau_p$) is given as:

$$\tau_p = \frac{d_p^2 \rho_p}{18 \mu} \quad (2.2)$$

where $d_p$ is the diameter and $\rho_p$ is the density of a particle. As it can be seen from the equation 2.2, the relaxation time is proportional to the square of the particle diameter. Thus, smaller the particle, faster it would react to velocity jumps.

In summary, the mechanical properties suggest that the particle should be as small as possible.

Light scattering properties

For a given constant light intensity of the source (laser), the light intensity received by a camera is equivalent to light scattered by the particles. Thus the particles play a major role in the contrast of PIV recordings.

The scattered light intensity depends on the ratio of refractive indices of the particle and the surrounding medium, size of the particle, their shape and orientation (Raffel et al. [31]). Additionally, the light scattering also depends on the polarisation and the observation angle. For spherical particles with the diameter ($d_p$) larger than the wavelength of the incident light ($\lambda$), Mie's scattering theory can be applied. Mie's theory uses normalised diameter ($q$) to characterise particle scattering. The normalised diameter is defined as:

$$q = \frac{\pi d_p}{\lambda} \quad (2.3)$$

The scattered light intensity in the different direction is shown in Figure 2.19.
2.6. Particle Image Velocimetry (PIV)

With the increase in $q$ the ratio of forward to backward scattering increases, hence it is advantageous to record the forward scatter but the limited optical access and depth of field makes it difficult for PIV application. Typically for PIV, the side scatter is recorded by placing the camera normal to the light sheet. Unlike the mechanical properties, the light scattering properties favour the larger size of the particle. Thus optimal trade-off between the light scattering ability and flow following ability has to be made.

Finally, the density of the seeding particles must be appropriately set. Generally, the density of seeding particles for PIV ranges between $10^9$ to $10^{12}$ particles/m$^3$. Excessively high density would change the flow characteristics. The flow would exhibit multi-phase flow effects due to very high seeding particle density. Whereas at significantly small particle density would lead to an insufficient number of particles in the interrogation windows and introduce significant uncertainty in the measurement.

2.6.2. Illumination

The light source is required to illuminate the tracing particles making them distinguishable from the flow. A small and repetitive time duration of illumination is essential for PIV application. The particles are to be illuminated and observed twice during the observation time($\Delta t$). If the particles are illuminated for a longer time, they would appear as streaks rather than circular dots. To obtain the particle images as dots, the imaged particle displacement must be smaller than the particle diameter. It can be mathematically formulated as:

$$\delta t = \frac{d_r}{VM}$$  \hspace{1cm} (2.4)

where $\delta t$ is the pulse duration, $d_r$ resultant particle diameter, $V$ is the flow velocity and $M$ is the magnification ratio. Only the particles lying in within the thin sheet of light is illuminated hence the position in the depth direction is controlled by the position of the light sheet. The intensity of light source must be sufficient enough so that the light scattered by the particles would be recorded by the imaging device. Given all these requirements, Lasers are usually chosen as the illumination source since they can produce a pulsed, collimated and monochromatic light beam. This beam can easily be produced into a light sheet with the help of optics.

The three important time duration for illumination is as follows:

1. **Pulse duration ($\delta t$):** The time duration for which the particles stay illuminated. It is in the order of nanoseconds.
2. **Pulse separation** ($\Delta t$): The time duration between the two pulses hence it is the time after which the particles are imaged again. Two images taken with a time lapse of $\Delta t$ form an image pair. Usually, the pulse separation time is in the order of microseconds.

3. **Repetition rate** ($T$): The time between the acquisition of two uncorrelated image pair samples. It is usually expressed in terms of acquisition frequency ($\frac{1}{T}$).

The figure 2.20 shows the schematic diagram of the laser and camera synchronisation for normal PIV (non-time resolved). The above listed time parameters are also indicated in Figure 2.20 for illustration purpose.

![Schematic diagram showing synchronisation between the laser pulses and camera acquisition.](image)

2.6.3. Imaging

Figure 2.21 shows the schematics of a PIV imaging system. The image of tracing particles present inside the light sheet is formed at the camera sensor via lenses. The imaging system is characterised by three parameters, lens focal length ($f$), f-number or f-stop ($f_\#$) and the image magnification($M$).

![Schematic diagram showing PIV imaging.](image)

The object distance ($d_o$), image distance($d_i$) and the focal length of the lens can be expressed as follows using the thin lens formula:

$$\frac{1}{f} = \frac{1}{d_i} + \frac{1}{d_o} \quad (2.5)$$

f-stop($f_\#$) is defined as $f_\# = f / D$, where D is the aperture diameter of the lens. f-stop controls the amount of light entering the camera lens. Higher the f-stop lower would be the amount of light, darker the image and larger depth of view. Image magnification is defined as the ratio of image distance to object distance.

$$M = \frac{d_i}{d_o} \quad (2.6)$$
Magnification ratio can also be written in terms of sensor size and field of view as:

\[ M = \frac{\text{sensor size}}{\text{imaged object size}} = \frac{\text{no of pixels} \times \text{individual pixel size}}{\text{Field of view}} \] (2.7)

Since \( f_o \) is connected to the depth of view, it should be chosen in such a way that the particles inside the light sheet should form sharp images at the sensor. In other words, the depth of focus should be larger than the thickness of the light sheet. Excessively small \( f_o \) would produce a too bright image to distinguish the particles from background whereas the images would be significantly dark if the \( f_o \) is way too big.

### 2.6.4. Evaluation of the particle motion

The following four operations are to be carried out sequentially in order to obtain instantaneous velocity field from the data.

1. **Image windowing**: The image samples are divided into small partitions known as ‘interrogation windows’. Each window should contain a significant number of tracing particles. Lower particle number would increase the uncertainty in the measurement. Typically the size of interrogation window varies from 16×16 pixels to 128×128 pixels. For the current experiment, the velocity field is computed using 16×16 pixel window size.

2. **Cross-correlation analysis**: Images belonging to an image pair are taken and a statistical cross-correlation operator is applied to corresponding interrogation windows. The images are nothing but a two-dimensional light intensity array. The correlation operator correlates these two arrays and produces a correlation peak whose distance relative to the origin represents the average particle displacement in the interrogation window.

3. **Correlation peak sub-pixel interpolation**: As the correlation map is produced in terms of discrete pixel size, the distance of peak from the origin is also found in integer pixel shift. To find more accurate peak position the interpolation of the peak is required.

4. **Divide by time and scaling**: Once the average displacement of the particles in an interrogation window is obtained. The average velocity value is calculated using the image magnification, individual pixel size and the known time period between the recorded images.

\[ \text{Physical particle displacement} = \frac{\text{particle displacement in image}}{\text{image magnification}} = \frac{\text{Average displacement (in pixel) \times pixel size}}{M} \] (2.8)

\[ \text{Velocity} = \frac{\text{Average displacement (in pixel) \times pixel size}}{M \times \Delta t} \] (2.9)

The above described image evaluation process is pictorially presented in Figure 2.22.

PIV has become a standard tool for flow evaluation for scientific research and industrial applications. The velocity field data acquired using PIV can be used for various purposes like load determination on an object, studying the flow topology etc. Terra et al. [42] evaluated aerodynamic drag force on a full-scale cyclist mannequin using large-scale tomographic PIV. Figure 2.23(a) shows the setup for the experiment conducted by Terra et al. [42] and Figure 2.23(b) shows the velocity field in the wake of the cyclist mannequin.
2. Research background

Figure 2.22: Particle displacement determination using cross-correlation. Reproduced from Sciacchitano [37].

Figure 2.23: (a) PIV setup schematic diagram (b) Vector field in the wake of mannequin. Reproduced from: Terra et al. [42].

2.7. Load determination from PIV and Control surface approach

Traditionally, the integral loads are evaluated with the use of mechanical balance systems. Whereas using PIV, the load determination can directly be done using the flow field information which eliminates the need for intrusive instrumentation (van Oudheusden et al. [49]). PIV enables one to determine the aerodynamic loads along with the visual understanding of the flow field and the flow features responsible for the load generation.

The quantification of drag acting on a model is carried out by the application of momentum conservation equation in the integral form on a defined control surface. A control surface enveloping the model is selected and the momentum equation is evaluated on it. The difference between influx and outflux of fluid momentum is the force acting on the fluid due to the presence of the model. The equal and opposite reaction force is applied to the model by the fluid which is known as Drag. Figure 2.24 illustrates the choice of a control surface around the model.
In principle, the control surface is defined in such a way that:

1. The upper and lower boundaries (ac and bd in Figure 2.24) are sufficiently far away from the model such that they coincide with the streamlines. The pressure on upper and lower boundaries of control volume is equal to free stream static pressure ($p_\infty$).

2. The boundaries perpendicular to the flow (ab and cd in Figure 2.24) are far upstream and downstream of the model and again the pressure on these boundaries is assumed to be uniform and equal to $p_\infty$.

3. The inflow velocity on ab is uniform across the boundary and equal to free stream velocity $u_\infty$, whereas the velocity at the outlet boundary(cd) is non-uniform due to the wake of the model. The velocity at the outlet is a function of y i.e.($u(y)$).

For steady flow, the drag force per unit span is given as:

$$D = -\int_{acdb} (pV \cdot dA)u - \int_{acdb} p \, dA$$  \hspace{1cm} (2.10)

where $dA$ is the line normal pointing outwards the surface at every boundary. The first term on the RHS represents the momentum flux through the control surface and the second term represents the pressure force acting on the model. According to the assumption of equal pressure everywhere on the boundaries, the last term on RHS in the above equation reduces to 0. Although due to experimental limitations, the outlet boundary cannot always be chosen sufficiently far enough and hence the pressure term must be taken into account for drag computation. The pressure force term for an outlet boundary not sufficiently far from the body due to experimental limitation is written as:

$$-\int_{acdb} p \, dA = \int_{d}^{c} (p_\infty - p) \, dy$$  \hspace{1cm} (2.11)

Since the upper and lower boundaries represent the streamlines of the flow, they have zero mo-
mentum flux through them. Hence the first term on RHS of the equation 2.10 can be written as:

\[- \int_{acdb} (\rho V \cdot dA)u = \rho \int_{a}^{d} u_{\infty}^{2} dy - \rho \int_{b}^{c} u^{2} dy \]  (2.12)

The change of sign for the first term on RHS is due to the \(dA\) and \(V\) pointing in opposite direction along \(ab\). According to continuity equation in integral form,

\[\rho \int_{b}^{a} u_{\infty} dy = \rho \int_{d}^{c} u dy\]  (2.13)

Using equation 2.13 and equation 2.12,

\[- \int_{acdb} (\rho V \cdot dA)u = \int_{d}^{c} u_{\infty} - u dy\]  (2.14)

Hence, the drag force per unit span acting on the model is given by:

\[D = \int_{d}^{c} u_{\infty} - u dy + \int_{d}^{c} p_{\infty} - p dy\]  (2.15)

here \(u\) and \(p\) are the velocity and pressure at the outlet respectively.

Decomposing the drag, velocity and pressure term into mean and fluctuating components and taking Reynolds average on the equation 2.15 gives:

\[\overline{D} = \rho \int_{d}^{c} u_{\infty} - \overline{u} dy - \rho \int_{d}^{c} u' u' dy + \int_{d}^{c} (p_{\infty} - \overline{p}) dy\]  (2.16)

Where \(\overline{u}\) and \(\overline{p}\) represents mean velocity and mean pressure respectively and \(u'\) is the fluctuation in the velocity with time. The equation 2.16 gives the time-averaged drag force per unit span on a body placed in incompressible, two-dimensional flow.

As indicated, three terms namely momentum deficit, pressure deficit and Reynolds stress contribute to the total time-averaged drag. According to van Oudheusden et al. [49], the contribution of viscous stresses is neglected when the control surface contour is taken sufficiently far enough from the model. First two terms on the right-hand side in equation 2.16 can directly be evaluated from the velocity field obtained from PIV. Whereas the pressure term calculation is non-trivial since pressure cannot be extracted from PIV data directly. The reconstruction of pressure field from PIV velocity field is carried out by solving Pressure Poisson Equation (PPE). The pressure reconstruction is discussed in detail in the further section.

### 2.8. Pressure reconstruction using velocity field

Traditionally, the surface pressure distribution is obtained by means of pressure tapping on the model or pressure sensitive paints. The pressure in the flow field is measured using a single pressure probe or multiple probes mounted on a wake rake. In this section, the pressure computation form the velocity information is discussed since pressure term is essential for accurate drag calculation.
The recent developments by van Oudheusden et al. [49], De Kat et al. [11] and Ragni et al. [32] in the pressure computation from PIV velocity fields have made load determination using PIV more reliable.

Figure 2.25 shows the comparison between the PIV integrated pressure and RANS computed pressure contours on the tip of a propeller blade.

Van Oudheusden et al. [50] carried out non-intrusive load characterisation of NACA642A015 aerofoil using PIV. Moment and lift coefficients were obtained using surface force integration approach. The surface pressure values were measured using conventional pressure tapping. While the drag coefficient is measured with pitot-tube wake rake survey. The load coefficients obtain with the PIV is validated with the experimental data and the comparison is shown in Figure 2.26. As it can be inferred from Figure 2.26, load measurements using PIV closely resemble the results from classical intrusive techniques.

For incompressible flow, the relation between time-mean pressure and velocity is established using momentum conservation equation as:

$$-\nabla p = \rho \frac{\partial \mathbf{u}^0}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} - \mu \nabla^2 \mathbf{u} \quad (2.17)$$

For application purpose, the viscous term having a significantly low contribution in the pressure gradient, it can be neglected. Also, the second-order discrete derivative would introduce numerical error in the estimation of the pressure gradient (van Oudheusden et al. [49]).

Writing the momentum equation for the 2-D coordinate system and ignoring the viscous term,

$$-\frac{\partial p}{\partial x} = \rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial y}$$

$$-\frac{\partial p}{\partial y} = \rho u \frac{\partial v}{\partial x} + \rho v \frac{\partial v}{\partial y} \quad (2.18)$$

decomposing the above equation into a mean and fluctuating component and taking Reynolds average yields:

$$\frac{\partial \bar{p}}{\partial x} = -\rho \left\{ \frac{\partial u}{\partial x} + \frac{\partial u'}{\partial x} + \frac{\partial u''}{\partial x} + \frac{\partial u'v'}{\partial x} \right\}$$

$$\frac{\partial \bar{p}}{\partial y} = -\rho \left\{ \frac{\partial v}{\partial x} + \frac{\partial v'}{\partial x} + \frac{\partial v''}{\partial y} + \frac{\partial v'v'}{\partial y} \right\} \quad (2.19)$$
All the terms on the RHS of the equation 2.19 can be delivered by planar PIV. The time-mean pressure field for an unsteady flow can be inferred from the standard, time-uncorrelated PIV acquisition (van Oudheusden et al. [49]). Mean velocity gradient terms are the major contributor to the pressure gradient. However, there is one interesting observation about the fluctuating velocity gradient terms. These gradients play a minor role in the computation of pressure gradients in the wake of streamline bodies but their contribution is dramatically large in the case of separated flows behind a bluff body.

After obtaining the pressure gradients using velocity information, these gradients are numerically integrated to compute the numeric pressure values in the domain. The first approach is direct numerical integration with spatial marching starting from a reference point using Dirichlet boundary condition. The second is 2-D Poisson’s approach which incorporates the in-plane divergence with Neumann and Dirichlet boundary conditions (de Kat and van Oudheusden [10]). The latter one is independent of the integration path whereas the former one isn’t.
Poisson pressure equation is formulated by taking the divergence of the momentum equation:

$$\nabla \cdot \nabla p = \rho \nabla \cdot (\bar{u} \cdot \nabla) \bar{u}$$  \hspace{1cm} (2.20)

Expanding the equation 2.20 and setting terms to 0 using incompressible continuity equation ($\rho \nabla \cdot \bar{u}$) yields the following relation:

$$\frac{\partial^2 p}{\partial x^2} + \frac{\partial^2 p}{\partial y^2} = -\rho \left\{ \left( \frac{\partial u}{\partial x} \right)^2 + 2 \frac{\partial u}{\partial x} \frac{\partial v}{\partial y} \frac{\partial v}{\partial x} + \left( \frac{\partial v}{\partial y} \right)^2 + 2 \frac{\partial^2 u'}{\partial x \partial y} + \frac{\partial^2 v'}{\partial y^2} \right\}$$  \hspace{1cm} (2.21)

The equation 2.21 is a two-dimensional pressure Poisson equation (PPE). Since PPE is an elliptical partial differential equation, the solution (pressure in this case) has no preferred path of information propagation and that the domain of dependence and range of influence for any arbitrary point is the entire domain. Thus, the solution of PPE can not have any discontinuity. The solution at any point in the domain depends on all other points in the domain and especially it depends on the immediately neighbouring points (Hoffman and Frankel [15]). Thus the partial derivatives are discretized using second-order centred-difference scheme. The figure 2.27 shows the stencil for second order centred-difference scheme also known as 5-point scheme.

The discretised equation for second order centred-difference scheme at grid point ($i, j$) with equal grid spacing in x and y-direction ($\Delta x = \Delta y$), is written as:

$$f_{i+1,j} + f_{i,j+1} + f_{i-1,j} + f_{i,j-1} - 4f_{i,j} = S$$  \hspace{1cm} (2.22)

Since there are two second-order partial derivatives in the equation the 2-D PPE requires 4 boundary condition for the well-posed problem. The boundary conditions to be specified on the computational domain can be Dirichlet, Neumann boundary condition or both. Once the boundary conditions and source term are specified, the partial differential equation reduces to a system of linear equation of form $Ax = B$, where $A$ is the coefficient matrix, $x$ is the variable to be solved for (Pressure in this case) and $B$ is a source term. The velocity gradients are often noisy but fortunately, the pressure reconstruction can still be carried out due to the smoothening nature of Poisson’s equation. However, discontinuities in the PIV data must be avoided and spatial resolution must be treated with special care (Gurka et al. [14]).
Model design & Experimental setup

This chapter is divided into two parts, first part explains the design procedure of test models and the second part focuses on the experiment setup, data acquisition and post-processing of data.

Part-1

3.1. Model design

As described in the chapter 1, the skier’s lower leg is assumed as a cylinder for simplification. Thus, the base geometry for which drag reduction is sought is a circular cylinder. To evaluate the extent of drag reduction achieved with the modified design, the drag quantification on base geometry is mandatory. Thus, the circular cylinder is also included in the wind tunnel experiment.

The wind tunnel tests are carried out using a test section of $40 \times 40 \text{ cm}^2$. Aiming to achieve largest possible Reynolds number, the diameter of the cylinder is kept as large as possible while also taking area blockage ratio into account. The diameter of the cylinder is decided as 5 cm which leads to acceptable 12% area blockage. The area blockage ratio is defined as:

$$\text{Area blockage ratio} = \frac{\text{Frontal area of the model}}{\text{Cross sectional area of the test-section}}$$  (3.1)

The diameter of skier’s leg (Joost Vandendries) is approximately 10cm and the maximum fairing depth allowed as per the regulations is 30cm. Assuming that the diameter of skier’s leg is located at the maximum thickness of the fairing, the maximum possible chord-to-thickness ratio of the fairing aerofoil is 3. In real life condition, assuming free stream velocity of 200 kmph (55 mps) and diameter of skier’s leg as 10cm. Reynolds number of the flow is $3.75 \times 10^5$. However, the experiments in this research are carried out at lower Reynolds number ($8.45 \times 10^4$) compared to real life flow condition due to experimental limitations like the size of the test section, maximum flow velocity in the tunnel. For the current experiment, the models are scaled down by the factor of 2. The 2-D profile sketches of models with the dimensions are shown in Figure 3.1. For the ease of reference, the cylinder-splitter plate assembly is named “Cylinder B” (Figure 3.1b), wing with sharp trailing edge is named “Wing A” (Figure 3.1c) and lastly, the wing with finitely bluff trailing edge is named as “Wing B” (Figure 3.1d).

The first modified design of the base cylinder is a cylinder-splitter plate assembly as shown in Figure 3.1(a). Length of the plate measured from cylinder base protruding in downstream is 10cm. Hence the total streamwise length of the assembly is 15cm. This assembly hence meets the chord-to-thickness ratio requirement of 3.
3. Model design & Experimental setup

(a) Baseline cylinder.

(b) Cylinder with a splitter plate (Cylinder B).

(c) Aerofoil with sharp trailing edge (Wing A).

(d) Aerofoil with bluff trailing edge (Wing B).

Figure 3.1: The models tested in the wind tunnel experiment. All dimensions are in cm.

According to O’Connor et al. [27], the geometrical modification of cylinder into NACA symmetrical aerofoils is the most effective passive drag reduction technique. NACA 4-series aerofoils are
3.1. Model design

designed to have their maximum thickness located at 30% of the chord length from the leading edge. According to the design constraints for this project (refer chapter 1), no geometrical modifications are allowed in the front half and upstream the leading of the cylinder. Hence keeping the front half intact and replacing the apt half by the ‘tail’ of NACA0028 forms the preliminary aerofoil design which is shown in Figure 3.1(c). By tail of NACA0028, the author refers to the aerofoil profile downstream the maximum thickness (i.e. from $\frac{x}{c} = 0.3$ to 1).

![Semi-circle NACA0028 'tail'](image)

Figure 3.2: Wing A design illustration.

For the ease of understanding, wing A design is highlighted as the combination of a semi-circle and NACA airfoil tail in Figure 3.2. The reason behind the selection of NACA0028 tail specifically is that the addition of chord length of this tail and cylinder radius is three times the diameter of the inscribed cylinder. For illustration purpose, the original chord length of NACA0028 inscribing the 5cm diameter cylinder at its maximum thickness would be 17.85cm. The length of the tail would be $(17.95 \times 0.7)$ equal to 12.5cm. Hence, the total length of wingA is $12.5+2.5 = 15cm$ which maintains the chord-to-thickness ratio of 3.

The idea behind the design of Wing A is to utilise maximum chord length allowed by design constraints and fit a suitable NACA aerofoil tail. However, it is not guaranteed that wing A is the optimal aerofoil for drag reduction. Since front half of the cylinder has to unchanged, an optimum design for the ‘tail’ must be found. To obtain this optimum ‘tail’ profile, a batch analysis of aerofoils from NACA0005 to NACA0050 is carried out. When thin aerofoil like NACA0005 and thick aerofoil like NACA0050 inscribe identical cylinder at their respective maximum thickness, their chord lengths are drastically different. For example, if NACA0005 and NACA0050 inscribe the cylinder of 5cm diameter at their maximum thickness, then the chord length of NACA0005 is 100cm whereas that of NACA0050 is 10cm. However, the maximum thickness of both these aerofoils is identical i.e. 5cm.

As moving from thin to thick symmetrical aerofoil at zero angle of attack, the pressure drag increases whereas the skin-friction drag decreases. Thus, an optimum geometry should be found by performing a batch analysis using XFOIL.

XFOIL is a standard tool used for load analysis on subsonic aerofoils. While doing the drag analysis, XFOIL uses chord-based parameters like Reynolds number and coefficient of drag. Since all the aerofoils ranging from NACA0005 to NACA0050 have identical maximum thickness i.e diameter of the inscribed cylinder, thickness-based Reynolds number is used as similarity parameter. The equation 3.2 represents the mathematical definition of diameter-based Reynolds number ($Re_D$) and coefficient of drag ($C_{d,D}$) and relates them to their counterparts defined using chord length.

\[
Re_D = \frac{\rho V D}{\mu} = \frac{D}{c} Re
\]

\[
C_{d,D} = \frac{F_d}{\frac{1}{2} \rho V^2 D} = \frac{c}{D} C_d
\]

(3.2)
The Reynolds number corresponding to a flow around 5cm diameter cylinder at free stream velocity of 25 m/s is approximately 84,500. Thus, the batch analysis carried out using XFOIL is conducted at $Re_D = 84,500$. The $C_{d,D}$ acting on this range of aerofoils is plotted in Figure 3.3.

![Figure 3.3: Drag coefficient as a function of chord-to-thickness ratio for NACA aerofoils at $Re_D = 84,500$.](image)

As observed from the Figure 3.3, the thicker aerofoils ($\frac{c}{D} = 1-4$) have significantly higher drag which is because of large pressure drag component. Initially, the $C_{d,D}$ value decreases as the chord-to-thickness ratio increases and reaching minimum at $\frac{c}{D} = 6.67$ (indicated with a ‘diamond’ marker in the plot). The $C_{d,D}$ does not increase considerably and remains close to the minimum till $\frac{c}{D} = 10$. Thereafter, for very thin aerofoil ($\frac{c}{D} = 10$ and beyond), the $C_{d,D}$ starts increasing which is due to the dominant skin friction drag on dramatically thin and long aerofoils.

The long-thin aerofoils have smooth pressure distribution over the surface which facilitates attached flow and keeps the pressure drag component minimal. However, these aerofoils have larger wet area and hence considerable skin-friction drag. Contrarily, in the case of short-thick aerofoils, the streamwise adverse-pressure gradient on the surface is aggressive which facilitates the flow separation. Due to the separated flow, these aerofoils have higher pressure drag and comparatively negligible skin-friction drag.

The aerofoil with minimum $C_{d,D}$ value has a chord-to-thickness ratio of 6.67 which corresponds to NACA0015. If this aerofoil has to inscribe a 5cm diameter cylinder at its maximum thickness then its chord length is 33.33cm. Even after trimming the front portion of the aerofoil upstream the maximum thickness and fitting a cylinder, the resultant chord length is 25.83cm.

Fitting a NACA0015 tail with the circular head would be an optimal approach except for the fact that the chord-to-thickness ratio is beyond acceptable in order to meet the constraints. Though the geometry of wing A (circular head + NACA0028 tail) is perfectly acceptable by the constraints, it has steep adverse pressure gradient which makes it prone towards flow separation. To have a balanced trade-off between the smoother streamwise pressure gradient and a chord length of the aerofoil, second-order polynomial curve fitting is performed between the tail profiles of NACA0015 and NACA0028. The resultant tail profile is truncated at the two and a half cylinder diameter length and the circular head is attached to it at the front which yields the geometry of wing B. The profile of wing B is shown in Figure 3.1(d).
Part-2

3.2. Experimental setup

Thus far in this chapter, design process behind the model geometry is discussed. These models are fabricated and tested in the wind tunnel for drag quantification. This part describes the details of PIV setup and instruments used during the experiment.

3.2.1. Wind tunnel, Test section and Model mounting

The experiment is carried out in the W-tunnel, at the high-speed laboratory, TU Delft. It is an open-circuit, low-speed wind tunnel with a centrifugal fan. The maximum flow velocity is about 35 m/s. W-tunnel can be operated with both open and closed test sections. In the current experiment, the closed test section of dimensions 0.4m × 0.4m × 0.4m made with plexiglass sheets is used. Being transparent, the plexiglass provides the full optical access for PIV measurements. The test models span over the entire width of the test section and are mounted horizontally, clamped to the side walls. Since both the spanwise ends are attached to the side walls, the formation of tip vortices from both the ends is prevented. Though, there would be a formation of horseshoe vortex due to the junction flow in the vicinity of side walls of the test section. The PIV measurements are taken in a 2D plane bisecting the model spanwise. The effect of the junction flow vortices at the measurement plane is practically negligible and flow is dominantly two-dimensional. The two-dimensionality of the flow at the location of measurement plane is confirmed using woollen tufts. The global setup of PIV experiment is shown in Figure 3.4(a) and Figure 3.4(b) shows the cylinder model mounted inside the test section.

![Global setup](image1)

![Cylinder mounted inside the test section in the W-tunnel](image2)

Figure 3.4: Setup of the PIV experiment. The position of the camera and the laser sheet wrt to the model is highlighted.

3.2.2. Seeding particles and Seeder

The fog particles used to trace the flow are generated by Safex Fog 2010 using nebelfuild. The size of a fog particle is approximately 1 µm. The seeder is placed in the plenum of the tunnel, next to the centrifugal fan. Along with the flow, the particles are accelerated by the fan and travels through the wind tunnel. They are illuminated (by the laser) and observed inside the test section by the camera.
3.2.3. Laser

The source of illumination for the experiment is Quantel Evergreen 200 laser. It is a Nd:YAG dual cavity laser has a wavelength of 532nm and repetition rate of 15Hz. Inferring from the repetition rate, this particular laser can shoot 15 pulse couples in a second allowing 15 image pair acquisition per second.

The laser setup is shown in Figure 3.5. The laser head is horizontally fixed on an X95 structural beam and another similar beam supports the mirrors and the lenses. The laser beam is reflected by two mirrors before passing through the lenses. The combination of cylindrical and spherical lenses is used to transform the laser beam into a thin sheet. The thickness of the sheet in the experiment is approximately 2 mm.

This laser assembly is placed downstream of the test section, the laser sheet travels upstream illuminating the tracer particles and strikes the model at mid-span. Laser assembly is placed sufficiently downstream of the model to have a negligibly small intrusive effect.

3.2.4. Field of view and Recording device

The particles illuminated by the short-timed pulse of laser have to be recorded to mark their spatial position at an instance of time. To carry out this recording task, Imperx Bobcat IGV-B1610 camera is used. It is a digital camera with interline transfer CCD image sensor. The camera has the maximum resolution of 1628×1236 pixels with an individual pixel size of 4.4µm. The maximum sampling frequency of the camera is 8.3Hz.

<table>
<thead>
<tr>
<th>Imaging parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Magnification ratio</td>
</tr>
<tr>
<td>0.021</td>
</tr>
</tbody>
</table>

Table 3.1: Imaging parameters

As shown in the Figure 3.6, the camera is positioned normal to the laser sheet. On close observation, one can notice that the camera is slightly downstream the test model. The camera perpendicularly faces the wake of a model to minimise image distortion in the wake and capture the
3.2. Experimental setup

wake velocities with better accuracy. The normal distance between the camera and the laser sheet is approximately around 2m. The various parameter regarding imaging is listed in the Table 3.1.

Using a 35mm lens, the camera is focused at the mid-span plane of the model where the particles are illuminated by the laser sheet. Lower f-stop is used to receive a greater amount of light by the camera since it is placed relatively away from the light sheet. The physical size of the sensor is 7.16mm × 5.44mm and the size of the field of view is approximately 330mm × 250mm. Thus, the Magnification ratio is 0.021.

Among the camera acquisition rate (i.e. 8.3Hz) and the laser repetition rate (i.e 15Hz), former one determines the upper bound of image acquisition rate. Hence, the image acquisition rate for the experiment is decided as 8Hz. The time between two images of an image pair (\(\Delta t\)) depends upon the free stream velocity, Magnification ratio and the particle displacement in pixels. To decide \(\Delta t\) for the experiment, the particle displacement in the free stream is taken as 10 pixels. The pulse separation is calculated using the following relation:

\[
\Delta t = \frac{10 \times 4.4 \times 10^{-6}}{MV}
\]  

(3.3)

Following the equation 3.3, the \(\Delta t\) is selected as 103 \(\mu s\), 83 \(\mu s\) and 69 \(\mu s\) for the free stream velocity of 20, 25 and 30 m/s respectively.

**Zoomed-in study on flow separation**

To closely inspect the effect of applied roughness on flow separation, acquisition on a smaller field of view enclosing the separation point is carried out. Size of the field of view is around 30mm × 25mm. The magnification factor is 0.227. The camera-laser sheet distance is reduced to 30cm approximately and a camera lens of 105mm focal length is used for higher magnification. Since the camera is closer to the laser sheet, higher f-stop value (\(f_h = 5.8\)) is used compared to the previous case. The imaging parameters for zoomed-in study is listed in Table 3.2.

<table>
<thead>
<tr>
<th>Imaging parameters</th>
<th>Magnification ratio</th>
<th>lens focal length (mm)</th>
<th>f-stop((f_h))</th>
<th>acquisition frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.227</td>
<td>105</td>
<td>5.6</td>
<td>8</td>
</tr>
</tbody>
</table>

Table 3.2: Imaging parameters
3.2.5. Pressure measurement setup

The pressure measurements using the pitot-static tube is performed to validate PIV-pressure. The experiment is carried out in the W-tunnel using the same test section as PIV experiment. The experimental setup is shown in Figure 3.7. The pitot-static probe is mounted on a mechanical traverse system attached to a vertical X95 support beam such that the initial probe position is extremely close to the model base point. The pressure values are measured by traversing the probe in a streamwise direction along the centreline of the model. The pressure measurements are performed in the wake of the cylinder and wing A. The probe is traversed downstream in the steps of a cylinder radius length (2.5cm). The acquisition is carried out with the digital pressure gauge connected to a computer system. The acquisition frequency is around 4Hz with acquisition time period of 30 seconds.

![Figure 3.7: The experimental setup for pressure measurements.](image)

3.3. PIV Data acquisition

Mean velocity flow field is the primary interest of this project hence time uncorrelated data acquisition is performed. To perform averaging using time-correlated data, one would require substantially high sampling frequency but smaller acquisition time period. Whereas in case of time-uncorrelated sampling, the acquisition is carried out at fairly low frequency but for a longer period of time. The integral time scale is the measure of a time for which the data shows correlation. If the time period between two consecutive acquisition is greater than integral time scale, the samples are uncorrelated in time. The integral time scale is mathematically defined as:

\[
T_{int} = \int_{0}^{\infty} \rho(\tau) d\tau
\]

where \( \rho \) is the velocity autocorrelation coefficient. For uncorrelated sampling, the time period between two consecutive samples should be larger than \( T_{int} \). In the present case, the frequency of the cylinder flow oscillation considering Strouhal number of 0.2, free stream velocity of 20 m/s and cylinder diameter 0.05m, would be 80Hz. The sampling frequency is 8Hz. Thus, it is evident that the data sampling is uncorrelated in time. The number of total sample pairs recorded is 1000 hence the time period for acquisition is approximately 2 minutes.

Data acquisition with all 4 models described earlier in this chapter, is performed at different flow
velocities and angles of attack. The implementation of passive flow control devices like roughness strip and vortex generators is examined for drag reduction on the models.

3.4. PIV data processing

PIV acquisitions are in the form of image pairs. The images belonging to a pair are taken with a short time interval between them. The cross-correlation analysis is performed on the positions of particles present in an image pair to determine particle displacement. Knowing the displacement of particles between two images and the time interval at which the images are recorded, one can construct a velocity field.

LaVision Davis 8.4.0 software is used to process the PIV data. Before performing cross-correlation analysis, two filters are applied for the enhancement of particle contrast. ‘Subtract time factor’ is the first filter applied. This filter eliminates the background noise by subtracting the minimum intensity in the defined neighbourhood of the image, from the source image (LaVision [21]). It also helps in minimising the laser reflection on the objects. The filter length used is 11 images with a symmetrical operation. Thus, the minimum intensity from 5 preceding, 5 succeeding and image itself is subtracted from the image. Intensity normalisation is the second filter used. Application of this particular filter considerably enhances the contrast (bright particles and dark background). The uniform illuminated particle images are achieved after the application of this filter. An example of using intensity normalisation filter is shown in Figure 3.8.

![Original particle image vs. After normalization](image)

*Figure 3.8: example of intensity normalisation filter. Reproduced from (LaVision [22])*

The outcome after the application of these two filters is high contrast particle images which are desired for cross-correlation analysis. The Figure 3.9 shows a comparison between the processed and raw PIV sample image of the flow around a cylinder. As it can be seen, the processed image has a higher contrast between the particles and the background. Also, the reflection of the laser on the cylinder model is weakened out in the processed image.

To construct instantaneous velocity fields using cross-correlation analysis, the images are divided into smaller interrogation windows. Table 3.3 presents the parameters used in the analysis. A geometric mask is applied to cover the test model (since no particle present) and its shadow (particles not visible) before initiating the cross-correlation analysis.

<table>
<thead>
<tr>
<th>Cross-correlation parameters</th>
<th>Iterations</th>
<th>Multi pass with decreasing window size</th>
</tr>
</thead>
<tbody>
<tr>
<td>Windows size</td>
<td>64×64 (1 pass) to 16×16 (3 pass)</td>
<td></td>
</tr>
<tr>
<td>overlap</td>
<td>75 %</td>
<td></td>
</tr>
<tr>
<td>window shape</td>
<td>Round</td>
<td></td>
</tr>
</tbody>
</table>

*Table 3.3: Parameters in cross correlation analysis.*
After the velocity vector fields are computed, vector post-processing is carried out to increase the accuracy by eliminating the noisy vectors. Also, the median filter with universal outlier detection is implemented. According to this filter, the vectors in a filter region \((7 \times 7\) pixels\) are examined for their deviation from the median in that region. If the deviation is found to be larger than the specified threshold, then the vector is deleted (Westerweel and Scarano [52]). The deleted vectors are filled up by the interpolation.

As mentioned already, the time-averaged flow is of the primary interest in this project. Thus, these instantaneous velocity fields in time are averaged to obtain the mean flow. For accurate averaging, it is to be taken care of that the biased samples are not included. Among these instantaneous field samples, there might be few samples where the entire vector fields are unphysical or some regions of the field are noisy. The noisy region or entirely unphysical fields usually occur due to the reasons like insufficient/non-uniform seeding particles, high laser reflection from models. Thus, to exclude such biased samples from the set of averaging samples, only the vectors lying inside the \(\pm 2\) standard deviation range are considered for averaging. The Figure 3.10 shows an instantaneous vector field and a mean vector field of velocity. Since the flow oscillations are symmetrical around the cylinder centreline, the time-averaging leads to a symmetrical mean flow as shown in Figure 3.10(b). The coloured vectors in the instantaneous field indicate the interpolated vectors resulted from the vector processing from Davis.
3.5. Flow control devices used

As discussed in section 2.4, the flow control devices can aid achieving drag reduction. The implementation of active flow control devices on skiing boot-fairing is considered impractical. Thus, only the passive techniques like zigzag strips and vortex generators are applied.

3.5.1. Zigzag strips

The sketch of the zigzag strip used for this thesis is shown in Figure 2.16(a). The width of the strip is 6 mm with the angle of 70° between consecutive peaks. The inspection of minimum strip thickness required for triggering the transition is not performed. The strips with thickness 0.4 mm and 0.6 mm are examined for their ability to force the flow transition. Two zigzag strips are applied at ± 45° from the leading edge point on the cylinder. Sound measurement close to the model wall, using microphone probe is conducted to detect the presence of turbulent boundary layer. Unlike in the case of 0.6 mm thick strip, the microphone probe failed to detect the presence of turbulent boundary layer downstream 0.4 mm thick zigzag strip. Thus, the zigzag strip with the thickness of 0.6 mm is used henceforth. The figure 3.11 shows the location of a strip on the cylinder.

![Zigzag strip applied on the reference cylinder.](image)

The relative surface roughness \(k_s\) for the reference cylinder with the zigzag strip is

\[
k_s = \frac{\epsilon}{D} = \frac{0.6\text{mm}}{5\text{cm}} = 0.012
\]

where \(\epsilon\) is relative surface thickness and \(D\) is the cylinder diameter.

3.5.2. Vortex generators (VG)

The boundary layers on the models are tripped to reduce the drag as explained in the previous section. To seek further drag reduction, vortex generator strips are used along with the zigzag strips. The sketch of a vortex generator strip is shown and parameters are indicated in Figure 3.12. The height (h) of the VGs is 3 mm, chord (c) is 10 mm, width (w) of the strip is 15 mm and step size is 1 mm approximately. The VGs are inclined at an angle of approx. 15° w.r.t. the flow direction.

The bottom side of the strip has a glue-surface and it can be stick to the model. The strip material is stiff and the width of the strip is too large to follow the curvature of the cylinder without bending.
Hence, the VG strips are applied only to the wings. For wings at a non-zero angle of attack, when the flow separation is not prevented by the zigzag strips alone then the VGs are mounted at the maximum thickness. Also, the VGs are mounted on wing B trailing edge at zero angle of attack to examine its effect on the wake deficits behind the finite thick edge.
Data reduction techniques

The previous chapter described the procedure to obtain mean velocity field around the models. After a velocity field is obtained, instantaneous flow properties like forces, pressure, vorticity, velocity divergence, acceleration, turbulence dissipation rate or statistical quantities like time average velocity field and Reynolds stresses can be derived from it. This chapter describes how the velocity field information obtained from PIV is used to compute pressure field and the drag force acting on a model. The main objective of PIV experiment is drag quantification. As discussed in section 2.8, reconstruction of pressure field from PIV is an integral part of drag computation. Thus, this chapter elaborates on the pressure reconstruction and boundary conditions used, followed by drag computation and uncertainty quantification of drag contributing terms.

4.1. Pressure reconstruction and the boundary conditions

The two-dimensional pressure Poisson’s equation (4.1) is solved over the domain using second-order centred difference spatial discretisation scheme. The theoretical description of pressure reconstruction is given in the section 2.8. PPE is an elliptical partial differential equation and contains two second-order derivatives, one in each direction x and y. In order to solve this equation, appropriate boundary conditions are necessary.

\[
\frac{\partial^2 p}{\partial x^2} + \frac{\partial^2 p}{\partial y^2} = -\rho \left\{ \left( \frac{\partial \bar{u}}{\partial x} \right)^2 + 2 \frac{\partial \bar{u}}{\partial x} \frac{\partial \bar{v}}{\partial y} + \frac{\partial^2 \bar{u}' u'}{\partial x^2} + 2 \frac{\partial \bar{u}' u'}{\partial x \partial y} + \frac{\partial^2 \bar{v}' v'}{\partial y^2} \right\}
\]  

(4.1)

Source term (S)

The pressure gradients in x and y directions are obtained from the Reynolds’ averaging of the momentum equation and neglecting the viscous terms. These pressure gradients are written in equation 4.2:

\[
\frac{\partial \vec{p}}{\partial x} = -\rho \left\{ \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{u}' u'}{\partial x} + \frac{\partial \bar{v}' v'}{\partial y} \right\}
\]

\[
\frac{\partial \vec{p}}{\partial y} = -\rho \left\{ \frac{\partial \bar{v}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{u}' v'}{\partial x} + \frac{\partial \bar{v}' v'}{\partial y} \right\}
\]

(4.2)

All the terms on RHS in equation 4.2 can be derived from PIV data. The velocity gradients are numerically computed on the discrete PIV grid by central difference method for the interior point and
4. Data reduction techniques

by a direct-difference method for the boundary points.

For interior points:

$$\frac{\Delta u}{\Delta x} = \frac{u_{i+1,j} - u_{i-1,j}}{2\Delta x}$$

For boundary points:

$$\left(\frac{\Delta u}{\Delta x}\right)_i = \frac{u_{i+1,j} - u_{i,j}}{\Delta x}$$ (4.3)

After the pressure gradients are known at every grid point and the boundary conditions are prescribed, numerical integration on the domain produces the pressure field. Figure 4.1 shows the computational domain of PPE and the boundary conditions to be applied are highlighted on the edges of the domain. The boundary conditions are also listed in Table 4.1.

![Figure 4.1: Mean velocity field on which pressure computation is carried out with pressure boundary conditions.](image)

<table>
<thead>
<tr>
<th>Boundary conditions</th>
<th>Top</th>
<th>Bottom</th>
<th>Left</th>
<th>Right</th>
<th>Wall</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Dirichlet BC</td>
<td>Dirichlet BC</td>
<td>Neumann BC</td>
<td>Neumann BC</td>
<td>Wall normal Neumann BC</td>
</tr>
<tr>
<td></td>
<td>$P = P_{bernoulli}$</td>
<td>$P = P_{bernoulli}$</td>
<td>$\frac{\partial P}{\partial x}$ (equation 4.2)</td>
<td>$\frac{\partial P}{\partial x}$ (equation 4.2)</td>
<td>$\frac{\partial P}{\partial n}$ (calculated using $\frac{\partial P}{\partial x}$ and $\frac{\partial P}{\partial y}$)</td>
</tr>
</tbody>
</table>

Table 4.1: Boundary conditions

At the top and bottom boundaries, the flow is mainly unaffected by the presence of the model and is streamlined. Thus, Dirichlet boundary condition is used to assign a specific value of pressure. The pressure values are computed using Bernoulli’s equation with the assumption of constant total pressure along a streamline. Since the top and bottom boundaries lay in external flow, the velocity gradients are weaker and hence the application of Dirichlet condition is justified.

The left boundary mainly deals with the potential flow and the right boundary deals with the viscous flow which is reflected in the stronger velocity gradient along this boundary, especially the gradient along the y-direction. Thus, it is advisable to implement the Neumann (gradient boundary) condition. A wall normal boundary condition at the model wall is prescribed. The pressure gradients in
x and y-direction are known at the wall. Thus, the wall-normal pressure gradient is calculated from
the known pressure gradients and is prescribed at the wall.
The velocity fields from PIV usually contain discontinuities at the boundaries and close to the model
wall. These discontinuities act as sources or sinks in the pressure computation. Due to the diffusive
nature of PPE, local regions around these discontinuities will have non-physical pressure values. To
avoid the occurrence of such regions in the pressure field computation, few grid points (≈ 8 – 10)
close to the model wall and outer boundaries are ignored.

PIV-pressure is validated by comparative assessment of the results obtained using Pitot-static
tube. The Pitot-static pressure measurements are performed along the wake centreline of the cylin-
der and wing A. Figure 4.2 shows the comparison between PIV-pressure and the Pitot-static tube
pressure. PIV-pressure and Pitot-static measurements exhibit near-identical trend for both the mod-
}

![Figure 4.2: Comparison between the PIV-pressure and Pitot-static pressure values in the wake centerline of cylinder and wing A.](image)

els. The pressure in the near wake of a cylinder is lower than the free stream and it gradually recovers
as moving further downstream. Whereas, at zero angle of attack, wing A does not exhibit flow separ-
ation and the flow resembles potential flow. The pressure in the vicinity of the trailing edge is higher
than the free stream pressure and it starts to drop towards free stream value while moving further
downstream. Thus, the coefficient of pressure shows an opposing trend in Figure 4.2 for cylinder
and wing A.

However, the Pitot-static measurements slightly underestimate static pressure compared to PIV-
pressure. This minor difference in the static pressure prediction by the use of two different methods
is characterised by the dissimilar nature of measurement techniques. Unlike PIV, Pitot-static probe
measurements are intrusive. The insertion of the probe disturbs the flow it measures. Moreover,
the diameter of measurement probe is larger than the trailing edge thickness of wing A. The vertical
holes for static pressure measurement lies outside the thin wake of wing A and introduce error to
the measurement. Thus, insertion of the probe has spatial modulation effect in case of wing A.
4. Data reduction techniques

4.2. Drag computation

The drag computation is straightforward after velocity and pressure field around the object is known. In the current work, the control surface approach is used to compute the drag force acting on the object. The benefit of this method over the traditional balance measurement is that along with the total drag it also provides an understanding of how the quantities contributing to drag varies in space. The three terms which constitute the time-averaged drag force on an object are expressed in equation 4.4.

\[ D = \rho \int_0^\infty \overline{u}(u_\infty - \overline{u}) \, dy - \rho \int_0^\infty \overline{u'u'} \, dy + \int_0^\infty (p_\infty - \overline{p}) \, dy \]  

(4.4)

Here \( y \) is direction normal to the object and flow velocity. \( \overline{u} \) is mean streamwise velocity averaged over time, \( u' \) is its fluctuating component. \( u' \) is the standard deviation from the mean value and is expressed as \( u' = u - \overline{u} \).

1. **Momentum deficit**: This term represents the reduction of fluid momentum in the wake due to the presence of object opposing the fluid motion. The calculation of this term depends entirely on the mean flow velocity which is available from PIV experiment. After the instantaneous flow velocity fields are known, the averaging is done in the following way to obtain the mean velocity field:

\[ \overline{u} = \frac{1}{N} \sum_{i=1}^{N} u_i \]  

(4.5)

Here \( N \) is number of instantaneous fields. These fields are captured at different instants of time with uniform time spacing between consecutive fields. Thus, \( \overline{u} \) represents the time-averaged velocity.

2. **Reynolds stress**: Reynolds averaged Navier-Stokes (RANS) equations are primarily used to describe turbulent flows. In RANS, the instantaneous quantities are decomposed into mean and fluctuation components then the time-averaging is performed. Reynolds stress terms emerge from Reynolds averaging of the non-linear term in the Navier-Stokes equations. Reynolds stresses are mathematically represented as:

\[ Re_{i,j} = -\rho \overline{u'u'} \]  

(4.6)

Reynolds stresses are classified into normal stresses \( (i = j) \) and shear stresses \( (i \neq j) \). \( \overline{u'u'} \) represents the Reynolds normal stress in the streamwise direction. Consideration of Reynolds stresses in the drag calculation is important since they account for the rate of mean momentum transfer due to the turbulent fluctuations. Statistically, normal stress for streamwise velocity component \( \overline{u'^2} \) is defined as the variance of corresponding velocity component (Sciaccitano [38]).

\[ \overline{u'^2} = \frac{1}{N-1} \sum_{i=1}^{N} (u_i - \overline{u})^2 \]  

(4.7)

This term has its maxima in the region corresponding to high turbulent fluctuations and reduces to zero where the fluctuations are absent.

3. **Pressure deficit**: The flow region downstream an object where local pressure is lower than the free stream pressure, contributes to drag on the object. The pressure deficit term quantifies the difference between free stream pressure and local pressure. This term is integrated over a linear path in the wake, normal to the free stream velocity. In case of bluff bodies,
integrals have higher positive values in the near wake due to the presence of low-pressure region formed because of the separated flow. The low-pressure region in the near wake of the cylinder can be observed in figure 5.1(b). Integrals taken outside of this low-pressure region will have comparatively lower value. The local pressure in the wake gradually recovers to free stream value after the flow reattachment and the pressure deficit term decays to 0.

### 4.3. Drag uncertainty analysis

To determine the interval which contains the error in measurement, uncertainty quantification is performed (Sciachitano [38]). PIV experiment only yields velocity vector field information. The physical quantities like mean pressure and drag force are derived from PIV measurements. According to Sciachitano [38], the uncertainties contained in the instantaneous field quantities propagate to the quantities derived from the former one. The uncertainties of derived instantaneous flow field quantities can be affected by the correlation of velocity components. While for statistical quantities, the uncertainty is mainly caused due to the finite number of samples.

Following the work of Pattnaik [30] and Sciachitano [38], the uncertainty in computed drag by using control volume approach on PIV data is given as:

\[
U_{\text{Drag}}^2 = \left( \frac{\partial F}{\partial u} \right)^2 U_{\pi}^2 + \left( \frac{\partial F}{\partial p} \right)^2 U_{\bar{p}}^2 + \left( \frac{\partial F}{\partial u' u'} \right)^2 U_{u' u'}^2
\]  

(4.8)

Here \( U_{\pi} \), \( U_{\bar{p}} \) and \( U_{u' u'} \) are uncertainty in mean velocity, in mean pressure and normal Reynolds stress respectively. These uncertainties in mean quantities are discussed in the forthcoming subsections. The terms inside the brackets on RHS are obtained by differentiating the equation 4.4.

\[
\frac{\partial F}{\partial u} = \rho \int_0^{\infty} (u_\infty - 2\bar{u}) \, dy
\]

\[
\frac{\partial F}{\partial p} = -\rho \int_a^b dy
\]

\[
\frac{\partial F}{\partial u' u'} = -\rho \int_a^b dy
\]

(4.9)

where \( a \) and \( b \) in the equation 4.9 represent two boundary points of a drag station line. The uncertainty contribution due to individual terms in the total drag force is presented by equations 4.8 and 4.9. The uncertainty in the computation of all three terms depends on the distance normal to the wake (y-direction) over which deficit integrals are taken. As seen from equation 4.9, the pressure term and Reynolds term uncertainties are only dependent on this distance. However, the momentum deficit uncertainty also depends on free stream velocity and mean flow velocity. The negative sign in front of the mean velocity \( \bar{u} \) in equation 4.9 reflects that the lower the mean velocity, higher would be the uncertainty in the computation of momentum deficit.

#### 4.3.1. Uncertainty of the mean velocity (\( U_{\pi} \))

Assuming that the instantaneous velocity measurements show Gaussian distribution around the mean value, the uncertainty related to the mean velocity is given as:

\[
U_{\pi} = \frac{\sigma_{\pi}}{\sqrt{N}}
\]

(4.10)
Here, $\sigma_u$ is the standard deviation of velocity from the mean and $N$ is a number of samples. The mean velocity is a statistical quantity and hence the uncertainty depends on finite sample size. The accuracy of the mean velocity increase with the number of samples. The mean estimation would match the exact value if $N \to \infty$. The uncertainty in mean is directly proportional to the standard deviation. For a given $N$, the flow region with higher fluctuations (near wake of bluff bodies) will have larger uncertainties compared to outer flow region where the fluctuations are smaller. As shown in Figure 4.3, the uncertainty in the mean $x$-velocity is computed at three stations and presented in Table 4.2.

![Figure 4.3: The $x$-velocity standard deviation in the flow field.](image)

The station A lies closer to the cylinder where fluctuations are largest due to the free shear layer springing from the body (Bearman [7]). Hence, stations A reports the largest value of standard deviation in $x$-velocity and in turn, the largest uncertainty in the mean $x$-velocity. The station B is located in far wake where the fluctuations are smaller compared to station A. The standard deviation and uncertainty at station B is smaller by a factor of 2. Since the station C is chosen in the free stream, the standard deviation and uncertainty are found to be the least. The values of $\sigma_u$ and $U_u$ are measured lower than that of station A by a factor of 18.

<table>
<thead>
<tr>
<th>Stations</th>
<th>Standard deviation $\sigma_u$ (m/s)</th>
<th>Uncertainty $U_u$ (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>13.23</td>
<td>0.42</td>
</tr>
<tr>
<td>B</td>
<td>6.07</td>
<td>0.19</td>
</tr>
<tr>
<td>C</td>
<td>0.73</td>
<td>0.02</td>
</tr>
</tbody>
</table>

Table 4.2: Uncertainty in the mean streamwise velocity at different stations.
### 4.3.2. Uncertainty of the Reynolds normal stress \((U_{uu})\)

Reynolds normal stress on which the drag determination depends in this thesis work is the normal stress for the x-velocity component. It is defined as the variance of the u-velocity component \(\sigma^2_u\).

\[
Re_{uu} = \overline{u'u'} = \sigma^2_u = \frac{1}{N-1} \sum_{i=1}^{N} (u_i - \overline{u})^2 \tag{4.11}
\]

Assuming that the samples are uncorrelated in time and follow a normal distribution of standard deviation. The uncertainty of the variance i.e. Reynolds normal stress is given as:

\[
U_{u'u'} = \sigma^2_u \sqrt{\frac{2}{N-1}} \approx \frac{u'u'}{\sqrt{2N}} \tag{4.12}
\]

According to Sciacchitano [38], the uncertainty in the Reynolds stress composes of the measurement uncertainty and the uncertainty due to the spurious fluctuations. For application purpose, the uncertainty due to the latter can be ignored when the actual flow velocity fluctuations are larger than the measurement error. Hence, equation 4.12 is competent to represent the uncertainty in Reynolds stress.

The uncertainty analysis for Reynolds stress is carried out at the stations shown in Figure 4.3 and the results are listed in the Table 4.3.

<table>
<thead>
<tr>
<th>Stations</th>
<th>Reynolds normal stress (\overline{u'u'}(m^2/s^2))</th>
<th>uncertainty (U_{u'u'}(m^2/s^2))</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>164</td>
<td>7.33</td>
</tr>
<tr>
<td>B</td>
<td>36.41</td>
<td>1.63</td>
</tr>
<tr>
<td>C</td>
<td>0.7</td>
<td>0.031</td>
</tr>
</tbody>
</table>

Table 4.3: Uncertainty in the Reynolds stress \(u'u'\) at different stations.

### 4.3.3. Uncertainty in time-averaged pressure computation \((U_p)\)

Thus far the uncertainty quantification of mean x-velocity and Reynolds normal stress is carried out. Both of these are statistical quantities obtained from instantaneous PIV fields. Whereas the pressure is a derived quantity and is computed by solving Poisson’s equation on PIV field. For uncertainty quantification in pressure computation, the linear uncertainty propagation formula is applied to all individual terms of equation 4.1.

Following the work of Pattnaik [30], the uncertainty in mean pressure \(U_p\) is given as:

\[
U_p = \frac{\rho \Delta x}{\sqrt{12}} \sqrt{2(U_p)^2 \left(\frac{\partial U_p}{\partial x}\right)^2 + (\frac{\partial U_p}{\partial y})^2 + \left(\frac{\partial U_p}{\partial y}\right)^2 + \left(\frac{\partial U_p}{\partial y}\right)^2} + \frac{12}{\Delta x^2} (U_{u'u'}^2)^2 + \frac{1}{\Delta x^2} (U_{v'v'})^2 \tag{4.13}
\]

where \(\Delta x\) is grid length, \(U_p\) is uncertainty in the velocity component. This method involves an assumption that the uncertainties in all the velocity component are equal to \(U_p\) and uncertainties in pressure is \(U_p\), at all grid points. Also, the mean velocity component errors are uncorrelated to each other. The uncertainty quantification of mean pressure is performed at station A, B and C as shown in Figure 4.3 and resulting values are listed in the Table 4.4.
### 4. Data reduction techniques

#### Table 4.4: Uncertainty in the mean pressure $\overline{p}$ at different stations.

<table>
<thead>
<tr>
<th>Stations</th>
<th>Uncertainty $U_p$ (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>5.89</td>
</tr>
<tr>
<td>B</td>
<td>0.10</td>
</tr>
<tr>
<td>C</td>
<td>0.12</td>
</tr>
</tbody>
</table>

#### 4.4 Grid independence study

The objective of the grid independence study is to examine the effect of interrogation window (IW) size on the resultant velocity field. Two different window sizes $16 \times 16$ pixels and $32 \times 32$ pixels are considered for this analysis. The data processing using $16 \times 16$ pixels window size is computationally expensive but provides higher spatial resolution compared to $32 \times 32$ pixels windows. Figure 4.4 shows, the mean $x$-velocity contour and standard deviation in $x$-velocity contour obtained using two window sizes.

![Mean x-velocity fields around a cylinder obtained using two different window sizes.](image)

Looking at the contours of Figure 4.4, one can find them identical. Thus, for a clear understanding, the velocity and standard deviation values along the centerline and a vertical line are plotted. These lines are indicated in the Figure 4.4 with yellow lines. The centreline runs from $x = -35$ mm to $x = 200$ mm, while the vertical line extends along the width of contour at $x = 0$ mm. The Figure 4.5 (a) and (b) show the plots for mean velocity values along these lines whereas 4.5 (c) and (d) present the plots for standard deviation values on these yellow lines.

It can be seen from the Figure 4.5 (a) that the mean $x$-velocity along the centreline obtained using two different window sizes is near-identical. The maximum difference between the $x$-velocity
4.5. Free stream velocity correction

The experiments are carried out at the velocities of 20, 25 and 30 m/s in a closed test section. The area blockage ratio is 12.5% which has a considerable effect on free stream velocity. Due to the presence of test-section walls and the model, the flow experiences acceleration. The Figure 4.6 values obtained using these two window sizes is 0.57 m/s for free stream velocity of 27.5 m/s. This accounts for less than 0.5% of the free stream velocity value. This maximum difference occurs in the recirculating region behind the cylinder where velocity gradients are larger. The smaller window size is competent of resolving sudden velocity gradient more effectively compared to larger windows. Figure 4.5(b) shows the mean x-velocity profiles extracted at $x = 0$ position. Both the profiles are near-identical and coinciding on each other. Figure 4.5 (c) displays the standard deviation in x-velocity along the centreline. Among the four plots in figure 4.5, (c) shows the highest disagreement between the values obtained using two different window sizes. The maximum difference in standard deviation is 0.72 m/s. The profile of standard deviation in x-velocity at $x = 0$ is shown in Figure 4.5(d) and the plots are near-identical. However, the major deviation is shown at the middle part which lies in the near wake. Note that the deviation at the bottom of the profile is due to the noises present in both the cases.

Figure 4.5: Mean x-velocity fields around a cylinder obtained using two different window sizes.
shows the free stream velocity on the top boundary for inlet velocity of 25 m/s.

![Figure 4.6: Velocity variation on the top boundary.](image)

As seen in the Figure 4.6, the flow has acceleration and attains the highest velocity of nearly 29 m/s. This peak in the velocity occurs at the streamwise location roughly corresponding to the cylinder location. To find the effective free stream velocity, the average over a range shown in Figure 4.6 (inside the box) is taken. The PIV-pressure is obtained using the effective free stream velocity values and it is in good agreement with the Pitot-static tube measurements. The effective velocities for all three cases are listed in the table 4.5. It can be observed that the effective free stream velocities are roughly 10% higher than the pre-set velocities.

<table>
<thead>
<tr>
<th>Free stream velocity (m/s)</th>
<th>Effective free stream velocity (m/s)</th>
<th>Percentage increase in velocity (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>22.2</td>
<td>11</td>
</tr>
<tr>
<td>25</td>
<td>27.5</td>
<td>10</td>
</tr>
<tr>
<td>30</td>
<td>33.2</td>
<td>10.7</td>
</tr>
</tbody>
</table>

Table 4.5: Effective free stream velocities for different inlet velocities.
The previous chapter described the method of computing drag using time-averaged PIV velocity field. Thus far, it was argued that the drag reduction on the cylinder can be achieved by geometric modification. This chapter presents the findings of this thesis i.e. drag force acting on the reference model and drag reduction achieved by implementing different geometric modifications. The analysis also takes different velocity and angles of attack into account and the effect of implementation of passive flow control techniques.

5.1. Drag quantification on the reference geometry

Following the methodology described in the previous chapter, the computed drag on reference cylinder is presented in this section. The contribution of three terms in the total drag and their trend as a function of downstream distance is established. Figure 5.1 presents the mean x-velocity field, mean pressure field and the plots of drag contributing terms. These plots correspond to free stream velocity of 27.5 m/s and Reynolds number of $9.8 \times 10^4$.

As observed from the mean pressure field in Figure 5.1(b), the base pressure coefficient of the cylinder is -1.2. The value of base pressure coefficient at Reynolds number of $1 \times 10^5$ found by Roshko [35] and Norberg [26] was -1.16. At the same Reynolds number Moll [25] found base pressure coefficient of -1.3. The discrepancies between these values could be because of difference in experimental setup, blockage ratio and incoming turbulence level.

5.1.1. Trend analysis of drag contributing terms for cylinder:

Knowing the velocity and pressure field around a model, the deficit terms can be computed. Figure 5.1(c) shows the trend of these terms in the wake of a cylinder with free stream velocity of 27.5 m/s. As seen in Figure 5.1(c), the momentum deficit term contributes to negative $C_d$ for slightly less than a diameter length. This negative contribution is due to the reverse flow in recirculation bubble. Thus, it can be deduced that the length of the recirculation bubble measured from cylinder base is slightly lower than a cylinder diameter. The momentum deficit rapidly grows from $\approx -1$ to $\approx +0.8$ in approximately 2.5 diameter length, but the increment thereafter is gradual.

The pressure deficit term exhibits decreasing trend with the downstream distance. The term attends its maxima in the near wake of the cylinder due to the presence of low-pressure region i.e. the recirculation bubble. Alike momentum deficit term, this term also shows a rapid change in the...
2.5 diameter length region where it decreases from \( \approx 2.6 \) to \( \approx 0.6 \). Moving further downstream, this term slowly approaches towards null value, attainment of the null value reflects free stream pressure recovery. Due to the limited field of view (FOV), pressure recovery is not observed here. Nevertheless, the negative slope of pressure deficit term ensures that the pressure recovery would be achieved further downstream. Ultimately, this pressure deficit term would plateau at 0.

As inferred from equation 4.4, the Reynolds stress term contributes negatively to the total drag. As mentioned earlier, it accounts for the mean flow momentum transfer due to turbulent fluctuations. Hence attains its lowest (or highest negative) value in the region where the amplitude of turbulent fluctuations are larger. The Reynolds stress term attains its minimum value in the cylinder near wake and then gradually increases, tending towards a null value. At a sufficiently downstream location where turbulent fluctuations have faded out, the value of this term would become 0.

![Mean x-velocity field around cylinder](a)

![Mean pressure field around cylinder](b)

![Plot showing the trends of drag terms with streamwise distance](c)

Figure 5.1: Mean fields and drag terms plot for reference cylinder for free stream velocity of 25 m/s.

The resultant drag coefficient is computed by subtracting the \( C_d \) due to Reynolds stress term from the algebraic sum of \( C_d \) due to pressure and momentum terms (refer equation 4.4). In principle, the total drag on a model should be constant and independent of the location of drag measure-
ment station. However, the total drag in Figure 5.1(c) varies slightly with the streamwise distance. The total drag varies comparatively more in the near wake region where the fluctuations in velocity are larger. Due to these larger fluctuations, the uncertainty in the mean velocity and hence in PIV-mean pressure is higher (Terra et al. [43]). Thus, the drag measurements performed at the stations fairly close to the cylinder base are less accurate. At a downstream location where the turbulent fluctuations have faded out and the free stream pressure is recovered, the total drag force on the object will be given by the momentum deficit term.

The coefficient of drag is calculated by taking a mean of $C_d$ values measured for the drag stations lying between $x/D = 2.5 − 4.5$. The $C_d$ values for cylinder at different velocities are listed in Table 5.1.

<table>
<thead>
<tr>
<th>Effective free stream velocity (m/s)</th>
<th>Reynolds number</th>
<th>Coefficient of drag $C_d$</th>
</tr>
</thead>
<tbody>
<tr>
<td>22.2</td>
<td>$\approx 79500$</td>
<td>1.07</td>
</tr>
<tr>
<td>27.5</td>
<td>$\approx 98500$</td>
<td>1.06</td>
</tr>
<tr>
<td>33.2</td>
<td>$\approx 120000$</td>
<td>1.1</td>
</tr>
</tbody>
</table>

Table 5.1: Cylinder drag coefficient at different free stream velocities.

Reynolds numbers corresponding to the three different velocities belong to the same cylinder flow regime i.e. sub-critical. Thus, the coefficient of drag does not change significantly and remains close to unity. For a smooth cylinder subjected to cross-flow in sub-critical flow regime, Achenbach and Heinecke [2] found drag coefficient to be 1.4. The well-known $C_d(R)$ curve by Wieselsberger [53] suggested $C_d$ value of 1.2. Fage [13] found $C_d$ to be approximately equal to 1.0. Again the discrepancies in these values are due to different experimental setup, blockage ratios and blockage correction formulae used.

### 5.2. Effect of surface roughness on cylinder drag

The surface roughness of a body has a significant effect on the boundary layer characteristics and in turn, affects the overall aerodynamic performance of the body. Surface roughness greater than a critical height can trigger the boundary layer transition. As discussed in section 3.5, zigzag strips are efficient tools and are used to trip the boundary layer on the cylinder. These strips are placed at $\pm 45^\circ$ from the leading point on the cylinder. In this section, the effect of boundary layer tripping the drag on the cylinder is discussed.

The drag coefficient value for a cylinder with and without the roughness is listed in Table 5.2. Note that the same thickness of roughness element is used for these three cases. Higher the free stream velocity earlier would be the attainment of critical Reynolds number for the transition. Thus, the case with higher free stream velocity has a larger proportion of turbulent boundary layer and has higher tendency to delay flow separation. Hence, the drag reduction achieved is proportional to the free stream velocity. The highest percentage of drag reduction is achieved for the free stream velocity of 33.2 m/s.

The $C_d$ value corresponding to the Reynolds number of $9.85 \times 10^4$ for a rough cylinder (zigzag strip) is found to be 0.69 which is in good agreement with Moll [25]'s experiment. Moll [25] found $C_d$ equal to $\approx 0.7$ for a cylinder with zigzag strip at Reynolds number of $1 \times 10^5$.

Figure 5.2 shows the velocity and pressure on the centreline behind a rough cylinder for three different free stream velocities. From Figure 5.2, the length of recirculation region is defined as the distance on the x-axis for which the velocity on the centreline reaches a numeric value of 0. Looking at the velocity centreline plots, it can be inferred that the recirculation region length is
Effective free stream velocity (m/s) & Smooth cylinder $C_d$ & Rough cylinder $C_d$ & Percentage decrease in $C_d$ (%) \\
\hline
22.2 & 1.07 & 0.81 & 24 \\
27.5 & 1.06 & 0.69 & 35 \\
33.2 & 1.1 & 0.59 & 46 \\
\hline
Table 5.2: Effect of roughness on cylinder drag coefficient.

The pressure coefficient at the most downstream point/ trailing edge point of the cylinder is referred as base pressure coefficient ($C_{pb}$). Looking at the pressure centreline plots in Figure 5.2, the base pressure coefficient values of the rough cylinder for three free stream velocities are different. Higher the free stream velocity, larger is the value of $C_{pb}$. The larger value of $C_{pb}$ indicates that the pressure behind the cylinder is closer to the free stream value and hence the pressure drag on the cylinder is lower. The lowest $C_{pb}$ is approximately equal to -1, for free stream velocity of 22.2 m/s. As per the Table 5.2, the rough cylinder has the highest drag at this particular free stream velocity. Thus, the base pressure coefficient gives an indicative idea of the drag coefficient.
5.2. Effect of surface roughness on cylinder drag

5.2.1. Flow field comparison for smooth and rough cylinder for $u_\infty = 27.5 \text{m/s}$

Figure 5.3 shows the streamlines close to the smooth and rough cylinder for free stream velocity of 27.5 m/s. It can be observed that the flow separation line where outer flow and reverse flow meet, is inclined at a higher angle for a smooth cylinder. Here the inclination angle of separation line is defined from the positive x-axis. Unlike the smooth cylinder, the streamlines close to the rough cylinder appear to be pushed towards the body.

![Streamline comparison](image)

(a) Smooth cylinder (zigzag strip absent)  
(b) Rough cylinder (zigzag strip present)

Figure 5.3: Streamline highlighting the flow separation line for smooth and rough cylinder mean fields.

To inspect the effect of roughness on recirculation region, x-velocity contour lines around smooth and rough cylinder are shown in Figure 5.4. It can be observed that the lengths of recirculation region in both the cases are different. The streamlines close to the rough cylinder appear squeezed towards the body in Figure 5.5 which is consistent with the observation made from Figure 5.3. The
5. Results and discussion

(a) Smooth cylinder (zigzag strip absent)  
(b) Rough cylinder (zigzag strip present)

Figure 5.5: Streamline highlighting the difference in recirculation region for a smooth and a rough cylinder mean fields.

Squeezing of streamlines slightly elongates the length of recirculation region behind the rough cylinder. As observed from Figure 5.5, the length of recirculation region for the smooth cylinder is roughly 0.6 diameter-length whereas it is approximately 0.9 diameter-length in case of a rough cylinder. Note that the lengths of these recirculation regions are measured from the cylinder base. The reconstruction of cylinder geometry in Figure 5.5 possesses slight error due to perspective error from PIV data.

Figure 5.6 shows the velocity and pressure profiles for a smooth and a rough cylinder at a distance of one diameter in wake of the cylinder. The velocity profile for smooth cylinder attains only positive values which indicate that the station lies outside the recirculation region. Whereas the peak value for rough cylinder velocity profile has negative numeric value. Thus, it can be concluded that the recirculation region for a rough cylinder is larger than one diameter-distance measured from the base point. Measured from the cylinder base-point, Moll [25] found the lengths of recirculation region for a smooth and rough cylinder to be 1.14 and 1.54 diameter-length. The recirculation region lengths for a smooth and rough cylinder is determined as 0.75 and 1.05 cylinder diameter in this research (refer Figure 5.5). Though the Reynolds number for Moll [25]'s and current experiment is close to $1 \times 10^5$, the different values of recirculation region lengths are observed. This is mainly
5.2. Effect of surface roughness on cylinder drag

due to different experimental setup and the blockage ratios. The blockage ratio in Moll [25]’s experiment is 17.5% whereas it is equal to 12.5% for the current experiment. The minimum value of velocity profile for a rough cylinder is lower compared to that of a smooth cylinder. This means that the velocity recovery in the wake of a smooth cylinder occurs earlier when traversing downstream. However, the velocity profile for a smooth cylinder is fuller compared to that of a rough cylinder which indicates that the smooth cylinder has a wider wake.

While comparing the pressure profiles in Figure 5.6, the minimum value for smooth cylinder profile is lower than that of a rough cylinder. This indicates that the pressure in the wake of a smooth cylinder is lower than its counterpart in case of a rough cylinder. Due to this lower wake pressure, the smooth cylinder has high-pressure drag component. This can be observed from the Figure 5.7. The drag coefficient due to pressure deficit term attains the value of 2.5 for a smooth cylinder whereas the peak value for its counterpart in case of a rough cylinder is slightly less than 2. Additionally, it can be inferred from the pressure term plot in Figure 5.7, that the free stream pressure recovery in the wake of the rough cylinder is faster. The Reynolds stress contributions to drag coefficient in both

Figure 5.7: Plots highlighting the difference in pressure deficit term for a smooth and a rough cylinder.

Figure 5.8: Centreline Pressure and velocity for a smooth and a rough cylinder at free stream velocity of 27.5 m/s (Re = 9.85 × 10⁴).
the cases are comparably equal but the drag coefficient due to momentum deficit term in case of a rough cylinder is smaller. Thus, due to a smaller contribution from momentum and pressure deficit terms, the total drag on a rough cylinder is smaller than that on a smooth cylinder.

In conclusion, due to the presence of zigzag strips (rough cylinder), the recirculation is slightly elongated in the streamwise direction. Whereas the wake is made narrower. The base pressure attains higher value due to delayed flow separation and results into lower pressure drag. The Figure 5.8 clearly depicts the difference in base pressure behind a rough and a smooth cylinder at Reynolds number of $9.85 \times 10^4$. The base pressure coefficient for a rough cylinder is approximately -0.75. Moll [25] found the same value of base pressure coefficient behind a cylinder with the zigzag strip at Re = $1 \times 10^5$. The value of $C_{pb}$ for a smooth cylinder is -1.2 which is lower compared to a rough cylinder. Thus, even after the slightly elongated recirculation region, the total drag acting on a rough cylinder is lower than that on a smooth cylinder.

5.3. Effect of splitter plate on cylinder drag

In this section, the effect of splitter plate on the drag coefficient of reference cylinder is investigated. Also, the changes made in the flow field due to the splitter plate presence are described. Later, the effect of boundary layer tripping on the cylinder B is examined. As mentioned earlier, Cylinder B is an assembly of reference cylinder with splitter plate attached to its base point.

<table>
<thead>
<tr>
<th>Free stream velocity (m/s)</th>
<th>Cylinder A $C_d$</th>
<th>Cylinder B $C_d$</th>
</tr>
</thead>
<tbody>
<tr>
<td>27.5</td>
<td>1.06</td>
<td>0.86</td>
</tr>
</tbody>
</table>

Table 5.3: Comparison between drag coefficient of cylinder A and cylinder B.

5.3.1. Comparison between cylinder A and cylinder B at $u_\infty = 27.5 m/s$

According to Apelt et al. [5], drag reduction on a cylinder can be achieved by attaching a downstream splitter plate. Table 5.3 presents the coefficient of drag for cylinder A and cylinder B at free stream velocity of 27.5 m/s. Due to the presence of splitter plate, the $C_d$ value drops from 1.06 to 0.86 indicating drag reduction by approximately 20%.

Figure 5.9 shows the mean streamwise velocity field around cylinder A and cylinder B. The flow streamlines clearly indicate the difference in size of the vortices formed behind the cylinders. The vortices behind cylinder A are compact in size and thus leading to a smaller recirculation region. Whereas in case of cylinder B, a pair of larger vortices are formed, one on either side of the splitter plate. The recirculation region behind cylinder B is roughly 3 times that of cylinder A. Hence, it gives an intuition that the drag force acting on cylinder B should be higher than that on cylinder A. However, the wake width in mean flow downstream the cylinder B is smaller compared to cylinder A. Figure 5.10 shows wake profile comparison for cylinder A and cylinder B at 3 diameter-distance downstream the cylinder base point. It is evident from Figure 5.10 that the wake width in the latter case (due to the presence of splitter plate) is smaller.

Figure 5.11 presents the mean pressure fields around cylinder A and cylinder B. The base pressure coefficient ($C_{pb}$) for cylinder B (-0.8) is comparatively higher than that of cylinder A (-1.2). The increment in the base pressure due to the presence of the splitter plate is in good agreement with the literature. According to Roshko [35], the base pressure coefficient increased from -1.0 to -0.5 in presence of a splitter plate with roughly 3-diameter length for Reynolds number in order of $10^4$. 
5.3. Effect of splitter plate on cylinder drag

Figure 5.9: Mean x-velocity fields for cylinder A and cylinder B.

Figure 5.10: Wake profile comparison for cylinder A and cylinder B depicting smaller wake width behind the cylinder B. The profiles are taken at $\frac{x}{D} = 3$ as shown with green lines in Figure 5.9.

Even though Roshko [35] carried out the experiment at different Reynolds number and with different splitter-plate length, the trend exhibited by $C_{pb}$ is observed to be similar. If the drag on a cylinder is computed with surface pressure integration method, cylinder B would have lower pressure drag due to the higher base pressure.

5.3.2. Effect of roughness on cylinder-splitter plate assembly

Like cylinder A, the roughness on cylinder B elongates the recirculation bubble in streamwise direction while suppressing its transverse width due to delayed flow separation. Figure 5.12 shows the x-velocity contours around the cylinder-splitter plate (‘cylinder B’) without and with the application of zigzag strips. As mentioned earlier in the section 3.5, the zigzag strips are applied to the circular cylinder at ±45° from the leading point.

As observed from Figure 5.12, the vortices are wider in the y-direction for the smooth cylinder.
5. Results and discussion

(a) Pressure field around cylinder A

(b) Pressure field around cylinder B

Figure 5.11: Pressure field around cylinder A and cylinder B indicating the difference in the base pressures.

(a) Smooth cylinder B

(b) Rough cylinder B

Figure 5.12: Mean x-velocity fields for smooth and rough cylinder B.

The same vortices for rough cylinder B look restricted in y-direction since the streamlines springing from the circular cylinder are closer to the plate. As highlighted with yellow coloured lines in Figure 5.12, two vertical lines are selected at $x/D = 1$ and $x/D = 3$ to extract x-velocity profiles. The vertical line at $x/D = 1$ starts from the splitter plate and extends to the end of the domain whereas the other line is in the wake extending from one to the other end of the domain. The mean x-velocity profiles on this lines are extracted and plotted in Figure 5.13. Figure 5.13(a) shows the profiles for line $x/D = 1$ and Figure 5.13(b) presents the profile for vertical line $x/D = 3$.

In Figure 5.13(a), the maximum velocity value is attained immediately outside the recirculation region. Thus, looking at the Figure 5.13(a), one can conclude that the width of recirculation region is larger for smooth cylinder B. Also the far field velocity value in case of a smooth cylinder is higher than rough cylinder because of the higher blockage due to larger recirculation region. The wake width is larger for a smooth cylinder, this can be inferred from Figure 5.13(b). The velocity profile for smooth cylinder B is more widely spread than that of a rough cylinder. The flow acceleration due to higher wake blockage is also observed here, a smooth cylinder having wider wake has
5.4. Wing A

According to the literature study, enveloping a cylinder by an aerofoil is the optimum way of drag reduction. Thus, two aerofoils shapes are designed and tested in the wind tunnel. This section presents the results of drag quantification on wing A for different flow velocities and at various angles of attack. The section also includes the discussion about the effect of passive flow controls on drag force. As mentioned earlier, the Reynolds number and drag coefficient are defined based on the maximum thickness of the wing which is identical to the diameter of the inscribed cylinder.

First of all, the wing A is tested at 0° angle of attack with three different free stream velocities to check for Reynolds number effect. Table 5.5 shows the coefficient of drag acting on wing A at different flow velocities. The mean x-velocity and pressure field are presented in Figure 5.14 for free stream velocity of 25.8 m/s.

The drag coefficient of wing A for the velocity of 20.8 and 25.8 m/s has no significant difference. While the $C_d$ for a free stream of 30.9 m/s is slightly lower. Wing exhibits minor Reynolds number effect as the $C_d$ drops from 0.085 to 0.072. This drop in $C_d$ is caused due to lower momentum deficit.

1For wings the coefficient of drag is calculated based on the maximum thickness instead of chord length.
5. Results and discussion

<table>
<thead>
<tr>
<th>Pre-set velocity</th>
<th>Effective free stream velocity (m/s)</th>
<th>Wing A $C_d$</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>20.8</td>
<td>0.088</td>
</tr>
<tr>
<td>25</td>
<td>25.8</td>
<td>0.085</td>
</tr>
<tr>
<td>30</td>
<td>30.9</td>
<td>0.072</td>
</tr>
</tbody>
</table>

Table 5.5: Drag coefficient for wing A at 0° angle of attack for three different free stream velocities.

![Velocity and Pressure Fields](image)

(a) Mean x-velocity field.  
(b) Mean pressure field.

Figure 5.14: Mean fields around wing A for free stream velocity of 25.8 m/s.

![Drag Contributions](image)

Figure 5.15: Drag contributing terms plot for three different free stream velocities. The ‘red’-momentum deficit term, ‘blue’- pressure deficit term, ‘green’- Reynolds stress term and ‘black’- total coefficient of drag. ‘solid lines’ represents the $C_d$ for 20.8 m/s, ‘dashed lines’ represents the $C_d$ for 25.9 m/s and ‘dotted lines’ represents the $C_d$ for 30.9 m/s.

The Figure 5.15, shows the three drag contributing terms and total drag as a function of downstream distance for 3 different flow velocities.

As evident from Figure 5.15, the contribution of pressure and Reynolds stress terms towards the drag for all three velocities are significantly identical. Whereas for momentum term, the $C_d$ is near-identical only for free stream velocities of 20.8 and 25.8 m/s. The momentum term in case of 30.9 m/s free stream shows slightly lower contribution in drag. Thus, the total $C_d$ is lower in case of 30.9
5.4. Wing A

m/s case. Figure 5.16 shows the velocity profiles at a station 2 diameter-distance downstream in the wake of wing A for all three velocities.

The trend exhibited by the pressure deficit term for wing A (Figure 5.15) is dissimilar to the trend for cylinder (Figure 5.1(c)). This dissimilarity is due to the fact that wing A is a streamlined body and experiences no flow separation at 0°. The flow around wing A mainly resembles the potential flow and has an aft stagnation point (Figure 5.14(b)) with local pressure value higher than that of the free stream. While the cylinder being a bluff body has separated flow in the wake with lower local pressure. For both the objects, the pressure gradually recovers to free stream value by moving downstream of the object. Thus, the pressure deficit term tends to approach to null value in both Figure 5.15 and 5.1(c).

5.4.1. Wing A at non-zero angles of attack

Though the wings are designed to operate at 0° angle of attack, the assessment at non-zero angles of attack is also carried out to inspect the performance. The flow fields around the wing at angles of attack of 3° and 5° are nearly identical. For both these non-zero angles, wing A experiences flow separation close to the leading edge and a recirculation region is formed (Figure 5.17(a)). The length of the region is of the order of wing chord-length. Thus, the wing experiences large increment in the drag force.

As the flow fields for these two angles of attack are nearly identical, the coefficients of drag ($C_d$) are equal to 0.48. Notice the dramatic increase in the $C_d$ value compared to zero angle of attack. It increases from 0.085 to 0.48, showing the increment by a factor of more than 4. To avoid this enormous increase in drag, the zigzag strips are applied on the circular nose of the wing at an angle of 70°–75° from the leading edge point. As mentioned earlier, these zigzag strips initiate the boundary layer transition and thus can delay or avoid the flow separation. The positions of zigzag strips on wings are determined by trial and error method.

Figure 5.17 highlights the difference in mean x-velocity and pressure fields with and without boundary layer tripping. To compare the effect of boundary layer tripping on the coefficient of drag, the $C_d$ values for a wing with and without the tripping is presented in Table 5.6.

An interesting observation can be made by comparing the first row of Table 5.6 with the second row of Table 5.5. The drag coefficient of wing A at velocity of 25.8 m/s for $\alpha = 3°$ with tripping is
5. Results and discussion

(a) Mean x-velocity field, $\alpha = 3^\circ$.

(b) Mean x-velocity field with boundary layer tripping, $\alpha = 3^\circ$.

(c) Mean pressure field, $\alpha = 3^\circ$.

(d) Mean pressure field with boundary layer tripping, $\alpha = 3^\circ$.

Figure 5.17: Mean fields around wing A at $\alpha = 3^\circ$ with and without boundary layer tripping for free stream velocity of 25.8 m/s.

Table 5.6: Drag coefficient for wing A at different angles of attack with and without tripping.

<table>
<thead>
<tr>
<th>Angle of attack</th>
<th>$C_d$ without tripping</th>
<th>$C_d$ with tripping</th>
<th>$C_d$ with tripping and VGs</th>
</tr>
</thead>
<tbody>
<tr>
<td>$3^\circ$</td>
<td>0.48</td>
<td>0.075</td>
<td>-</td>
</tr>
<tr>
<td>$5^\circ$</td>
<td>0.48</td>
<td>0.18</td>
<td>-</td>
</tr>
<tr>
<td>$8^\circ$</td>
<td>-</td>
<td>0.3</td>
<td>0.53</td>
</tr>
</tbody>
</table>

slightly lesser than that for $\alpha = 0^\circ$. In the latter case, there is no flow separation hence the use of tripping is avoided. The boundary layer is turbulent in case of $3^\circ$ angle of attack due to the presence of zigzag strips. It is discovered that the momentum deficit is lower due to the presence of turbulent boundary layer. The tripping of boundary layer not only avoids the flow separation but it also decreases the momentum deficit in the wake. Consequently, the drag coefficient ($C_d$) value is reduced. The velocity profiles at half-chord distance (1.5 diameter-distance) downstream the wing for both these configurations are shown in Figure 5.18. This Figure advocates the argument made about the tripping reducing the momentum deficit in the wake, since the profile without tripping has deeper
Figure 5.18: The velocity profiles taken at 0.5 chord-distance in the wake of wing A.

Figure 5.19: (a) and (b): Mean x-velocity fields around wing A with zigzag tripping and with/without vortex generators, at $\alpha = 8^\circ$, (c) and (d): Mean x-velocity field close to the maximum thickness location in presence and absence of VGs. The free stream velocity of 25.8 m/s.
peak showing higher momentum deficit. Also, the peaks in the velocity profiles are offset from one another because of the difference in angles of attack of the wing.

Even though the boundary layer is tripped, the wing starts exhibiting trailing edge flow separation at $\alpha = 5^\circ$ and $C_d$ attains a value of $0.18$. To suppress flow separation, vortex generator strips along with the zigzag tripping are implemented. The vortex generator strips are applied at the maximum thickness location of the wing.

Figure 5.19(a) and (b) shows mean x-velocity fields around wing A without and with VGs. Note that both fields have boundary layer tripping in common. Though Vortex generators are supposed to bring high momentum fluid close to the body and delay/avoid flow separation, as it can be seen from Figure 5.19(b) flow separation is encouraged due to the presence of vortex generators.

To further inspect the uncharacteristic behaviour of vortex generators promoting the flow separation, zoomed-in flow study near the VG location is carried out. Figure 5.19(c) and (d) show the velocity field in the vicinity of maximum thickness location of wing A at an angle of attack $8^\circ$. As illustrated by the streamlines in the figure 5.19 (d), the flow separation starts at the leading edge of the vortex generator leaving it in the separated region, thus the VGs are non-operational and cannot prevent flow separation. To the best understanding of the author, this could have happened due to the significant step size of the vortex generator strips as shown in Figure 3.12. To better utilise vortex generators, the author recommends having them embedded into the surface.

### 5.5. Wing B

The profile of wing A has a sharper gradient which makes it prone to flow separation at non-zero angles of attack. The wing B is designed to have a smoother slope and perform without flow separation for a larger range of angles of attack. In this section, the drag quantification on wing B is discussed at zero and non-zero angles.

#### 5.5.1. Wing B at zero angle of attack with three different free stream velocities.

The wing B is tested for Reynolds number effect by performing the drag determination at three different velocities like the other models. The drag coefficient values at these velocities are presented in Table 5.7. The flow field around the model for all three velocities is near-identical and thus the drag coefficient is also significantly the same. The time-averaged x-velocity and pressure for velocity of $25.45$ m/s is shown in Figure 5.20. As observed from Figure 5.20(a), small recirculation region at the trailing edge is formed. The counterpart of this region in the pressure field has lower local pressure. The flow closes after this region and the reattachment is a stagnation point which is reflected in higher pressure region behind the trailing edge in Figure 5.20(b). The velocity and pressure in the wake approach towards free stream values as moving further downstream.

The flow detaches from the bluff trailing edge leading to higher momentum and pressure deficit. Further drag reduction is possible by suppressing this flow separated region. The vortex generators are implemented close to the trailing edge with the intention of entrapping higher momentum fluid
in the near wake and thus reducing the drag. There are two configurations taken into consideration, VGs at 10% chord-distance upstream from the trailing edge and VGs at the trailing edge. The drag coefficient due to the presence of trailing edge (TE) vortex generators is computed for free stream velocity of 25.45 m/s and the values are presented in Table 5.8.

<table>
<thead>
<tr>
<th>VGs position</th>
<th>Drag coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>No TE VGs</td>
<td>0.16</td>
</tr>
<tr>
<td>10% upstream TE</td>
<td>0.087</td>
</tr>
<tr>
<td>At TE</td>
<td>0.085</td>
</tr>
</tbody>
</table>

Table 5.8: Effect of trailing edge vortex generators on drag coefficient.

It is evident from the Table 5.8 that drag coefficient values are halved in the presence of trailing edge vortex generator. However, there is no significant difference observed in $C_d$ due to two different positions of VGs. The difference made in the x-velocity field due to the presence of trailing edge vortex generators is highlighted in the figure 5.21 and Figure 5.22. The figure 5.21 shows x-velocity contours in the vicinity of the wing B trailing edge, without and with the trailing edge vortex generators (TE-VG). To highlight the difference in these contours, the contour lines are plotted in a plot shown in Figure 5.22. It is evident that the recirculation region is elongated in the streamwise and transverse directions due to the presence of the TE-VG. However, the effect of vortex generators is predominant after the recirculation bubble where the outer flow is closing towards the wake leading to a smaller wake width.

The wake velocity profile is extracted at a cylinder diameter-distance in the wake (at x/D = 4 in Figure 5.22) for wing B with and without the trailing edge VG implemented. The resulting profile is shown in Figure 5.23.

Both the profiles have only positive numeric values indicating the presence of drag station outside of the recirculating region. The profile for ‘no TE-VG’ configuration is fuller indicating larger wake compared. Thus, it is concluded that the presence of trailing edge vortex generators transfers momentum into the wake from the outer flow leading to smaller wake size and consequently lower drag force on the model.
5. Results and discussion

(a) Trailing edge VGs absent.  (b) Trailing edge VGs present.

Figure 5.21: Mean x-velocity around wing B in absence and presence of trailing edge vortex generators.

Figure 5.22: Contours showing the effect of trailing edge vortex generators.

Figure 5.23: Wake profiles at $x/D = 1$ for wing B with and without the trailing edge VGs.
5.5.2. Drag determination on wing B at non-zero angles of attack.

Like the wing A, Wing B is also designed to operate ideally, at zero angle of attack. Though, in real life scenario wing might face non-zero angles of attack. Thus to quantify the drag force at these angles the analysis at various non-zero angles is carried out at the pre-set velocity of 25 m/s.

<table>
<thead>
<tr>
<th>Angle of attack $\alpha$ [°]</th>
<th>Drag coefficient $C_d$ [−]</th>
<th>Flow control technique</th>
<th>Angle of attack $\alpha$ [°]</th>
<th>Drag coefficient $C_d$ [−]</th>
<th>Flow control technique</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.16</td>
<td>-</td>
<td>5</td>
<td>0.62</td>
<td>-</td>
</tr>
<tr>
<td>3</td>
<td>0.12</td>
<td>-</td>
<td>5</td>
<td>0.62</td>
<td>-</td>
</tr>
<tr>
<td>5</td>
<td>0.24</td>
<td>zz</td>
<td>8</td>
<td>0.82</td>
<td>zz</td>
</tr>
<tr>
<td>8</td>
<td>0.3</td>
<td>zz+vg</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>10</td>
<td>0.31</td>
<td>zz+vg</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>12</td>
<td>0.4</td>
<td>zz+vg</td>
<td>15</td>
<td>0.78</td>
<td>zz+vg</td>
</tr>
</tbody>
</table>

Table 5.9: Drag coefficient for wing B at different angles of attack with and without flow control techniques. zz: zigzag trips and vg: Vortex generators.

Unlike wing A, the flow stays attached to the wing B till 3° angle of attack but the wing B experiences higher value of $C_{d_{\alpha=0}}$. Due to the gentle slope after the maximum thickness location, wing B shows flow attachment for a wider range of angles of attack. The flow separation appears for the first time at $\alpha = 5^\circ$. To widen the range of operational angles of attack without flow separating from the body, the boundary layer is tripped using the zigzag strips. This tripping of boundary layer adds 3 more degrees in the range. It is the $8^\circ$ of angle where flow begins to separate even after the tripping of the boundary layer. The range of operational angles of attack can be further extended to $12^\circ$ without flow separation using the vortex generators. However, the drag coefficient values keep on increasing with the angle of attack.

The coefficient of drag at 3° angle of attack is lower compared to at 0°. It is counter-intuitive since the drag force on a symmetrical aerofoil at a non-zero angle of attack is usually higher because of the larger component of resultant aerodynamic force in the streamwise direction. The plots of drag contributing terms for wing B, at 0 and 3 degree angles of attack are shown in Figure 5.24. Comparing Figure 5.24(a) and (b), the contributions of Reynolds stress terms are practically similar. The momentum deficit in case of $\alpha = 3^\circ$ configuration is slightly higher than its counterpart in case of $\alpha = 0^\circ$. Whereas the pressure deficit for 0° configuration is significantly higher and thus is a major reason behind the larger value of drag coefficient at 0°. For example at $\frac{x}{D} = 2.5$ in the Figure 5.24, the $C_d$ due to momentum term is larger by 0.066 for $\alpha = 3^\circ$ whereas $C_d$ due pressure term is larger by the margin of 0.113 for $\alpha = 0^\circ$. The flow symmetry does not exist at 3° angle of attack and the flow closes earlier at the bluff trailing edge. Thus, the wing B experiences lower drag compared to 0°.

Figure 5.25 presents the mean x-velocity and pressure field for wing B at an angle of attack 3°. Comparing Figure 5.25(b) and Figure 5.20(b), the value of pressure coefficient behind the trailing edge of wing B is higher when $\alpha = 3^\circ$. Thus the pressure deficit is lower and so is the coefficient of drag.

During the angle of attack sweep, the flow separates when $\alpha = 5^\circ$. The zigzag strips are implemented at this angle of attack to with intention of preventing flow separation. Figure 5.26(a) and (b) demonstrate the effect of boundary layer tripping on the flow separation at 5° angle of attack. Being turbulent, the boundary layer has higher capabilities of withstanding adverse pressure gradient and the flow separation is avoided. As the angle of attack increases, the adverse pressure gradient on the suction side keeps on increasing. After a particular angle of attack, even the turbulent boundary
layer cannot withstand the adverse pressure gradient and separates from the surface. In case of wing B, this angle of attack is $8^\circ$. To achieve further drag reduction, vortex generators strips are attached at the maximum thickness location. The streamwise trailing vortices created by the vortex generators brings higher momentum fluid closer to the surface, re-energising the boundary layer and thus, avoiding/delaying flow separation. Figure 5.27 presents the mean velocity fields with streamlines to highlight the effect of vortex generators on flow separation.

The drag determination is carried out by the wake analysis method. Thus, only the flow field quantities in the wake region affect the drag calculation. As described in the section 5.1.1, for accurate measurements, the drag determination should be performed few characteristic lengths downstream. The uncertainties in the drag measurement performed in the separated region close to the body are significantly large due to large-scale flow fluctuations. In case of wing A and wing B, the wake portion captured in the field of view is limited and thus the credibility of the drag determination in case of higher angles of attack is constrained. At these higher angles, the fluctuations close to the suction side and in the wake are significantly larger. Figure 5.28 shows drag coefficient computed on wing B at $\alpha = 10^\circ$. The drag coefficient values (indicated with black cut-line) is varying as a function of downstream distance. In principle, the drag coefficient is independent of the location.
of drag station. This unphysical behaviour of drag value is due to the large fluctuations introducing errors in the pressure and velocity measurement. Whereas, the drag coefficient values at lower angles of attack like $0^\circ$ and $3^\circ$ is practically independent of the drag station location (refer Figure 5.24). Thus, in order to accurately compute the drag on wings at a higher angle of attack, a streamwise extension of the field of view is desired. At a larger streamwise distances, the fluctuations would be reduced and the total drag plot in Figure 5.28 would plateaul.

To summarise this section, the plot of drag coefficient as a function of the angle of attack is shown in Figure 5.29. As discussed in the section, the flow is kept attached to the body in order to reduce drag, utilising flow control techniques. The separated flow results in higher drag coefficient on the wing.
Thus far the drag analysis is performed on different models at various angles of attack and three different flow velocities. To conclude the comparison of the tested model in terms of the drag coefficient, Figure 5.30 shows the values of $C_d$ experienced by the models in reference configurations. This comparison is made using the drag measurements for pre-set inlet flow velocity of 25 m/s.

The drag coefficients on wings are found to be significantly smaller compared to drag coefficients of cylinder A and cylinder B. The minimum $C_d$ on the wing A is measured at 3° angle of attack with boundary layers tripped. Whereas the minimum $C_d$ for wing B is also measured at the same angle of attack but without the tripping of the boundary layer.

At 0° angle of attack, the wing B experiences nearly twice the drag force compared to wing A. Along with having lowest $C_{d_{\alpha=0}}$ value, wing A is also highly sensitive to flow separation at non-zero angles of attack. At $\alpha = 0^\circ$, the values of $C_d$ predicted by XFOIL analysis for wing A and wing B are
0.093 and 0.105 respectively. The $C_d$ prediction from XFOIL for the wing A is considerably closer to the experimentally measured $C_d$ value. Whereas XFOIL underestimates the $C_d$ for wing B due to the separated flow at the bluff trailing edge of the wing B. XFOIL uses vortex panel method which is not sufficiently accurate to model separated flows.
Conclusions and Recommendations

This master thesis research investigated the drag reduction on a cylinder by geometrical modifications in a constrained space of 3 diameter-length downstream the cylinder. Three geometries along with the reference cylinder are designed and tested experimentally for drag quantification using Particle Image Velocimetry (PIV) as the measurement technique. Additionally, the effect of passive flow control devices like zigzag strips and vortex generators on the drag force of different geometries is investigated. The experiments are carried out at high Reynolds number ranging between $7.95 \times 10^4$ to $1.2 \times 10^5$. The configuration with free stream velocity of 27.5 m/s ($Re = 9.85 \times 10^4$) and zero degree angle of attack (defined only in case of wings) is considered as reference configuration. The following conclusions can be drawn from the findings of this research.

6.1. Conclusions

The drag coefficient determination is performed in the sub-critical flow regime and its value is found close to unity. Primary drag reduction is achieved by the application of roughness (zigzag strips) on the cylinder. These strips force the early boundary layer transition and the higher Reynolds number flow is experimentally simulated. The characteristics of flow like the location of a transition point, the location of separation point, base pressure and recirculation region length appear similar to that of critical-regime flow. Due to the presence of zigzag strips, the flow separation is delayed, the wake deficits are lower and consequently, the drag reduction is achieved. Percentage drag reduction by the application of roughness element depends on the flow free stream velocity. In the reference configuration, nearly 35% drag reduction can be achieved by the application of roughness element.

A flat plate attached to the cylinder base (‘cylinder B’) can aid drag reduction. The presence of downstream splitter plate reduces the amplitude of oscillation in the cylinder flow leading to a narrower wake in the time-averaged flow. Additionally, it also helps in base pressure recovery i.e. the pressure at the cylinder base point is higher in presence of the splitter plate. Consequently, the drag force acting on a cylinder in presence of the downstream splitter plate is lower. In the reference configuration, 20% drag reduction is attained by attaching the downstream splitter plate of 2 cylinder diameter-length. Further drag reduction by 30% is achieved by application of zigzag strip. The application of zigzag strips on a bluff body causes delayed flow separation. The recirculation region formed in the wake of bluff body is elongated in the streamwise direction and restricted in the transverse direction.
The drag force experienced by all three modified geometries in the reference configuration is smaller than the cylinder drag. In the reference configuration, wing A experiences the smallest amount of drag force. However, due to the larger adverse pressure gradient in the streamwise direction, the wing A is extremely sensitive to flow separation at non-zero angles of attack. In the reference configuration wing, A achieves dramatic 92% drag reduction but exhibits flow separation for 3° angle of attack and onwards. The flow can be kept attached to the surface using zigzag strips for angles of attack less than 8°. However, the drag coefficient increases with increasing angle of attack. The least drag on the wing A is measured at $\alpha = 3^\circ$ with the zigzag strips placed closer to the aerofoil nose. The vortex generator strips are found to be ineffective due to significantly higher step size.

In the reference configuration, wing B experiences higher drag compared to wing A. This is due to the higher pressure and momentum deficits in the wake due to the bluff trailing edge of wing B. However, wing B has a higher operational range of angles of attack without flow getting separated from the surface. The flow separation is observed at an angle of 5°. Application of zigzag strip extends the operational range of angle of attack and the flow separation reoccurs at 8°. Unlike for wing A, the vortex generators are proven useful in case of wing B. With the simultaneous application of zigzag strips and vortex generators the wing B has an operational range of 12°-14°. Like wing A, the drag coefficient on wing B also increases with the angle of attack.

Comparing wing A and wing B in the reference configuration, latter has higher drag. However, the drag force on both these geometries can be made comparable by implementing the vortex generators at the bluff trailing edge of wing B. The vortex generators entraps higher momentum fluid into the wake region reducing the wake deficits.

For safety purposes, the speed skiing events are conducted only when the side winds are practically negligible. Thus, it is acceptable to assume that the apparent angle of attack is 0°. Thus, implementation of wing A as the boot fairing can be beneficial. However, for the other applications like drag reduction on wind turbine tower, a wise decision has to be made depending upon the willingness to make trade between the drag coefficient and operational angles range.

During the angle of attack sweep, the aerofoils have higher tendency to experience earlier flow separation at lower Reynolds number. Whereas at higher Reynolds number the boundary layer transition resists flow separation. Thus, at higher Reynolds number the extension in the operational range of angles of attack is expected.

The passive flow control techniques like zigzag strips and vortex generators are used in this research but the fairings are worn under the skiing suit. To the best knowledge of the author, the skiers are not permitted to stick or install anything on the skiing suit surface. Thus if the flow control devices are to be used, they must be installed on the wing and would be covered by the skiing suit. Thus, the practicality of the configuration has to be judged by the skier. The skiing suit has extremely smooth surface thus it might be helpful to use extra thick zigzag strip underneath it. The surface of the fairing could be made non-uniformly smooth especially close to the nose, to trigger flow transition. Additionally, it should be assured that the vortex generators and zigzag strips should not temper or tear the skiing suit.
6.2. Recommendations and future work

Although the current research successfully reduces the drag force on the cylinder, the author believes that further investigation may increase the extent of drag reduction.

The aerofoil design approach used in the current research is rudimentary. Extensive aerofoil optimisation routine may be implemented to design superior aerofoil with least possible adverse gradient in the streamwise direction. Implementation of a genetic algorithm for optimised aerofoil design can be performed.

To investigate higher Reynolds number flow around the geometries, a bigger test section or test models can be used. The field of view can be extended in the streamwise direction so that the drag quantification on wings at higher angles of attack can be performed more accurately. In the current experiment, the captured wake length is roughly one chord-length which is enough to perform drag quantification at zero angle of attack. Whereas at larger angles, the large recirculation region has higher uncertainty in the mean velocity and mean pressure due to the finite number of samples. Thus, the measured drag is dependent on the location of drag station. Additionally, the vortex generators should be embedded into the surface for optimum utilisation. The study regarding the optimal location of the zigzag strips can be performed.

The skier accelerates going downhill, hence a study of drag force as a function of time can be performed for better enhancement of the overall performance. This can be made possible by studying the aerodynamics of transiting skier using large-scale PIV ('The ring of fire concept') (Spoelstra et al. [39]).

In order to extrapolate the results of the experiment to higher Reynolds number, 2-D RANS simulation using Fluent was performed. The simulation failed to produce dependable results. For better numerical simulation with accurate prediction of separation location, base pressure coefficient and the drag coefficient, the author recommends 3D detached eddy simulation (DES) or LES. However, LES at such higher Reynolds number would be computationally expensive.

The maximum thickness of the aerofoils lies closer to the leading edge roughly at 17% of the chord length from the leading edge. Thus, the aerofoils have self-aligning characteristics i.e negative \( \frac{\partial C_m}{\partial \alpha} \) around the cylinder centre (O’Connor et al. [27]). However, the study regarding moment coefficient is not performed in the current research.

Finally, for optimum fairing design, realistic boot-fairing assembly model can be studied. This would include the study of flow in-between the bottom end of the fairing and skies.


