A new experimental setup for studies on wake flow instability and its control

by

Bengt E. G. Fallenius

April 2009
Technical Reports from Royal Institute of Technology
KTH Mechanics
SE-100 44 Stockholm, Sweden
Akademisk avhandling som med tillstånd av Kungliga Tekniska högskolan i Stockholm framlägges till offentlig granskning för avläggande av teknologie licentiatsexamen onsdagen den 6 maj 2009 kl 10.15 i sal E51, Osqurs backe 14, Kungliga Tekniska högskolan, Stockholm.

(C)Bengt E. G. Fallenius 2009
Universitetsservice US-AB, Stockholm 2009
Abstract

A new experimental setup for studies on wake flow instability and its control, which has been designed and manufactured, is introduced and described. The main body is a dual-sided flat plate with an elliptic leading edge and a blunt trailing edge. Permeable surfaces enable boundary layer suction and/or blowing that introduce the feature of adjusting the inlet condition of the wake created behind the plate. This, in combination with a trailing edge that is easily modified, makes it an ideal experiment for studies of different control methods for the wake flow instability. Additionally, a vortex detection program have been developed in order to detect, analyse and compare small-scale vortical structures in the wake behind the plate for different inlet conditions and control methods applied to the wake flow. Instantaneous velocity fields behind a cylinder subjected to suction or blowing through the entire cylinder surface have been analysed with this program. The results of the analysis show that the major change for different levels of blowing or suction is the location of vortices while the most common vortex size and strength are essentially unchanged.
Preface

This Licentiate thesis in fluid mechanics treats the design of a new experimental setup, from which enhanced knowledge about how the hydrodynamic stability of the flow in the wake of a bluff body can be controlled. The thesis is divided into two parts, where the first part starts with a brief overview of bluff bodies and wake flows. This is followed by a description of the new setup and a Matlab® program that has been developed for vortex analyses of the particle image velocimetry measurements. The second part consists of two papers.

April 2009, Stockholm

Bengt Fallenius
Contents

Abstract iii
Preface iv

Part I. Overview and new setup

Chapter 1. Introduction 1

Chapter 2. Bluff bodies and wake flow 2
  2.1. Bluff bodies 2
  2.2. Wake flow 4

Chapter 3. New experimental setup 7
  3.1. Boundary layer control 7
  3.2. The Boundary Layer wind tunnel 8
  3.3. Test section 9

Chapter 4. Vortex detection program 18
  4.1. Definitions of a vortex 18
  4.2. Velocity field filtering 21

Chapter 5. Summary 26

Chapter 6. Papers and authors contributions 27

Acknowledgements 28

References 29

Part II. Papers

Paper 1. Vortex analysis in the wake of a porous cylinder subject to suction or blowing 35
Paper 2. On the vortex generation behind a passive V-shaped mixer in a pipe flow
Part I

Overview and new setup
CHAPTER 1

Introduction

The flow around and in the near downstream region of bluff bodies is a research area that has caught the interest of many people for a long period of time. The main reason is the challenge to understand the physics of the flow. Another reason for the large interest is the large number of technical applications, where different flow phenomena occur. An example is the low pressure region behind vehicles travelling at high speeds that gives a drag force on the vehicle and hence, has a direct impact on fuel efficiency and road stability. Another is vibrations and fatigue in structures caused by periodic vortex shedding.

Learning how to control these phenomena, and the vortex shedding in particular, can lead to improved energy efficiency and reduction of noise and vibrations in high aspect ratio structures. In industrial processes such as papermaking, control of the wake instability would enable the manufacturing of multiple layer paper, which by use of rough unbleached fibres in the middle of the paper sheet would maintain or improve the quality while the cost and the load on the environment are reduced.

This thesis treats the design of a new experimental setup that aims at studies and the development of different methods to control the hydrodynamic instability of the wake behind bluff bodies. In addition, a tool to analyse the flow structures has been developed by means of a vortex detection program.

In times when computer capacity increases exponentially and the cost decreases, more complex events may be simulated and one might question the need for expensive experiments. Though, as the accuracy of the simulations increases, it is even more important to be able to validate the computer codes and find the real value of different physical parameters.

However, for high Reynolds number flows today’s computers are still not fast enough to perform direct numerical simulations of the governing equations. Thus, turbulence modelling is required, which always have to be validated against accurate experimental data. Furthermore, at low Reynolds numbers, where interesting flow stability problems occur, there is today the possibility to numerically perform global mode stability analyses, which is performed on entire velocity fields and gives rise to large eigenvalue problems to be solved. There is thus a search for new experimental setups, which can produce data to test and validate numerical stability codes and physical boundary conditions.
CHAPTER 2

Bluff bodies and wake flow

In this chapter an overview of what can be called a bluff body is given and how a uniform stream is affected by its presence.

A two-dimensional body is a body with an arbitrary cross-sectional area, which is extended to infinity in the direction perpendicular to this area. The flow around a body can be considered two-dimensional if the aspect ratio, defined as the extension width over the equivalent diameter of the cross-section, is large enough. This means that the end-effects do not influence the flow at the centre of the body. However, in a mean velocity perspective the flow will always be two-dimensional provided that the aspect ratio is large enough (see e.g. Norberg 1994). In the following we are assuming two-dimensional flow.

2.1. Bluff bodies

The drag force on objects placed in a flow can be divided into two parts, namely skin friction drag $F_{D,f}$, which is due to the boundary layer formation and the pressure drag $F_{D,p}$, also denoted form drag, which is due to the pressure distribution around the object.

In figure 2.1 a simple example is shown. A two dimensional cylinder with the perimeter $c = c(r, \theta)$, is subjected to a uniform velocity field $(U_0)$, which causes tangential wall-shear stress $\tau_w = \tau_w(r, \theta)$ and a pressure distribution $p = p(r, \theta)$ around the body. The corresponding skin-friction drag per unit width ($F'_{D,f}$) is given by the integrated wall-shear stress projected in the direction parallel to the oncoming uniform velocity field as

$$F'_{D,f} = \int_{c(r,\theta)} \tau_w(r, \theta) \sin \theta \, ds , \quad (2.1)$$

where $ds = rd\theta$ is the path along the perimeter $c$. The pressure difference in the streamwise direction is obtained through direct integration around the body after projecting $pds$ in the direction parallel to the oncoming uniform velocity field. The result is the form drag per unit width ($F'_{D,p}$), which is obtained as

---

1Note, for high enough Reynolds numbers (see section 2.1) physical flow phenomena will introduce 3D effects and consequently the two-dimensionality will be altered.
Figure 2.1. Drag forces acting on the cylinder. The drag on a cylinder may be divided into a pressure/form ($p$) and skin-friction ($\tau_w$) contribution.

\[ F_{D,p} = \int_{c(r,\theta)} p(r,\theta) \cos \theta \, ds . \]  

(2.2)

The sum of the two drag contributions gives the total drag force of the body as,

\[ F_{D,tot} = (F_{D,f} + F_{D,p}) \cdot L_w , \]

(2.3)

where $L_w$ is the extension width of the two-dimensional body. Generally speaking there are two types of bodies with very different flow characteristics. The first type are aerodynamically smooth bodies, which are found in many engineering applications where a low drag and/or high lift is desired, such as airplane wings and similar bodies that end with a continuously decreasing thickness in the streamwise direction. The second type is the opposite of an aerodynamically shaped and body, i.e. bluff bodies, which typically has a blunt trailing edge and separated flow somewhere around the surface of the body. This makes the pressure recovery around the body incomplete and hence gives a contribution to the pressure drag. To illustrate how different the total drag is between these two types of bodies an example is given below.

**Example**

Consider two objects placed in a free stream with the velocity $U_\infty$, kinematic viscosity $\nu$ and density $\rho$. One of the bodies is a cylinder with the diameter $d$ and the other is a NACA 0018 airfoil\(^2\).

The Reynolds number ($Re$) is a flow parameter, which is defined as a characteristic velocity times a characteristic length of the flow over the kinematic viscosity. This parameter can be seen as the ratio between flow destabilising

\(^2\)The number 0018 states that the maximum thickness of the airfoil is 18% of the chord.
forces, i.e. inertia forces, and flow stabilising forces, i.e. viscous forces. For the
flow around a cylinder the Reynolds number becomes

\[ Re = \frac{U_\infty d}{\nu} , \]

where \( d \) is the diameter. If \( Re \) is set to 2000, the drag coefficient \( C_D = F_D'/(q_\infty d) \) is about unity. Here, \( q_\infty = 0.5\rho U_\infty^2 \) is the dynamic pressure and \( F_D' \) denotes drag force per unit width. To obtain the same \( F_D' \) of the airfoil the chord length has to be about 100d, which gives a Reynolds number based on the chord of about \( 2 \times 10^5 \). Hence, the thickness of the airfoil that gives the same drag force per unit width as the cylinder is 18d. In figure 2.2 the two bodies are shown according to scale.

\[ \text{Figure 2.2. The drag force caused by the flow on a cylinder} \]

\[ \text{with diameter } d \text{ is the same as for a airfoil with a maximum} \]

\[ \text{thickness of } 18d \text{ and a chord of } 100d. \text{ Note, the figure is ac} \]

\[ \text{cording to scale.} \]

2.2. Wake flow

As illustrated in the previous section, the drag force on an object is not solely
determined by its frontal area. It is rather the shape of its trailing edge, which
is important. A blunt trailing edge gives rise to a low-pressure region, also
called the near wake.

The size of such a wake is governed by the Reynolds number that depends
on the free stream velocity and the size of the object. Depending on the
geometry of the body, there can be sudden changes of the wake flow properties
at different Reynolds numbers (see section 2.1). As an example, one can look
at the drag coefficient for a circular cylinder. In the range \( 10^2 < Re < 10^3 \)
it is close to unity, while at \( Re \sim 10^3 \) it drops suddenly down to about 0.3,
whereafter it subsequently increases and levels out again at \( Re \sim 10^7 \) to about
0.6. For a smooth cylinder, the drop is due to formation of separation bubbles and boundary layer transition to turbulence on the cylinder surface. The drag coefficient for a square cylinder behaves differently, it is constant equal to 2 for Reynolds numbers within the range of incompressible flow, since the separation points do not move from the sharp leading edges. Although, rounding of the corners of the square cylinder makes the drag coefficient change towards that of a circular cylinder.

Another important phenomenon is the periodic vortex shedding that occurs behind objects with a blunt trailing edge. It is characterised by the alternating shedding of vortices from the two sides of the object. This phenomenon is also Reynolds number dependent and sets in at around $Re = 40 - 50$ for a circular cylinder and around $Re = 115 - 125$ for a square cylinder, depending on the flow quality. The frequency of the periodic shedding can be estimated through the Strouhal number ($St$), a non-dimensional frequency, given as

$$St = \frac{fd}{U_\infty},$$

where $f$ is the frequency of the periodic shedding. The Strouhal number varies with geometry and $Re$. However, for $Re$ in the range $10^2 - 10^5$ the Strouhal number is almost constant, 0.2 and 0.13 for the circular and square cylinder, respectively. For rectangular cylinders it depends on the aspect ratio $l/d$, where $l$ is the streamwise length of the object.

The periodic vortex shedding induces alternating positive and negative side forces, which can induce vibrations of the object. This can be a source of noise and other interferences. In worst case the shedding frequency coincides with the eigenfrequency of the structure/object, which can lead to material fatigue and structural failure. An example of this was the Tacoma Narrows bridge in the state of Washington, USA, which soon after its construction in 1940 started to oscillate with an increasing amplitude until the central part of the bridge broke apart and fell down into the water.

The vortex shedding phenomenon is purely two-dimensional and is called the von Kármán vortex street after Theodore von Kármán (see Kármán 1912), who first studied and described this phenomenon. Figure 2.3 shows a NASA satellite image that captures an area of $365 \times 150$ km$^2$ near the island of Jan Mayen in the north Atlantic ocean. In the image one may observe a von Kármán vortex street evolving downstream of the Beerenberg volcano that raises 2200 m above the sea level. The stratified layers in the atmosphere makes the flow locally two-dimensional around the otherwise three-dimensional volcano.
Figure 2.3. A more than 300 km long von Kármán vortex street near the island of Jan Mayen. (NASA)
CHAPTER 3

New experimental setup

A new experimental setup for studies on wake flow instability and control, including a new test section, has been designed and built at the department of Mechanics, KTH. The setup consists of a flat plate, from here on denoted rectangular forebody, with an elliptic leading edge and a blunt trailing edge. Permeable surfaces on both sides of the rectangular forebody add the unique feature of being able to vary the boundary layer along the body and thereby the inlet flow condition of the wake. In addition, the blunt trailing edge is interchangeable, enabling various means of base flow control.

3.1. Boundary layer control

Modulation of the boundary layer profile at the trailing edge of the rectangular forebody, i.e. the inlet condition of the wake, is managed by withdrawal and/or injection of air through the permeable surfaces by applying suction and/or blowing, respectively. Suction reduces the boundary layer thickness and at sufficiently high levels it will completely disappear. A special case of boundary layer suction is the so-called asymptotic suction boundary layer (ASBL). The analytical expression for the ASBL profile was first derived by Griffith & Meredith (1936), but was not fully experimentally verified until 2003 by Fransson & Alfredsson. The expression for the streamwise velocity component in the ASBL reads

\[ U(y) = U_\infty \left( 1 - e^{V_0/y/\nu} \right), \quad (3.1) \]

where \( U_\infty \) is the free stream velocity, \( V_0 \) = const. is the suction velocity, \( y \) is the wall-normal direction and \( \nu \) is the kinematic viscosity. Note, here \( V_0 \) is defined as being negative, implying that \( U \to U_\infty \) as \( y \to \infty \). The analytical expression (3.1) enables direct integrations of the integrands corresponding to the displacement (\( \delta_1 \)) and the momentum loss thickness (\( \delta_2 \)), which result in

\[ \delta_1 = \int_0^\infty \left( 1 - \frac{U(y)}{U_\infty} \right) dy = -\frac{\nu}{V_0} \quad (3.2) \]

and
respectively. This means that the shape factor $H = \delta_1/\delta_2$ is equal to 2. The boundary layer thickness $\delta_{99}$, defined as the wall distance where $U$ reaches 99% of $U_\infty$, becomes

$$\delta_{99} = \frac{\nu}{V_0} \log(0.01) = \delta_1 \log(100).$$

(3.4)

In the asymptotic suction region the Reynolds number based on the displacement thickness becomes

$$Re = \frac{U_\infty \delta_1}{\nu} = -\frac{U_\infty}{V_0}.$$ 

(3.5)

This provides the ASBL with a unique feature, namely, that one may vary the Reynolds number and the boundary layer thickness independent of each other.

### 3.2. The Boundary Layer wind tunnel

The Boundary Layer (BL\textsuperscript{1}) wind tunnel is located at the department of Mechanics at KTH, and has been chosen as the experimental facility to host the new setup. The idea with the BL tunnel is to have a short time swap between different experiments by having exchangeable test sections. Below a brief description of the wind tunnel is given. For a more thorough description the interested reader is referred to Lindgren (2002).

The BL wind tunnel is a closed circuit tunnel, powered by an 15 kW axial fan. It was the first tunnel where expanding corners were utilised, making it possible to have a 9:1 contraction ratio together with an short overall wind tunnel length. The space for the testsection if 4.2 m and the cross sectional area of the contraction outlet is $0.5 \times 0.75 \text{ m}^2$. The maximum flow velocity is 48 m/s and the turbulence levels\textsuperscript{2} are 0.04%, 0.06% and 0.04% in the streamwise, wall-normal and spanwise directions, respectively, at the nominal\textsuperscript{3} free stream velocity of 25 m s\textsuperscript{-1}. At this nominal velocity the variation in total pressure is less than $\pm 0.1\%$ and the variation in temperature is less than $\pm 0.07 \degree C$ over the cross sectional area.

\textsuperscript{1}BL also corresponds to the initials of the wind tunnel designer Björn Lindgren.

\textsuperscript{2}The following turbulence levels correspond to the high-pass filtered intensities, with a cut-off frequency of 20 Hz.

\textsuperscript{3}During the design of the BL-windtunnel, most of the planned experiments were aimed for a free stream velocity of 25 m s\textsuperscript{-1}. 
3.3. Test section

The new experimental setup consists of a main body, which is mounted into a new exchangeable test section, see figure 3.1. The test section is based on two steel frames and has a total length of 4 m. Plexiglas together with plywood have been used for the walls since a high level of optical access is desired for measurements with high speed Stereoscopic Particle Image Velocimetry (S-PIV). Top and bottom walls have hatches for easy access into the test section important both for adjustments and cleaning.

3.3.1. Main body

The main body in the new experimental setup is symmetric, i.e. a dual-sided plate with permeable surfaces. Changing the pressure difference across the permeable surfaces will affect the boundary layer growth along the plate and consequently the inlet condition of the wake.

In figure 3.2 a schematic of the main body shows the principle and its feasibility. Note, the dimensions are not according to scale due to the high aspect ratio of the body. The co-ordinates are defined as, $x$ in the streamwise direction, $y$ normal to the body and $z$ in the spanwise direction. The corresponding velocity components are $U$, $V$ and $W$, respectively. A free stream flows past the body with the streamwise velocity $U_\infty$. In the schematic the
upper side of the body is subjected to suction through the permeable surface, with the uniform suction velocity $V_0$. By varying the direction and velocity of the flow through the permeable surfaces, different boundary layer profiles can be obtained. Moderate levels of suction gives an asymptotic suction boundary layer (I), while a high suction velocity can result in an inviscid slip-condition (II). Blowing can also be applied through the full length or only partly (III). Depending on the chosen flow through the surfaces the initial condition of the wake and consequently the shape of the wake profile (IV) may be varied.

Figure 3.3 shows the assembly of the main body, which starts with a 300 mm long leading edge (1), followed by a 2 m long flat plate (2–4) and ending with a trailing edge (5, 6).

**Leading edge.** The leading edge has been milled out from a solid block of aluminium and is symmetric in the $xz$-plane. The symmetric profile is described by a modified super ellipse,

$$\left(\frac{a - x}{a}\right)^{m(x)} + \left(\frac{y}{b}\right)^n = 1, \quad 0 < x < a,$$

where $m(x) = 2 + (x/a)^2$ and $n = 2$, see Saric et al. (2002). The aspect ratio, given by the quotient between the semi-major axis $a$ and the semi-minor axis $b$, is 12.5. The advantage with a modified super ellipse versus an ordinary super ellipse is the continuous derivative of the curvature at the junction between the leading edge and the following flat plate.

**Flat plate.** The flat plate is a sandwich construction with identical top and bottom sections (2) and (3). These are decoupled from each other by a separating aluminium sheet (4). In between the separating sheet (4) and the
permeable plates (3) there are two frames consisting of 33 T-profiles directed in the spanwise direction with a spacing of 60 mm in the streamwise direction. The T-profiles are clamped in between two solid aluminium bars along the streamwise direction. Between every T-profile two 10 mm holes are drilled through the bars and where pipes and tubing are connected for withdrawal and/or injection of air. In the case of excess pressure, i.e. injection, there is nothing holding the permeable plates in position, therefore these plates are point-wise glued to the frame.

\textit{Trailing edge}. The three last sections of the T-profile arrays are exchangable in order to allow for different control devices such as, an internal chamber (5) for base-bleed through a slotted plate (6), air-jets perpendicular to the main stream or simply different types of splitter plates.
Figure 3.3: The main body showing the sandwich construction in (a) and the side view in (b). All dimensions are in [mm]: 1. Leading edge; 2. Supporting frame; 3. Permeable plate; 4. Separating sheet; 5. Rear chamber and 6. Slotted end plate.
3.3.2. Permeable plate

Boundary layer control and drag reduction by suction through permeable sheets have been studied over several decades, mainly with applications within the aeronautical field. This extensive research has led to many investigations about permeable materials, which are suitable for boundary layer control.

Studies on the asymptotic suction boundary layer as well as the flow past a circular porous cylinder have been carried out at the department of Mechanics, KTH, see Fransson & Alfredsson (2003); Yoshioka et al. (2004) and Fransson et al. (2004), respectively. In those experiments, a sintered plastic material with a maze-like structure were used. However, experiences from the setups have shown that such a material is not suitable for extensive particle image velocimetry measurements, since the smoke that is used as seeding clogs the pores. Furthermore, such material has problems with ageing, which results in undesirable cracks and changes of the material properties and must typically be replaced regularly.

For the new setup, a material without the above drawbacks was preferable, which resulted in the choice of laser drilled titanium sheets. This material has several advantages; it is stiff, it can stand strong detergent used for cleaning, it is resistant to corrosion and it has a smooth surface after the perforation process. The discrete holes of the perforation makes the inspection of the cleanliness easy simply by placing a light source behind the sheet. Similar materials such as aluminium and stainless steel were also considered, however, the former is difficult to perforate and the latter has problem with debris in the perforation process, which results in a rugged surface.

There are two main types of permeable materials, granular and fibrous materials with a labyrinth-like structure and materials perforated straight through. A major difference between these two types of materials is the relation between the flow velocity and the pressure drop over the material. The flow velocity through a labyrinth-like material increases linearly with the pressure difference, following Darcy’s law, which reads

\[
V = \frac{\kappa}{\mu} \frac{\Delta p}{t},
\]

where \( V \) is the flow velocity through the material of thickness \( t \), \( \Delta p \) is the pressure difference across the material and \( \mu \) and \( \kappa \) is the dynamic viscosity and permeability, respectively.

However, for perforated materials the relation between \( V \) and \( \Delta p \) is different. The velocity, through such material, may be approximated by the flow through individual pipes. A theoretical analysis was carried out in Goldstein (1938), yielding narrow pipes, which may be applied on perforated materials. Provided that \( t/(r \cdot Re_{2r}) \ll 1 \), where \( r \) and \( Re_{2r} \) correspond to the radius of the holes and the Reynolds number based on the hole diameter and the mean
velocity through a single hole, respectively, the relation reads

\[
V = \left( \frac{2\Delta p}{\rho \left( \beta + 32t/(r \cdot Re_{2r}) \right)} \right)^{\alpha}.
\]

Here, \( \alpha = 1/2 \) and \( \beta = 2.41 \). This theoretical relation has been verified experimentally by Poll, Danks & Humphreys (1992).

The titanium sheets used in the present study are denoted 2TAl and have a thickness of \( t = 0.9 \) mm. The hole diameter is \( 2r = 60 \) \( \mu \)m in average and the distance from centre to centre is 0.75 mm which gives a distance to diameter ratio of 12.5. The row to row distance was also chosen to be 0.75 mm and each row in the streamwise direction is shifted. Figure 3.4 shows pictures of the holes on both sides of the sheets. Due to the focus of the laser beam during drilling, the holes have a taper angle of approximately 3 degrees. This makes the hole larger on the drilling side compared to the back, which has the desired diameter of 60 \( \mu \)m.

\textbf{FIGURE 3.4.} Images of the laser drilled titanium sheet. (a) the smaller hole diameter on the outlet side of the laser beam. (b) the larger hole diameter on the inlet side. At the bottom of the images a reference ruler is shown.

Measurements to determine the permeability of the material were also performed. Figure 3.5 shows the setup. A sample of the titanium sheet was clamped between two pipes with an inner diameter of 5 cm (a). One end was connected to a vacuum cleaner (b) and the other end to a flowmeter (c). The flow velocity was varied by changing the voltage (d) supplied to the vacuum cleaner. By measuring the pressure drop over the sample (e) for different flow velocities the permeability could be determined. A barometer (f) and a thermometer (g) were used to calculate the air density. The result of the permeability measurement is shown in figure 3.6(a) where (•) and (□) represent suction and blowing, respectively. For the present perforated sheet with a typical cross flow velocity of 3 cm s\(^{-1}\) we get \( t/(r \cdot Re_{2r}) > 1 \) and, hence,
relation (3.7) does not hold. However, the constant $\beta$ and the exponent $\alpha$ in relation (3.7) may be determined through curve fitting to the data in a least square sense. We obtain $\beta = 42.0$ and $\alpha = 6/7$ and $9/11$ for the suction and the blowing case, respectively, which are shown with solid and dashed lines in figure 3.6(a). The difference is attributed to the different inlet conditions (see figure 3.4). During the laser drilling process, the pressure drop over the sheet for a chosen flow velocity of $5 \text{ cm s}^{-1}$ was monitored in order to secure the quality of the perforation. The drilling was conducted in the spanwise direction and the pressure drop was measured at the end of each row and in figure 3.6(b – c) the pressure drop along the sheets is shown.

The streamwise distance between the T-profiles extending in the spanwise direction has carefully been chosen. There must be enough space for tubing and at the same time the titanium sheets have to withstand the imposed pressure difference across them. A study in the commercially available multihysics and finite element solver program Comsol® was performed in order to ensure that neither bulging nor curving of the sheet will occur when blowing and/or suction are applied. To compensate for the porosity of the material, the elasticity module was reduced based on the diameter of the holes. This gives an

---

**Figure 3.5.** Setup of permeability measurements. (a) sample mount, (b) vacuum cleaner, (c) flowmeter, (d) transformer, (e) difference pressure gauge, (f) barometer and (g) thermometer.
Figure 3.6. (a) permeability measurement on the sample. (c) suction and (□) blowing configuration, with least square fits with the relation $(3.7)$ in $(-)$ and $(--)$, respectively. $(b-c)$ pressure drop along the sheets in the streamwise direction at a cross flow velocity of $5$ cm s$^{-1}$. $(--)$ and $(---)$ represents the left and right side of the sheets, respectively.

...elasticity module of $102$ GPa instead of $106$ GPa, which was supplied by the manufacturer.

The average of the pressure drop measured during the manufacturing was $535$ Pa. This was used as a load in the displacement calculations using Comsol®...
with the argument that the contemplated velocity will be less than 3 cm s\(^{-1}\). Figure 3.7 shows that the maximum displacement is less than 50 µm for the blowing as well as the suction case. This is considered to be an acceptable displacement considering both the flatness of the material and the boundary layer thickness. The suction case in figure 3.7 shows, regularly, regions extending in the spanwise direction where there is no displacement (white regions). These regions correspond to the location of the spanwise pointing T-profiles. In contrary, the blowing case shows, locally, regions of no displacement (i.e. white regions), which correspond to the point-wise glued elements holding the sheet in place (see section 3.3, *Flat plate*).
Vortex detection program

A search of the literature gives many different definitions of a vortex, see e.g. Jeong & Hussain (1995). In order to develop a vortex detection program one has to come up with some criteria that are characteristic for a vortex. A major drawback with experiments is that most available data is two-dimensional, i.e. the velocity field is only known in one plane. This seriously limits the definition of a vortex, which leads to a less restrictive definition compared to a vortex defined in a three-dimensional velocity field.

This chapter will therefore start with a brief summary of different vortex definitions and discuss their pro and cons in order to shed some light on the limitations of the present vortex detection algorithm.

4.1. Definitions of a vortex

How to define a vortex is an issue that have been discussed for several decades due to the complexity of turbulent flow fields. Numerical modelling has made the dynamics of flow fields more accessible for studies and different methods can more easily be compared. Jeong & Hussain (1995) summarise and discuss the most frequently used definitions of a vortex. Roughly speaking, the different definitions can be divided into intuitive and analytical approaches. The former is based on local properties of the flow such as local pressure minima $p$, the paths of the streamlines

$$\frac{dx}{u} = \frac{dy}{v} = \frac{dz}{w}, \quad (4.1)$$

and the magnitude of vorticity

$$|\omega| \equiv |\nabla \times u| . \quad (4.2)$$

The latter is based on properties of the velocity gradient tensor $\nabla u$.

Jeong & Hussain (1995) states two requirements for a vortex core as a preliminary check for an evaluation of the different methods. Firstly, a vortex core must have a net vorticity and hence, net circulation, and secondly, the geometry of the identified vortex should be invariant in a Galilean transformation. A short summary of the review follows.
4.1. DEFINITIONS OF A VORTEX

4.1.1. Intuitive approaches

**Pressure minima.** When the centrifugal force is balanced by the pressure force, a local pressure minimum is located at the axis for the swirling motion. This is shown to be true only in a steady inviscid planar flow, why this is not a valid condition in general.

**Closed or spiral pathlines or streamlines.** Closed or spiral pathlines or streamlines have been proposed to be used to identify the swirling motion of a vortex. The lifetime of a vortex might however not be long enough for a particle to complete a full revolution that is required for the closed pathline, which means that vortices will occur without being detected. Furthermore, neither closed or spiral pathlines or streamlines are invariant with respect to Galilean transformation, so only vortices that are translated within a certain range of velocity will be detected.

**Vorticity magnitude.** Defining a vortex as a region where its vorticity magnitude is higher than some threshold has also been suggested as a method. This method turns out to be quite arbitrary since, firstly, it depends on the threshold and secondly, as soon as the background shear is within the same magnitude a distinction between the shear and the vortex may be unfeasible.

4.1.2. Analytical approaches

Considering the more analytical approaches, the drawbacks are fewer, although there are still cases where those approaches are unsuitable. Two older methods are presented and compared to the most accepted and used method, namely the $\lambda_2$-method.

**Complex eigenvalues of velocity gradient tensor, the $\Delta$-method.** This approach considers the eigenvalues, $\lambda$, of the velocity gradient tensor $\nabla \mathbf{u}$, which satisfies the characteristic equation

$$\lambda^3 - P\lambda^2 + Q\lambda - R = 0 \quad \text{(4.3)}$$

Considering an incompressible flow ($u_{i,i} = 0$) the three invariants of $\nabla \mathbf{u}$ above become

$$P \equiv u_{i,i} = 0 \quad \text{(4.4)}$$

$$Q \equiv \frac{1}{2}(u_{i,i}^2 - u_{i,j}u_{j,i}) = -\frac{1}{2}u_{i,j}u_{j,i} \quad \text{(4.5)}$$

and

$$R = \det(u_{i,j}) \quad \text{(4.6)}$$

Chong et al. (1990) showed that complex eigenvalues imply that the local streamline pattern is closed or spiral in a reference frame moving with the
point, i.e. when the discriminant

\[ \Delta = \left( \frac{1}{3} Q \right)^3 + \left( \frac{1}{2} R \right)^2 \]  

(4.7)

is positive. Although this method is Galilean invariant, it shows when trying the method on some special cases, such as mixing layers and swirling jets, that \( \Delta \) is slightly positive even outside vortex cores resulting in that the boundary of the vortices becomes noisy and the size of the vortices are overestimated.

The second invariant of the velocity gradient tensor, the \( Q \)-method. It has been suggested to define vortices as regions where \( Q > 0 \), with the additional condition that the pressure is lower than the ambient value. One may rewrite \( Q \) in terms of the symmetric and the antisymmetric parts of \( \nabla \mathbf{u} \), i.e. the strain rate tensor and the rotational tensor, respectively. Hence, \( Q \) represents the local balance between shear strain rate and vorticity magnitude. According to (4.7) the \( Q \)-method is more restrictive than the \( \Delta \)-method, however, the most appropriate method is not obvious \textit{a priori}.

\( \lambda_2 \)-method. The frequently used \( \lambda_2 \)-method (Jeong & Hussain 1995) comes from inspection of the acceleration gradient

\[ a_{i,j} = -\frac{1}{\rho} p_{,ij} + \nu u_{i,jkk} , \]  

(4.8)

which is derived by taking the gradient of Navier-Stokes equations. Pressure minimum has been used as a starting point without being used as a requirement. The left hand side of (4.8) can be divided into a symmetric and an antisymmetric part where the antisymmetric part is the vorticity transport equation. Leaving out the unsteady irrotational straining and viscous effects in the symmetric part one gets

\[ -\frac{1}{\rho} p_{,ij} = S_{ik} S_{kj} + \Omega_{ik} \Omega_{kj} = S^2 + \Omega^2 , \]  

(4.9)

where \( S_{ij} = (u_{i,j} + u_{j,i})/2 \) and \( \Omega_{ij} = (u_{i,j} - u_{j,i})/2 \) are the symmetric and the antisymmetric parts of \( \nabla \mathbf{u} \), and defined as the strain rate tensor and the rotational tensor, respectively. Local pressure minima existing only due to vortical motion, are then present if two of the eigenvalues of \( S^2 + \Omega^2 \) are negative. Since \( S^2 + \Omega^2 \) is symmetric its eigenvalues \( \lambda_1 \), \( \lambda_2 \) and \( \lambda_3 \) are real, which requires that \( \lambda_2 < 0 \) within the vortex core if \( \lambda_1 \geq \lambda_2 \geq \lambda_3 \).

This definition is then compared, by Jeong & Hussain (1995), with the two methods above for various cases and it is found to be the most general method to identify vortices. It is referred to as the \( \lambda_2 \)-method and has been widely accepted and is even implemented in many numerical codes. The method, however, requires that the Hessian of the pressure is known, i.e. all three
components of the velocity gradient tensor, which are never available in experiments. For this reason the $\Delta$- as well as the $Q$-method are the most widely used on two-dimensional experimental data.

### 4.1.3. Two-dimensional velocity fields

In order to detect vortices that are embedded in two-dimensional velocity fields, commonly acquired through PIV-measurements, Adrian et al. (2000) suggested that decomposition of the velocity field by low-pass filtering is an adequate way to visualise small-scale vortices. In their study, a Gaussian filter was used for the decomposition and the vortices were then detected by using the approach suggested by Chong et al. (1990), i.e. identifying closed or spiral streamline patterns by looking at the complex eigenvalues of the high-pass filtered two-dimensional velocity gradient tensor,

$$\nabla u''_2 = \begin{bmatrix} \frac{\partial u''}{\partial x} & \frac{\partial u''}{\partial y} \\ \frac{\partial v''}{\partial x} & \frac{\partial v''}{\partial y} \end{bmatrix}. \quad (4.10)$$

Regions where the imaginary eigenvalues are positive and greater than a threshold are then defined as a vortex. Agrawal & Prasad (2002) also used a Gaussian filter to perform the decomposition suggested by Adrian et al. (2000), while vortices were identified by looking at the neighbouring vectors of each point. If the angular orientation of the surrounding vectors experienced a monotonically angular variation from 0 to $2\pi$ the point was considered to be a vortex centre. The same decomposition will be used here, while the $\Delta$-method according to Chong et al. (1990) will be used for the vortex identification.

### 4.2. Velocity field filtering

#### 4.2.1. Decomposition

A turbulent flow field consists of a spectrum of different scales, from the largest geometrically allowed down to the smallest viscous scale, namely the Kolmogorov scale. To reveal the small-scale structures that are embedded in the measured turbulent flow field, the latter is decomposed into a low-pass filtered velocity field and a high-pass velocity field, corresponding to the spatially large and small scale structures, respectively. If these velocity fields are added together, one recovers the fully measured flow field, see figure 4.1. The decomposition is performed in the same manner as in e.g. Agrawal & Prasad (2002), i.e. convolving a low-pass filter on the full velocity field $u$, and thereby get a velocity field $\bar{u}$ that contains the larger scales of the full velocity field. To get the small scale velocity field $u''$, the large-scale field is then subtracted from the full velocity field as

$$u'' = u - \bar{u}. \quad (4.11)$$
4. VORTEX DETECTION PROGRAM

Figure 4.1. (a) An instantaneous velocity field behind a porous cylinder with continuous suction through the surface of 2.6% of the oncoming velocity. (b) and (c) show the low- and high-pass filtered velocity fields, respectively.

4.2.2. Gaussian filter

The filter used for the decomposition is a Gaussian filter that averages the single point \((m, n)\) with the surrounding points. This will give a smeared out velocity field which will emphasise and keep the large scale structures according to

\[
\bar{u}(m, n) = \frac{\sum_{i=-k}^{k} \sum_{j=-k}^{k} g(i, j) u(m - i, n - j)}{\sum_{i=-k}^{k} \sum_{j=-k}^{k} g(i, j)},
\]

(4.12)

where \((i, j)\) is the indices in \(x\) and \(y\), respectively. Here, \(k\) is defined as the radius of the filter and since a discrete velocity field is considered, it has a quadratic shape, with each point \((m, n)\) being affected by a surrounding squared region. The Gaussian kernel \(g\) is defined as
\[
g(i, j) = \exp \left[ -\frac{(i \Delta x)^2 + (j \Delta y)^2}{2 \sigma^2} \right],
\]

where \(\Delta x\) and \(\Delta y\) are the grid spacing and \(\sigma\) is the padding of the filter. The parameters \(k\) and \(\sigma\) can then be chosen by introducing an anisotropy measure \(d_{rms}^2\), which is defined as the absolute value of the normalised difference between the velocity variance components,

\[
d_{rms}^2 = \left| \frac{v_{rms}^2 - u_{rms}^2}{U_\infty^2} \right| .
\]

In figure 4.2(a) the maximum value of the anisotropy is shown as contour lines for varying \(k\) and \(\sigma\) for the velocity field downstream of a cylinder. A consistent requirement for the choice of filter would be to allow a certain amount of anisotropy in the final high-pass filtered velocity field. Typically one here chooses a \(\max\{d_{rms}^2\} \leq 0.01\), which is in the order of one magnitude lower than for the unfiltered case. Figure 4.2(b) shows the shape of the Gaussian filter for \(k = 5\) and \(\sigma = 5\). The filter is normalised, i.e. the total weight is equal to 1, and the small difference in weight between the minima and maxima implies that this filter will smear out smaller scales, while larger scales will remain, which is the purpose with low-pass filtering.

### 4.2.3. Statistics

In each instantaneous PIV-image, contours where the imaginary part of the complex eigenvalues, \(\lambda_{ci}\), corresponds to a threshold value is defined as a vortex. Each contour is then examined in order to determine relevant properties such as location, size, circulation and swirl strength of the vortex. This is executed in the following manner. The centre of the vortex is identified by finding the \(x\)- and \(y\)-coordinates of the maximum imaginary eigenvalue, \(\lambda_{ci,max}\), within the contour. This eigenvalue is also stored as a measure of the swirl strength of the vortex (see e.g. Zhou et al. 1999). The size of the vortex is then determined by first calculating the area inside the threshold contour level. An equivalent radius to a corresponding circle \((C)\) with its origin at \(\lambda_{ci,max}\) is then used as a starting radius for calculating the circulation, \(\gamma\), through direct integration along \(C\)’s perimeter \(\Gamma\) according to

\[
\gamma = \oint_C \mathbf{u}' \cdot d\Gamma .
\]

This process is repeated while stepping outwards from the vortex centre until the maximum value of the circulation is reached, which then is stored. The corresponding radius is also stored as the vortex size. The two different vortex size measures are shown in figure 4.3(b). Note, that the background velocity
4. VORTEX DETECTION PROGRAM

Figure 4.2. The maximum values of the anisotropy measure $d_{rms,max}^2$ for different values of $k$ and $\sigma$ when filtering the flow field behind a circular cylinder with the diameter $D = 50$ mm. (b) The corresponding shape of the normalised Gaussian filter for $k = 5$ and $\sigma = 5$.

vector field is the full velocity field in where the vortices are not necessarily shown.
4.2. VELOCITY FIELD FILTERING

\[ \frac{V_0}{U_\infty} \times 100 = -2.6 \]

Figure 4.3. Instantaneous velocity field of the flow behind a circular cylinder subjected to suction. (a) shows the small scale velocity field where contour lines are regions of \( \lambda_{ci} > 15 \) and (b) the unfiltered velocity field where (-) corresponds to the equivalent radius of the threshold contour and (---) corresponds to the radius of the vortex, based on the definition of maximum circulation.
CHAPTER 5

Summary

In this thesis a new experimental setup for studies on wake flow instabilities and control is introduced. A main body consisting of a flat plate, with an elliptic leading edge and a blunt trailing edge, was designed as a sandwich construction with an hollow interior and has been manufactured. Permeable surfaces on both sides give the unique possibility to perform boundary layer suction or blowing along the plate and thus, mastering the inlet profile of the wake. The dual layer design enables an asymmetric wake to be created, by independently adjusting the pressure difference across the surfaces on the two sides. Furthermore, the plate has separate compartments, which makes local manipulation of the boundary layer possible.

An exchangeable trailing edge of the plate adds the possibility to implement various types of active control devices, such as feedback controlled jets or base-bleed. Passive control devices such as splitter plates and other obstacles for manipulation of the periodic separation is also easily mounted.

The new test section is designed for the use of modern measurement techniques such as high-speed stereoscopic PIV, which generates a high amount of data about the flow field. To effectively handle all the acquired data, a Matlab® program that automatically filters a two-dimensional velocity field and identifies small-scale vortices has been developed. The program stores information about vortex location, size, strength and circulation, which makes statistical analyses for different flow conditions a straightforward process.

Combining the new experimental setup with the developed tool for velocity field analyses, the understanding of the wake flow behaviour for different inlet conditions as well as control methods, will be enhanced. The aim is that this will contribute to the efforts in finding new means to reduce drag and oscillating structural forces on bluff bodies in different technical applications.
CHAPTER 6

Papers and authors contributions

Paper 1

*Vortex analysis in the wake of a porous cylinder subject to suction or blowing.*


This work is based on experiments on a porous cylinder subjected to suction or blowing, performed by JF. A computer program that was initiated by JF is used to identify small-scale vortices. BF debugged and restructured the program and ran it on the experimental data. The results have been produced by BF and the article has been written jointly by the authors. Parts of this work have been presented at the European Fluid Mechanics Conference 2008, Manchester, Great Britain, and at the XXII International Congress of Theoretical and Applied Mechanics 2008, Adelaide, Australia.

Paper 2

*On the vortex generation behind a passive V-shaped mixer in a pipe flow.*


The experiments were carried out by BF and JF in collaboration with the Marcus Wallenberg Laboratory. BF has extracted the results from the measurements and the paper has been written jointly by the authors. Parts of this work are published in AIAA Paper 2008-3057, and have been presented at the 14th AIAA/CEAS Aeroacoustics Conference 2008, Vancouver, British Columbia, Canada.
Acknowledgements

First of all I would like to express my gratitude to my supervisor Docent Jens Fransson for giving me the opportunity to perform the graduate studies as well as the professional guidance you are giving me. Also, I would like to thank my co-advisor Prof. Henrik Alfredsson for the nice and inspiring working environment in the fluid physics laboratory.

I also would like to thank Docent Fredrik Lundell, Dr. Nils Tillmark, Prof. Alessandro Talamelli and Prof. Laszlo Fuchs for encouraging advices, comments on my work and for introducing interesting references to me. Göran Rådberg and Joakim Karlström are greatly acknowledged for the skilful manufacturing of the new setup and for patiently making last minute changes, as well as for giving constructive suggestions of improvements in the design. Thanks also to Marcus Gällstedt and Ulf Landén for introducing me to toolmaking as well as my very first experimental setup.

Thanks to Ola, Malte and Shahab for the fun and relaxing atmosphere we have in our office, which is essential for my work.

Special thanks to Veronica, Ramis, Gabriele, Thomas, Allan and Outi for the reception and support that has continued ever since I started. Thank you also Fredrik H., Anna, Shiho, Monika, Olle, Fredrik L., Emma, Charlotte, Ulrika and Niklas for contributing to all the joy I have during the days in the lab. Linus, Niklas, Robert, Mattias, Tobias, Espen and all the other colleagues and staff at OB18 have also made the days at the department of mechanics to a pleasure.

Last but not least I would also thank my family and friends outside KTH for their endless support.

The Swedish Research Council (VR) and the Göran Gustafsson Foundation is acknowledged for the financial support.
References


Part II
Papers