Development and optimization of synthetic jets for active flow control

by

Patrick Bennani
871109-0350
paben@kth.se

Performed at
ISAE Ensica, DAEP
1 Place Emile Blouin
31500, Toulouse
France

Report of final project study internship, September –February, 2011
Date Submitted: 19 March 2011
The purpose of this report is to describe the studies and results obtained during my internship period at the Institut Supérieur de l’Aéronautique et de l’Espace (ISAE) in the aerodynamic, energy and propulsion department (DAEP). This report is also a requirement for the double degree Kungliga Tekniska Högskolan (KTH) internship program and for the Masters degree.

I have written this report myself and have not received any previous academic credit for it at this or any other institution.

I would like to thank all of the people who have supported me during this double degree exchange program in Sweden and in France. I especially thank all my friends, the employees of Ensica and KTH and in particular Mr. Alfredsson Henrik, supervisor of this project and teacher of two of my courses, for his interest in my work, competence and kindness.

I would also like to thank the director of my project, Mr. Yannick Bury, who helped me to learn new points, to progress in many fields both personally and intellectually, and who has always been willing to advise me concerning my future. I would also like to extend my gratitude to Mr. Thierry Jardin for useful suggestions on my manuscript, interesting discussions and his great kindness. Both of them were always available to answer my questions.

Un grand merci à mes parents et à ma femme Alexandra pour m’avoir supporté dans les moments difficiles du projet et avoir toujours été derrière moi pour me suivre et m’encourager.
Content

Introduction ............................................................................................................................................. 6

1. Bibliographic review .......................................................................................................................... 8
   1.1. Control methods Classification ............................................................................................... 8
   1.2. Vortex shedding behind a cylinder ............................................................................................ 10
   1.3. Flow Control examples ......................................................................................................... 12
   1.4. 3D forcing and synthetic jets ................................................................................................ 14
   1.5. Conclusion ............................................................................................................................. 16

2. Numerical investigation methods ..................................................................................................... 18
   2.1 Presentation of Fluent .............................................................................................................. 18
   2.2 Possible ways to simulate turbulence ......................................................................................... 19
      2.2.1 Direct numerical simulation ................................................................................................. 19
      2.2.2 Large eddy simulation ...................................................................................................... 19
      2.2.3 RANS modeling ................................................................................................................ 20
   2.3 Pressure velocity coupling ........................................................................................................ 21
   2.4 Discretization description ......................................................................................................... 22
      2.4.1 Temporal discretization .................................................................................................. 22
      2.4.2 Spatial discretization ...................................................................................................... 24
   2.5 Post-processing ........................................................................................................................ 24

3. Results ............................................................................................................................................. 26
   3.1 Two-dimensional cases .............................................................................................................. 26
      3.1.1 2D cylinder ................................................................................................................... 26
      3.1.2 2D ogive-cylinder ......................................................................................................... 28
   3.2 Three-dimensional cylinder cases .............................................................................................. 30
      3.2.1 Investigation at Re=300 .............................................................................................. 32
      3.2.2 Investigation at Re=220 .............................................................................................. 36
   3.3 Three-dimensional cylinder cases with 2D control method applied ............................................. 40
      3.3.1 Blowing at 90° ............................................................................................................. 40
      3.3.2 Blowing at 110° ......................................................................................................... 41
      3.3.3 Suction at 110° ............................................................................................................. 43
   3.4 Three-dimensional cylinder cases with 3D control method applied ............................................. 45
      3.4.1 Blowing/suction at 90° ................................................................................................ 45
3.4.2 Synthetic jets at Re=300 ................................................................. 51
3.4.3 Synthetic jets at Re=500 ................................................................. 53
3.4 Three-dimensional ogive-cylinder case .............................................. 59
4. Conclusion ...................................................................................... 64
REFERENCES ..................................................................................... 66
Introduction

ISAE is counted among the best aeronautical universities in Europe and benefits directly from being located in Toulouse, the heart of the aerospace industry in France. The aerodynamic, propulsion and energetic department (DAEP) is located in the Ensica campus and boasts 15 researcher-professors and a technical team composed of 21 people. Among the many facilities available to staff and students, the DAEP is equipped with one shock tube and seven wind tunnels; four small subsonic, two small supersonic and a medium subsonic wind tunnel (3m*2m). The wind tunnels are used by students in the framework of their project and by the researchers in the framework of the various partnerships that exist with aerospace industries.

Among the many projects that exist, my final project study was within the Airflow Influence on Airdrop (AIA).

The previous project started in July 2002 and is a multi-national project which involve various countries; the United States with the Natick Soldier Research, Development and Engineering Center and the United States Air Force Academy (USAFA), Germany with the Federal Office for Defense Technology and Procurement (IABG), the United Kingdom with the Air Warfare Center (AWF) and the Joint Air Transport Evaluation Unit (JATEU) with the Royal Air Force (RAF) and France with ISAE. The goal of the project is to simulate and analyze the near C-130 Hercules flow field and then evaluate the influence of passive and active flow control method. This carrier aircraft has been extensively used and is essential while cargos, troops or material supplies has to be delivered rapidly in difficult access areas to military troops or during disasters that require humanitarian supply. The upsweep region in the aft fuselage leads to an increased drag, due to the highly detached flow and to strong upsweep vortices. The latter can be dangerous, by several means, during airdrop operations. For instance parachutes, while caught on the upsweep vortices can touch the empennage if they exit from the rear door. A good knowledge of the flow pattern around the aircraft is thus of particular interest to be able to control this flow by, for instance, attenuating the upsweep vortices. The ultimate goal of control device is to cancel any undesirable effect in the aft fuselage and enhance airdrop capability. In this sense it was proved that the aft fuselage region is the site of highly complex flow. Indeed this region is characterized by a massive separation, highly three-dimensional, where one can observe a pair of counter-rotating vortices also called upsweep vortices. These vortices then interact with the lower side of the empennage and cause the flow to detach resulting in counter-rotating vortices also called induced vortices.

In order to efficiently control such a complex flow, characterized by a highly unsteady behavior, both in closed configuration and in open cargo bay, it is particularly important to have a strong and clear knowledge of the dynamics associated with the use of control methods. As it will be explained in the bibliographic review, a strong theory behind control method is still lacking. Thus acquiring a good understanding on how control methods interact with the flow, one must go back to simple and well-known configurations, for which the dynamic of the wake is already mastered. With these configurations, one can investigate how 3D forcing or synthetic jets affect the instabilities present in the wake, and more importantly how they affect the origin of these instabilities. Once control methods effects at this level is better understood and efficiently implemented, in terms of drag...
reduction, vortex shedding cancelation, lock-on regime ..., one would eventually be able to analyze their effect on more complex flow with already a strong background on the area.

Military transport aircraft (C-130)

Circular cylinders offer excellent opportunities to test active flow control method. Indeed the flow around circular cylinders has been extensively studied for many years and became the archetype for the study of unsteady flows. Although simple from a geometrical point of view, it offers a wide variety of flow phenomena, clearly visible, well understood and broadly documented in the literature. The circular cylinder also presents a highly detached flow, which is of direct relevance in the AIA project contest.

This report presents numerical investigations conducted on the cylinder and ogive-cylinder at low Re (100-1500). As these cases have not been investigated before in this project, the first goal was to validate an optimal mesh. Once validated, different control methods were tested and a detailed study of the wake was conducted to understand how the control method interacts with the flow. The desired outcome is that these investigations will give a better understanding of active flow control method for future people working on this project.
1. Bibliographic review

Flow control behind aerodynamic and bluff bodies has been investigated since decades and has been one of the main areas of research for the past ten years. Since then many new steps were performed all around the world allowing new knowledge on the field resulting in some cases at a drag reduction ($C_d$) of 70%. Still, this is an open topic and despite an increasing number of projects on the flow control field, the theory behind flow control remains unclear.

1.1. Control methods Classification

In this chapter, the main insights on the area are presented.

Flow control methods are usually divided into two groups, namely active and passive methods. Passive flow control methods refer to mechanism where there is no power input. Among them, one can mention surface modifications such as change in surface roughness, presence of dimples or longitudinal grooves. One can also mention geometric modification in the spanwise direction such as a segmented or wavy trailing edge.

On the other hand active flow control methods refer to mechanisms using a power supply. High-frequency rotation of the circular cylinder, base bleed, synthetic jets, single dielectric barrier discharge plasma all belong among these methods. When well implemented, these methods can turn out to produce drastic reduction of $C_d$. However, even though these methods proved to perform well, their implementation in an industrial context is one of its main challenges, due to their lack of efficiency.

Active flow control method can themselves be divided into open-loop or closed-loop methods. The latter refers to control methods that require sensing and actuation while active open-loop control methods do not require feedback sensors. Feedback control methods present the advantage that it continuously modify the control input depending on the flow system response. Flow past bluff bodies usually present multiple global modes; each of these modes becomes unstable at a certain point. At low Re, when only one of these mode becomes unstable, linear feedback control is possible using a single-sensor actuator feedback loop. Conversely, at high Re the wake has multiple unstable modes, which require multiple feedback sensors to have complete flow-field information. However, not all of these information are required to have feedback control. In other words, low-dimensional description of the flow features may be sufficient to control the flow. This is precisely the idea behind control methods based on reduced-order models. Finally optimal and suboptimal control theory represent the last closed loop active flow control category. The idea is to minimize a “cost function” defined differently in each case. The “cost function” can be the lift or drag coefficient, the difference between the velocity field and the steady laminar flow... The efficiency of these methods is closely linked with the choice of the cost function, and a clear knowledge of the flow is required to apply such a control method. For instance Min & Choi (1999) showed that drag is reduced more when the cost function is the difference between the real and potential-flow surface pressures than when the cost function is the drag itself.
Another, more related to fluid mechanics, classification of control method is the division based on which part of the flow the control wants to modify.

If the control method aims to delay separation, either through boundary-layer transition or though early separation and reattachment, the literature refers to “Boundary-layer control methods”. On the other hand if flow control is performed directly through wake-modification not caused by separation delay, the literature refers as “Direct-wake control methods”.

Boundary-layer control methods are certainly the most spread, well-known, and famous method, in the sense that triggering boundary-layer instability in order to decrease drag coefficient comes immediately to mind for anyone who has little knowledge in fluid mechanics. In fact it is well-known that a laminar boundary-layer, has little resistance to an adverse pressure gradient, due to its low near-wall momentum. As the flow evolves, the adverse pressure gradient will cause the boundary layer to separate. If the Re number is high enough, one can trigger transition to turbulence before the boundary-layer separates. The new turbulent boundary layer has a strong near-wall momentum. As a consequence the friction drag will increase in a turbulent boundary-layer, but the pressure drag which has a considerable effect on the drag coefficient will decrease.

Forced transition can be achieved by a trip device such as zigzag or plastic tape, dimples and surface roughness. Another way is to use a vortex generator, such as jet turbulators, which will produce strong near-wall momentum and thus delay main boundary layer separation.

These ideas have been extensively used in all kind of fields. In sports for instance, most golf balls are covered by approximately 350 dimples. The dimples cause the boundary layer to transition from laminar to turbulent, and hence reduce pressure drag. Even in football, the presence of seams around the ball encourages turbulent behavior, resulting in drag reduction. In these two cases transition lower the wake thickness and enable the ball, for certain configuration, to fly higher and longer than a smooth ball.

In aerospace, some airplanes are equipped with vortex generators. They consist of small plates about an inch deep in a row spanwise along the wing (Fig.1). When implemented on aircrafts they delay flow separation and thus aerodynamic stalling and ensure aileron effectiveness. They were for instance used in the two seat one engine Symphony SA-160, the C-17 Globe master III and the EMBRAER EMB-120.
Boundary layer control methods can also consist of early separation and reattachment as mentioned previously. For instance, at a certain Re, the drag coefficient over a circular cylinder or a sphere significantly decreases, and one can observe the so-called “drag crisis”. In fact in the case of the separating laminar boundary layer, the flow often transitions just behind the separation point. The high-momentum shear-layer is entrained towards the surface, so that a normal, attached turbulent boundary layer forms, resulting in the delay of the main separation. Because there is a small region of reverse flow, streamlines form a bubble-shaped pattern.

A few researchers investigated the control input to generate early separation and reattachment. Jeon et al. (2004) used a local time-periodic blowing and suction over a sphere at the subcritical Re. They showed that disturbances from high-frequency forcing (much higher than the vortex shedding frequency) rapidly grows along the separated shear-layer which leads to the reattachment of the boundary layer. The main separation is thus delayed, i.e. the drag coefficient is significantly decreased.

After having presented the main insights of the boundary layer control, it is of particular interest to discuss direct-wake control. It has the advantage to act directly on one of the main contributions to the bluff-body drag; the wake. It is important to point out that the control changes the wake field directly and not through separation delay. Thus it can be used on any bluff body whether it has a fixed or movable separation point.

Before presenting different methods, one should first have a glimpse at the wake dynamics. Indeed a clear knowledge of the latter plays a crucial role in building up a direct-wake control method. As the numerical cases investigated during this project mainly concern flows around a cylinder, the vortex shedding process is described below.

### 1.2. Vortex shedding behind a cylinder
The phenomenon of vortex shedding has been extensively studied since the early drawings of Leonardo da Vinci, by Karman (1948), Föppl, Roshko (1955), Gerrard (1966), Berger (1967) and Williamson (1986) among many others. For instance the cylinder case has received a great deal of attention. Flow over a circular cylinder is a simple and useful model for many applications concerning obstacles to flow.

In the case of a smooth cylinder before Re= 46, the wake is symmetric and eddies behind the cylinder are steady. Twin vortices spinning in opposite directions form behind the cylinder, and become more elongated as the Re increases. When Re reaches 46, a highly-energetic wake appears, characterized by the presence of a double row of alternate concentrated vortices, known as a Karman Vortex Street. The break-up of the recirculating and symmetric bubble in an unsteady wake flow has been the subject of a large number of investigations. Gerrard (1966) described the onset of vortex shedding as follows: “The growing vortex continues to be fed by circulation from the shear layer until the vortex becomes strong enough to draw the other shear layer across the wake. The approach of oppositely-signed vorticity in sufficient concentration cuts off further supply of circulation to the vortex, which then ceases to increase in strength. We may speak of the vortex as being shed from the body at this stage.” More recent studies, based on global stability approach, suggested that the symmetric bubble wake flow located behind a circular cylinder is globally unstable above the critical Re, in the sense that a perturbation located at any distance from the body grows. The lower vortex moves upstream and penetrates the flow between the upper vortex and the body while the upper vortex tends to move downstream and upwards. This dynamics constitute the origin of the shedding process as it brings instability to the wake, which is globally unstable for Re>46.

At this state the separation point oscillates around its average location. A vortex is generated as the separation moves, it is then shed from the cylinder and the separation point moves back. When Re reaches 150, three dimensional effects cannot be neglected anymore. In fact, the Re range 150-300 represents the transition to turbulence in the cylinder’s wake. Again this transition has received a great deal of attention. Williamson (1996) and others achieved thorough analysis of the set-up of turbulent-wake transition. It is known that during the three-dimensional transition regime, there are two modes of formation of streamwise vorticity in the near wake, each of which occurring in a different range of Reynolds number, associated with a different scale of streamwise vortex structure and with a different vortex shedding frequency. Mode A instability is associated with the inception of vortex loops and the formation of streamwise vortex pairs due to the deformation of primary vortices as they are shed. Those streamwise vortex pairs reside and are stretched in the braid region, between primary Karman vortex structures. This mode scales on the larger physical feature in the wake flow, namely the primary vortex cores, and has a spanwise wavelength of 3-4 diameters. Mode A instability occurs at Re of 180 and can be characterized as a Strouhal-Reynolds relationship discontinuity. This first discontinuity marks the passage from a laminar wake to a 3D wake. Mode B is associated with finer-scale streamwise vortices and occurs around Re of 240. This mode, on the other hand, scales on the smaller physical length scales, namely the braid shear layer and has a spanwise wavelength of 1 diameter. Again mode B instability is associated with a St-Re discontinuity. However this second discontinuity is by no mean similar to the previous one as it involved a gradual transfer of energy from one mode of shedding to the other. All these differences between those two modes tend to prove that their formation mechanisms are clearly distinct. The first instability is self sustaining. At some of the vortex loop ‘sites’, two sided vortex dislocations appear. They then grow rapidly into large-scale structures downstream and are responsible for much of the large-scale
distortion and break-up to turbulence of the vortex wake. When mode A instability and dislocation appears, one can thus observe a reduction of fluctuation level, a growth in the size of the wake formation region, a reduction in base suction and thereby a reduction in Strouhal number. The second instability is influenced by the existence of a reverse flow in the bluff-body wake (Fig.2). In fact the forming braid shear layer lies in proximity to the previously formed braid which comprises the streamwise vortices brought upstream by the reverse flow. The disturbances thus imposed on the forming braid sets the preferred locations of the new braid-vortices. The mode B has thus an in-phase symmetry for the streamwise vortex oppositely at the mode A that has an out-of-phase symmetry.

Fig.2 Physical mechanism in the braid shear layer to produce mode B streamwise vortices (Figure from Journal of Fluid mechanics 1996, Williamson)

After the ‘wake’ transition in the Re range 150-300, which involves small-scale streamwise vortices and large-scale vortex dislocations, one can observe a ‘shear layer’ transition for Re in the range 400-200 000, which involves small-scale shear-layer instability vortices. At Re number of around 1000, secondary, or Kelvin-Helmholtz, vortices begin to form in the shear layer. Finally one can also observe a ‘boundary layer’ transition at Re= 200 000, from which the consequence, such as the drag crisis, are well-known and were developed in the boundary-layer control method part.

All these features, i.e. the streamwise wavelength, the formation mechanism of each mode and the Strouhal number associated, are of particular interest when building wake-control method.

1.3. Flow Control examples

First steps on the wake control method are 3D geometric modification in the spanwise direction. Bluff-body front edge waviness proved to be particularly efficient in reducing drag. Darekar & Sherwin (2001) investigated numerically the flow past a square cylinder presenting a wavy stagnation face at low Re. They achieved brilliant result by suppressing Karman vortex shedding behind the cylinder. Instead of a non symmetric unsteady wake, they observed a steady and symmetric wake. The waviness of the leading edge distorted the 2D character of the flow, thus of the shear layer, and
the new 3D shear layer was less susceptible to degenerate into a Karman vortex street. It is interesting to note that the optimal wavelength of spanwise waviness was close to the mode A instability spanwise wavelength. Yoon (2005) investigated the effect of a small-size tab, located near the separation point at Re=100 on a circular cylinder. The dimensions of the table were \( \frac{L_y}{2} = \frac{L_x}{2} = 0.2 \). He found that this control method reduces drag and also attenuates the Karman vortex shedding. Again it is interesting to note that the optimal spacing between the adjacent tabs was close to the mode A instability spanwise wavelength. For a spacing \( \lambda=4d \), the vortex shedding was completely suppressed and the drag coefficient was 1.1 instead of 1.34. Strykowski & Sreenivasan (1990) investigated the presence of a small secondary cylinder at low Re, Re<250, and succeeded in suppressing the vortex shedding. They attributed the phenomenon to subtle changes in the near wake of the main cylinder, the alteration of global stability and thus the temporarily suppression of evolving modes responsible for the formation of vortex shedding. They also added that to achieve drag reduction the secondary cylinder should be properly located. Another example of control method acting on the global instability is the splitter plate. Roshko (1955) and many others (Bearman (1965), Hwang et al. (2003), Ozono (1999)) investigated the effects of the disturbance imposed by introducing a splitter plate in the near wake region. He discovered that a critical region exists downstream of the cylinder where interaction between opposing shear layers needed to be cut off to prevent vortex formation.

The notion that wake dynamics are determined by the global instability of the near wake velocity field and not by details of the separated flow at the body surface is an interesting concept that receives a lot of attention. The body itself is necessary to create the shear layer but it is the interaction of the shear layers through opposing signs of vorticity which gives rise to the Karman vortex street. In other words, the existence of a region having absolute instability is a necessary condition for vortex shedding. Sakamoto et al. (1994) conducted experiments to investigate the effect of a control cylinder placed near the outer boundary of the separated shear layer originated by the wake behind a square prism at Re=4.2*10^4. They observed significant drag reduction and attributed that to the alteration in the separated shear layer affected by the control cylinder. More recently Hwang & Choi (2006) studied the effect of a control cylinder on global instability and found that the region of attenuating instability shrinks with increasing the ratio between the main and secondary cylinder.

Another way of acting on the local instability is to introduce base bleed (Wood, Bearman (1967), Monkewitz (1988)) or wake heating. Leu & Ho (2000) who investigated the latter method found that the decrease of the fluid density in the near wake, due to the heating, affects the absolute instability growth rates.

In the range of direct-wake control, one can mention the single dielectric barrier discharge. They consist of plasma actuators generating a body force as a consequence of the air ionization by ac input. Thomas et al. (2008) implemented four plasma actuators on the downstream half of the cylinder and studied their effect at Re=30000. They observed that the vortex shedding had been completely suppressed and that the wake width significantly decreased. One of the great advantages of such a flow control actuator is that it comprises no moving part, requires no implementation of holes and is efficient in terms of power input to flow momentum ratio.
1.4. **3D forcing and synthetic jets**

Before ending this chapter, it is of particular interest to present a last control method, namely synthetic jets. The idea is born in the 1990s when Glezer (2002), among others, investigated for the first time this control method. Glezer (2002) describes this device as follows: “An isolated synthetic jet is produced by the interactions of a train of vortices that are typically formed by alternating momentary ejection and suction of fluid across an orifice such that the net mass flux is zero”. The idea seems quite attractive, especially since it uses the working fluid of the flow to generate synthetic jets resulting in a nonzero linear momentum transfer to the flow without net mass injection. Furthermore our attention to such a device is motivated by previous encouraging results that can be found in the literature (Smith et al. (1998), Amitay et al. (2001), Glezer & Amitay (2002). All these studies were concluded by stressing that synthetic jet actuators significantly decrease drag, by suppressing separation at moderate Reynolds numbers \([O(10^6)]) or by suppressing von Karman shedding at a Re range when this phenomenon is usually observed.

![Fig.3 Structure of synthetic jet](Figure from electronicdesign.com)

Many synthetic jets configurations can be implemented:

- First the pressure variation across the orifice which is needed to create a blowing/suction mechanism can be imposed by several means. Typically, one can generate an acoustic field, either by creating a standing wave in an acoustically driven tube to induce an oscillating velocity filed, or by transmitting high-amplitude sound waves in a tube. Other methods consist of oscillating the boundary of a quiescent medium, such as an oscillating diaphragm or piston mounted on a sealed shallow cavity, facing the orifice (Fig.3).

- Secondly the implementation of synthetic jets around bluff body can lead to many configurations. Some searchers decided to investigate the effect of orifices mounted on the upper part of the bluff body, other on the lower parts and others on both parts. When mounted on a bluff body in the spanwise direction, one can vary the space step between each orifice, the wavelength, the strength, the frequency and/or the orifice’s diameter.
associated with the blowing/suction profiles. When mounted on both parts of the bluff body, one can investigate the effect of an in-phase or an out-of-phase forcing between the upper and lower part.

- Finally the direction of the jet can vary, i.e. one can decide to turn the jets toward the normal direction of the local flow, toward its tangential direction or between those two directions. Last but not least the location of the jets at various azimuthal positions around the bluff body, where local pressure gradients are different, can produce different results. The control method can be located before or after the unforced separation point and even where it takes place. Synthetic jets can be used alone or in pair. Thus when implementing synthetic jets, one needs to take into account in all these elements to find the optimal configuration.

Despite the various numbers of studies on this topic, it seems fair enough to admit that the physical understanding of perturbations associated with the presence of synthetic jets remains unclear. However, some important insights can already be presented.

The effect of a synthetic jet on the flow can be categorised following three parameters; the dimensionless frequency, the momentum coefficient and the Reynolds number based on the impulse:

\[
\hat{f} = St \times \frac{U_j}{U_\infty} \tag{1.3.1}
\]

\[
C_m = \frac{I_j}{\frac{1}{2} \rho U_\infty^2 D} \tag{1.3.2}
\]

\[
Re_{I_0} = \frac{\rho I_0}{\mu a} \tag{1.3.3}
\]

Where St is the Strouhal number based on the jet width and \(U_\infty\) the free-stream velocity; \(U_j\) the jet flow, \(I_0\) the momentum associated with the discharge, \(I_j\) the time-averaged jet momentum per unit length during the outstroke.

Glezer & Amitay (2002) showed that there are two distinct interaction domains, depending on the above parameters. These interactions regions alter the flow above the surface of the cylinder and thus its apparent aerodynamic shape. Before \(\hat{f} = 0.1\), discrete vortices exist, whereas after \(\hat{f} = 0.1\), a closed recirculation region can be observed. Another important point to underline is that the Re stresses and more generally the power spectra in the cylinder wake are substantially reduced by the presence of the actuation. This suggests that the actuated flow causes an enhanced dissipation within the wake. In fact, all the properties of the flow seem to be modified in an actuated case; starting from the pressure distribution (Williams et al. (1991)) to the vorticity distributions (Glezer & Amitay (2002)). Another important point is that synthetic jets can be used to modify, and thus control, flow that scales one to two orders of magnitude larger that the characteristic length scale of the jets themselves.

Amitay et al. (1997) investigated the 2D interaction between the jet and cross flow for different azimuthal jet position at Re=4000. The cylinder was instrumented with a pair of adjacent rectangular synthetic jet actuators (0.5*140 mm) that was spaced 205mm apart and flushed collinear with its axis (or normal to the cylinder surface). With a \(C_m\) of 10^{-1} and when located at 180°, the external flow appeared to be almost attached to the surface. Rediniotis et al. (1999) used a tangential synthetic jet,
located just downstream of the non actuated separation line for delaying separation over a circular cylinder at Re=6000. By setting the natural unstable frequency of the cylinder to the jets, they delayed separation and argued that this was due to an increased mixing on the boundary layer.

Concerning 3D forcing distribution, Kim & Choi (2005) achieved thorough results after a numerical investigation on the blowing and suction effect on the drag and lift forces on a circular cylinder and on the vortical evolution in the wake behind the cylinder for a range of Re from 40 to 3900. The slots were placed both on the upper and lower surface and 90° and -90°. The orifices were spaced of one cylindrical diameter in the spanwise direction and had a sinusoidal, but steady in time, spanwise profile. The optimal configuration for Re=100, which includes a spanwise spacing of 4D, a forcing amplitude of $0.08U_{\infty}$, completely suppressed the vortex shedding and the drag coefficient was reduced by up to 20%. For Re=220 and 300, and with the same configurations, the drag coefficient was reduced by up to 21 and 23% respectively. However the Karman vortices were not completely suppressed but weakened and the wake now displays a reorganised vortical structure. Three points are interesting to stress:

- The results obtained are quite similar to those of the wavy square cylinder case mentioned previously. Thus the same idea remains behind 3D forcing; the phase mismatch along the spanwise direction in the vortex shedding process. The mismatch introduces incoherence in the flow, which as a way of consequence becomes more three-dimensional. This process weakens the shedding vortex formation and decreases the drag.
- The spanwise length recalls the mode A instability spanwise length. As mentioned previously the wake presents an elliptical instability with a wavelength of 4 diameters that degenerates only for Re=180. Introducing a disturbance having the same spanwise wavelength, at lower Re, seems to significantly alter the wake.
- Kim et al. (2006) found that the associated disturbance here reduces the local absolute growth rate of the model wake profile suggested by Monkewitz and thus suppresses vortex shedding.

1.5. Conclusion

As depicted in this bibliographic review, flow control method devices are numerous. Passive control methods are clearly easiest to implement, but present several limits in their application. In real on-board applications, many configurations are encountered; passive flow control method can have a strong influence on the flow in certain configurations, but no influence at all on others. Conversely, active flow control may be harder to implement, but it is often seen as the best solution for flow control. Indeed, it presents several advantages as it acts directly on the origin of instabilities and can be used on demand. Active closed-loop provides the best results but its implementation in practical situation is much harder than an Active open-loop control method. Many articles have been published on active open-loop method. However, this number must not hide the fact that numerous studies are still needed to understand clearly in which configuration active control methods can perform well. Indeed articles and research in this field present results obtained while applying active control method, but very little conduct a deep physical analysis of the resulting wake. Most often researchers who perform thorough investigations use well-known shapes, such as the cylinder, the square-cylinder or even the sphere at low Re. This project aims to conduct investigations in continuity to those already done. With the use of previous papers, the numerical investigations
presented in this report aim to acquire a better knowledge concerning active control methods, namely synthetic jet and 3D forcing. Eventually this knowledge is to enable person who will be working on the AIA project, to develop efficient active control method in the more complex case of the up-sweep region of the C-130 Hercules.
2. Numerical investigation methods

In contrast to numerous studies on wake control method, there are relatively few publications of 3D numerical investigation of synthetic jets and 3D forcing. Kim & Choi (2008) referred to the latter point as follows: “Nevertheless, we feel that a detailed and systematic analysis of the effect of a synthetic jet on the flow past a bluff body such as circular cylinder is still lacking in the literature, and thus further studies should be conducted in the near future”. The current investigation aims to first study the wake of basic shapes, namely the cylinder and the ogive-cylinder, at low Re, covering the interesting range Re=200 to Re=1500. The ultimate goal of such an investigation is then to develop a methodology for active flow control, efficient enough so it can be tested. The computation was performed using Fluent, a CFD software ideally suited for incompressible and mildly compressible flows and that will be presented next.

2.1 Presentation of Fluent

Fluent is the world leader in CFD and is present in all the continents. One of the reasons of such a success is that it is a general-purpose CFD, in the sense that it displays a large variety of model able to simulate different aspect of flow mechanic; Combustion, chemical mixing, multiphase flow and more importantly in our context turbulence. Although it does not contain any mesh generation software, the mesh can be generated by, for instance, Gambit, a software distributed with Fluent in which the geometry and mesh generation tools are integrated in a single Graphical User Interface (GUI). This non-trivial task may consume days of effort. Indeed the best choice for a grid system depends on convenience in generation, numerical accuracy, memory requirements and flexibility for localized regions of high or low resolution. Size control, structured (quad/hexahedral)/ unstructured (triangle/tetrahedral)/ hybrid meshing, boundary layer meshing, Cooper and paver meshing tools and built in mesh quality examination are among other the powerful tools provided by Gambit to build a successful grid.

The Navier-Stokes equation that is aimed to resolve in the framework of this project (incompressible case) is:

- Conservation of momentum:

\[
\frac{d(\rho u)}{dt} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \nabla \cdot (2\mu \mathbf{D}) + F_{st} + \rho \mathbf{g} \tag{2.1.1}
\]

- Conservation of mass, or continuity equation:

\[
\frac{dp}{dt} + \nabla \cdot (\rho \mathbf{u}) = 0 \tag{2.1.2}
\]

Where \( \mathbf{u} \) is the velocity vector, \( \rho \) is the density, \( t \) is the time, \( p \) is the pressure, \( \mu \) is the dynamic viscosity, \( \mathbf{D} \) is the rate of deformation tensor with the components \( D_{ij} = \frac{1}{2}(u_{i,j} + u_{j,i}) \), \( F_{st} \) is the body force due to the surface tension, and \( \mathbf{g} \) is the gravity vector. These equations are rather complex to solve and Fluent often uses approximations to evaluate some terms. These approximations will be the subject of the next chapters.


2.2 Possible ways to simulate turbulence

Turbulent flows are often encountered in engineering problem. Such flows are characterized by eddies with a wide range of length and time scales. It is well known that large eddies scales with the characteristic length of the flow and the smallest scales, more universal, are responsible for the dissipation of turbulent kinetic energy. A first way to numerically solve the Navier-Stokes equations is to resolve the whole spectrum of spatial and temporal scales involved in the turbulent flow. This method is known as the direct numerical simulation (DNS) and will be discussed below.

2.2.1 Direct numerical simulation

DNS, as indicated by its name, uses no turbulence model. Instead it solves the Navier-Stokes equations without any averaging or approximations, other than the ones associated with the numerical discretizations. All the motions in the flow are resolved and thus the result is accurate enough so that the entire flow being studied is controlled. It can provide more information than experimental measurements, and is used to understand the mechanisms of turbulent production and dissipation. However this simulation has its downside. In fact, the computational cost required for DNS becomes quickly restrictive. The smallest dissipative scale, i.e. the Kolmogorov micro scales, \( \eta \), corresponds to the scale in which the viscous stresses are comparable to the inertial processes, i.e. \( Re = \frac{u' \eta}{\nu} = 1 \), where \( u' \) is the smallest eddies velocity and \( \nu \) the viscosity. It thus depends on the viscosity and the rate of kinetic energy dissipation, \( \varepsilon \). A quick analytical analysis gives:

\[
\eta = (u^3/\varepsilon)^{1/4}
\] (2.2.1)

The smallest resolved length scale is required to be of \( O(\eta) \). By assuming the small scales are in quasi-equilibrium with the large scales, in other words that the rate at which energy is dissipated into heat in the small eddies will be determined by the rate at which energy is transferred from the large eddies to the small eddies, \( \varepsilon_f \), one can write:

\[
\varepsilon = \frac{u'^2}{t_{vis}} \approx \varepsilon_f = \frac{u_T^2}{t_{large}}
\] (2.2.2)

Where \( u_T, t_{large} = l_T/u_T \) and \( l_T \) represents the large velocity, time and length scales respectively and \( t'_{vis} = \eta^2/\nu \) the small time scale. The contrast between large and small scales is thus an increasing function of the Re, based on large scales:

\[
\frac{l_T}{\eta} = Re^{3/4}
\] (2.2.3)

This previous result, valid for free turbulence (no wall) leads to a number of grid points, in a 3D case, of \( N_{nodes} \sim Re^{9/4} \). For instance for a modest Re number, Re=3000, the above ratio is equal to 405, in this case the number of control volumes needed to resolve all the eddies in a three-dimensional computation would be greater than \( 6.6 \times 10^7 \). Clearly, DNS does not suit a large range of Re, and Fluent offers different models to simulate turbulence which will be explained next.

2.2.2 Large eddy simulation

Large eddy simulation (LES) is a three dimensional simulation, in which the large unsteady turbulent motions are directly resolved, while the smaller scale are modelled. The premise of this point is that
large scale motions contain the larger fraction of energy in the flow responsible for transport of conserved properties. This allows the use of coarser mesh and larger time-step than in DNS. A low-pass filtering is used to separate the large-scale motions from small-scales ones. This filtering process filters out the eddies whose scales are smaller than the grid spacing used in the computations. The unresolved eddies are treated by approximating their effect through a remaining term after the filtering process, called the subgrid-scale turbulent stresses. Fluent models this with a Boussinesq hypothesis. The stress is modelled using a local eddy viscosity (proportional to the local grid size) and the rate-of-strain tensor. However LES still needs a large number of control volume, computational cost involved is also still high in terms of memory (RAM) and CPU time.

2.2.3 RANS modeling

Thankfully Fluent offers other turbulence models, known as Reynolds Average Navier-Stokes (RANS) modelling. This is probably the most used modelling, as in an engineering contest, one is more interested in the macroscopic quantitative properties of the flow. As previous simulation, this modelling is motivated by the fact that passive scalars, such as momentum, mass or energy is transported mostly by large eddies. In the RANS equations, the Reynolds stress terms are unknown and need to be solved. The equation for the Reynolds stresses can be derived from the Navier-Stokes equations but will involve higher order moments which in turn will involve higher order moments. Thus this term needs to be modelled, in terms of mean flow quantities, at some stage. This is referred to as the closure problem. In other words, where DNS and LES can produce an overwhelming quantity of detailed information about a flow structure, RANS models generally gives the appropriate amount of information needed in an engineering context. However in the framework of this project, a high level of detail is needed to have a clear understanding of the effect of the control method when applied on a well-known configuration. A clear knowledge of the latter will eventually allow an efficient use of control method on more complex cases. Thus a DNS was used, indeed this approach is largely justified while considering the range of Re numbers (100-1500) investigated, as explained in §2.2.1, and ideally suits to have a maximum number of information on the wake. Therefore no deep details about RANS modelling will be given.

Fluent offers several models (one-equation model, two equation model, seven equation Reynolds stress...) to model the Reynolds stresses. Each of them suits for particular regimes, and the choice of the model depends on the case investigated. There is not yet a single turbulence model that can reliably predict all turbulent flows found in industrial applications with sufficient accuracy.

The choice to use DNS,LES or RANS depends on the problem needed to be solved as was discussed previously. Figure 1 briefly summarizes the differences between these three simulations.
2.3 Pressure velocity coupling

Fluent is a collocated finite volume code. That means that pressure and velocity (and more generally all the flow-field variables) are discretized on the same grid points. The variables are then stored at the central node of the control volumes. This arrangement presents several advantages; solving the equations is simpler, all geometric data are stored once and it allows the ease of transfer of information between various grid levels. One now needs to solve the integral form of Eq. (2.1.1) over one control volume resulting in a discretized equation of the form:

\[ a_P^U U_p = \sum a_i^U U_i + S_{up} + S_u \]  \hspace{1cm} (2.3.1)

Where \( U \) are the velocity components, \( a_P^U \) are coefficients that depend on the grid space and on the viscosity, little indices represents cell volumes around the control volume, \( S_{up} \) a source term arising from integrating the pressure gradient over the cell and \( S_u \) represents any other source terms.

However the pressure field is not known \textit{a priori}. Fluent provides several pressure-velocity coupling algorithms such as SIMPLE (Semi-Implicit Pressure Linked Equations), SIMPLEC (SIMPLE Consistent) and PISO (Pressure Implicit with Split of Operators). The SIMPLE algorithm is a guess-and-correct procedure on the staggered grid arrangement.

First the calculation begins by guessing the pressure field, and then the discretized momentum equations are solved using the guessed pressure field.

At this stage the continuity equation is in general not satisfied and correction terms need to be added so that they satisfy both the momentum and continuity equation:

\[ u^* = u + u' \]  \hspace{1cm} (2.3.2)

\[ p^* = p + p' \]  \hspace{1cm} (2.3.3)
This leads to:

\[ u' = u_1' + u_2' \] (2.3.4)

\( u_1' \) and \( u_2' \) are terms that depend on the pressure and the velocity on the adjacent cells. This choice of split ensures a simple linkage between the main variables. The more complicated terms involving velocity corrections at neighbouring nodes are put into the second part of the correction. \( u^* \) and \( v^* \) must now satisfy the continuity equation Eq. (2.1.2), which is integrated over the control volume.

When using the Simple scheme, only the first correction part, i.e. \( u_1' \), is taken into account. Once the pressure correction terms are calculated, the velocity correction components terms are calculated and, after updating all the flow field variables and solving other scalar equations, convergence is checked. This procedure is repeated until convergence is achieved. SIMPLE scheme can be rather low to converge, mainly because as mentioned above it neglects some term in the correction procedure. SIMPLEC and PISO scheme are better scheme in the sense that they take into account more correctional step.

Those coupling schemes are part of the so-called pressure-based segregated algorithms, meaning that the governing equations are solved sequentially, and are thus segregated from each other. Another pressure-based code consists of coupled algorithms. As indicated by its name, the momentum equations and the pressure-based continuity equation are solved together, forming a coupled system. The rate of solution convergence significantly improves but on the same time, the memory requirement is widely increased. Comparison between these both algorithms is presented in Fig.2.

### 2.4 Discretization description

#### 2.4.1 Temporal discretization

Fluent offers first order and second order discretization for the time-dependant equations. For more accuracy, a second order discretization was used during this project, which can be given by:

\[ \frac{d\varphi}{dt} = \frac{3\varphi^{n+1} - 4\varphi^n + \varphi^{n-1}}{2\Delta t} \] (2.4.1)

Where \( \varphi \) represents a scalar quantity, \( \Delta t \) the time step and \( t = n\Delta t \) the current time.

For incompressible case, which is the case adopted in the framework of this project, Fluent uses only an Implicit Scheme to evaluate function that incorporate any spatial discretization. Although it requires more computational effort in each solution step, it allows for large-time step sizes, without
threatening the stability of the solution, although much care must be taken to ensure accuracy of the solution.

A Von-Neumann analysis provides that the previous requirement is equivalent at:

\[
\frac{u\Delta t}{\Delta x^2} < \frac{1}{2} \quad (2.4.2)
\]

\[
1 > 2 \frac{u\Delta t}{\Delta x^2} - \frac{u\Delta t}{\Delta x} \quad (2.4.3)
\]

Where \(\Delta x\) represents the grid step, and \(\Delta t\) the time step.

A necessary condition for previous requirement to be fulfilled thus leads to the so-called Courant-Friedrichs-Lewy or CFL condition:

\[
\frac{u\Delta t}{\Delta x} < 1 \quad (2.4.4)
\]

The quantity \(\frac{u\Delta t}{\Delta x}\) is called the Courant Number and represents the ratio of the time step to the time required for a disturbance to be convected across the cell. Its value can be checked in Fluent. For explicit scheme, the conditions present a serious restriction on the time-step. On the other hand, an implicit scheme solve a system of algebraic equations in order to calculate the values at all grid points simultaneously, they will thus not have such time step restrictions and they allow a much
larger Courant number. However Implicit schemes are not unconditionally stable as the Navier Stokes equations present non-linearity. This is why, even for an Implicit scheme, much care must be taken concerning the Courant number and its value should not be too large. The specific value of the maximum allowable Courant number depends on the case investigated. In the numerical cases investigated, a maximum CFL number of 5 was used.

2.4.2 Spatial discretization

The spatial terms in Eq. (2.4.1) still needs to be investigated. After integration over a control volume, the Green-Gauss theorem is applied for the convective and diffusive term, and the volume integral is transformed into a surface integral. Fluent uses a mid-point rule integration of the surface which is second-order accurate. To numerically evaluate the diffusive term Fluent uses a second-order accurate central-differences scheme.

To evaluate the face values, Fluent provides several upwind schemes. Upwind is used here because the face values are derived from quantities in the cell upstream, or upwind, relative to the direction of the normal velocity, and not relative to the general motion of fluid.

The schemes provided by Fluent all differ by the accuracy precision that they offer, the nature of the error introduced through the numerical viscosity, and by their stability criteria. However those schemes are not to be treated apart but together when launching a Fluent calculation. Indeed, it is common to first practice a calculation with a first order scheme, for its stability, and then switch to a higher order one once the solution is closer to convergence.

In the next chapters settings A refer to:

- Pressure based solver
- 2nd order Implicit in time formulation
- Second Order Pressure discretization
- Second Order Upwind discretization
- Simple Pressure-Velocity Coupling
- Absolute Criteria for convergence of 10^-6 for all variables.
- A time step of 0.05 and a maximum number of iteration per time step of 20

2.5 Post-processing

Fluent displays a large variety of tools to analyse the result once computed. One can choose to display the static pressure, the pressure coefficient, the velocity, the vorticity, the helicity, the flow rate, the Courant Number, the Strain rate, the pathlines, iso-surfaces... One can also choose to apply a FFT, to plot the drag, lift or moment coefficient and even \( Y^+ \), to refine or coarsen the mesh, among many others tool. Unfortunately for a deeper analysis, some Fluent tools have their own limit. The vorticity display counts among them. Vorticity, \( \omega \), is defined as the curl of the velocity. This approach
is fairly successful in some free shear flows. But in reality vortices do not always represent regions of high vorticity. For instance vorticity may be high in parallel shear flows where no vortices are present. Also it does not identify cores in a shear flow, especially if the background shear is comparable to the vorticity magnitude within the vortex. In fact even in free shear flows, this approach seems to give a misrepresentation of vortices. In an intense dynamic element, a vorticity sheet does not represent a vortex and the visualisation of the flow can display several vortices whereas in reality only one vortex is present.

A fundamental issue is the lack of a unique mathematical of physical definition of vortex as admitted by Jeong & Hussain (1995). Even the intuitive pressure minimum criterion to detect vortex cores is not suitable for all cases, such as the so-called “Karman’s viscous pump”, in which the pressure is constant in planes perpendicular to the vortex axis and the swirling motion is due to viscous effect. In this project the kinematic vorticity number, $N_k$, criterion, initially developed by Truesdell (1953), was used. He defined $N_k$ as follow:

$$N_k = \frac{\|\omega\|}{\|s\|}$$

(2.5.1)

Where $\|\omega\| = [tr(\Omega \Omega^T)]^{1/2}$, $\|s\| = [tr(SS^T)]^{1/2}$, $S$ and $\Omega$ are the symmetric and anti-symmetric components of $\omega$. Melander & Hussain (1993) identified the core of an axisymmetric vortex column as ‘a maximally connected spatial region with $N_k > 1$’. This approach has its downsides, as it gives a measure of the quality of rotation, regardless of the vorticity magnitude. However this approach suits well for the cases to be studied during this project, and was thus adopted.
3. Results

As mentioned earlier, the ultimate goal of the numerical investigation is to test the effect of 3D forcing and synthetic jets on a bluff-body. Before reaching that goal, there are a few steps to ensure that the final cases, representing a complex 3D geometry which requires weeks of Fluent calculation, are worth to be tested, and that the results found are valid enough to be trusted. In this sense, first 2D configurations were tested for the cylinder and the ogive-cylinder at different Re number. Once the meshing was configuration validated, the 3D configurations without control method implemented were tested preceding the 3D configuration with applied forcing.

3.1 Two-dimensional cases

3.1.1 2D cylinder

First the 2D cylinder was tested. The geometry configuration is summarized in Fig.3 and the meshing configuration in Table 1 and Fig.4. Triangular elements were used to mesh the domain.

![Fig.3 2D cylinder geometric configuration.](image)

<table>
<thead>
<tr>
<th>Edge</th>
<th>Interval Count</th>
<th>Ratio</th>
<th>Boundary Types</th>
</tr>
</thead>
<tbody>
<tr>
<td>BCD&amp;DB</td>
<td>50(each)</td>
<td>1</td>
<td>Wall</td>
</tr>
<tr>
<td>ED</td>
<td>50</td>
<td>0.94</td>
<td>In</td>
</tr>
<tr>
<td>BI</td>
<td>120</td>
<td>1.03</td>
<td>In</td>
</tr>
<tr>
<td>EF&amp;EL</td>
<td>25</td>
<td>1.05</td>
<td>Velocity-Inlet</td>
</tr>
<tr>
<td>IH&amp;IJ</td>
<td>20</td>
<td>1</td>
<td>Pressure-Inlet</td>
</tr>
<tr>
<td>FG&amp;LK</td>
<td>15</td>
<td>1</td>
<td>Wall</td>
</tr>
<tr>
<td>GH&amp;KJ</td>
<td>36</td>
<td>1</td>
<td>Wall</td>
</tr>
</tbody>
</table>

Table 1 2D cylinder meshing details.
The case was tested for Re=100 and Re=300. Only the case Re=300 will be detailed here as these two cases are basic. The value of the density of the fluid used for the calculation was set to 3 and proper viscosity to 0.01, in order to have Re=300. As the flow is unsteady for Re>47, an unsteady formulation was chosen combined with a 2nd Order Implicit formulation in time. Other parameters were left at the default setups of Fluent.

The time step size was 0.05, resulting in a maximum Courant Number of 1.6 and 3000 time step were computed. The drag coefficient ($C_d$) and the lift coefficient ($C_l$) are plotted in Fig.5. The mean value of $C_d$ was found to be equal to 1.43 in good accordance with Persillon & Braza (1998) who found 1.41 for the 2D case. The mean value of $C_l$ is 0, as it can be expected. The Strouhal number was found to be equal to 0.213, in good accordance with Persillon (1998) who found 0.209, and is plotted in Fig.6. The Strouhal number observed is the manifestation of the Von Karman vortex street that develops behind the cylinder.

![Fig.4 Meshing details of the 2D cylinder.](image)

![Fig.5 Lift (up) and Drag (down) coefficient for the 2D cylinder at Re=300.](image)
3.1.2 2D ogive-cylinder

The 2D ogive-cylinder was also tested for Re=1500. The full analysis of this case does not present much interest in a 2D problem as it does not take into account 3D effects, and no values are available in the literature to perform a comparison. However, before a 3D investigation of the ogive-cylinder, it is always interesting to first perform a 2D study. First one can make sure that Von Karman Vortex Street is observed, secondly it gives some meshing detail to be applied in a 3D case and a glimpse of the number of cells that will be required. It also allowed to test several configuration, first some turbulence models were used, then a fully DNS was computed and finally a boundary layer meshing was tested. The best configuration is displayed in Fig.7.

The ogive-cylinder has an aspect ratio of 10 with a conical part of 1.5D and a cylindrical part of 8.5D. The total number of mesh was 60 898. The choice to use a DNS is then justified; in fact if one uses Eq. (2.2.3), the required number would be 58094 cells. The smallest scale using relation (2.2.3) is O(10^{-2}) and the smallest scale in the grid is 0.011, which is the same order of magnitude.

![Fig.6 Strouhal number for the 2D cylinder at Re=300](image)

The plane was divided into three regions as it can be seen in Fig.7, in order to concentrate the meshing in the rear part, which is the region of interest, where the fixed separation takes place. The transition between the conical and cylindrical part also displays a concentrated meshing. In Fluent, settings A were applied. The density was set to 1.5, the viscosity to 0.001, and the upstream velocity to 1, in order to have Re=1500. Time step was set to 0.05, which leads to a maximum Courant Number of 4.54. With this configuration the Von Karman vortex street is clearly visible in Fig.8.
Fig. 7 2D ogive-cylinder mesh.

Fig. 8 2D ogive-cylinder pressure field flow visualization
3.2 Three-dimensional cylinder cases

Once the 2D cases investigated, the 3D cylinder wake was analyzed for Re=200 and Re=300.

The meshing required at this stage must both bring out the Von Karman vortex street and the 3D instability that develops at this range of Re. To fulfill previous requirement, a sensitivity study was conducted to get the optimal configuration. In that sense it was proved that a too coarse near cylinder mesh did not allow the capture neither of the mode B instability at Re=300, neither of the mode A instability at Re=200, because of an inheriting high numerical viscosity. Even though it is hard to evaluate, it is well-known that it slightly “lowers” the effective Re encountered by the flow. This explain why mode A can be observed at Re=300 and no instability at all can be observed at Re=200.

The validated grid details are presented in Table 2, Fig.9 and 10. An interval count of 200 was set in the spanwise direction, for a spanwise length of 12D. The cylinder is composed of 36 arcs, of 10° each, which were swept into the $-z$ direction and which represents the future control method location. Each arc was meshed with an interval count of 3 resulting in 108 nodes around the cylinder. The final grid is composed of 4 799 200 elements, which of whom maximum volume is 0.012 and maximum equisize skew is 0.3. The latter is a measure of the skewness of the tetrahedral elements, and must be kept lower than 0.6.

The mesh is displayed in Fig.11. As one can see the mesh in the wake at a distance of 3.5D from the rear cylinder remains fine.

Fig.9 3D cylinder geometric configuration. Plane x,y.
<table>
<thead>
<tr>
<th>Edge</th>
<th>Interval Count</th>
<th>Ratio</th>
<th>Size function (Growth rate)</th>
</tr>
</thead>
<tbody>
<tr>
<td>FG&amp;CD</td>
<td>35(each)</td>
<td>1</td>
<td>1.09</td>
</tr>
<tr>
<td>GH&amp;DE</td>
<td>18</td>
<td>1</td>
<td>1.09</td>
</tr>
<tr>
<td>GD</td>
<td>60</td>
<td>1</td>
<td>None</td>
</tr>
<tr>
<td>HE &amp; FBC</td>
<td>45</td>
<td>1</td>
<td>None-1.065</td>
</tr>
<tr>
<td>EK&amp;HN</td>
<td>55</td>
<td>1</td>
<td>1.09</td>
</tr>
<tr>
<td>IL</td>
<td>15</td>
<td>1</td>
<td>None</td>
</tr>
<tr>
<td>NK</td>
<td>25</td>
<td>1</td>
<td>None</td>
</tr>
<tr>
<td>IJ&amp;LM</td>
<td>6</td>
<td>1</td>
<td>None</td>
</tr>
<tr>
<td>JK&amp;MN</td>
<td>20</td>
<td>1</td>
<td>None</td>
</tr>
</tbody>
</table>

Table 2 3D cylinder geometric meshing details.

Fig. 10 3D cylinder geometric details

Fig. 11 Meshing detail, (up) Meshing volume visualized from the back of the wake at 3.5D and (down) Meshing volume visualized from the right of the cylinder.
3.2.1 Investigation at Re=300

In Fluent, settings A were applied resulting in a maximum Courant Number of 5. After 2500 iterations, i.e. 125 s. of flow time, the mode B instability was clearly visible.

Figure 12 shows the instantaneous x-vorticity field in which the flow associated with a fully developed mode-B instability can be observed. Indeed one can count 12 vortexes in the spanwise direction, resulting in a spanwise wavelength of 1D, main characteristic of the mode B instability (Williamson (1992)). Another interesting point here is that this instability is not found in the far downstream locations and is restricted to the very near wake, which is consistent with the observation made by Henderson (1995).

The mean value of $C_d$, plotted in Figure 13, was found to be equal to 1.342, in good accordance with Henderson (1995) who found 1.36 and Persillon & Braza (1998) who found 1.366. The Strouhal number was found to be equal to 0.202 in good accordance with Persillon (1998) who found 0.206. The separation point, based on the wall shear stress, was found to be located between 104° and 106° depending on the spanwise location.

The 3D behavior of the flow is also clearly visible in Fig.14. The drag coefficient for the 2D and 3D cases were plotted together and display different results. The amplitude of the oscillations and the mean drag coefficient is decreased in the 3D case. It is well-known that a general tendency of the two-dimensional computations is to over predict the drag coefficient. This already suggests that an increase of the flow three-dimensionality, following the use of control method, can be helpful for drag reduction.
Fig. 13 Time-history of the drag coefficient over the 3D cylinder for Re=300

Fig. 14 Time history of the drag coefficient over the 2D and 3D cylinder for Re=300
Fig. 15 Time histories of the drag coefficient due to the in-phase forcing at Re=300: –, unforced; - - - , $\lambda_z = 2d$; - - - - - , $\lambda_z = 4d$.
(Figure from Physics of fluids 17 Kim & Choi (2005))

Fig. 13 is to be compared with data from Kim & Choi (2005) in Fig. 15, as it can be seen, results are very similar.

A better representation of the x-vorticity using the $N_k$ method (§2.5), using Tecplot, is displayed in Fig. 16. One can compare this picture to experimental flow visualizations conducted by Williamson. In both cases one can observe an ordered fine-scale three dimensional structure. One can also observe in Fig. 16 the symmetry of the mode B instability. In fact if one observes two consecutive vortex filaments, they would appear in the same color, i.e. they comprises an in-phase arrangement. The latter point was already proved by Williamson as a main characteristic of the mode B instability compared to the mode A instability.
Fig. 16 Spanwise undulation of the main vortex rows and streamwise vortices for Re=300; iso-contours at values $N_k = 1.5$, Width=8D. In red, positive $x$-vorticity, in blue, negative $x$-vorticity.

Fig. 17 Mode B instability associated with the formation of finer-scale streamwise vortex pairs. (Picture from Journal of Fluid mechanics, Volume 328, Williamson)

The mode B was thus successfully observed. The mesh used here can thus be validated. However before applying control methods on the cylinder, the mode A instability was also investigated.
3.2.2 Investigation at Re=220

The second three-dimensional case investigated was the wake of the cylinder to detect the mode A instability. The same grid used previously for the investigation at Re=300 was used, as it has provided successful results to detect the mode B instability.

At the beginning of the investigation, the Strouhal number and the drag coefficient remains constant at 0.19 and 1.39 respectively (Fig.18). Meanwhile the streamwise vortices begin to form and a streamwise wavelength can be observed. These small-scale streamwise vortices display a spanwise wavelength of 4D, corresponding to the mode A instability (Fig.20). One can compare this picture to experimental flow visualizations conducted by Williamson (1996). In both cases one can observe vortex loops between the primary vortices (Fig.21). After 70 seconds there is a sudden drop in the drag coefficient and the oscillations amplitude of the lift coefficient are lowered by 20% (Fig.19). This is associated to the development of two-sided dislocations in the near-wake (Fig.20). The inception of dislocations is also clearly visible when a probe is placed downstream in the wake (Fig.22). The mean Strouhal number is now equal to 0.173 and sometimes reaches 0.13. This sudden drop does not last long and after about 30 seconds, the wake display a more regular pattern and the drag and lift coefficient take a more regular form. However after 40 seconds, there is a new drop in the drag coefficient and in the lift coefficient oscillation amplitude. At the same time, one can observe the inception of two-sided vortex dislocation in the near-wake. Finally after 30 seconds the wake displays a more regular pattern.

Fig. 18 $C_d$ at Re=220; left $C_d$ before dislocation; right $C_d$ during the inception of dislocations.
Fig. 19 $C_l$ at $Re=220$; The wake at time (a) and (b) is displayed in next figure.
Fig. 20 Wake at Re=220; (i): The wake at time (a) in Fig.19, iso-contours at values $N_k=1.4$ Length=15D and flow is from left to right. The inception of dislocations completely disorganizes the wake; (j): The wake at time (b) in Fig.19, iso-contours at values $N_k=1.4$ Width=10D. Pure mode A associated with the inception of streamwise vortex loops. The spanwise wavelength is here equal to 3.9D.

Fig. 21 Experimental investigation at Re=200 associated with mode A instability. (Picture from Journal of Fluid mechanics, Volume 328, Williamson)
Fig. 22 Data provided by a probe placed at x/D=3 and y/D=0.5. The frequency irregularity at 195s is indicative of dislocations.

This duality in the wake while the mode A instability takes place has already been observed in the past. Williamson (1996) presented two modes inside the mode A instability; a first mode where the flow mode follows a transition corresponding to mode A instability small-scale structure, without dislocations. Then when dislocations develop at some of the vortex loop ‘sites’ of mode A, the flow reverts to a state A* which is in fact a mix of both mode A structures and dislocations. The first state is stable whereas the second one is unstable or transient. There are intermittent periods where the small-scale structures are predominant corresponding to a higher frequency and periods where the dislocations are predominant. The inception of dislocations leads to a growing size of the vortex formation region. The base suction is decreased and the vortex formation frequency is decreased. The three-dimensionality associated with the random inception of dislocations decreases the two-dimensional stresses and thus the drag coefficient. This is why the inception of dislocations is associated with a drop in the drag coefficient as suggested earlier. The fact that increasing the three dimensionality of the wake decreases many aerodynamic coefficients is a main point while considering control method and this point will be discussed and treated later on when control method will be applied. Mittal & Behara (2009) made the same observations during their numerical investigation. They referred to a “pure mode-A instability” as a first stage when the wake is associated with periodic streamwise vortices. The next phase corresponds to the inception of dislocations that ‘contaminate’ the wake and destroy the spanwise periodicity of the vorticity.

Table 3 summarizes all the values obtained for the moment and compares them to other values found in the literature.

<table>
<thead>
<tr>
<th>Edge</th>
<th>Re</th>
<th>St, C_d (from this project)</th>
<th>St, C_d (from Persillon(1998))</th>
<th>St, C_d (from Williamson (1996))</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D case</td>
<td>200</td>
<td>0.202, 1.38</td>
<td>0.198, 1.32</td>
<td></td>
</tr>
<tr>
<td></td>
<td>300</td>
<td>0.215, 1.43</td>
<td>0.209, 1.40</td>
<td></td>
</tr>
<tr>
<td>3D case</td>
<td>200</td>
<td>0.183, 1.31</td>
<td>0.181, 1.30</td>
<td>0.18, 1.31</td>
</tr>
<tr>
<td></td>
<td>300</td>
<td>0.202, 1.33</td>
<td>0.206, 1.36</td>
<td>0.21, 1.36</td>
</tr>
</tbody>
</table>

Table 3 Global parameters of the flow and comparison with other values.
3.3 Three-dimensional cylinder cases with 2D control method applied

Once the previous mesh validated, it was of particular interest to meet this project aim and apply different type of control method. Before testing 3D forcing on the cylinder, it was important to first test the effect of blowing and suction slots located at 90°/-90° and 110°/-110°.

The forcing was applied on the whole length of the cylinder, i.e. on 12D, and each slot had an angular width of 10° centered on the indicated location. The fluid was either sucked or blew in phase and parallel to the normal at the indicated location (not normal to the surface), at a speed of 0.1U continuously along the slot, where U represents the upstream velocity. These settings are based on studies led by Kim & Choi (2005).

Even though these tests are not expected to result on a significant decrease of the drag coefficient, these cases play a major role. In fact they serve as references to the synthetic jets investigation which will be conducted latter on and they can also help to understand how the suction and blowing of the fluid interacts with the incoming fluid, whether or not it modifies the separation point, the pressure distribution, the cylinder’s wake and the mode B instability that usually develops behind the cylinder at Re=300. It was also important to ensure that the tools used to set the blowing and suction area on the cylinder works as expected and that the use of user defined function, udf, gives reasonable results in terms of what can be found in the literature. Finally it was useful to demonstrate the intuitive idea that the effect of synthetic jets does not only consist on the effects of the blowing and suction part linearly added one to another but that it was more complex.

In the next chapters the normal case refers to the investigation of the cylinder at Re=300 without any control method applied with the settings A in Fluent.

3.3.1 Blowing at 90°

The first case investigated, as a reference case, was the blowing at the top and bottom surfaces of the circular cylinder at velocity of 0.1U.

In fluent settings A were applied. For the slot at 90°/-90°, the boundary conditions were changed to “Velocity-inlet”, the velocity specification method to “Components” where the Y-velocity was set to 0.1 for the slot at 90° and -0.1 for the slot at -90°.

The mean value of the drag coefficient (Fig. 23) was found to be equal to 1.41, approximately 0.09 higher than the no control case but very close to the two-dimensional simulation, i.e. only approximately 0.01 lower. The Strouhal number is 0.193.

The rise of the drag coefficient compared to the case without control method applied was expected. In fact, it is legitimate to expect the separation point to move even though the power input of the control method, i.e. 0.1U, is not excessively important. An analyze of the average, both in time and space, separation point reveals that this point is moved upstream and is now located at 103.5° where as in the normal case it was located at 106°, i.e. 3.5° ahead. By forcing the slot placed at the top and bottom of the cylinder to blow continuously, the flow is forced to deflect from its natural path line and take a more vertical trajectory. The flow behind the slot where the forcing is applied is thus
encountered to move away from the cylinder. Furthermore, because of the blowing applied in-phase at 90°/-90°, the two shear-layers slightly separates from the upper and lower cylinder surfaces are pushed away from each other.

From this study it results that the 2D control method applied upstream from the detachment point leads to a rise in the drag coefficient. The new drag coefficient is approximately the same as the 2D simulation of the cylinder at the same Re, and the structure of the wake is less three-dimensional, confirming that the flow becomes more two dimensional.

![Graph of drag coefficient](image)

**Fig.23** Drag coefficient with the blowing applied at -90/90°. The red curve (blowing at 90°) leads the blue curve.

### 3.3.2 Blowing at 110°

After investigating the forcing at 90°, it was interesting to investigate another two-dimensional control method case to compare the results obtained. The choice to apply an in-phase blowing at the slots located at 110°/-110° was highly motivated by the results that one can find in the literature. In fact, the most significant results are often obtained when control methods are applied at the separation location. For the cylinder at Re=300, the average detachment point along the span was found to be at 106°, which belongs to the slot located at 110°. The two slots at 110°/-110° were thus forced to blow in phase at 0.1U.

In fluent, for the slot at 110°/-110°, the boundary conditions were changed to “Velocity-inlet”, the velocity specification method to “Components” where the X-velocity was set to 0.0342 and the Y-velocity to 0.094 for the slot at 110°, 0.034 and -0.094 respectively for the slot at -110°.

For this case, the mean drag value slightly increases, i.e. 0.015 higher than the normal case (Fig.24), and the mode B instability was still clearly visible.

It can be concluded that the 2D blowing with the previous settings at 110° does not give any significant change to the aerodynamics of the cylinder. Is this result really surprising? It is well-known
that one of the main drawbacks of 2D forcing is that they require relatively high power input for an effective drag reduction. Leu & Ho (2000) proved that to suppress vortex shedding the level of input velocity must be approximately of 0.46U. In the framework of this project, and more generally in an industrial contest, such device would consume too much power and is not interesting to implement in real on-board situations. This is why the numerical studies were investigated with relatively low power input which was proved to be efficient while 3D forcing was applied. The previous results are thus not surprising and confirm that to have an efficient control method with low power input, able to control the absolute instability behind a cylinder, one must apply 3D forcing and thus make the flow “more” three-dimensional. Kim & Choi (2008) quoted: “Conversely, the high effectiveness of 3D forcing such as distributed forcing or tab attracts our attention [...] 3D forcing should be applied to flow over any 2D bluff body, which contains 2D Karman vortex shedding, for drag reduction at various Reynolds numbers”.

In the case investigated here, the forcing at 110° is seen by the flow as an obstacle that it must overcome. As a consequence the pressure distribution along the cylinder is altered as it can be shown in Fig.25. In result to the blowing the flow has a lower velocity before the slot where the forcing is applied but, as it overcomes the blowing, its velocity increases and the pressure just behind the slot is decreased. Furthermore the momentum brought to the fluid in the recirculation region, through the forcing, also helps to increase the velocity in that area. The slight decrease in the base pressure observed in Fig.25 is sufficient to explain why the drag coefficient is slightly increased.

![Fig.24 Drag coefficient with the blowing applied at -110/110°.](image)
Before testing 3D forcing on the cylinder, a last 2D forcing was investigated.

### 3.3.3 Suction at 110°

In Fluent, for the slot at 110°/-110°, the boundary conditions were changed to “Velocity-inlet”, the velocity specification method to “Components” where the X-velocity was set to -0.034 and the Y-velocity to -0.094 for the slot at 110°, -0.0342 and 0.094 respectively for the slot at -110°.

The mean drag coefficient, after stabilization, was found to be equal to 1.21, i.e. the amount of drag reduction is about 7.5% (Fig.26). The same discussion conducted in previous chapter can be made here. The suction at 110° results in an acceleration of the flow near the cylinder before it reaches the slot at 110° and a reduction of the pressure. As it overcomes this slot, its velocity is decreased and its momentum brought to another component seems to enter in count. As suction is applied at 110°/-110°, the two shear layers separating from the upper and lower cylinder surfaces are pulled down toward the centerline and the distance between them decrease as it can be observed in Fig.27. The two shear layers are lowered by 4.6 cm on each side. As a consequence the eddies have a shorter period and this explains why the Strouhal number slightly increases. Also from Fig.26, it seems that while suction is applied at 110°, the vortex strength is lowered by the forcing. The slight increase in the base pressure combined to the weakening of the vortex strength in the near wake explains why the $C_d$ is lowered.
Numerical investigation

The drag coefficient with the suction applied at -110/110° is shown in Fig. 26. Fig. 27 illustrates the width of the wake measured at x/D=1 and averaged over 10 periods.

Table 4 summarizes the results obtained while investigating the 2D forcing. Further 2D forcing will not be investigated as it is not the aim of this project and previous investigations served as reference cases and clearly indicated that 3D forcing and synthetic jets are not the linear addition of blowing and suction sequences added one to another, but that their consequences on the flow are more complex. Another point is that the suction at 110° is the only case that gave a decrease in the $C_d$. This decrease is not due to separation delay but to a modification of the wake field. Indeed the vortex strength is weakened and the shear layers are pulled down.

However in all the case tested here, the fundamental nature of the wake, i.e. the mode B instability is not broken.
### Table 4 2D forcing results summarized

<table>
<thead>
<tr>
<th>Case</th>
<th>$C_d$</th>
<th>Strouhal number</th>
<th>Separation point</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normal</td>
<td>1.33</td>
<td>0.202</td>
<td>106</td>
</tr>
<tr>
<td>Blowing at 90°</td>
<td>1.41</td>
<td>0.190</td>
<td>103.5</td>
</tr>
<tr>
<td>Blowing at 110°</td>
<td>1.33</td>
<td>0.206</td>
<td>105</td>
</tr>
<tr>
<td>Suction at 110°</td>
<td>1.19</td>
<td>0.211</td>
<td>105</td>
</tr>
</tbody>
</table>

3.4 **Three-dimensional cylinder cases with 3D control method applied**

3.4.1 **Blowing/suction at 90°**

This case was the first one where 3D forcing was applied. The distributed forcing consisted of two blowing/suction slots placed at the top and bottom areas of the cylinder (Fig.28). These slots were forced in phase and had a sinusoidal profile in the spanwise direction. The angular width was of 10°. The spanwise wavelength was set to 4D and the forcing amplitude to 0.1U. The previous settings are based on optimal configuration that can be found in the literature (see bibliographic review). Finally the forcing profile (1st configuration) for the upper and lower slot is given as:

$$
\varphi_1(z) = \varphi_2(z) = 0.1 \sin \left( 2\pi \frac{z}{4} \right)
$$

(2.6.1)

Fig.28 Schematic diagram of the forcing; up: side view; down: front view of the in-phase forcing.

In Fluent, settings A were applied. Only the maximum number of iteration per time step was set to 30 to have a convergence of at least $10^{-6}$ concerning the continuity residual.

Contrarily to the 2D forcing, for which a long physical (and thus computational) time was needed to reach convergence, i.e. about 120 physical seconds, in the case investigated here the aerodynamic coefficient converged rather rapidly. As it can be seen in Figure 29, after two oscillations, the drag coefficient is significantly decreased, about 19% of drag reduction. The lift oscillations amplitude is also decreased by about 81% in average and at 67s it is reduced by 98%. However the small
Numerical investigation of oscillations in the drag and lift coefficient shows that the vortex shedding is not totally suppressed, even though its effect on the aerodynamic coefficient is significantly lowered. The Strouhal number associated with those small amplitude oscillations is 0.203. Also one can observe the presence of a small frequency, probably due to presence of intermittent disturbance.

To have other points of comparison and better understand the vortex dynamic behind the cylinder, it was decided to test two other cases. First a square forcing profile (2nd configuration) was tested:

$$\varphi_1(x) = \varphi_2(x) = 0.1 \text{ if } 0.1 \sin \left( \frac{2\pi x}{4} \right) > 0$$

$$\varphi_1(x) = \varphi_2(x) = -0.1 \text{ if } 0.1 \sin \left( \frac{2\pi x}{4} \right) < 0$$

This profile presents two main differences with the previous one; first it injects and pumps up twice more momentum in the blowing and suction part respectively and secondly the difference between the suction and blowing part is now abrupt.

![Graph](image)

Fig. 29 Aerodynamic coefficient at Re=300 when 3D forcing is applied; up: lift coefficient; down: drag coefficient.
The second profile (3rd configuration) tested was:

\[ \varphi_1(x) = \varphi_2(x) = 0.1\sin \left(2\pi \frac{x}{3.6}\right) \]  

(2.6.4)

3.6D corresponds to the exact wavelength of the mode A instability, measured in this project.

By applying a wavelength of 4D, the drag coefficient obtained still presented low amplitudes oscillations (Fig.29), whereas Kim & Choi (2005), in their numerical investigation, found the complete suppression of oscillations (Fig.15) while applying the same settings. Only in their case the mode A instability had a spanwise wavelength of exactly 4D. Thus by applying a forcing with a wavelength of 3.6D, one would be able to determine whether it is the exact mode A instability wavelength or the wavelength at 4D that gives the better results.

Figure 30 presents the drag coefficient for the three 3D forcing cases investigated. The square signal input with a wavelength of 4D (red curve) leads to a \( C_d \) that converges more rapidly. The low-frequency that was present in the first configuration is not longer visible and the amplitude of the oscillations is smaller. At \( t=79s. \) the oscillations seem to be totally cancelled for about 5s., before taking place again. The Strouhal number associated with these little oscillations is 2.1. The sinus signal input with a wavelength of 3.6D (green curve) leads to a \( C_d \) that converges without oscillations. The curve pattern is approximately the same than the first configuration curve especially after \( t=86s. \), both of these curves converges toward a \( C_d=1.09 \). The difference is that the green curve presents no oscillation at all.

![Drag coefficient comparison at Re=300 for three different signal input.](image)

**Fig.30** Drag coefficient comparison at Re=300 for three different signal input.

Figure 31 displays the instantaneous z-vorticity on different X-Y plane at different time instant for the normal case and two 3D forcing configuration; the sinus input at \( \lambda_z = 3.6D \) and at \( \lambda_z = 4D \). From these figure it is clear that the forcing retards the initial vortex formation. In the normal case one can observe the strong Karman vortices rolling up in the near wake location. In the actuated case, the vortex formation takes place approximately 1D downstream this location. The effect of the Karman vortex rollers on the rear cylinder is thus significantly decreased which explains the drag coefficient behavior.
Another interesting point is that the initial vortex formation site does not change between the different 3D forcing cases and thus cannot explain the difference between the three cases tested. The reason lies behind the vortex formation process. Indeed Fig. 31(b) and extensive visualizations show that, in the case where $\lambda_2 = 3.6D$, the formation process is similar to a synchronized shedding process. Two opposite sign vortices are shed from the top and bottom of the cylinder, almost in a synchronized manner. Conversely, for $\lambda_2 = 4D$, the formation consists of alternative shedding. In both cases the wake does not display a single row of alternative opposite signs vortices, as in the normal case, but a double row of alternative same sign vortices (see dashed line in Fig 31(b)).

Figure 32 (a) displays the instantaneous vortical structure of a part of the wake. The effect of the in-phase forcing is manifest from this visualization. Not only is the mode B instability broken but the Karman vortex rollers are broken.

Just downstream the cylinder the Karman vortex rollers undergo sinusoidal deformation. At the maximum blowing sites the vortex rollers are bent in the upstream direction and at the suction sites the vortex rollers are bent in the downstream direction. This can be explained as the flow is accelerated at the suction sites and decelerated and the blowing sites. This is confirmed by Fig. 31 (b) which shows the spanwise variation of the separation angle due to the forcing. At the suction sites the separation angle is delayed and at the blowing sites it is advanced. In fact the suction applied pull down the high-momentum fluid toward the wall and re-energizes the boundary layer whereas the blowing decelerates the boundary layer flow and thus advance separation. It is also interesting to quote that the suction has a higher influence on the separation point than the blowing. The maximum separation angle is located at the maximum suction site, in the spanwise direction, and at $120.5^\circ$, i.e. $14^\circ$ downstream of the normal case separation point. The minimum separation angle is located at the maximum blowing sites, in the spanwise direction, and at $99.5^\circ$, i.e. $7^\circ$ upstream of the normal case separation point. This results remind previous investigation (§2.6.2.2), in which 2D forcing was applied. It has been proved that while suction and blowing is applied alone at 0.1U, the separation angle is more influenced in the suction case that in the blowing case. However, the drag reduction observed here does not come from the separation delay but, as it can be seen in Fig. 30 and 31 (a), from the drastic change in the wake structure.
Fig. 31 Instantaneous 2D z-vorticity field at Re=300 for the normal case, the sinusoidal input signal $\lambda_2 = 3.6D$ and at $\lambda_2 = 4D$ respectively. Vorticity magnitude comprise between $w_2 = -2$ and $w_2 = 2$. (a): 2D plane located at $z=-2$ (b): 2D plane located at $z=-8$. 
Numerical investigation

The interaction of the forcing and the vortical evolution leads to a reorganization of the vortical structures. The 3D forcing breaks down the Karman vortex roller by increasing the three-dimensionality of the flow. The resulting wake is composed of one sinusoidal Karman vortex roller just downstream the cylinder and loop-like structure behind the first vortex roller. These investigations also show that to cancel any oscillations at Re=300, one must apply a 3D forcing at the exact mode A wavelength, proper to the studied object. Indeed, these investigations showed that only the forcing at $\lambda_2 = 3.6D$ cancelled durably the oscillations, by changing the vortex formation process and location. As a consequence the wake displays a new pattern.

Other 3D forcing investigations were performed with the different configuration:
- 3D forcing applied tangentially to the surface at 90°
- 3D forcing applied normal to the surface at 110°
- 3D forcing applied tangentially to the surface at 110°

These blowing/suction profile cases led to no interesting results (Table5), as the mean $C_d$ was found to be equal to 1.28, and the shedding frequency was still clearly visible. Thus, the optimal configuration for this active control method seems to be when the slots are located at 90°.
The last control method investigated was the synthetic jet. The forcing profile used was:

\[ \phi(t) = 0.1 \sin \left( 2\pi f^+ t \right) \]  

(2.6.5)

Where the non-dimensional frequency \( f^+ = \frac{fD}{U} \).

The forcing was applied at the detachment location, i.e. at the slot located between 105 and 115°. Indeed in the literature forcing at this location seems to produce the most efficient results. Three non-dimensional frequency were investigated; 0.2, 0.5 and 6. The settings A were applied with a maximum number of iterations per time step of 10^6. For the case \( f^+ = 6 \), the time step was set to 0.01 in order to have more than 15 data per period. To calculate the momentum coefficient \( C_\mu \) (Eq. (1.3.2)) associated to the present forcing, one must first calculate the time average jet momentum per unit length during the outstroke:

\[ \bar{T}_j = \langle \frac{1}{2} \rho \varphi(t)^2 h \rangle = \frac{C_D}{\tau} \int_0^\tau \sin(2\pi f^+ t)^2 \, dt \]  

(2.6.6)

Where \( \tau = \frac{1}{2f} \) is the time of discharge during the outstroke, \( h = \frac{\pi}{36} \) the jet width and \( C = \frac{0.1^2}{2} h \).

By recalling that \( 2 \sin^2 x = 1 - \cos(2x) \), and that \( \frac{D}{U} = 1 \) in the present investigation, one has:

\[ \bar{T}_j = \frac{C_D}{2} - \frac{C_D}{\tau} \frac{1}{4\pi f^+} \sin(4\pi f^+) = \frac{C_D}{2} \]  

(2.6.7)

Finally the momentum coefficient is:

\[ C_\mu = 2 * \frac{\bar{T}_j}{\rho UD} = 2C = 8.7310^{-4} = 0.08% \]  

(2.6.8)

Figure 33 presents the drag coefficient for the three cases investigated. In the case where \( f^+ = 6 \), the general shape of the \( C_d \) curve (green) is not modified, the vortex shedding is still observable, with a period of 2.38s. However the curve presents small amplitude oscillations, at the dimensionless frequency \( f^+ = 6 \), corresponding to the signature of the applied forcing. The Strouhal number, based on the lift coefficient is not changed and is equal to 0.215. In the case where \( f^+ = 0.5 \), the drag coefficient (red curve) had a period of 2.08s. The Strouhal number associated is equal to 0.23. Finally in the case where \( f^+ = 0.2 \), the drag coefficient (blue curve) presents a different shape than the normal case. First the amplitude of the oscillations is three times higher than in the normal case. Also the signature of the applied force is more visible. Indeed, the power
spectral density of $C_d$ exhibits two peaks, one at 0.4 but also another at 0.2. However the Strouhal number based on the lift coefficient is equal to 0.219 and the wake aspect remains the same as in the normal case.

More generally in these three cases no major difference is observed compared to the normal case. The drag coefficient mean value in these three cases is the same and is equal to 1.27, i.e. approximately the same than the normal case. The mode B instability is still observed, the wake presents no major difference and no lock-on phenomenon is observed. The level of instabilities and turbulence in the near wake seem to be not high enough to allow the synthetic jets to influence the wake. Internal discussion revealed that same conclusion was drawn in 2D investigations, even with a $C_{\mu}$ of one order of magnitude higher. Same 2D investigations revealed that synthetic jets begin to play an important role in the wake from Re=1000 (Fig.34). At equal $C_{\mu}$, the 2D investigation revealed that the vortex shedding process was clearly modified by the forcing for certain values of $f^+$. In order to evaluate when synthetic jets begin to play an important role, a 3D investigation at Re=500 was conducted.

Fig.33 Drag coefficient comparison, at Re=300, between different $f^+$ input signal.

Fig.34 2D simulation of synthetic jets at Re=1000 with $f^+ = 0.5$

(Figure kindly provided by Thierry Jardin)
3.4.3 Synthetic jets at Re=500

The DNS approach at this Re with this mesh was justified as the grid is composed of 4.7 Millions elements and the number of Nodes required using Eq. (2.2.3) is 1.2 Million. However before applying synthetic jets, it was important to validate the mesh at this Re to make sure that the results are conformed to what is expected and to have a point of comparison. The calculation was first made with a steady in time option to have rapid convergence and then with the settings A.

The mean value of $C_d$ was found to be 1.19 and the Strouhal number 0.21. The structure of the wake is presented in Fig. 34. As it can be seen the 3D wake pattern becomes more dominated by stream-wise vortices, i.e. the mode B instability is more significantly observed. These results are in good concordance with the experimental investigation conducted by Scarano & Poelma (2009).

Once the mesh was validated, the forcing was applied with the same forcing profile as in Eq.(2.6.5). First the forcing input was set at 0.1U. These configurations did not lead to any great modification of the wake. However the influence of the forcing was more visible on the aerodynamic coefficient. It was then decided to apply the same settings but with a forcing input of 0.3U. The momentum coefficient is now equal to $C'_\mu = 3.92 \times 10^{-3} = 0.39\%$.

With these settings, for $f^+ = 6$, the only signature of the forcing frequency in the lift coefficient consists of small scale oscillations at the forcing frequency (Fig.36(a)). Figure 36(b) presents the power spectrum of the longitudinal velocity at $x/D=1$ and $x/D=4$. The presence of this forcing frequency in the wake is visible at $x/D=1$ (Fig.36). This represents a small difference compared to previous cases with $C'_\mu = 0.08\%$ as no influence was observed in the near wake. However at $x/D=4$, the presence of the forcing frequency is no more visible at all. Same conclusion can be drawn for $f^+ = 0.5$. These two cases did not change the structure of the wake, however the forcing influence on the drag and lift coefficient and on the near wake is increased compared to the previous cases.

Fig.35 Instantaneous vorticity field. Iso-surface at $N_k=1.2$. 

---

53
Numerical investigation

The VIP_R software characterized by the presence of numerous vortices, direct consequence of the applied forcing. Thus, in the wake between the two previous cases, a vortex tracking was used to follow the creation and behavior of the vortices.

The VIP_R software was used to track the vortices. Indeed, the wake in the actuated case is characterized by the presence of numerous vortices, direct consequence of the applied forcing. Thus, in the wake width, are present.

In order to determine what triggers the lock-on regime, i.e. what is the difference in the wake between the two previous cases, a vortex tracking was used to follow the creation and behavior of the vortices.

For $f^+ = 0.2$ the vortex shedding locked on to the forcing frequency (Fig.37). The fact that lock-on regime is observed only at the lower frequency used is in good concordance with Munska & McLaughlin (2005). Indeed, in their experimental investigation, using plasma actuators in the same configuration as in this project, they found that the frequency band for which the lock-on occurred increased with applied voltage, beginning from the lower frequency. The applied voltage in their case is equivalent to the momentum coefficient in the present case.

Figure 38 presents the drag coefficient for the un-actuated, $C_d = 0.08\%$ and $C_d = 0.39\%$ cases at Re=500 for $f^+ = 0.2$. The mean $C_d$ value was found to be equal to 1.19 and 1.17 in the two first cases while it was found to be equal to 1.26 in the third case. In fact the lock-on regime at $f^+ = 0.2$ is followed by an increase in the mean $C_d$ and in the drag amplitude oscillation. The increase is of 11\% when $C_d = 0.08\%$ and of 45\% when $C_d = 0.39\%$. This behavior was also observed at Re=300 in Fig.33. With $C_d = 0.08\%$ the increase in the drag amplitude oscillation was of 8\%. Another important point concerns the dimensions of the wake. In the un-actuated case the dimensions of the wake at x/D=2 were 0.84m in the un-actuated case and 0.91m in the forced case, i.e. an increase of 7 cm in the wake width. The latter is consistent with the observed decrease of the Strouhal number. From these observations it seems that the lock-on regime at $f^+ = 0.2$, is characterized by an increase in $C_d$ mean value and amplitude oscillations and an increase in the dimensions of the wake. The case in which $C_d = 0.08\%$ and $f^+ = 0.2$ can be associated to a transient regime, in which the lock-on is not observed but the characteristic points, such as the increase in drag amplitude oscillations and wake width, are present. In order to determine what triggers the lock-on regime, i.e. what is the difference in the wake between the two previous cases, a vortex tracking was used to follow the creation and behavior of the vortices.

The VIP_R software was used to track the vortices. Indeed, the wake in the actuated case is characterized by the presence of numerous vortices, direct consequence of the applied forcing. Thus, in the wake width, are present.

Fig.36 Data at Re=500 with $C_d = 0.39\%$ for $f^+ = 6$. (a) Lift coefficient (b) FFT of the longitudinal velocity measured at y/D=0.5. The red curve (x/D=1) presents a clear pick at f=6.
to be able to understand what they role are in the establishment of the lock-on regime, a software which clearly separates different vortices and gives their level of circulation was required.

Fig.37 Strouhal number, based on $C_f$, for different forcing frequency at $Re=500$ and $C_\mu = 0.39\%$

Fig.38 Drag coefficient at $Re=500$ for $f^+ = 0.2$ and different $C_\mu$

As it is well-known, during the blowing stroke, a counter-rotating vortex pair is formed at the edges of the orifice. In the right side of the orifice, the created vortex due to the blowing has the same sign than the natural primary forming vortex. For instance on the top of the cylinder, the natural Von-
Karman forming vortex and the created vortex at the right edge of the orifice are both clockwise vortices, and oppositely on the bottom of the cylinder.

Figure 39 presents the wake evolution over one shedding period. As it can be seen the natural vortex shedding process is substantially altered. Normally in the time interval presented here the rolling-up shear layer from the top of the cylinder is drawn down (black arrow in Fig.39 (b)) across the wake, when it becomes strong enough to draw the other shear layer from the bottom of the cylinder across the wake, it is shed downstream, as in a normal Von-Karman vortex formation process. However in Fig.40 this is not the case. The large scale vortex initiated from the top of the cylinder meets another one initiated from the bottom of the cylinder (Fig.39 (e) (f)). This observation is the result of the additional vorticity brought to the fluid through the orifices of the synthetic jets, as described before. Indeed, the created vortices at the edge of the orifices interact directly with the separated shear layer and supplement the vorticity shed (black arrow in Fig.39 (f)). This explains why the bottom shear layer gains strength and is drawn across the wake more rapidly than in the un-actuated case. At this time instant large-scale vortices are formed on both side of the cylinder at the same instant, similarly to synchronized shedding.

This phenomenon is also visible from the drag coefficient time history (Fig.40). Indeed, at time instant (e) and (f), the wake of the cylinder consists of synchronized shedding, as described previously (Fig.39 (e) and (f)) and the level of circulation is just slightly increased during these two time instant. The pressure fluctuation on the aft of the cylinder is thus almost constant, which explains why the top of the drag coefficient does not have a sinus shape as in the un-actuated case (Fig.38) but a more rounded shape. This more rounded shape is the direct consequence of the synchronized shedding.

At this stage the resulting wake consist of two cohabiting vortices. As it could be expected the top vortex has more strength that the lower one, with an average circulation of 3.35 and 2.45 respectively. As a consequence the bottom vortex is broken into two vortices, one of them is shed downstream while the other is shed upstream toward the top of the cylinder (Fig.39 (i) (j)). Finally the near wake is characterized by the presence of three main vortices, whereas in the unforced case, only two main vortices are present in the near wake (Fig.41). The presence of these three strong vortices explains both the observed increase in amplitude and mean value of the drag coefficient in the actuated case.

During the investigation conducted here, it was observed that the lock-on regime was established without drag reduction. The latter point gives new insights on the application of control methods. In fact, synthetic jets, and more generally active control method are often depicted as devices which mainly aim to reduce the pressure fluctuations induced by the Vortex Street that develops above a critical Reynolds number. However this is not always the case as mentioned previously and confirmed by other research on the area. Jukes & Choi (2009) found that, while using dielectric barrier discharge (DBD) plasma actuators to control the flow over a circular cylinder at Re=15000, the lock-on regime was observed for $f^+ < 0.6$ but that it was always accompanied by an increase in drag fluctuation and mean value. However such control methods still found many applications, particularly in the acoustic field. Indeed, if one is able to change the vortex formation frequency, even though the mean drag coefficient is not reduced, the acoustic level can be significantly altered. Thomas et al. (2008) applied four plasma actuators around a circular cylinder at Re=30000.
Fig. 39 Vortex shedding evolution over one period $T=5s.$ at $Re=500$ and for $f^+=0.3$. Time in frames (a)-(j), $t/T=-0.4$, -0.1, 0.1, 0.25, 0.4, 0.52, 0.59, 0.67, 0.74, 0.8.
Fig.40 Zoom of the drag coefficient history at Re=500 with $f^+ = 0.2$ and $C_{\mu} = 0.39\%$; c,d,e and f corresponds to the time instant in which Fig.39 (c), (d), (e) and (f) were taken. The black dashed line shows the flat top of the drag coefficient.

He described the experience as follow: “The application is to lower the acoustic levels originating from the main landing gear of commercial aircraft.” Thus, it is stressed out that active control method displays a variety of application, and depending on the configuration in which it is forced, the resulting wake can present different advantages.

Fig.41 Wake structure at Re=500; (left) unforced case, (right) forced case.
3.4 Three-dimensional ogive-cylinder case

The second goal of this project was to create a valid grid concerning the ogive-cylinder. Indeed, in the framework of the AIA project, numerical investigation of the effect of active control method on the wake needs to be done, as described in the introduction. In this contest a valid mesh at Re=1500 of an ogive-cylinder is first required. While validated, investigation on slanted ogive-cylinder, which shape is closer to the Hercules C 130, will then be possible.

After a sensitive study to get the optimal configuration the final grid details are given in Fig.42,43,44 and Table 6.

As it can be seen in Fig.44, the transition between the ogive part and the cylindrical part, plus the rear part of the ogive-cylinder are meshed with more precaution, to catch all the effect that exists in these zones. A boundary layer mesh type was used over the cylindrical part, as the boundary layer thickness at separation plays an important role.

The unstructured grid is composed of approximately 7.5M cells. The simulation was performed at Re=1500, based on the body diameter, with a velocity input of 1m.s\(^{-1}\). The settings A were applied with a time step of 0.03s. Indeed, the natural shedding frequency found in the literature is around 0.17, resulting in a temporal resolution of 196 time steps per shedding cycle.
<table>
<thead>
<tr>
<th>Edge</th>
<th>Interval Count</th>
<th>1st length/last length</th>
<th>Boundary type of the surface generated after rotation</th>
</tr>
</thead>
<tbody>
<tr>
<td>AB</td>
<td>50</td>
<td>0.04/0.002</td>
<td>Wall</td>
</tr>
<tr>
<td>BC</td>
<td>90</td>
<td>0.01/0.002</td>
<td>Wall</td>
</tr>
<tr>
<td>CD</td>
<td>50</td>
<td>0.002/0.01</td>
<td>Wall</td>
</tr>
<tr>
<td>DE</td>
<td>90</td>
<td>0.01/0.5</td>
<td>In</td>
</tr>
<tr>
<td>EF</td>
<td>15</td>
<td>-/0.6</td>
<td>In</td>
</tr>
<tr>
<td>FG</td>
<td>90</td>
<td>-</td>
<td>In</td>
</tr>
<tr>
<td>GH</td>
<td>15</td>
<td>-</td>
<td>In</td>
</tr>
<tr>
<td>HI</td>
<td>12</td>
<td>-</td>
<td>In</td>
</tr>
<tr>
<td>IJ</td>
<td>19</td>
<td>-/0.6</td>
<td>In</td>
</tr>
<tr>
<td>JK</td>
<td>9</td>
<td>-</td>
<td>Velocity Inlet</td>
</tr>
<tr>
<td>LM</td>
<td>9</td>
<td>-</td>
<td>Pressure Outlet</td>
</tr>
<tr>
<td>KL</td>
<td>25</td>
<td>-</td>
<td>Wall</td>
</tr>
<tr>
<td>BG&amp;CF</td>
<td>80</td>
<td>-/0.002</td>
<td>In</td>
</tr>
<tr>
<td>EM</td>
<td>28</td>
<td>0.6/-</td>
<td>In</td>
</tr>
<tr>
<td>AI</td>
<td>25</td>
<td>0.6/0.04</td>
<td>In</td>
</tr>
<tr>
<td>LM&amp;KO</td>
<td>157</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>FN&amp;PQ</td>
<td>100</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 6 3D ogive-cylinder meshing details

Fig.43 3D view of the ogive-cylinder grid.
The drag and lift coefficient are presented in Fig.45. The lift coefficient behavior shows periodic fluctuation, but the amplitude variations indicate that the shedding is not fully periodic. The normal shedding frequency was found to be equal to 0.16 (Fig.46) as it can be found in the literature (Seidel et al. (2006)). This mode is similar to the Von Karman vortex street that develops behind a circular cylinder. However other peaks are displayed in Fig.46, around a Strouhal number of 0.04. These peaks were already observed by Seidel et al. (2006). They suggested that this was due to the slow shift of the symmetry plane of the shed vortices due to slight frequency variations of the Von Karman instability. This is confirmed by Fig.47, which displays the streamwise vorticity component at several downstream locations, and Fig.48. As it can be seen the symmetry plane is not the same and is slightly shifted between the three downstream locations.

Figure 49 gives an impression of the near wake behavior. The streamlines issued from the aft of the ogive-cylinder meet at a certain point downstream and eventually curls up. Among the multiple mode present in the wake, two are predominant; the Von Karman mode and the helical mode. Indeed the wake exhibits two counter rotating vortices which are the dominant structures. These structures present a helical mode as the resulting flow field exhibits a plane of symmetry that rotates over time in a random fashion.
Fig. 45 Aerodynamic coefficient for the ogive-cylinder at Re=1500. Left: $C_d$; Right: $C_l$.

Fig. 46 Strouhal Number at Re=1500 for the ogive-cylinder.

Fig. 47 Streamwise vorticity component at Re=1500. The 2D plans are located (from left to right) at: x/D=9, 10, and 11.
Fig. 48 Iso-surface at $N_x=1.45$ colored by the total vorticity magnitude at Re=1500. Flow from left to right between x/D=8 and x/D=15. View from –y direction.

Fig. 49 Streamlines present in the rear of the ogive-cylinder in an x-z plane. View from –y direction.
4. Conclusion

The primary goal of this project and this manuscript has been to investigate the effect of active flow control on the wake of a circular cylinder at low Re. To that end, numerical investigations were performed in Fluent, based on a DNS approach, and results were analysed through Matlab, Tecplot and VIP_R. The active flow control tested consisted of 2D and 3D forcing. These forcing brought altogether new insights on the active control field. First the vortex formation process was dramatically altered at Re=300 when the exact mode A instability wavelength forcing was applied. As a consequence the drag and lift coefficient exhibits no more oscillation. Secondly synthetic jets were proved to significantly affect the aerodynamic coefficient from Re=500. At this level, the lock-on regime was observed while the synthetic jets were forced close to the natural shedding frequency. However the lock-on regime was proved to be accompanied by a significant increase in lift and drag oscillation amplitude and an alteration of the shedding process. Finally it was proved that 3D forcing effect on the flow does not consist neither on a simple superposition of a simple blowing and suction part nor on a delay of the separation point but on more subtle change on the wake leading to an alteration of the vortex formation process.

In parallel a valid grid was created and computed for the ogive-cylinder at Re=1500. Two modes are observed in the wake, the Von-Karman mode and the helical mode. The unforced-wake computation can now serve to initialize the active control methods.

All the knowledge gained from the investigation around the circular cylinder is to benefit to future investigation on active control method effect on the ogive-cylinder. Once investigated, the ogive-cylinder can then be slanted and investigated, in order to have a more similar shape form than the Hercules C-130. Thus it is hoped that the ultimate goal of the AIA project, i.e. to enhance airdrop capability on the rear Hercules C-130, would be achieved.

Finally the results provide scope for future research such as noise control. Therefore, more studies should be conducted in this area to obtain significant noise reduction. Also it was found that a clear knowledge of the unstable modes present in the wake is of prime importance while building up control method. Thus before applying control method at any ends, one should first concentrate on the most unstable mode on the wake and try to cancel them.
REFERENCES


Corke T. C., Post M.L., Orlov D. M. 2009 *Single dielectric barrier discharge plasma enhanced aerodynamics: physics, modeling and applications*, Exp Fluids 46 1:26


Henderson R.D., 1995 *Details of the drag curve near the onset of vortex shedding*, Phys. Fluids 7 (9) 2102-2104


Truesdell C., 1953 *The Kinematics of Vorticity*, Indiana University