TRANSPORT AND MIXING ENHANCEMENT IN FLUID-THERMAL MICROSYSTEMS

DOCTORAL THESIS

By:
Miguel Reyes Mata
Ingeniero Aeronáutico

Mentored by:
Ángel Velázquez López Juan Ramón Arias Pérez
Doctor Ingeniero Industrial Doctor Ingeniero Aeronáutico

2013
Acknowledgements

First and foremost, this thesis would not have been possible without the unqualified and relentless support of my advisors Professor Ángel Velázquez López and Professor Juan Ramón Arias Pérez. I am also indebted to my colleagues, specially Diego Alonso Fernández, Elliott Bache, Sergio de Lucas Bodas, Marcos Antonio Rodríguez Jiménez and Unai Iradier Gutiérrez, for their support, assistance and friendship. Finally, I would like to give thanks to my parents, Miguel and Adi, my sister Rocio, and my girlfriend Pilar for their understanding, endless patience and encouragement when it was most required.
Abstract

In this thesis, experimental research focused on passive scalar transport is performed in micro-systems with marked sense of industrial application, using innovative methods in order to obtain better performances optimizing critical design parameters or finding new utilities. Part of the results obtained in these experiments have been published into high impact factor journals belonged to the first quarter of the Journal Citation Reports (JCR).

First of all the effect of tip clearance in a micro-channel based heat sink is analyzed. Leaving a gap between channels and top cover, letting the channels communicate each other causes three-dimensional effects which improve the heat transfer between fluid and heat sink and also reducing the pressure drop caused by the fluid passing through the micro-channels which has a great impact on the total cooling pumping power needed.

It is also analyzed the enhancement produced in terms of dissipated heat in a micro-processor cooling system by improving the predominantly used fin plate with a vapour chamber based heat spreader which contains a two-phase fluid inside. It has also been developed at the same time a numerical model to optimize the new fin plate dimensions compatible with a series of design requirements in which both size and weight plays a very restrictive role.

On the other hand, fluid-dynamics phenomena that appears downstream of a bluff body in the bosom of a fluid flow with high blockage ratio has been studied. This research experimentally confirms the existence of an intermediate regime characterized by an oscillating closed recirculation bubble intermediate regime between the steady closed recirculation bubble regime and the vortex shedding regime (Karman street like regime) as a function of the incoming flow Reynolds number. A particle image velocimetry technique (PIV) has been used in order to obtain, analyze and post-process the fluid-dynamic data.

Finally and as an addition to the last point, a study on the vortex-induced vibrations (VIV) of a bluff body inside a high blockage ratio channel has been carried out taking advantage of the results obtained with the fixed square prism. The prism moves with simple harmonic motion for a Reynolds number interval and this movement becomes vibrational around its axial axis after overcoming at definite Reynolds number. Regarding the fluid, vortex shedding regime is reached at Reynolds numbers lower than the previous
critical ones. Merging both movement spectra and varying the square prism to fluid mass ratio, a map with different global states is reached. This is not only applicable as a mixing enhancement technique but as an energy harvesting method.
Resumen

En esta tesis se investiga de forma experimental el transporte pasivo de magnitudes físicas en micro-sistemas con carácter de inmediata aplicación industrial, usando métodos innovadores para mejorar la eficiencia de los mismos optimizando parámetros críticos del diseño o encontrar nuevos destinos de posible aplicación. Parte de los resultados obtenidos en estos experimentos han sido publicados en revistas con un índice de impacto tal que pertenecen al primer cuarto del JCR.

Primero de todo se ha analizado el efecto que produce en un intercambiador de calor basado en micro-canales el hecho de dejar un espacio entre canales y tapa superior para la interconexión de los mismos. Esto genera efectos tridimensionales que mejoran la extracción de calor del intercambiador y reducen la caída de presión que aparece por el transcurso del fluido a través de los micro-canales, lo que tiene un gran impacto en la potencia que ha de suministrar la bomba de refrigerante.

Se ha analizado también la mejora producida en términos de calor disipado de un micro-procesador refrigerado con un ampliamente usado plato de aletas al implementar en éste una cámara de vapor que almacena un fluido bifásico. Se ha desarrollado de forma paralela un modelo numérico para optimizar las nuevas dimensiones del plato de aletas modificado compatibles con una serie de requerimientos de diseño en el que tanto las dimensiones como el peso juegan un papel esencial.

Por otro lado, se han estudiado los fenómenos fluido-dinámicos que aparecen aguas abajo de un cuerpo romo en el seno de un fluido fluyendo por un canal con una alta relación de bloqueo. Los resultados de este estudio confirman, de forma experimental, la existencia de un régimen intermedio, caracterizado por el desarrollo de una burbuja de recirculación oscilante entre los regímenes, bien diferenciados, de burbuja de recirculación estacionaria y calle de torbellinos de Karman, como función del número de Reynolds del flujo incidente. Para la obtención, análisis y post-proceso de los datos, se ha contado con la ayuda de un sistema de Velocimetría por Imágenes de Partículas (PIV).

Finalmente y como adición a este último punto, se ha estudiado las vibraciones de un cuerpo romo producidas por el desprendimiento de torbellinos en un canal de alta relación de bloqueo con la base obtenida del estudio
anterior. El prisma se mueve con un movimiento armónico simple para un intervalo de números de Reynolds y este movimiento se transforma en vibración alrededor de su eje a partir de un cierto número de Reynolds. En relación al fluido, el régimen de desprendimiento de torbellinos se alcanza a menores números de Reynolds que en el caso de tener el cuerpo romo fijo. Uniendo estos dos registros de movimientos y variando la relación de masas entre prisma y fluido se obtiene un mapa con diferentes estados globales del sistema. Esto no solo tiene aplicación como método para promover el mezclado sino también como método para obtener energía a partir del movimiento del cuerpo en el seno del fluido.
Contents

Acknowledgements iii
Abstract v
Resumen vii

1 INTRODUCTION 1

2 LOW PRESSURE DROP MICRO-HEAT SINKS 11
  2.1 Problem description and experimental setup . . . . . . . . . . 11
  2.2 Experimental results . . . . . . . . . . . . . . . . . . . . . 18
  2.3 Conclusions . . . . . . . . . . . . . . . . . . . . . . . . . . . 25
  2.4 Tables of experimental data . . . . . . . . . . . . . . . . . . 26

3 MICRO-EVAPORATOR BASED HEAT SPREADERS 29
  3.1 Problem description and experimental setup . . . . . . . . . . 30
  3.2 Experimental results . . . . . . . . . . . . . . . . . . . . . . 35
  3.3 Theoretical model and system optimization . . . . . . . . . . 39
    3.3.1 Module 1: the heat dissipating electronic component . 40
    3.3.2 Module 2: the hot plate . . . . . . . . . . . . . . . . 40
    3.3.3 Module 3: the vapour chamber . . . . . . . . . . . . . 41
    3.3.4 Module 4: the cold plate . . . . . . . . . . . . . . . . 42
    3.3.5 Module 5: the fins . . . . . . . . . . . . . . . . . . . 42
    3.3.6 Module 6: the air . . . . . . . . . . . . . . . . . . . . 42
    3.3.7 Resolution . . . . . . . . . . . . . . . . . . . . . . . . 43
    3.3.8 System optimization . . . . . . . . . . . . . . . . . . . 44
  3.4 Conclusions . . . . . . . . . . . . . . . . . . . . . . . . . . . 45

4 LOW REYNOLDS NUMBER VORTEX STUDIES 47
  4.1 Confined 3D laminar flow . . . . . . . . . . . . . . . . . . . 47
    4.1.1 Experimental Details . . . . . . . . . . . . . . . . . . 48
    4.1.2 Results and discussion of the PIV measurements . . . 57
    4.1.3 Experimental results in the downstream region . . . . 68
    4.1.4 Conclusions . . . . . . . . . . . . . . . . . . . . . . . . 94
Chapter 1

INTRODUCTION

Micro-electromechanical systems (MEMS) refer to devices whose characteristic length is less than 1 mm but more than 1 micron and that combine electrical and/or mechanical components ([79]). The beginning, in mid-twentieth century, was merely for fun and maybe for satisfying curiosity, and I quote the famous physicist Richard Feynman’s talk given on December 29th 1959 at the annual meeting of the American Physical Society at the California Institute of Technology (Caltech): "What are the possibilities of small but moveable machines? They may or may not be useful, but they surely would be fun to make", in which he even offered a sum of money to start the race for those whose motivation was not enough.

Notwithstanding, that micro-systems are finding increased applications in a huge variety of fields, and have suffered from an explosive growth during the last two decades. Just to quantify this fact, Yole Développement has shown this rapid increase to be from less than $600 million in 2002 to $3.8 billion in 2012 and to a potential $6 billion in 2022, mainly due to the automotive and mobile phone industries which demand accelerometers, gyroscopes, microphones, pressure sensors... A rising part of MEMS are the ones involving fluid flows and are intended for heat dissipation, mixing enhancement or energy harvesting, which are highly demanded by the electronics, biological, chemical and energy industries to name a few. These micro-systems are the ones this thesis is focused on.

Particularly this dissertation comprises a study of three different micro-systems with innovative solutions to improve performances of the associate device. Two of them are dedicated to heat transfer studies and the third and last one to fluid dynamics structures.

Heat dissipation has been always in the spotlight due to the fact that it is the main power restriction to microprocessors. A passive heat exchanger component that cools a device by dissipating heat into the surrounding air or liquid is the highly extended application of a heat sink. The problems come when customers and clients demand more and more processing speed,
memory transfer... and therefore power. The present situation is that for a microprocessor not larger than a square centimeter, the dissipation device is at least 5 times the size of the microprocessor.

During the past few years, a big research effort has been devoted to the study of micro-heat sinks. The reason is that practical application of these micro devices is expected to have a significant impact in electronics area as well as many other industrial sectors; see Yoo [1], Hassan et al. [2], and Obot [3] for comprehensive reviews in this field. Concerning engineering applications, it is to be noted that engineering products are seldom designed having just one objective in mind. Most often, the industrial viability of a given product depends on whether a compromise has been reached between conflicting objectives. For example, a good technical performance does not guarantee market acceptance unless cost is competitive as well. In the field of micro-heat sinks, the main emphasis has been traditionally placed on the thermal performance of the system, although there are other issues that influence viability. One of these is the pressure drop, which affects both the power required by the pump and the weight and size of the device. These are quite relevant in, e.g., the aerospace sector, where micro-heat-sink devices are increasingly used to control temperature in on-board avionics. Decreasing weight has a multiplicative effect on reducing fuel consumption, and the increasing space limitations in both the cockpit and the avionics bay impose strong constraints to the size of the various on-board devices. Then, it could be said that thermal efficiency (namely, the total heat that must be evacuated per unit time) is a natural requirement but pressure drop is a strong design constraint to be reckoned with. For example, in modern fighter aircraft designs in which micro-heat sink devices are used to cool electronics systems (like radar) that dissipate a large amount of power, the on-board fluid management system provides a fixed flow rate of cooling fluid with a prescribed pressure drop. Therefore, it is important to minimize the local pressure drop associated to the different micro-cooling devices.

A comprehensive review of the literature dealing with heat sink optimization with regard to heat transfer and pressure drop appears in the introduction of a recent article published by Khan et al. [4]. In this introduction, the authors stress the importance of accounting with these two effects when practical engineering applications are foresighted. In particular, in the technical chapters, the authors numerically assess combined thermal resistance and pressure drop behavior when optimizing a heat sink accounting for channel aspect ratio, fin spacing ratio, heat sink material, and Knudsen number. Optimization of micro-channel heat sinks has also been addressed by Kim and Kim [5] using asymptotic solutions for velocity and temperature distributions. The authors focused on the case of high channel aspect ratio (height/width > 4), high ratio of solid to fluid thermal conductivity (>20), and low Reynolds number (<690 based on the channel hydraulic diameter). In this regime, they provided closed form correlations that relate geometry
to heat transfer and pressure drop (pumping power). It was reported that, according to the analysis, optimum thickness of the wall separating channels depends on channel height and solid and fluid thermal conductivities, but not on pumping power, fluid viscosity and micro-channel length. On the contrary, optimum channel width is a function of fluid and solid properties and pumping power. Micro-heat sink optimization has also been considered by Husain and Kim [6], who used an evolutionary algorithm for optimization purposes, and defined an objective function depending on both heat transfer and pumping power. In particular, they choose to optimize two design variables: wall thickness and channel width, and found that a clearly defined Pareto front exists. This fact suggests that, in their problem, there is a trade-off between thermal resistance and pumping power on the selected space of design parameters. Foli et al. [7] and Ryu et al. [8] followed a somewhat similar approach but, instead, they used the pumping power as a constraint in the optimization algorithms. A very detailed experimental study on the pressure drop and heat transfer in a micro-channel has been published by Qu and Mudawar [9], who considered an array of rectangular micro-channels 231 microns wide and 713 microns deep in the Reynolds number span from 139 to 1672, for two different heat fluxes: 100 and 200 W/cm2. They provided an interesting set of conclusions. Namely: a) contrary to what other articles have suggested, the conventional Navier-Stokes equations adequately predict fluid flow and heat transfer behavior inside micro-channel heat sinks; b) early laminar to turbulent transition, also reported in other papers, was not observed in the range up to Reynolds number equal to 1672; c) higher Reynolds number are beneficial for the heat transfer standpoint at the expense of a greater pressure drop; and d) the channel top wall, made up of polycarbonate plastic, can be considered as adiabatic for all practical purposes.

The pressure drop in a micro-channel depends on a number of factors. For example, Croce et al. [10] have reported a significant influence of surface roughness on pressured drop and provided correlations among the Nusselt and Reynolds numbers, the friction factor, and various geometry parameters; this study was numerical and surface roughness was modeled as set of 3-D conical shapes distributed over a smooth surface. On the other hand, Pence [11] has reported on the use of fractal-like channel networks to reduce the pumping power. In particular, this author stated that, according to her analytical study, if the remaining parameters such as wall temperature and total length of the micro-channel network are kept constant, the use of fractal-like set-ups yields a pressure drop that is of the order of 60% of the one obtained when using conventional configurations.

In this context of devising methods to reduce the pressure drop while keeping a reasonable thermal performance, a relatively simple approach consists of using the tip clearance as a control parameter. It is clear that the resulting flow bypass affects the thermal performance of the system but the overall effect (pressure drop plus heat transfer) might be favorable. This
physical effect has long been considered and related studies have been published in the specialized literature. Sparrow et al. [12], back in the seventies, presented an analytical study on the laminar heat transfer associated to shrouded thin arrays where they concluded that conventional uniform heat transfer coefficient models are not applicable to this type of configuration. An experimental study including the same type of geometry was published by Sparrow and Kaddle [13]. The authors considered air as the cooling fluid and the flow was turbulent. They reported that for clearances in the range $10 - 30\%$ of the fin height, the heat transfer coefficients were $85 - 64\%$ of those for the zero clearance case, and that the ratio of the with tip-clearance to no-clearance heat transfer coefficient was a function of only the clearance-to-fin-height ratio, independent of both the flow rate and fin height. Also, it was mentioned that the presence of clearance slowed the rate of thermal development of the flow. It is to be noted that no information about pressure drop was provided in references [12, 13]. Similar experimental correlations were provided by Wirtz et al. [14] for air cooling devices in the case when flow bypass is present. Development of a new semi-analytical model for the accurate prediction of pressure losses in configurations exhibiting bypass has been reported by Coetzer and Visser [15]. A specific study on the effect of tip clearance on the cooling performance of an array of micro-channels for a fixed pumping power bounding condition has been reported by Min et al. [16], who used a numerical model under the assumption of fully developed laminar flow. The conclusion was that for any prescribed pumping power there exists an optimum tip clearance that minimizes thermal resistance. Also, it is noteworthy to mention the work by Dogruoz et al. [17], Jeng [18], Moores et al. [19], and Rozati et el [20], who considered the effect of tip clearance in micro-heat sinks that use pin fin configurations instead of channels.

Another example of industrial application in which other parameters apart from performance take side in the design, are the thermal control systems of avionics boxes. In this context fin plates is the widely used system to dissipate heat from microprocessors. Due to the growth of power consumption, new methods have been developed to support relatively low temperature microprocessors such as heat pipes. Vapour chambers are conceptually similar to heat pipes as both use an enclosed fluid traveling from a hot to a cold spot through a phase change and returns by gravity or capillary action. Vapour chamber based heat spreaders are thermal control systems characterized by their robustness and somewhat simple design. These two aspects allow for their use in a wide spectrum of industrial applications in which reliable performance under a variety of operating conditions is more important than peak efficiency. One of the product areas that is suitable for the use of this type of heat spreaders is avionics. The reason is that one of the current design limitations of electronics equipment aboard airplanes, helicopters, et cetera, is the thermal dissipation of the components rather
than the electronics aspects themselves. Typically, avionics systems inside aircraft are placed in the so-called “avionics bay” storage area that, because of the fact that free space is a very valuable commodity in aeronautics, tends to be as small as possible. The standard avionics bay contains a series of racks in which avionics boxes are tightly packaged. Boards, motherboards, etcetera, containing all kind of electronic components are assembled inside these boxes. Normally, the cooling of the electronics components is carried out via forced convection generated by air flow supplied by the aircraft that passes through grids of holes that are manufactured in the avionics boxes. This cooling method is robust and reliable, has been used for many years, and is still the approach preferred by the aircraft manufacturers for avionics thermal control. However, its limitations are twofold: the air mass flow rate supplied by the aircraft cannot be increased indefinitely, and the method cannot deal efficiently with hot spots caused by high power components. Therefore, it is in this context where heat spreaders can play a role because of their capability to effectively transfer heat from localized high temperature areas to regions where the heat can be dissipated using standard means. Furthermore, heat spreaders are attractive for avionics applications because they are self-contained passive systems, which is important when looking towards the minimization of operational and maintenance costs.

In the field of heat spreaders for generic applications, Ming et al. [21] have recently presented an experimental and numerical investigation on a new concept of grooved vapour chamber that is able to homogenize heat more efficiently than other conventional designs. Wang et al. [22] have explored the effect of heat source size on a vapour chamber heat spreader. Another example of micro grooved heat spreader for fuel cell cooling applications has been reported by Rulliere et al. [23]. A high heat removal capacity (220W/cm²) vapour chamber heat spreader has been studied by Hsieh et al. [24]. In the introduction of their article, the authors also point out the comparatively limited number of studies dealing with flat vapour chamber heat sinks. Other four different concepts of vapour chamber heat spreaders have been studied experimentally by Koito et al. [25], Shen et al. [26], Go [27] and Murthy et al. [28]. On the CFD and theoretical modeling side of the problems, it is worth mentioning the works of Boukhanouf and Haddad [29], Chen et al. [30], Chen et al. [31] and Revellin et al. [32], although in this last article the authors also consider an additional porous media so that their concept could be labeled as “intermediate” between a vapour chamber heat spreader and a flat heat pipe. Along this line, it is also important to mention the work of Kang et al. [33], Min et al. [34] and Hwang et al. [35].

On the other hand, the problem of laminar flow around bluff bodies has been and still is the subject of a very large research activity. The reasons are twofold: a) there are many Fluid Mechanics aspects involved that cause the problem to be extremely rich, and b) there are, also, many related engineering implications that are of industrial interest. In this last regard,
bluff bodies are increasingly being considered a means to generate vorticity in internal flows at low Reynolds numbers so as to promote mixing and, for example, enhance heat transfer. The basic idea is that embedded vortices in confined flow contribute to transfer heat from the hot channel walls into the main body of fluid. This approach has the advantage of being self-sustaining so the engineering complexity is kept to a minimum, although the penalty to pay is a larger pressure drop that needs to be compensated by a larger pumping power. A generic overview of the physical aspects involved in this type of systems could be found in the works of Fiebig [36], Turki et al. [38], Sharma and Eswaran [39], Dhiman [40], Meis et al. [41] and in the references therein. As a matter of illustration, Meis et al. [41] considered the 2D confined laminar flow past a series of obstacles of different shape, aspect ratio and inclination towards the inflow direction when there is a difference of 60 K between the hotter channel flow temperature and the incoming flow temperature. In particular, they showed that the heat transfer increases as a function of the blockage ratio although at the expense of a larger pressure drop which requires larger pumping power. For the case of a circular obstacle and a blockage ratio of 1/2, the authors reported 40% increase in the Nusselt number as compared with the clean channel case. However, these conclusions were based on a 2D numerical flow solver, so variations might be expected in the case of an experimentally tested 3D geometry. Some other recent experimental studies on vortex generators, not necessarily of square prism shape, have been reported by Shi et al [42], Zhang et al. [43], Liu et al [44], Henze et al [45], and Min et al [49].

In practice, convective mixing is not decoupled from thermal effects. This is apparent when considering, for example, the strong temperature dependency that water viscosity exhibits in the range from, say, 20°C to 80°C that is typical of many thermal control applications. Nevertheless, there are aspects that are of a purely fluid mechanics nature that should be studied, in a first approximation, decoupled from the thermal aspects. One of these aspects is the influence that channel aspect and blockage ratios have on the flow topology. In the case of a square cylinder this has been pointed out, for example, by Camarri and Giannetti [50] that showed that in 2D confined flow past a square prism there is a downstream inversion of the position of the shed vortices with respect to the symmetry line as compared to the sequence that takes place right after their shedding, and this is different from what occurs in free stream conditions. In a more recent work, Camarri and Giannetti [51] extended their numerical study to the case of a 3D confined flow around a circular cylinder. Also Patil and Tiwari [52] have shown numerically the influence of the blockage ratio on the onset of the Karman street that develops downstream of a 2D square cylinder in the range of Reynolds numbers from 30 to 250. In particular, these authors studied how confinement in the cross-flow wise direction affects wake characteristics, recirculation bubble size and the onset of the wake transition.
to the Von Karman regime. Specifically, they found that: a) the onset of planar vortex shedding in terms of the critical Reynolds number is delayed as the blockage ratio increases, b) for a given blockage ratio, the Strouhal associated to the wake is slightly dependent only on the Reynolds number, and 3) the length of the steady wake recirculation bubble decreases when the blockage ratio increases. Rehimi et al. [54] have published an experimental work, based on PIV, on the effect of wall confinement on the wake formation past a circular cylinder. They considered a blockage ratio of 1/3 and a span wise channel aspect ratio of 30/1 so the flow could be considered basically 2D even though the experiment was nominally 3D. The Reynolds number was changed in the range from 30 to 277 and the authors reported that even moderate confinement significantly affected the critical Reynolds number at which transition to the Karman street takes place. In particular, they reported a critical Reynolds number of 108 as compared to the value of 47 in the unconfined case. Also, regarding the rms of the velocity, they found a stabilizing effect induced by the walls as compared with the free stream case. In addition, they found span wise instabilities similar to Modes A and B reported by Williamson [63] in the unconfined case. Another experimental study on wall effects, this time on a cantilevered square prism in an isolate wall, has been reported by Wang and Zhou [55].

On the numerical side, the number of articles dealing with this 3D problem is, as expected, much lower than those addressing its 2D counterpart. In this context Martin and Velazquez [56] have pointed out that in highly confined flow (both isothermal and non-isothermal) around a square section prism located in a square section channel with a blockage ratio of 1/2.5, the transition from a closed recirculation bubble regime to a Karman street type of vortex shedding is not abrupt as in the unconfined case. In particular, they identified an intermediate regime in which the closed recirculation bubble oscillates before entering into the next vortex shedding regime. Specifically, they identified a steady recirculation bubble for Reynolds numbers less than 110, an oscillating recirculation bubble for Reynolds numbers between 110 and 170, and a Karman street for Reynolds numbers greater than 170. This clearly differs from the 2D (or quasi 2D) unconfined case in which vortex shedding is reported to start in the range of Reynolds numbers from 50 to 60 (depending in the author). A broad idea of the R & D status in this field from the numerical point of view could be obtained from the discussions and references present in Martin and Velazquez [56], Saha et al. [57], Schafer and Turek [58].

The idea of the experiments that are going to be carried out in following chapters is to identify the different flow regimes that appear as a function of the Reynolds number and to study how they differ from unconfined and 2D cases. In particular, these regimes are: a steady recirculation bubble, an unsteady recirculation bubble and a vortex shedding regime. In this context it is important to refer to the work of Jirka [59] and Jirka and Seol
which, among many other aspects, describe three similar regimes in a problem (shallow turbulent wake flow) that is very different from the one addressed in the present article. Specifically, Jirka [59] shows experimental evidence of these three regimes and links them to three different types of generation mechanisms. In the opinion of the author of this thesis, it is remarkable that problems so different might have what appears to be a similar cascade of events leading to instability and, eventually, generating flow patterns with qualitatively similar features.

When looking at the specialized literature on flow induced vibrations, it could be observed that only a small fraction of the associated research articles is devoted to the issue of tethered bodies. Among these, most of them deal with unconfined flow past circular tethered cylinders or spheres in which buoyancy forces play a critical role in the characterization of the motion. As pointed out, for example, by Ryan et al [67] this problem and its variants are not only of scientific relevance but, also, of practical interest in situations that involve, among others, submerged pipelines, ocean spars, and tethered lighter-than-air craft. Furthermore, since implementation of tethered systems is relatively simple and leads to robust engineering designs (that is an obvious advantage) it is important to try to understand their underlying physical mechanisms so that their behavior can be predicted with reasonable accuracy.

Regarding numerical approaches to the problem being considered, Ryan et al [67] performed a 2D numerical analysis on the problem of a vertically tethered buoyant circular cylinder for a range of reduced velocities of 1 to 22, at a fixed mass ratio of 0.833 and with a tether length to cylinder diameter ratio of 5.05. Because of the flow configuration (flow velocity perpendicular to the tether at rest) the tethered cylinder oscillated around a mean layover angle from the vertical direction. This oscillation was generated by the simultaneous presence of lift, drag, and buoyancy forces. In their results, the authors reported that they found the presence of three different oscillation regimes corresponding to an in-line oscillation branch, a transverse oscillation branch, and a transition in-between the two. In a later article, Ryan et al [68] studied numerically the flow-induced vibration on a circular cylinder held free to oscillate transverse to the free stream (an idealized version of a tethered cylinder). The Reynolds number varied in the range from 30 to 200 and two different flow oscillation regimes were observed characterized by the amplitude of the oscillations. The effect of the mass ratio and the tether length was analyzed by Ryan et al [69] in a configuration similar to that of Ryan et al [67]. In particular, they found a critical mass ratio below which large amplitude oscillations are observed. Shortening the tether length caused the critical mass ratio to increase and vice versa. A detailed numerical analysis of the wake states of this very same problem has been reported by Ryan [70].

While all the previous references dealt with unconfined flow, Sanchez-
Sanz and Velazquez [71] considered the vortex induced oscillation of a square section prism placed inside a 2D channel. The prism had neither structural damping nor spring restoring force, so the body equation of motion contained inertia and aerodynamics forces only (again, an idealized tethered prism situation). The channel blockage ratio was 2.5:1 and the Reynolds number, based on the prism cross section height, varied in the range from 50 to 200. It was found in this numerical study that for each Reynolds number there is a critical mass ratio that acts as a boundary between two different flow regimes. If the actual mass ratio is larger than the critical one, the prism oscillates harmonically. If the actual mass ratio is smaller than the critical one, the prism oscillation assumes an irregular pattern that is characterized by multiple harmonics that appear to belong to a uniform (chaotic) spectrum. The transition between the two regimes as a function of the mass ratio was found to be abrupt. The effect of body shape on this type of abrupt transition between the periodic and uniform spectrum regimes has been analyzed numerically by Sanchez-Sanz and Velazquez [72].

Regarding experimental studies, Van Hout et al [73] considered the case of a tethered sphere in a closed loop water channel. The sphere at rest was suspended vertically while the flow direction was horizontal. The authors considered a range of dimensionless reduced velocities from 2.8 to 31.1 (range of Reynolds numbers from 486 to 5655) and were able to identify three different flow regimes. Carberry and Sheridan [64] considered a buoyant tethered cylinder in a configuration similar to one of the numerical cases mentioned above, Ryan [67]. In the experimental setup, the cylinder had a diameter and a span on 16.2 mm and 594 mm respectively so the flow motion could be considered as two dimensional (the aspect ratio was 36). The rigid tethers had a length of 75 mm and their motion could be considered as one dimensional. Experiments were performed for mass ratios in the range from 0.54 to 0.97, and for each run flow velocity increased from zero to 0.46 m/s that corresponds to a Reynolds number of 7390. Regarding the results, the authors were able to identify two distinct states in the cylinder oscillation around the mean layover angle, although in both cases the motion was typically periodic. For any given mass ratio in the range from 0.54 to 0.72, the amplitude of the oscillation was small below certain threshold of the incoming flow velocity and the wake was consistent with the 2D Karman shedding mode. Above the threshold, the oscillations were significantly larger and the wake was different from the typical Karman wake observed at lower values of the critical mass. When the mass ratio was larger than 0.76, there was no jump in the behavior of the oscillation amplitude and it remained small for all tested flow velocities. Wang et al [74] studied experimentally the same type of geometry but they were also able to implement a piezoelectric load cell so as to directly measure lift and drag forces. In particular, they found a rather good single fit of drag as a function of the mass ratio and the buoyancy Froude number.
Chapter 2

LOW PRESSURE DROP MICRO-HEAT SINKS

This chapter presents an experimental study of the optimization of micro-heat sink configurations when both thermal effects and pressure drop are accounted for. The interest of the latter is that the practical engineering viability of some of these systems also depends on the required pumping power. The working fluid was water and, according to typical power dissipation and system size requirements, the considered fluid regime was either laminar or transitional, and not fully developed from the hydrodynamics point of view. Five configurations were considered: a reference geometry (selected for comparison purposes) made up of square section micro-channels, and four alternative configurations that involved the presence of a variable tip clearance in the design. The performance of the different configurations was compared with regard to both cooling efficiency and pressure drop. Finally, we also provide some practical guidelines for the engineering design of these types of systems. The objective is to perform an experimental study of a series of configurations that involve arrays of micro-channels so as to infer information about what is the specific setup that provides an optimum combination of thermal performance and pumping power.

With regard to the organization of the work to be presented hereafter, the following section deals with the statement of the problem. Next, the experimental test bench and results are described. Finally, conclusions and engineering design guidelines are provided.

2.1 Problem description and experimental setup

We have studied the behavior of five different configurations, using water as the cooling fluid. The basic setup is as shown in figure 2.1, with channel height, $B$, and width, $C$, equal to 500 $\mu m$ in all cases. The platform on which the micro-channels were manufactured had an area of 15 x 15 $mm^2$. This
CHAPTER 2. LOW PRESSURE DROP MICRO-HEAT SINKS

Figure 2.1. Generic view of the model setup.

means that we had 15 parallel micro-channels whose length was 15 mm each. The ratio of micro-channel length to hydraulic diameter was 30. This ratio is low if fully developed flow is sought along most of the micro-channel length. For example, if the inlet Reynolds number (based on the channel hydraulic diameter \(D_h\)) is of the order of 1000, the ratio of the entrance length to hydraulic diameter is of the order of 60 [80], which is double the micro-channel length, meaning that the flow is not developed in our series of experiments. This can be ratified if thermal issue is considered. When both velocity and temperature depend on entrance length the combined entry region becomes dependent on Prandtl number (\(Pr\)) as

\[ Gr^{-1} = \frac{(L_e/D_h) / (Re \times Pr)}{0.05} \]

with \(Gr\) as Graetz number and \(L_e\) entrance length of fully developed flow [81]. In this case, a Reynolds number of the order of 1000 corresponds a ratio entrance length to hydraulic diameter of 290 approximately. This is a typical practical situation because industrial applications impose limits on the actual length of micro-channels. For example, when dealing with the cooling of avionics equipments placed inside Arinc-type avionics racks, the maximum allowable dimension of the micro-cooler is of the order of 10-20 mm which is usually much shorter than the entrance length.

The material of the base where micro-channels were manufactured was aluminum alloy certified for aeronautics applications while the top plate was manufactured on polycarbonate. The ratio of the thermal conductivity of aluminum alloy to polycarbonate is 850, meaning that the top plate can be considered as adiabatic. Both were micro-machined on a CNC micro-milling machine (EMCO Concept Mill 105) with the software EMCO WinNC Sinumerik 810D/840D Milling.

The details of the tested configurations are as follows:

• Configuration #1 (see figure 2.2), which was considered to be the baseline. No tip clearance was allowed and the working fluid flowed parallel along the micro-channels.

• Configurations #2, #3, and #4 (see figure 2.3). The working fluid
also moved parallel along the micro-channels but three different tip clearances were allowed: 250 $\mu m$, 500 $\mu m$, and 1000 $\mu m$, respectively. These tip clearances represented 50 %, 100 %, and 200 % respectively of the channel height (500 $\mu m$).

- Configuration #5 (see figure 2.1). The flow motion was perpendicular to the micro-channels and the tip clearance was 500 $\mu m$.

Two stagnation flow chambers (see figure 2.5) were implemented, upstream and downstream of the micro-channels, to distribute the flow. The two chambers were micro-machined on the same base as the micro-channels. The flow of water came in perpendicular to the base. We estimated, using standard correlations [81], that heat transfer associated to these stagnation chambers was of the order of 5 % of the total.

We could have designed thermally insulated stagnation chambers and, also, we could have supplied the cooling flow as being uniform and parallel to the micro-channels. However, we opted for a configuration that is as close as possible to practical industrial applications. What happens is that in these practical applications, because of cost and available space, it is not always possible to use an inlet chamber that fully guarantees flow uniformity. Also, since the flow supply hoses should not be placed close to the base because of the need to pass vibration tests for certification purposes, we preferred to implement a perpendicular supply system (see figure 2.6 below). The inlet and outlet hoses were located at approximately the center of the stagnation chamber roof and had an internal diameter of 3 mm.
Figure 2.3. Close-up view of configurations #2, #3 and #4.

Figure 2.4. Close-up view of configuration #5.
2.1. PROBLEM DESCRIPTION AND EXPERIMENTAL SETUP

Figure 2.5. Schematic (top) and actual (bottom) views of the micro-channels and stagnation chambers in configurations #1-4.

Figure 2.6. Lateral view sketch of the experimental setup.
CHAPTER 2. LOW PRESSURE DROP MICRO-HEAT SINKS

The heating system consisted of a block of aluminum alloy with an insertion of two electrical resistances. A PID control system was in place to maintain the aluminum wall temperature right below the micro-channels, at $70\degree C$. Inlet water temperature was regulated to enter the micro-heat sink at $35\degree C$. An overview of the whole block is presented in figure 2.7.

In a preliminary set of experiments, the micro-heat sink and the heating system were separate components joined by pressing them together with a constant mechanical force, using thermal grease at the contact surface. However, we encountered difficulties in guaranteeing the repeatability of the experimental series. In particular, the grease film had a tendency to deteriorate over time, probably owing to heat and surface contact pressure. We could have used thermal pads to solve the difficulty, but instead, we decided to manufacture a new set of prototypes on which the heat exchange area was micro-machined directly on top of the heating block, which solved the issue of test’s repeatability. Ensuring a good contact between elements is, nevertheless, a critical design aspect that must be dealt with in practical industrial applications. As our experience has shown, integrated systems are superior to those that need additional means to achieve a robust thermal contact. This good contact avoids temperature drops across the interface but due to surface roughness effects contact spots are interspersed with gaps that are, in most instances, air filled. That results in a global conduction coefficient that may be viewed as two parallel resistances: that due to the contact spots and that due to the gaps. Increasing the joining pressure, decreasing the roughness of the interfaces or adding an interfacial fluid are the commonly used methods to decrease the gap negative effect on heat transfer. This thermal resistance is defined by [81] as $R_t = (T_A - T_B)/q''_x$. Although theories have been developed for predicting this value, the most reliable results are those that have been obtained experimentally.

Two T-type TC-SA thermocouples were inserted right below the micro-channels at a distance of $1.5\ mm$ under the surface. Two additional thermocouples were used to measure inlet and outlet flow temperature. We requested to the thermocouples supplier that they all belong to the same manufacturing series so that the measurements errors ($\pm 0.5\ C$) were all based in the same direction. Pressure drop was measured by using two pressure sensors (Ellison GS4101) that were located in the inlet and outlet hoses right outside the stagnation chambers. A flow-meter (GEMS FT110) was placed on the outlet hose. The total mass flux was maintained at a fixed value in each run by the pump itself. The current in the electrical resistance was controlled in such a way that the temperature of the metal base was maintained at the prescribed value. Heat losses from the base were calculated as the heat evacuated by the coolant liquid, which in turn resulted from the total mass flux and the inlet and outlet temperatures, taking into account heat losses in the stagnation chambers (estimated as indicated above). The data acquisition system was a Keithley KUSB 3108. It features a variety of ana-
log input/output channels, including a Cold Junction Compensation (CJC) channel, as well as digital input/output channels. The CJC channel provides $1E-2\ V/C$ with an accuracy of $1\ C$. The whole setup was controlled by using a Proportional-Integral-Derivative (PID) algorithm as a control system, whereby inlet fluid temperature and wall temperature were held at 35 and 70 $C$ respectively. All of this programmed in LabVIEW software (short for Laboratory Virtual Instrumentation Engineering Workbench) which is a system design platform and development environment for a visual programming language from National Instruments.

The PID controller compares the set-point ($SP$) to the process variable ($PV$) to obtain the error ($e = SP - PV$), then calculates the controller action:

$$CA(t) = K_c(e + \frac{1}{\tau_i} \int_0^t e dt + \tau_d \frac{de}{dt}),$$  \hspace{1cm} (2.1)

where $K_c$ is the controller gain, $\tau_i$ is the integral time and $\tau_d$ the derivative time. Proportional action is the controller gain times the error. Integral action is calculated via trapezoidal integration and is used to avoid sharp changes when there is a sudden change in $PV$ or $SP$. It is proportional to both the magnitude of the error and the duration of the error; this term
Figure 2.8. Schematics of the experimental setup.

accelerates the movement of the process towards set-point and eliminates the residual steady-state error that occurs with a pure proportional controller. However, since the integral term responds to accumulated errors from the past, it can cause the present value to overshoot the set-point value. The derivative term calculates the slope of the error over time and helps in the responsiveness of the controlled system, but on the other hand differentiation of a signal amplifies noise and thus this term in the controller is sensitive to noise in the error signal. The exit of PID controllers were attached to a relay to activate or deactivate the cooler fan power supply in the case of inlet water temperature controller, and the electrical resistances power supply in the case of wall temperature controller.

A schematics of the experimental test bench is presented in figure 2.8.

2.2 Experimental results

The average results for the baseline configuration #1 (no tip clearance) are given in table 2.1. Six different volume flow rates $G$ (liters per minute) were considered in the range from 0.16 to 1.00 l/min. Each of these volume flow rates had an associated Reynolds number, $Re$, based on the average inlet velocity and hydraulic diameter of the micro-channels. $Q$ is the evacuated heat per unit time, $Q_S$ is the ratio of $Q$ to the platform area of the micro-heat sink (15x15 mm$^2$), $\Delta P$ is the pressure drop, and $P_P$ is the pumping power (volume flow rate times $\Delta P$).

Thus, we have basically four laminar ($Re$ 416 to 1300) and two transitional ($Re$ 1959 and 2600) flows. Critical Reynolds number is found exper-
2.2. EXPERIMENTAL RESULTS

Table 2.1. Results of the baseline configuration #1 with no tip clearance.

<table>
<thead>
<tr>
<th>Configuration 1</th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>G(l/min)</td>
<td>Re</td>
<td>Q(W)</td>
<td>Q_s(W/cm²)</td>
<td>ΔP(Pa)</td>
<td>P_P(W)</td>
<td></td>
</tr>
<tr>
<td>0.16</td>
<td>416</td>
<td>117</td>
<td>52</td>
<td>4907</td>
<td>0.013</td>
<td></td>
</tr>
<tr>
<td>0.25</td>
<td>650</td>
<td>146</td>
<td>65</td>
<td>7251</td>
<td>0.030</td>
<td></td>
</tr>
<tr>
<td>0.33</td>
<td>858</td>
<td>171</td>
<td>76</td>
<td>10456</td>
<td>0.058</td>
<td></td>
</tr>
<tr>
<td>0.50</td>
<td>1300</td>
<td>221</td>
<td>98</td>
<td>16589</td>
<td>0.138</td>
<td></td>
</tr>
<tr>
<td>0.75</td>
<td>1959</td>
<td>259</td>
<td>115</td>
<td>29916</td>
<td>0.373</td>
<td></td>
</tr>
<tr>
<td>1.00</td>
<td>2600</td>
<td>272</td>
<td>121</td>
<td>55296</td>
<td>0.921</td>
<td></td>
</tr>
</tbody>
</table>

Table 2.2. Comparison between the results obtained for configurations #2 to #5 and those of configuration #1.

<table>
<thead>
<tr>
<th>Conf. 2</th>
<th>Conf. 3</th>
<th>Conf. 4</th>
<th>Conf. 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>G(l/min)</td>
<td>Q'</td>
<td>∆P' = P'_P</td>
<td>Q'</td>
</tr>
<tr>
<td>0.16</td>
<td>0.76</td>
<td>0.29</td>
<td>0.83</td>
</tr>
<tr>
<td>0.25</td>
<td>0.77</td>
<td>0.32</td>
<td>0.81</td>
</tr>
<tr>
<td>0.33</td>
<td>0.81</td>
<td>0.31</td>
<td>0.83</td>
</tr>
<tr>
<td>0.50</td>
<td>0.80</td>
<td>0.39</td>
<td>0.82</td>
</tr>
<tr>
<td>0.75</td>
<td>0.81</td>
<td>0.53</td>
<td>0.84</td>
</tr>
<tr>
<td>1.00</td>
<td>0.85</td>
<td>0.50</td>
<td>0.88</td>
</tr>
</tbody>
</table>

experimentally to be approximately 2200 (see [77] thereafter transition to turbulence starts to take place. When characteristic length of micro-channels is smaller than 1 mm, this classical theory is not valid evidenced by experiments. The transition to turbulent flow occurred at Re about 1500 to 2000 or even lower for smaller micro-channels [78].

The evacuated heat is in the range from 52 to 121 W/cm², which is representative of the foreseen evolution of industrial power electronics dissipation for the next decade or so. The chosen platform area fits the surface of the average heat dissipating electronic component in avionics applications. Examples of current high-end line of microprocessor are an Intel Core i7-960 at 3.2 GHz with a power consumption of 130 W or an AMD Phenom II X4 925 at 2.8 GHz with 95 W of thermal power dissipation, so actual micro-processors are inside the range of thermal power dissipation evaluated in the experiment.

Configurations #2-5 are considered in table 2.2, where the average results are provided as referred to the results of configuration #1 (the baseline). In other words, Q', ΔP', and P'_P are the ratios of evacuated heat, pressure drop, and pumping power in these configurations to their counterparts in the baseline.

The following remarks about the results presented in table 2.2 are in order:
• If the volume flow rate is kept constant, implementation of tip clearance always yields less heat transfer than in the baseline configuration, as appreciated in figure 2.9. Depending on the volume flow rate and tip clearance height, the ensuing heat transfer ranges from 65% to 85% of the baseline heat transfer.

• For a given volume flow rate, the heat transfer does not behave monotonically as the tip clearance is increased in configurations #2-4. Instead, it starts increasing, reaches a peak at some optimum tip clearance height, and goes down again. We think that this effect is associated to the generation of 3-D structures that enhance convective heat transfer and loom over the top of the channel separation walls while simultaneously developing downstream. If the tip clearance is very small, the growth of these 3-D structures is either inhibited or delayed by the close presence of the top horizontal wall. If the tip clearance is very large, the effect of the top wall is small and the flow attains a quasi 2-D character again with small 3-D effects over the walls that separate the channels. Then, it appears that there should be some optimum tip clearance height that maximizes the 3-D effects responsible for promoting heat transfer. This subject, which is still open, is the objective of some of Thermo-fluid Dynamics Systems and Micro-systems research group current study activity.

• If the volume flow rate is kept constant, the tip clearance always leads to a smaller pressure drop and, therefore, to a smaller pumping power. This is detected in figure 2.10. Depending on the volume flow rate and tip clearance, the ensuing pumping power is in the range of 10-53% of the baseline pumping power.

• For a given volume flow rate, the pumping power always decreases monotonically as a function of the tip clearance.

• The use of micro-channels perpendicular to the flow direction (configuration #5) could also be an acceptable compromise between heat transfer and pressure drop. However, for the same tip clearance height, the results are consistently better when the micro-channels are aligned to the flow (configuration #3).

• When looking for a reasonable design point for an actual thermal control system, it is important to account for the fact that neither the heat transfer nor the pressure drop behave linearly as a function of the volume flow rate. Even in the case of the baseline configuration, the fact that a significant portion of the flow is non-developed (owing to the short channel length) means that the laminar Nusselt number is not constant (the flow is not thermal fully developed), which is appreciated in figures 2.9 and 2.10.
When considering the combined behavior of heat transfer and pressure drop, the most favorable results are obtained with the lower volume flow rates. For example, configuration #3 (tip clearance height equal to 500 µm for a channel height of 500 µm) with the lowest volume flow rate of 0.16 l/min (equivalent to $Re = 416$ in the baseline configuration) yields a heat transfer that is 83% of the baseline configuration while the pumping power has been reduced to only 18% of the baseline (it has been divided by a factor of 5.5). In this situation, the heat transfer is 43 W/cm$^2$, which still represents a large improvement over conventional plates of fins used for thermal control purposes of avionics equipment. In these conventional systems, the typical heat transfer rate at the component level is in the range from 2 to 5 W/cm$^2$. When considering the same configuration at a higher volume flow rate (0.50 l/min, equivalent to $Re = 1300$ in the baseline configuration), the heat transfer is 83% of the baseline configuration while the pumping power has been divided by a factor of 3.3. In this case, the actual heat transfer is 81 W/cm$^2$. The benefit of the tip clearance is not so clear at the largest flow rate, as distinguished in figure 2.11, where the heat transfer rate is plotted vs. the required pumping power. This is due to the flow topology, which is quasi-two dimensional at the upper part of the channel. A better performance of configurations with a tip clearance at large flow rates would require enhancing three-dimensional convection at the upper region of the channel, which is the object of current research.

In any event, configuration #3 outperforms the baseline configuration for thermal efficiencies smaller than 90 W/cm$^2$. Figure 2.11 illustrates well the main advantage of the tip clearance, and the fact that configuration #3 seems to be the best choice. For instance, for a thermal efficiency of 65 W/cm$^2$, the pumping powers associated with configurations #1 and #3 are 3.1 and 2.0 W, respectively, which means that configuration #3 requires about 33% less pumping power than configurations #1 for the same heat evacuation target.

Regarding the repeatability of the results, we performed three experimental campaigns for each configuration and each flow rate. In each campaign we acquired thirty measurements timely equidistant. The dispersion of experimental heat transfer and pressure drop results for configurations #1 and #3 are presented in figures 2.12 and 2.13 below, where it is seen that the dispersion in the measured volume flow rate was of the order of ±5%. The actual dispersion of heat transfer results around configuration #1 was of the order of ±6%, while it was significantly smaller (±2.6%) for configuration #3 and the other configurations, not represented in figure 2.12. A closer look at the time series shows that the larger dispersion in the baseline configuration is associated with unsteady effects, which are enhanced to a larger level.
Figure 2.9. Heat transfer in configurations #1-5 vs the volume flow rate.

Figure 2.10. Heat transfer in configurations #1-5 vs the volume flow rate.
in configuration #1 because the effective Reynolds number is larger. The Fourier transformation of all time series peaks at a frequency that is slightly larger than that of the data acquisition system, which is 2.5 $Hz$. This is much smaller than the frequencies associated with relevant unsteady effects, namely the frequency of the volumetric pump, which ranges between 60 and 70 $Hz$, depending on the volume flow rate, and the inverse of the convective hydrodynamic time, $t_c^{-1} = Re\mu/(\rho CL)$ which ranges from 50 to 350 $Hz$ depending on the Reynolds number. The observed dispersion is seemingly due to hydrodynamic instabilities. This generates local eddies that increase viscous dissipation (and thus, also increase pressure drop) but mainly promote local mixing, instead of the overall transverse convection that would be necessary to increase heat transfer from the hot walls to the bulk.

At this stage, it is worth mentioning the conclusions drawn by Min et al. [16] in their numerical study of a somewhat similar problem. In particular, they found that for a fixed pumping power, thermal resistance is minimized (the optimum design point) when the ratio of tip clearance to channel width is 0.6 approximately. Our problem was different in the sense that we did not fix pumping power but measured it (it was not a constraint in our setup) and, also, because the experimental nature of our work did not allow for a nearly continuous variation of the tip clearance, as can be done in a numerical study. Nevertheless, we found that our optimum tip clearance (see table 2.2) would be somewhere inside the span of 0.5-1.0 of the ratio of tip clearance to channel width, thereby coinciding qualitatively with the results of Min et al. [16]. The case of pin tip clearance seems to be different for a number of reasons that have to do, mostly, with geometrical considerations. In this case, Rozati et al. [20] have reported that the optimum tip clearance in the low Reynolds number regime would be 0.3 times the pin diameter (somewhat

Figure 2.11. Heat transfer in configurations #1-5 vs the pumping power.
Figure 2.12. Dispersion of heat transfer experimental points for configurations #1 and #3 as a function of the volume flow rate.

Figure 2.13. As in figure 2.12 but regarding the pressure drop.
2.3 Conclusions

We have studied the effect of tip clearance on micro-channel flow based thermal control systems when, owing to engineering design restrictions, the flow itself cannot be considered as fully developed. The study has accounted for two parameters of practical interest, namely the heat transfer and the pressure drop (which is related to the pumping power). Four configurations involving a tip clearance have been analyzed and compared to a baseline configuration of micro-channel flow without tip clearance. The baseline configuration consists of a 15 parallel micro-channels of 15 mm of length and separated by a step of 1 mm. The height of the square section micro-channels was 500 µm. Tip clearances of 250 µm, 500 µm, and 1000 µm were considered. One additional configuration with the channels perpendicular to the main flow and a tip clearance of 500 µm was also studied. For each configuration, six different volume flow rates were considered. These flow rates, in the case of the baseline configuration, led to Reynolds numbers in the range from 416 to 2600, containing both laminar and transient regime flows.

The main conclusion of the work that has been presented is that the implementation of tip clearance in active micro-channel based thermal control systems is an attractive option from the practical industrial application standpoint owing to two arguments:

- The added manufacturing cost is negligible since most of the manufacturing complexity is associated to the micro-machining of the micro-channels, while the top wall can be easily set at a lower or higher height with no extra cost of maintenance.

- The deterioration in heat transfer caused by the tip clearance is small while the savings in pumping power are large. In our study, for the optimum tip clearance height, the heat transfer (at the lowest volume flow rate, $Re = 416$) was 83% of the baseline configuration. However, the required pumping power was only 18% of the baseline case. The advantage of introducing a tip clearance can also be illustrated noting that the required pumping power can almost be halved maintaining the thermal efficiency. At a larger volume flow rate ($Re = 1300$), the heat transfer behaved similarly while the pumping power was 36% of the baseline configuration.

Regarding future work, there are three related issues to be analyzed: a) the existence of an optimum tip clearance height that, seemingly, has to do with stability issues within the fluid; b) the feasibility of enhancing three-dimensional convection in the tip clearance flow region, which could be done
manufacturing some obstacles in the top wall aiming to generate 3-D flow disturbances that promote heat transfer with a limited pressure drop; c) to further understand the unsteady nature of the dispersion of results, which is higher in the baseline configuration than in those with tip clearance.

### 2.4 Tables of experimental data

In this appendix whole data obtained in the experiments is presented. Reynolds number ($Re$), Nusselt number ($Nu$) and convection heat transfer coefficient ($\bar{h}$) are also shown and has been calculated via these equations:

\[
Re = \frac{\rho f C v_{med}}{\mu f}, \quad (2.2)
\]

\[
\bar{h} = \frac{Q}{A_{wet} \Delta T_m}, \quad (2.3)
\]

\[
\Delta T_m = \frac{(T_{wall} - T_{out}) - (T_{wall} - T_{in})}{\ln \frac{T_{wall} - T_{out}}{T_{wall} - T_{in}}}, \quad (2.4)
\]

\[
Nu = \frac{\bar{h} C}{\kappa f}. \quad (2.5)
\]

In formulas 2.2 to 2.8, $v_{med}$ is the mean velocity of the fluid, $T_{wall}$, $T_{in}$ and $T_{out}$ are the wall temperature, inlet flow temperature and outlet flow temperature respectively, all of them measured by thermocouples as indicated in the experimental setup. $\rho f$, $\mu f$ and $\kappa f$ are the density, viscosity and the thermal conductivity of water respectively. Water density has been considered constant with a value of 995 kg/m$^3$ and viscosity and thermal conductivity have been treated as temperature dependent variables via these quadratic equations [81]:

\[
\mu = \mu(T_m)(1 + \mu_1 T + \mu_2 T^2), \quad (2.6)
\]

\[
\kappa = \kappa(T_m)(1 + \kappa_1 T + \kappa_2 T^2), \quad (2.7)
\]

where (in the temperature range considered), the coefficients are the followings:

\[
\mu_1 = 1.1292, \quad \mu_2 = -0.49036, \quad \kappa_1 = -0.1572, \quad \kappa_2 = 0.04704. \quad (2.8)
\]
Table 2.3. Summary of data obtained from experiments.

<table>
<thead>
<tr>
<th>$G$ ($l/m$)</th>
<th>$Re$</th>
<th>$T_{out}$ ($K$)</th>
<th>$Q$ (W)</th>
<th>$P_P$ (W)</th>
<th>$h$ (W/(m²K))</th>
<th>Nu</th>
</tr>
</thead>
<tbody>
<tr>
<td>Configuration #1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.16</td>
<td>415.9</td>
<td>318.5</td>
<td>117.0</td>
<td>4907.0</td>
<td>11775.1</td>
<td>9.2</td>
</tr>
<tr>
<td>0.25</td>
<td>649.8</td>
<td>316.4</td>
<td>146.0</td>
<td>7251.0</td>
<td>14128.9</td>
<td>11.0</td>
</tr>
<tr>
<td>0.33</td>
<td>857.7</td>
<td>315.4</td>
<td>171.0</td>
<td>10456</td>
<td>16274.2</td>
<td>12.7</td>
</tr>
<tr>
<td>0.50</td>
<td>1299.5</td>
<td>314.3</td>
<td>221.0</td>
<td>16589.0</td>
<td>20642.2</td>
<td>16.1</td>
</tr>
<tr>
<td>0.75</td>
<td>1949.3</td>
<td>313.0</td>
<td>259.0</td>
<td>29916.0</td>
<td>23642.7</td>
<td>18.5</td>
</tr>
<tr>
<td>1.00</td>
<td>2599.1</td>
<td>311.9</td>
<td>272.0</td>
<td>55296.0</td>
<td>24415.1</td>
<td>19.1</td>
</tr>
<tr>
<td>Configuration #2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.16</td>
<td>415.9</td>
<td>316.0</td>
<td>88.9</td>
<td>1423.0</td>
<td>6407.3</td>
<td>5.0</td>
</tr>
<tr>
<td>0.25</td>
<td>649.8</td>
<td>314.5</td>
<td>112.4</td>
<td>2320.3</td>
<td>7890.0</td>
<td>6.2</td>
</tr>
<tr>
<td>0.33</td>
<td>857.7</td>
<td>314.0</td>
<td>138.5</td>
<td>3241.4</td>
<td>9651.1</td>
<td>7.5</td>
</tr>
<tr>
<td>0.50</td>
<td>1299.5</td>
<td>313.1</td>
<td>176.8</td>
<td>6497.9</td>
<td>12127.7</td>
<td>9.5</td>
</tr>
<tr>
<td>0.75</td>
<td>1949.3</td>
<td>312.0</td>
<td>209.7</td>
<td>15855.5</td>
<td>14148.0</td>
<td>11.1</td>
</tr>
<tr>
<td>1.00</td>
<td>2599.1</td>
<td>311.3</td>
<td>231.2</td>
<td>27648.0</td>
<td>15422.7</td>
<td>12.0</td>
</tr>
<tr>
<td>Configuration #3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.16</td>
<td>415.9</td>
<td>316.7</td>
<td>97.1</td>
<td>883.3</td>
<td>7090.2</td>
<td>5.5</td>
</tr>
<tr>
<td>0.25</td>
<td>649.8</td>
<td>314.8</td>
<td>118.2</td>
<td>1740.2</td>
<td>8347.4</td>
<td>6.5</td>
</tr>
<tr>
<td>0.33</td>
<td>857.7</td>
<td>314.2</td>
<td>141.9</td>
<td>3136.8</td>
<td>9914.0</td>
<td>7.7</td>
</tr>
<tr>
<td>0.50</td>
<td>1299.5</td>
<td>313.2</td>
<td>181.2</td>
<td>5972.0</td>
<td>12456.6</td>
<td>9.7</td>
</tr>
<tr>
<td>0.75</td>
<td>1949.3</td>
<td>312.2</td>
<td>217.5</td>
<td>12564.7</td>
<td>14706.6</td>
<td>11.5</td>
</tr>
<tr>
<td>1.00</td>
<td>2599.1</td>
<td>311.4</td>
<td>239.3</td>
<td>22118.4</td>
<td>15996.1</td>
<td>12.5</td>
</tr>
<tr>
<td>Configuration #4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.16</td>
<td>415.9</td>
<td>314.7</td>
<td>74.9</td>
<td>490.7</td>
<td>5278.9</td>
<td>4.1</td>
</tr>
<tr>
<td>0.25</td>
<td>649.8</td>
<td>313.4</td>
<td>94.9</td>
<td>1087.7</td>
<td>6549.6</td>
<td>5.1</td>
</tr>
<tr>
<td>0.33</td>
<td>857.7</td>
<td>313.1</td>
<td>118.0</td>
<td>2404.9</td>
<td>8101.1</td>
<td>6.3</td>
</tr>
<tr>
<td>0.50</td>
<td>1299.5</td>
<td>312.1</td>
<td>143.6</td>
<td>5142.6</td>
<td>9704.3</td>
<td>7.6</td>
</tr>
<tr>
<td>0.75</td>
<td>1949.3</td>
<td>311.5</td>
<td>183.9</td>
<td>11966.4</td>
<td>12305.2</td>
<td>9.6</td>
</tr>
<tr>
<td>1.00</td>
<td>2599.1</td>
<td>310.9</td>
<td>204.0</td>
<td>22118.4</td>
<td>13526.5</td>
<td>10.6</td>
</tr>
<tr>
<td>Configuration #5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.16</td>
<td>415.9</td>
<td>315.8</td>
<td>86.6</td>
<td>834.2</td>
<td>6215.7</td>
<td>4.9</td>
</tr>
<tr>
<td>0.25</td>
<td>649.8</td>
<td>314.0</td>
<td>103.7</td>
<td>1812.75</td>
<td>7214.0</td>
<td>5.6</td>
</tr>
<tr>
<td>0.33</td>
<td>857.7</td>
<td>313.2</td>
<td>119.7</td>
<td>3450.5</td>
<td>8228.5</td>
<td>6.4</td>
</tr>
<tr>
<td>0.50</td>
<td>1299.5</td>
<td>312.5</td>
<td>156.9</td>
<td>6635.6</td>
<td>10664.6</td>
<td>8.3</td>
</tr>
<tr>
<td>0.75</td>
<td>1949.3</td>
<td>311.8</td>
<td>196.8</td>
<td>13462.2</td>
<td>13222.9</td>
<td>10.3</td>
</tr>
<tr>
<td>1.00</td>
<td>2599.1</td>
<td>311.0</td>
<td>212.2</td>
<td>24883.2</td>
<td>14092.9</td>
<td>11.0</td>
</tr>
</tbody>
</table>

Table 2.4. Constant parameters in the experiments.

<table>
<thead>
<tr>
<th>$T_{in}$ (K)</th>
<th>$T_{wall}$ (K)</th>
<th>$C$ (m)</th>
<th>$L$ (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>308</td>
<td>343</td>
<td>0.0005</td>
<td>0.015</td>
</tr>
</tbody>
</table>
Chapter 3

MICRO-EVAPORATOR BASED HEAT SPREADERS

The work presented in the present chapter deals with an experimental and theoretical/numerical study of a vapour chamber heat spreader intended for avionics applications. Three configurations were considered and compared among themselves. First, a finned metallic flat plate was considered as the reference configuration. This was because it represented the conventional industrial approach. Then, a vapour chamber heat spreader was studied having the same global dimensions as the reference configuration. The issue of trying to keep the same global volume is important because, in practice, the heat spreaders/heat sinks are inserted in between two Printed Circuit Boards (PCB) inside avionics boxes. Also, a second vapour chamber heat spreader with larger volume (that needed a larger separation distance between electronics boards) was studied for comparison purposes. Pertaining to the field of avionics applications, all configurations were tested and studied in a vertical position. However, in some cases, the off-design behavior of the system was studied in a horizontal position to simulate the situation of high operational angles of attack that may appear, for example, in helicopter flight. Boiling inside the vapour chamber was enhanced by implementing a mini-evaporator area made up of an array of mini-fin-pins having the dimensions of 1 cubic millimeter.

Additionally, the experimental results were also used to calibrate a theoretical and numerical model that was developed to assist in the engineering design of this type of heat spreaders. To illustrate the method capability, an optimization process was carried out to search for the minimum weight heat spreader (that is an important design criterion for avionics equipment) that is compatible with a series of design requirements.

The chapter organization is as follows: the following chapter deals with the description of the problem and the experimental setup; then, the experimental test bench and results are presented. Next, the theoretical/numerical
3.1 Problem description and experimental setup

The problem under consideration was the comparative study of one reference metallic heat sink, labeled $HS_0$, and two vapour chamber based heat spreaders, labeled $HS_1$ and $HS_2$ respectively, under a variety of conditions typical of avionics applications. The description of the three different configurations is as follows:

The reference configuration $HS_0$ was a finned plate (60 fins) manufactured on aluminum alloy (thermal conductivity equal to 170 $W/mK$) that weighed 480 g. A plate-fin heat exchanger has been chosen due to the fact that it is widely used in many industries, including the aerospace industry for its compact size and lightweight properties; emphasizing its relatively high heat transfer surface area to volume ratio. The dimensions of the rectangular fins were 140 mm x 12 mm x 1 mm. An overall impression of the reference configuration $HS_0$ is presented in figure 3.1 a and b. The total volume occupied by the reference configuration was 405,720 mm$^3$. The spacing between fins was chosen after a standard design used in the aeronautics industry. The pitch (2.2 mm) is small but it was decided to keep it as it is, and not to optimize it, so as to have a reference case as close as possible to an actual industrial application case. As it can be observed in figure 3.1b, a square prism of 10 mm x 10 mm area was placed on the back of the heat sink. This prism was hollow and an electrical resistance was inserted into it. In this way, by means of controlling the voltage passing through the resistance, a hot spot that simulated a high thermal dissipation electronics component was generated at the back of the heat sink. Following the standard practice in avionics applications, to allow for sufficient additional space for electric connections, a high conductivity aluminum plate (35 mm x 35 mm, see figure 3.1b) was located between the hot spot and the back of the heat sink, attempting to smooth the temperature distribution that the fin plate receive from the microprocessor. This part works as a heat spreader based on conduction and so it is most often simply a plate made of copper, which has a high thermal conductivity (around 400 $W/mK$) or another material with good thermal properties such as aluminum.

The first heat spreader configuration, $HS_1$, also manufactured in aluminum alloy, weighed 650 g and occupied the same volume as $HS_0$, was made up of two halves, see figure 3.2 a and b:

The global dimensions of this configuration $HS_1$ were the same as those of $HS_0$. The number and the spacing of fins were also the same. Accordingly, to keep the same total volume, fin height was 7 mm instead of 12 mm. Hot and cold plate thicknesses, $e_{hp}$ and $e_{cp}$ were 3 mm and 2 mm respectively (see figure 3.2b); hot plate is referred to the aluminum plate that is
3.1. PROBLEM DESCRIPTION AND EXPERIMENTAL SETUP

Figure 3.1. a) Front view of configuration HS_0 (distances are measured in mm). b) Back view of the reference configuration.

Figure 3.2. a) Front view of configuration HS_1. b) Top view of configuration HS_1.
in contact with the microprocessor (actually with the heat spreader used by the microprocessor) and thanks to the aluminum thermal conductivity, the global temperature is high; the cold plate is the opposite aluminum plate, both hold a volume that will work as a vapour chamber. Figure 3.2a and b show that the assembling of the two halves of configuration $H S_1$ left a closed empty space between the plates that forms the vapour chamber. The dimensions of this empty space (the vapour chamber) were $180$ mm x $130$ mm x $3$ mm. In the lower part of the vapour chamber, and located opposite the prism carrying the electrical resistance inside (the hot spot), a mini-evaporator area was manufactured (see figure 3.3). This region had a size of $35$ mm x $35$ mm (it matched the high conductivity aluminum plate located at the opposite side of the heat spreader wall) and it contained 324 mini-fin-pins. The dimensions of the prismatic pins were $1$ mm x $1$ mm x $1$ mm, and the spacing between them was, also, $1$ mm.

The working liquid used in the vapour chamber was HFE-7100 supplied by 3M Novec. Its boiling point temperature at $1$ bar was $61$ C. The vapour chamber was filled with liquid up to two thirds of its height ($90$ mm form the bottom of the chamber). The operations manual of the 3M Novec Engineered Fluid HFE-7100 provided by the manufacturer explicitly states its compatibility with aluminum. Air was extracted mechanically to ensure that the gas present in the vapour chamber was HFE-7100 vapour only. Some of the most important properties (liquid density, thermal conductivity, liquid specific heat and vapour pressure) of the fluid are given by the manufacturer and they are shown mathematically below:

$$\rho_{\text{HFE-7100}} = 1538.3 - 2.269(T - 273.15),$$
$$\kappa_{\text{HFE-7100}} = 0.073714 - 0.00019548(T - 273.15),$$
$$c_{\text{pf HFE-7100}} = 1133 + 2(T - 273.15),$$
$$\ln p_{\text{vapour HFE-7100}} = 22.415 - \frac{3641.9}{T}. \quad (3.1)$$

The second heat spreader configuration $H S_2$ was like $H S_1$ except for the fact that fin height was $12$ mm. Accordingly, this configuration, that weighed $780$ g, filled a volume that was $1.5$ times the volume of configurations $H S_0$ and $H S_1$. The wet area of this configuration is equal to the wet area of the finned plate but with the heat distribution of the first heat spreader ($H S_1$).

As explained above, all configurations were inserted and tested between two boards inside a mock-up avionics box. A top view and a general view of the experimental set up are shown in figure 3.4a and b. In the installation of the $H S_2$, the two boards have to be separated the pertinent distance leaving a gap between heat exchanger sides and boards of $1.5$ mm.

Four T-type TC-SA thermocouples (labeled T1 to T4) were inserted at the locations shown in figure 3.5. These temperature measurements were
3.1. PROBLEM DESCRIPTION AND EXPERIMENTAL SETUP

Figure 3.3. Micro-evaporator area made up of square pins. All dimensions in mm.

Figure 3.4. a) Top view of the avionics box with the inserted heat spreader. b) Side view of the experimental mock-up avionics box.

used afterward to calibrate the theoretical/numerical model being described below. They are also useful noticing the main change in the introduction of heat spreaders between fin plate and hot component by showing how uniform these temperatures are.

As it can be observed, the locations were: T1) in the center of the mini-evaporator on the fin plate side. T2) right above the mini-evaporator on the hot plate side, T3) upper left side of the heat spreader on the fin plate side, and T4) upper left side of the heat spreader on the hot plate side. We requested to the thermocouples supplier that they all belong to the same manufacturing series so that the measurements errors ($\pm 0.5 \degree C$) were all biased in the same direction. A portable hot wire Testo 425 anemometer was used to measure airflow velocity after it had passed along the fins. The anemometer accuracy was $\pm 0.03 \ m/s$. The velocity was measured at several locations along the heat exchanger span and integrated so as to have an
Two fans, Sunon PMD2406PMB1, were used in parallel to provide the required airflow as a function of the applied voltage, each of them provides a maximum of 1.58 $m^3/min$ at a voltage of 24 V. This airflow was chosen to simulate the ARINC aeronautics regulation (ARINC600 is the predominate avionics packaging standard introducing the avionics Modular Concept Unit, MCU) and, also, the case where a higher flow rate is needed because of the high thermal dissipation of the electronic components. Relation between air volume flow and voltage applied to the fans was done with several tests. Pressure inside the vapour chamber was measured by using a pressure sensor Ellison GS4101. After comparing the measured pressure with the theoretical vapour pressure at the fluid temperature (see equation 3.1) it was possible to verify the absence of air in the vapour chamber. The data acquisition system where all of the mentioned devices sent their output was a Keithley KUSB 3108. It features a variety of analog input/output channels, including a Cold Junction Compensation (CJC) channel, as well as digital input/output channels. The CJC channel provides $1E - 2 V/C$ with an accuracy of 1 C. An electric resistance of 265 $\Omega$ was the destined device to hold a thermal power dissipation or a constant component temperature by using a PID control system, explained in the previous chapter. The experimental setup was controlled by LabVIEW software. A schematics of the experimental test bench
3.2. EXPERIMENTAL RESULTS

Figure 3.6. Sketch of the experimental setup.

is presented in figure 3.6.

3.2 Experimental results

Cases considered were forced (the normal type of operation) and natural (to account for airflow system malfunctioning) convection conditions. The thermal power dissipation of the system was measured as a function of the component (hot spot) temperature and airflow volumetric rate.

Considering forced convection, figure 3.7a, b and c show the thermal power dissipation associated to three component temperatures: 80, 90 and 100°C, for the three experimental configurations $HS_0$, $HS_1$ and $HS_2$. The tests were repeated three times for each operating point in order to point out repeatability. Ambient temperature was 25°C. The spread of the measurements has been represented in figures 3.7a, b and c using vertical bars.

It can be observed that the vapour chamber heat spreader is always more efficient than the metallic fin plate. Also, when comparing any two configurations, the absolute increase in efficiency, measured in watts, does not depend on the airflow velocity. For example, when the component temperature is 80°C, configuration $HS_1$ allows for a heat removal rate that is 6 W higher than in the reference configuration $HS_0$, no matter what the air velocity is. When the component temperature is 100°C, configuration $HS_2$ removes 25 W more than the reference configuration $HS_0$ with no influence from the air velocity. The heat removal capability of the vapour chamber heat spreader is linked to the total fin area. Configuration $HS_1$ occupies the same volume as the reference configuration $HS_0$ but its fin area is smaller (by a factor of 0.5) because of the allowance for the vapour chamber volume. Then, in this case, $HS_1$ improves $HS_0$ by 6 W only, no matter what the component
Figure 3.7. a) Dissipated thermal power as a function of the heat spreader configuration and air flow volume rate, $G$, for a component temperature of $80 \, ^\circ C$. b) As in figure 3.7a, but with component temperature equal to $90 \, ^\circ C$. c) As in figure 3.7a, but with component temperature equal to $100 \, ^\circ C$. 
3.2. EXPERIMENTAL RESULTS

The temperature is. However, the fin area of configuration $HS_2$ is the same as that of the reference configuration $HS_0$ and, in this case, the improvement can be as high as $25 \, W$ for the higher component temperature. Regarding the dispersion of the results, the average spread was $\pm 2 \, W$ that, depending on the actual value of the heat transfer rate, represents a deviation that is of the order of $\pm 2 \%$ to $\pm 3 \%$ of the average values.

In regard to natural convection, figure 3.8 shows the thermal power dissipation results obtained in the natural convection case as a function of the component temperature for the three configurations. It could be observed that, in this case, the heat spreader performance is also superior to the fin plate. Furthermore, the relative gains are larger than in the case with forced convection. In fact, the heat transfer rate gain (similar for both configurations $HS_1$ and $HS_2$) varies between $35\%$ and $25\%$ depending on the component temperature when compared to the reference configuration $HS_0$. This fact suggests that a convenient application of vapour chamber based heat spreaders is for natural convection related thermal control. Specifically, this is important for avionics applications because, even though the normal mode of operation is in forced convection conditions, safety regulations require that the system has to survive a certain time span relying on natural convection only. The reason is to account for a possible accidental failure of the airflow supply system. Regarding figure 3.8, it is worth noting that configuration $HS_1$, that occupies the same volume that the reference configuration $HS_0$, is able to dissipate from $33 \, W$ to $38 \, W$ while keeping a component temperature in the range of $90-100 \, C$ that complies with the strict regulations of many on board avionics systems as the mentioned ARINC.

Additionally, the system behavior has been tested in the case when the avionics box is not placed vertically but it has fallen down either forward or backwards, off-design operation. That is: in this situation, the heat spreader is placed on a horizontal plane, not on a vertical plane. The results obtained in natural convection conditions for configuration $HS_1$ are presented in table 3.1 where it could be observed that the system behavior is still quite robust in these circumstances. For example, if the component temperature is $100 \, C$, the system still dissipates $30 \, W$ after having fallen forward, $\alpha = 90 \, C$ with fins pointing downwards, (or backwards, $\alpha = -90 \, C$) as compared to the $38 \, W$ that dissipates in the nominal vertical position.

Finally, configuration $HS_2$ has been tested under hard temperature conditions, in order to evaluate the performance loses due to a great increase of the ambient air temperature. Malfunctioning or a fire inside the avionics box room may be the source of that trouble. Figure 3.9 shows the dissipated thermal power of the heat spreader labeled as $HS_2$ as a function of air flow volume rate $G$ (note that natural convection is also plotted as a air flow volume rate equal to zero for simplicity) with an ambient air temperature of $70 \, C$. It can be concluded that such an increase in air temperature (180 \%) will lead to a decrease in thermal dissipation of 81 \% at the maximum air
Figure 3.8. Thermal power dissipation results obtained in the natural convection case as a function of the component temperature for the three configurations.

Table 3.1. System off-design behavior: dissipated thermal power as a function of the component temperature, position, and configuration.

<table>
<thead>
<tr>
<th>$T_{component}$ (°C)</th>
<th>$\alpha$ (deg)</th>
<th>$HS_0$ (W)</th>
<th>$HS_1$ (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td>80</td>
<td>-90</td>
<td>18.7</td>
<td>19.3</td>
</tr>
<tr>
<td>80</td>
<td>0</td>
<td>22.0</td>
<td>26.6</td>
</tr>
<tr>
<td>80</td>
<td>90</td>
<td>16.3</td>
<td>21.7</td>
</tr>
<tr>
<td>90</td>
<td>-90</td>
<td>22.4</td>
<td>25.0</td>
</tr>
<tr>
<td>90</td>
<td>0</td>
<td>27.2</td>
<td>31.2</td>
</tr>
<tr>
<td>90</td>
<td>90</td>
<td>19.3</td>
<td>26.3</td>
</tr>
<tr>
<td>100</td>
<td>-90</td>
<td>24.9</td>
<td>29.5</td>
</tr>
<tr>
<td>100</td>
<td>0</td>
<td>32.5</td>
<td>36.8</td>
</tr>
<tr>
<td>100</td>
<td>90</td>
<td>23.5</td>
<td>30.3</td>
</tr>
</tbody>
</table>
3.3. THEORETICAL MODEL AND SYSTEM OPTIMIZATION

Figure 3.9. Thermal power dissipation results for configuration $HS_2$ with an ambient temperature of 70°C.

flow rate and of 69% in natural convection for a component temperature of 90°C. With the dissipating device at 100°C, the percentages are 73% and 57% at maximum air flow and in natural convection respectively.

3.3 Theoretical model and system optimization

A theoretical model of the vapour chamber based heat spreader has been developed to be used for system optimization purposes. The aim is to perform a trade-off between heat dissipation rate and total weight that is very important for avionics applications. The theoretical model that has been developed divides the system in a series of modules that simulate: 1) the heat dissipating electronic component, 2) the hot plate, 3) the vapour chamber, 4) the cold plate, 5) the fins, and 6) the airflow passing along the fins. Each module is formulated separately and connected to the other modules via the heat sources, the sink terms, and the boundary conditions. A generic overview of the model is presented in the sketch given in figure 3.10 below:
Figure 3.10. Sketch of the heat spreader theoretical model.

The equations of the model, with the spatial coordinates defined in figure 3.5, are:

### 3.3.1 Module 1: the heat dissipating electronic component

The thermal power $P$ dissipated by the electronic component goes into the hot plate via thermal conduction. A thermal gel is used at the mechanical interface that exists between both elements. This interface has a thermal contact resistance $R_{\text{cont}}$ and its area is $A_{\text{cont}}$.

$$P = A_{\text{cont}} \frac{T_{\text{comp}} - \bar{T}_{\text{hp,cont}}}{R_{\text{cont}}}.$$  

(3.2)

In equation 3.2, $T_{\text{comp}}$ is the temperature of the electronic component and $\bar{T}_{\text{hp,cont}}$ is the spatially $x$-$y$ averaged hot plate temperature in the contact region.

### 3.3.2 Module 2: the hot plate

The hot plate is made up of an aluminum alloy that has a high thermal conductivity ($170 \ W/mK$). Furthermore, this hot plate is flat and very thin (the ratio of the square of the surface area to the thickness is 54) so it is possible to neglect the temperature spatial derivatives in the $z$ direction (normal to the plate). This means that the hot plate temperature $T_{hp}$ is function of $x$ and $y$ only.

Regarding the energy balance, the hot plate receives, on the one hand, heat from the contact region that is adjacent to the electronic component. On the other hand, heat leaves the hot plate via three mechanisms: a) boiling in the finned evaporator area, b) liquid convection and c) gas convection.
that inside the vapour chamber there is boiling and the working fluid exists in both gas and liquid phases). However, in this case, estimates of the gas convection term prove that it is much smaller than the other two terms and has not been included in balance equation 3.3:

\[
e_{hp}\kappa_p \left( \frac{\partial^2 T_{hp}}{\partial x^2} + \frac{\partial^2 T_{hp}}{\partial y^2} \right) = (Q_{boiling} - P)q(x, y) + q_l,
\]

where \(e_{hp}\) is the hot plate thickness (see figure 3.2b) and \(\kappa_p\) its thermal conductivity. \(Q_{boiling}\) is the total heat rate transferred in the boiling process. \(P\), as explained above, is the thermal power dissipated by the electronic component. \(q(x, y)\) is a step-like function that is equal to zero outside the interface that connects the electronic component to the hot plate and equal to \(1/A_{cont}\) at the interface. The dimensions of \(q(x, y)\) are \(m^{-2}\). The \(x\)-\(y\) integral of the function at the interface is 1. \(q_l\) represents the heat convection term per unit area associated to the working fluid in liquid phase that is present inside the vapour chamber and it is modeled using a thermal resistance that is adjusted using the experimental results. Thus, \(q_l = (T_{hp} - T_{cp})/R_l\) in the liquid wetted area and \(q_l = 0\) otherwise (\(T_{cp}\) is the cold plate temperature that is, also, a function of \(x\) and \(y\)). Equation 3.3 is integrated in a rectangular domain of dimensions \(H \times L\) (horizontal and vertical dimensions of the plate respectively) and its thermal boundary conditions at the four edges (\(x = 0\) and \(0 \leq y \leq L\), \(x = H\) and \(0 \leq y \leq L\), \(y = 0\), \(0 \leq x \leq H\), \(y = L\), \(0 \leq x \leq H\)) are adiabatic.

### 3.3.3 Module 3: the vapour chamber

The working fluid in the vapour chamber receives heat from the hot plate (via boiling and convection) and transfers it to the cold plate (via condensation and convection). Since there is mass conservation and a steady-state is assumed to hold the position of the interface liquid-vapour remains constant and total boiling heat transfer rate at the hot plate is equal to total condensation heat transfer rate at the cold plate. Therefore:

\[
Q_{boiling} = \int \int_{\Omega} h_{cond}(T_{vap} - T_{cp}) \, dx \, dy,
\]

where \(\Omega\) refer to the wetted vapour area, \(T_{vap} = (T_{hp, late} + T_{cpp, late})/2\) and \(h_{cond}\) is the condensation heat transfer coefficient inside the vapour chamber, \(T_{vap}\) is the vapour temperature, and the averages are performed over the whole plate surface.
3.3.4 Module 4: the cold plate

The model approach used for the cold plate is the same as the one used for the hot plate (subsection 3.3.2). In particular, the cold plate receives heat from the vapour chamber via condensation and liquid convection and delivers it to the fins via conduction. Accordingly, its balance equation is written as:

$$
\varepsilon_{cp} \kappa_p \left( \frac{\partial^2 T_{cp}}{\partial x^2} + \frac{\partial^2 T_{cp}}{\partial y^2} \right) = A_f \kappa_p \left( \frac{\partial T_f}{\partial z} \right)_{z=z_f} - q_{in},
$$

where $\varepsilon_{cp}$ is the cold plate thickness (see figure 3.2b). The term $-A_f \kappa_p (\partial T_f / \partial z)$ represents the thermal power that leaves the cold plate by conduction and goes into the fins. This term is evaluated at the root of the fin ($z = z_f$). $A_f$ is the ratio of the sum of the cross fin areas to the total plate area, and $T_f$ is the fin temperature. $q_{in}$ models the heat rate per unit area that comes from the vapour chamber into the cold plate. It is made up of two terms: $(T_{hp} - T_{cp}) / R_l$ in the region wetted by the liquid, and $h_{cond} (T_{vap} - T_{cp})$ in the region in contact with the vapour. The domain and boundary conditions for equation 3.5 are the same as for equation 3.3.

3.3.5 Module 5: the fins

The model equation for the fins represents a balance between heat conduction and convection to the ambient air. The information of the cold plate temperature enters the balance equation via the boundary conditions since fin root temperature is equal to the cold plate temperature at the same position. On the other side, the fin tip and the lateral edges are assumed to be adiabatic.

$$
\varepsilon_f \kappa_p \left( \frac{\partial^2 T_f}{\partial x^2} + \frac{\partial^2 T_f}{\partial z^2} \right) = 2h_a (T_f - T_a),
$$

here, $\varepsilon_f$ is the fin thickness, $h_a$ is the convection heat transfer coefficient with the surrounding air, and $T_a$ is the air temperature (25 $^\circ$C in this study). Equation 3.6 was integrated in rectangle of dimensions $B \times L$ where $B$ is the fin height (see figure 3.1.a).

3.3.6 Module 6: the air

The external air behavior is modeled as it passes along the channels formed by the cold plate, the fins, and the back of the next electronics board locate inside the avionics box (see figure 3.2 and figure 3.3). The average air mass flow rate $\dot{m}$ related with the air volume flow rate $G$ and it is assumed to be
prescribed since it is controlled by the fans. Then, the air is heated up as it passes along the fins and the balance equation 3.7 holds:

\[ \dot{m} c_{p,a} d \frac{\partial T_a}{\partial x} = 2h_a (T_f - T_a), \]  

(3.7)

where \( c_{p,a} \) is the air specific heat and \( d \) is the distance between fins (viscous effects have been neglected). The boundary condition for equation 3.7 is that \( T_a \) is equal to the ambient air temperature before entering into the fin area \( T_{a0} \) (25 C in this study).

### 3.3.7 Resolution

Summarizing, the model contains: a) six equations 3.2, 3.3, 3.4, 3.5, 3.6 and 3.7, b) six unknowns: \( T_{\text{comp}} \), \( T_{hp}(x, y) \), \( T_{vap} \), \( T_{cp}(x, y) \), \( T_f(x, z) \), \( T_a(x, z) \), c) four parameters to be adjusted via the experiments: \( R_{\text{cont}} \), \( R_l \), \( h_{\text{cond}} \), \( h_{a} \), d) four series of operation parameters to be selected: d1) the dimensions of the hot/cold plates (\( L \) and \( H \)), d2) the number, spacing and size (\( B \)) of the fins, d3) the thermal power dissipated (\( P \)) by the electronic component, and d4) the air mass flow rate \( \dot{m} \). Regarding the four parameters to be adjusted via the experiments, the following hypothesis, Incropera and DeWitt [82], were made regarding their behavior:

\[
R_{\text{cont}} = \lambda_1, \\
Nu_{\text{liquid}} = \frac{x_l}{R_l k_a} = \lambda_2 Ra^{\lambda_3} = \lambda_2 \left( \frac{\rho_l^2 c_{p,l} g \beta (T_{\text{comp}} - T_{a0})}{\mu_1 k_l} \right)^{\lambda_3}, \\
Nu_{\text{cond}} = \frac{h_{\text{cond}} (L - x_l)}{k_a} = \lambda_4 \left( \frac{\rho_l^2 g h_f g (1 + 0.68 Ja) (L - x_l^3)}{\mu_1 k_l (T_{vap} - T_{a0})} \right)^{\lambda_4}, \\
Nu_{a} = \frac{h_a D_h}{k_a} = \lambda_6 Re^{\lambda_7} = \lambda_6 \left( \frac{\dot{m} D_h}{\mu_a} \right)^{\lambda_7},
\]

where \( \lambda_1 \) to \( \lambda_7 \) are the seven coefficients to be adjusted, and \( Ra \), \( Re \), and \( Ja \) are the Rayleigh, Reynolds and Jacob numbers. \( Ra \) and \( Re \) are based on the hydraulic diameter \( D_h \) of the channel formed by adjacent fins and \( Ja \) defined as \( c_{p,l} (T_{\text{plate}} - T_{vap}) / h_f g \). The adjustment was carried out searching for the set of parameters \( \lambda_1 \) to \( \lambda_7 \) that minimized the total deviation in a series of experimentally measured points. The results obtained were: \( \lambda_1 = 39 \times 10^{-5} \ m^2 K/W \), \( \lambda_2 = 2.0 \), \( \lambda_3 = 0.18 \), \( \lambda_4 = 1.2 \), \( \lambda_5 = 0.25 \), \( \lambda_6 = 0.27 \), \( \lambda_7 = 0.51 \), and the root mean square of the deviation was 3.5\%. \( \lambda_1 \), that represents the thermal contact resistance, has a value of \( 39 \times 10^{-5} \ m^2 K/W \). For comparison purposes, contact resistances of aluminum metallic interfaces are in the range of \( 15 \times 10^{-5} \ m^2 K/W \) to \( 50 \times 10^{-5} \ m^2 K/W \), Incropera and
DeWitt [82]. $N_{u_{\text{liquid}}}$ was of the order of 160 that also agrees with the range provided in reference [82]. $\lambda_4$ and $\lambda_5$ were 1.2 and 0.25, respectively, while the values proposed for similar cases, reference [82], are 0.943 and 0.25, respectively. $N_{u_{a}}$ was in the range from 4 to 12 while empirical correlations [82], give values from 1.5 to 9. It is, nevertheless, to be noted that, in the present case, heat fluxes and temperature fields were not spatially constant and that the geometry differed from the idealized situation considered by many semi-empirical correlations provided in the literature, so deviations are to be expected.

3.3.8 System optimization

From the engineering standpoint, a theoretical/numerical model allows for the possibility to draw performance maps and carry out an optimization process. One performance map is presented in figure 3.11 where iso-contours of the system weight and component temperature are plotted as a function of heat spreader dimensions $H$ and $L$. In this case, the prescribed parameters were: dissipated power: 100 W, air flow rate: 15 $m^3/h$, heat spreader width: 15 mm (to fit between two adjacent electronic boards), and fin thickness, separation and height 1 mm, 2.2 mm and 5 mm respectively.
### 3.4 Conclusions

An experimental and theoretical/numerical study has been carried out on the behavior of vapour chamber heat spreaders for avionics thermal control.
purposes. In these systems, thermal efficiency has to be combined with a low system weight, a forced convection normal operating condition, and a natural convection accidental operating condition. From the thermal dissipation standpoint, it was found that vapour chamber based heat spreaders are more efficient than equivalent metallic fin plates that occupy the same volume. However, from the weight point of view, metallic fin plates are lighter than their heat spreaders counterparts. At the higher air flow rate that was considered (25 $m^3/h$), the heat spreader had a thermal efficiency 10 % better than the fin plate, but it was 35 % heavier. In natural convection conditions, the thermal efficiency of the heat spreader was 30 % higher than the fin plate. Higher thermal efficiencies, in forced flow conditions, of the vapour chamber heat spreader can be achieved using higher fin height. However this comes at the unwelcome expense of widening the gap existing between adjacent electronic boards inside avionics boxes, which translates into placing fewer boards per box. For example, increasing the heat spreader volume by a factor of 50 % would increase thermal efficiency by a maximum of 15 % only. Another attractive advantage of the vapour chamber heat spreader is its robust off-design behavior. For example, when the avionics box topples 90 $C$ either forwards or backwards, performance degradation is only of the order of 15 %. Also, this degradation is smaller than in the fin plate case. The development of a theoretical/numerical model of the heat spreader, coupled to an optimization algorithm showed that it is possible to save weight by a factor of the order of 20-30 % for the same heat dissipation rate. The model also showed that the weight reduction rate does not scale linearly with the increase in component temperature (and cost).
Chapter 4

LOW REYNOLDS NUMBER VORTEX STUDIES

Two particular experiments are done in this chapter. The first one deals with the confined 3D laminar flow behavior around a fixed square section bluff body. The second one deals with the same confined 3D laminar flow behavior but, this time, the square section bluff body is not fixed, the movement restriction perpendicular to the flow has been removed. These experiments focus on hydrodynamic topology by using Particle Image Velocimetry (PIV) methods. These methods rely on the presence of particles in the flow that not only follow all flow velocity fluctuations but are also sufficient in number to provide the desired spatial resolution of the measured flow velocity.

Both sections are organized with the same structure: first problem description and experimental details are explained, then experimental measurements are shown and discussed and finally conclusions are provided.

4.1 Experimental study on the confined 3D laminar flow past a square prism with a high blockage ratio.

As a summary from the previous literature discussion shown in chapter 1, it could be said that confinement plays an important role in the flow past an obstacle in the 2D laminar regime. Blockage ratio affects both local flow topology, even leading to an inversion in the sequence of vortices, and global flow parameters such as recirculation bubble size and the onset of vortex shedding. Although these results are well established, the author of the present dissertation believes that it is worth going a step farther so as to quantify the effect of the span-wise confinement as well. The reason is that, so far, this effect has deserved less attention in the literature even though it is, nevertheless, of interest from the industrial standpoint. In fact, the
typical configuration in many industrial applications is the channel having a nearly square cross section. As expected, confinement effects in this 3D case have large influence on flow topology.

An experimental PIV study is presented that addresses the confined 3D laminar water flow behavior past a square section prism. The Reynolds number, based on prism cross-section height varies between 100 and 256. The channel aspect ratio is 1/1 and the blockage ratio is 1/2.5. This geometry is representative of a passive method to enhance mixing in otherwise highly ordered laminar channel flow. It is found that the lateral walls exert a strong effect on the flow behavior with two main consequences: a) the onset of vortex shedding is delayed to a Reynolds number in the vicinity of 170, as opposed to the unconfined case where the critical Reynolds number is reported to be between 50 and 60, and b) transition from the steady closed recirculation bubble regime to the vortex shedding regime is not abrupt. In particular, there is a range of Reynolds numbers for which the closed recirculation bubble oscillates with increasing amplitude prior to the onset of the Karman street regime. The experimental results are supported by numerical computations that have been performed with the aim of cross checking results and conclusions (they have been carried out by other group members). The relation of the results with the practical design of engineering systems is also discussed.

4.1.1 Experimental Details

Experimental set-up

The experiments were conducted in a closed-loop circuit. The working fluid was water and the main components of the circuit were: a) a primary closed tank, b) a square section channel inserted into the tank that contained the test section with the square prism, c) a flow meter and a valve, d) a secondary open tank, and e) a pump. A schematics of the circuit is shown in figure 4.1.

The flow meter was a Siemens Sitrans FM MAG1100 electromagnetic flow-meter sensor with a transmitter Sitrans FM MAG5000 and it measured in the range from 0 to 5 liters/min with an uncertainty of less than ±5%. A magnetic flow meter is a volumetric flow meter which does not have any moving parts and is ideal for waste-water applications or any dirty liquid which is conductive or water based. The operation of a magnetic flow-meter or mag meter is based upon Faraday’s Law, which states that the voltage induced across any conductor as it moves at right angles through a magnetic field is proportional to the velocity of that conductor. As it will be shown below, in the characterization of the incoming flow section, the flow rate was also measured indirectly by integrating the upstream velocity profiles obtained via PIV. The valve was used to control the volumetric flow allowing the pump to work at their optimal power supply. The pump was an ITT
4.1. CONFINED 3D LAMINAR FLOW

Figure 4.1. Schematic view of the experimental set up.

Totton centrifugal pump magnetic drive DC15/5. The dimensions of both the main tank and the test section, manufactured on methacrylate, are given in figure 4.2. The total volume of water involved in the test was $20 \pm 1$ liters. A honeycomb section was inserted at the inlet of the test channel so as to homogenize the flow as much as possible. Actual pictures of the set-up are also presented in figure 4.2.

To quantify flow uniformity promoted by the honeycomb section, PIV measurements were also performed upstream of the prism. The room temperature of the experiments, that influences water viscosity, was $19^\circ C$. Distilled water used in the experiments was left for at least three days in the room to guarantee that it homogenized its temperature. Prior to the test, it was checked that water temperature was $19^\circ C$ as well. The volume flow rate was varied in the range from 0.387 to 0.962 liters/min. When the Reynolds number is defined based on the vortex generator cross section height and average inlet velocity, this volume flow yields a Reynolds number range from 100 to 256.

PIV measurements

Particle Image Velocimetry (PIV) working principle is based on the measurement of the displacement of small tracer particles that are carried by the fluid during a short time interval. These particles are small enough to accurately follow the flow motion and not alter the fluid properties and big
enough to scatter the sufficient light to get filmed by a camera. These tracer particles are illuminated by a thin light sheet generated from a pulsed light source (usually a double-head pulsed laser system) and the light scattered by them is recorded onto two subsequent image frames by a digital imaging device (typically a CCD camera). By means of stereoscopic imaging it is possible to determine the three components of the flow velocity within the planar field defined by the light sheet. The simplified one-camera systems suffers from the restriction that the optical axis must be aligned in the direction normal to the light-sheet plane, and moreover, this system only yields two velocity components within the measurements plane. PIV method is then a non-intrusive measurement method but it requires optical access for both the delivery of the light sheet and recording of the images. A schematic view of a PIV system is shown in figure 4.3.

The PIV set used on the next experiments was a Dantec Dynamics system. Flow illumination was provided by a pulsed Nd:YAG 800 mJ laser with wavelength of 532 nm. Each laser pulse lasted for 4 ns; that time is small enough to see particles as dots not as streaks because distance traveled by the particle in exposure times of 4 ns is much smaller than the size of the particle. This kind of laser, with its very short pulse duration and with pulse energies up to almost 1 J is suitable to illuminate flows without any limit of the flow speed. The standard architecture of these lasers consists of two sep-
4.1. CONFINED 3D LAMINAR FLOW

Figure 4.3. Schematic view of PIV system.

arate lasers firing independently at the required pulse separation. However its repetition rate ranges between 10 and 50 \( Hz \) (15 \( Hz \) in this case) posing the major limitation of Nd:YAG-based systems in performing time-resolved experiments. In the case study this is not going to be a limitation due to the very low-speed flow (\( v < 0.05 \text{ m/s} \)).

Images were taken using a Dantec Dynamics Flow Sense 2M camera with a resolution of 1600 x 1200 pixels. Data transmission was made directly to the PC RAM via a National Instruments PCIe-1427 image acquisition board which allows for an information transfer larger than 200 MB per second, which is sufficient for the information budget considered in this study. The frequency of images grabbing was 15 \( Hz \) so, the camera transfer 30 images per second which is at most 110 MB per second if the bit depth is 16 bit/pixel. CPU RAM reserved for data acquisition was 4 GB, that means a 537 pair of images per test batch. At a frequency of 15 \( Hz \) gives us a 35 seconds gap of images recording. The camera lens was a Zeiss Makro-Planar T* 2/50 mm ZF. The camera was oriented perpendicular to the laser sheet and normal to the duct side wall. That means only 2 velocity components are going to be yielded. After passing side walls, it was verified that the thickness of the laser sheet was smaller than 1 mm. Synchronization between image capturing and flow illumination was carried out using a timer box provided by Dantec Dynamics through another National Instrument Board PCI-6602 timer board.

Laser pulse, image acquisition, synchronization of all components and
post-processing was carried out by Dynamic Studio software of Dantec Dynamics.

The flow was seeded with Polyamide Seeding Particles (PSP-5) having a mean diameter of 5 \( \mu m \) (size between 1 to 10 \( \mu m \)). They are non-spherical but round, with a density of 1.03 \( g/cm^3 \). The seeding procedure consists of increasing the concentration gradually from a low level until reaching the desired one. If seeding exceeds the optimum level, multiple scattering and opacity compromise the setup of the experiment. It has been calculated to have at least 10 particles per window on average (high image density) doing some samples before each test. That means that a single particle cannot be tracked (PTV) because the displacement is higher than the mean distance between adjacent particles. With this approach one needs to rely on statistics to identify the most probable match of particle images in the two frames.

This type of seeding is in line with the one used in other related studies such as the one performed by Rehimi et al (2008). In any case, the particle response time was of the order of \( 10^{-6} \) seconds, as computed using the method described by Nguyen (2002) which is well below the typical threshold characteristic time of the tests. Within the approximation of low Reynolds number, the equation of motion of a small spherical particle immersed in a fluid flow is given as:

\[
\frac{4}{3} \pi a^3 \rho_p \frac{dv_p}{dt} - \frac{4}{3} \pi a^3 \rho_f \frac{dU}{dt} = -6 \pi \mu a \left[ (v_p - U) - \frac{1}{6} a^2 \nabla^2 U \right].
\]  

(4.1)
4.1. CONFINED 3D LAMINAR FLOW

This equation is known as the Basset-Boussinesq-Oseen (BBO) equation where only quasi-steady viscous term (Stokes drag) is evaluated. Thanks to the small size of the particle tracers used in PIV, the right hand side of the equation 4.1 is the term which dominates in the original BBO equation that can be found in [76]. In this equation \( v_p \) is the particle vector velocity, \( U \) is the velocity of the surrounding fluid, \( a \) is the particles mean radius and \( \rho_p \) and \( \rho_f \) are the densities of particles and fluid respectively. In the approximation that \( \frac{dU}{dt} = \frac{dv_p}{dt} \), the difference between the particle velocity \( v_p \) and that of the surrounding fluid \( U \) can be estimated as:

\[
(v_p - U) = \frac{2a^2(\rho_p - \rho_f)}{\mu} \frac{dv_p}{dt},
\]

(4.2)

where the velocity difference \( v_p - U \) is referred to as the slip velocity. Clearly from 4.2 the choice of neutrally buoyant particles \( (\rho_p - \rho_f)/\rho_f = 0 \) leads to particle tracers that accurately follow the flow. In our case, \( (\rho_p - \rho_f)/\rho_f = 0.03 \), that means our particles are going to have a good accuracy following water flow. The characteristic response time is defined as \( \tau = \frac{2a^2(\rho_p - \rho_f)}{\mu} \), and is equal to 4.1E-8 s with these particles, which is smaller than the smallest time scale of the flow that is of the order of tenths of second and the double pulse delay time (5 ms). The ratio between these characteristic times is known as particle Stokes number \( S_k \). From a practical point of view, it can be stated that the condition \( S_k < 0.1 \) returns an acceptable flow tracing accuracy with errors below 1 %. Regarding the scattering properties of tracer particles, particle diameter is larger than the light wavelength (532 nm) so it is the Mie regime where there is no problem with scattering of light. In this regime most of the light is scattered in the forward direction and an appreciable amount of light backward direction, while at an angle of 90 degrees the amplitude of scattered light is generally very low. That is why for a single-camera PIV with the optical axis normal to the light-sheet plane use a much stronger light source than other particle-based techniques.

PIV measurements were carried out in twelve different areas: six of them located upstream and another six downstream of the prism. Upstream and downstream areas were aligned among themselves. Their large size prevented the possibility of using a single area covering the upstream, the prism, and the downstream regions simultaneously; this is why upstream and downstream interrogation areas were treated separately. A front view (looking from the “y” direction) of their location is presented in figure 4.5. Interrogation areas were placed in the following planes: \( z = -12.5 \) mm, \( z = -6.5 \) mm, \( z = -1 \) mm, \( x = 5 \) mm, \( x = -1 \) mm and \( x = -6.5 \) mm respectively. Two of these planes (see figure 3) were aligned with the main axes of the channel section, another two were located near the walls, and the remaining two were placed in the intermediate region.
A 3D view of all six upstream areas is presented in figure 4.6. However, to keep a clear representation, only two of the downstream areas are shown in figure 4.7. In what follows, all distances will be made dimensionless using the prism cross section height, 10 mm, as the characteristic length (also in the numerical simulations to be described below). In figure 4.7, the dimensionless coordinates of test points are: \( P_1 \) (0, 5, -1.25), \( P_2 \) (0, 4, -1.25) and \( P_3 \) (0, 6, -1.25). The flow information associated to these three points will be used in a forthcoming section. All areas had the same physical dimensions: 80 mm x 25 mm, and each of them contained 1600 x 500 pixels. In all cases, each experiment was repeated three times to assess the repeatability of the results.

Once the recorded images are stored, it is time to process them. This consists essentially of a cross-correlation analysis of the particle-image patterns in small sub-domains, or interrogation regions, between the first and second image frame. The particle-image displacement divided by the image magnification and the time delay between the laser light pulses yields the local fluid velocity. This process is repeated for the entire image domain which yields the instantaneous velocity in a planar cross section of the observed flow. Two additional post-processing methods are implicitly done when images are being analyzed, image restoration and image enhancement. The first one attempts to repair undesirable effects due to the imaging, the second one accentuates certain image features. Such post-processing includes low-pass and high-pass filtering, histogram equalization, min/max filtering...

Sampling of the flow field was carried out at a frequency of 15 Hz. Each sample was generated by processing the information associated to two laser pulses separated 5 milliseconds in time. Then, in the case of the highest Reynolds number (256), where the mean flow velocity was 0.024 m/s, a par-
4.1. CONFINED 3D LAMINAR FLOW

Figure 4.6. View of the six upstream PIV areas.

Figure 4.7. View of two downstream PIV areas and the location of test points P1, P2 and P3.
particle would travel at the order of 0.12 mm between consecutive pulses in the streamwise direction that is a distance much smaller (by two orders of magnitude) than the characteristic length of the problem (the prism cross section height of 10 mm). Each PIV area was divided into smaller sub-interrogation areas of 6 mm x 6 mm containing 128 pixels x 128 pixels each one that correspond to 21 pixel per millimeter. The post-processing software allowed for a parallel self-consistent re-computation of the flow field in successive interrogation windows of 64 pixels x 64 pixels (3 mm x 3 mm), 32 pixels x 32 pixels (1.5 mm x 1.5 mm) and 16 pixels x 16 pixels (0.8 mm x 0.8 mm). That is, in the end, a velocity vector was computed for each window of 0.8 mm x 0.8 mm and this computation was consistent with the computations performed at the larger windows. Furthermore, the software allowed for the possibility of performing flow computations at smaller windows of 8 pixel x 8 pixel (0.4 mm x 0.4 mm) and so on. However, these finer computations were not carried out because, in this case, the typical distance traveled by flow particles at the highest Reynolds number (0.12 mm) was of the same order of magnitude as the window size. Then, it could be stated that the final flow resolution of the experimental tests was of the order of 1 mm that is considered to be sufficient to describe the phenomena being analyzed. An overlap of 50% of the sub-interrogation windows size was also implemented improving measurement accuracy and vector map resolution.

Regarding the time resolution, it will be shown in a later section that the typical frequency of the large scale vortical structures being observed was 0.7 Hz that correspond to a characteristic time of 1.5 s. In this regard, each recording sequence consisted of 500 frames separated 0.067 s in time from each other (15 Hz) and lasting for a total of 33.5 s. Then, the characteristic time of the observed phenomena was of the order of 20 times larger than the sampling time and, also, 20 times smaller than the total recording time.

For a given particle image in the first frame, each particle image in the second frame is a possible match candidate and each pair represents a possible displacement with equal likelihood. This procedure is repeated for all particle image in the interrogation domain in the first frame, and the probabilities in the displacement peak for each of the matching pairs will soon be dominant over the displacement peaks for unrelated pairs. Two important requirements for PIV interrogation are evident: a) the displacement of the particle images within the interrogation domain should be uniform; and b) the interrogation domain should contain a considerable number of particle image pairs (that is why these tests were done with high-image density).

It is clearly concluded from figure 4.8 in the right, that the displacement of particles goes unidirectional to the positive side of one axis. The method just described is called adaptive correlation, figure 4.8 shows an example of adaptive correlation on a pair of frames.
Figure 4.8. Histogram analysis for finding the most probable displacement.

4.1.2 Results and discussion of the PIV measurements

**Laminar duct flow in the clean configuration without the prism**

First of all, the flow in the duct has been characterized without the obstacle. The volume flow rate was measured using three different approaches: one direct and two indirect. They were:

- **Measurement 1**: direct flow meter lecture.

- **Measurement 2**: Integration of the PIV results (A). Specifically, intersection of the six upstream interrogation areas shown in figure 4.6, together with the fact that the incoming flow had two planes of symmetry, yielded experimental information on the stream wise “$v$” velocity “$v$” in a grid of 25 points inside the channel section, see figure 4.10. The PIV “$v$” velocity data at these points was used to generate a fifth order least squares fitted parabolic profile of the type shown in equation 4.3. Then, this profile was integrated along the “$x$” and “$z$” coordinates and a volume flow rate was computed.

- **Measurement 3**: integration of the PIV results (B). Instead of fitting a parabolic profile, a bi-directional natural neighbor interpolation scheme, Sibson [62], was used to compute the velocity profile. Then, the profile was integrated along the “$x$” and “$z$” coordinates and a volume flow rate was computed.

\[
V(x, z) = \lambda_0 + \lambda_1 x + \lambda_2 z + \lambda_3 x^2 + \lambda_4 xz + \lambda_5 z^2 + \lambda_6 x^3 + \lambda_7 x^2 z + \\
+ \lambda_8 x z^2 + \lambda_9 z^3 + \lambda_{10} x^4 + \lambda_{11} x^3 z + \lambda_{12} x^2 z^2 + \lambda_{13} x z^3 + \lambda_{14} z^4 + \\
+ \lambda_{15} x^5 + \lambda_{16} x^4 z + \lambda_{17} x z^4 + \lambda_{18} x^3 z^2 + \lambda_{19} x^2 z^3 + \lambda_{20} z^5. \tag{4.3}
\]
Figure 4.9. Velocity map of a fluid passing an obstacle obtained via adaptive correlation. In the top right corner, a 32x32 pixel interrogation area histogram of the pair of image corresponding to one figure 4.4 that yields to one vector of the velocity field shown in this figure.
4.1. CONFINED 3D LAMINAR FLOW

Figure 4.10. Front view, looking from the “y” direction, of the 25 grid points (shown as shaded squares).

Table 4.1. Comparison of the results obtained via direct and indirect measurement of the incoming flow Reynolds number.

<table>
<thead>
<tr>
<th>Flow meter lecture</th>
<th>PIV integration parabolic fitting of 5\textsuperscript{th} order</th>
<th>Discrep.</th>
<th>PIV integration bi-directional natural neighbor interpolation</th>
<th>Discrep.</th>
</tr>
</thead>
<tbody>
<tr>
<td>137</td>
<td>135.9</td>
<td>0.8 %</td>
<td>135.9</td>
<td>0.8 %</td>
</tr>
<tr>
<td>185</td>
<td>183.6</td>
<td>0.8 %</td>
<td>183.7</td>
<td>0.7 %</td>
</tr>
<tr>
<td>205</td>
<td>204.8</td>
<td>0.2 %</td>
<td>204.9</td>
<td>0.2 %</td>
</tr>
<tr>
<td>256</td>
<td>255.0</td>
<td>0.4 %</td>
<td>255.1</td>
<td>0.4 %</td>
</tr>
</tbody>
</table>

The results obtained with the three different techniques at four representative Reynolds numbers (maximum, minimum and two intermediate ones) are shown in table 4.1.

It could be observed that discrepancies between direct and indirect measurements in the range of Reynolds numbers being considered were of the order of 0.5 %. A third order parabolic fitting was also considered but it was found to be inappropriate because the associated discrepancies were, in that case, of the order of 5 %.

In order to obtain a theoretical approximation of the incoming flow, it can be assumed an incompressible fluid, with constant viscosity (as temperature is constant during all the experiments), so fluid-dynamic problem is decoupled from thermal problem. The system to be considered is:
\[ \nabla \mathbf{V} = 0, \]
\[ \rho \frac{D \mathbf{V}}{Dt} = \rho \mathbf{f} - \nabla p + \mu \Delta \mathbf{V}. \]  
\hspace{1cm} (4.4)

Namely, four scalar equations with four unknowns: \( u, v, w, p \). Gravity effects are often negligible or may be included in \( p \) as in these experiments, being \( P = p + \rho g z \). Primarily due to the nonlinear convective inertia forces, the problem is still very intricate one both analytically and numerically. In a steady state, inertia terms are vanishing, and that means that a fluid particle is subjected to zero acceleration, i.e. it moves in pure translation with constant velocity. It follows that path-lines, coinciding here with streamlines, must be straight lines. The velocity of each particle may depend only on coordinates perpendicular to the direction of the flow. Such a flow is called parallel. If the flow is taken to be in \( x \) direction:

\[ \frac{\partial u}{\partial x} = 0, \]
\[ 0 = -\frac{dP}{dx} + \mu \left( \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right), \]
\[ 0 = \frac{dP}{dy}, \]
\[ 0 = \frac{dP}{dz}, \]  
\hspace{1cm} (4.5)

as the pressure gradient is constant (due to continuity), one can make the following change of variable:

\[ \Phi = u - \frac{1}{4\mu} \frac{dP}{dx} (y^2 + z^2), \]  
\hspace{1cm} (4.6)

and then equation 4.5 becomes:

\[ \Delta \Phi = \frac{\partial^2 \Phi}{\partial y^2} + \frac{\partial^2 \Phi}{\partial z^2} = 0. \]  
\hspace{1cm} (4.7)

Our case is a particular case of a rectangular section \( (2a \times 2b) \) where \( a = b \). The analytical solution of a rectangular section duct is the following (written as a series of hyperbolic functions by Constantinescu [77]):
Figure 4.11. Comparison between Poiseuille velocity profile defined as in 4.8 and the PIV obtained profile for different Reynolds numbers.

\[ u = -\frac{16a^2}{\pi\mu} \frac{dP}{dx} \sum_{n=1}^{\infty} (-1)^{n-1} \left[ 1 - \frac{\cosh((2n-1)\pi z/2a)}{\cosh((2n-1)\pi b/2a)} \right] \frac{\cos((2n-1)\pi y/2a)}{(2n-1)^3}, \]

\[ Q = -\frac{4a^3 b}{3\mu} \frac{dP}{dx} \left[ 1 - \frac{192a}{\pi^3 b} \sum_{n=1}^{\infty} \frac{\tanh((2n-1)\pi b/2a)}{(2n-1)^5} \right]. \] (4.8)

The measurements were taken at 85 cm from the honeycomb. In these experiments no thermal problem was tested, so the fluid flow temperature was the same as wall temperature and then the entry region is only dependent on velocity profile. The honeycomb reduced the effect of the abrupt area contraction in the beginning of the channel between the cross section area of the main tank and the transverse area of the channel, similar to the step contraction of the cross-section channel. The reason for these pressure losses can be seen in the separation of the flow at the sharp convex edge of the
channel narrowing which the flow cannot follow. In bends or elbows, flow separation occurs usually at the inner side of the bend and with contraction of the main jet. That means a larger entrance length, in order to minimize this effect the channel begins with the corner rounded off and a honeycomb inserted. Then it could be said that honeycomb contributes to a) homogenize the incoming flow profile and b) anticipate the onset of viscous effects (the fluid inside the honeycomb moves along a set of narrow mini channels) so that the inviscid nucleus length of the entrance region might be shorter, see the discussion by Potter and Wiggert [53]. The entrance length needed in these experiments can be calculated from equation 4.9.

\[
\frac{le}{D_h} = 0.05Re_{D_h},
\]

(4.9)

where \(le\) is the entrance region where the flow is not fully developed, and \(D_h\) is the hydraulic diameter of the channel. In this case, a square section channel of 0.025 m side is considered. In experiments below, Reynolds numbers are based on the bluff body characteristic length, but in this calculation, Reynolds number should be based on channel hydraulic diameter, so, at maximum flow rate, Reynolds number 256 is reached. For that maximum Reynolds number, the ratio between entrance length and hydraulic diameter is 31.25, leading to an entry region no longer than 80 cm which is less than the length used in the experiment. In figure 4.12 the evolution for the last 5 cm of the entry region of the v-velocity in the middle of the channel for different Reynolds number can be seen. Figure 4.11 shows profiles obtained via experiments with PIV and the theoretical Poiseuille velocity profile for a square channel in the middle section \((z = -12.5 \text{ mm})\). Differences can be explained due to the vicinity of a narrowness downstream. In any case, the major differences are of the order of 10 %, they do not influence the total fluid flow and last, in the next experiments, fluid flow is going to have an obstacle with a high blockage ratio sooner than the narrowness. Therefore one can assume a fully developed fluid flow around the bluff body.

**Laminar duct flow upstream of the square prism**

After analyzing the flow without bluff body, the flow field was characterized upstream of the prism. This characterization was performed with regard to both the volume flow rate and the local velocity profiles in the incoming flow section.

Volume flow rate for different Reynolds number is shown in table 4.2. As in table 4.1, two different approaches were used; the first one by integrating a curve fitting of fifth order, and the second one by integrating a bi-directional natural neighbor interpolation. As it can be deduced from both tables, 4.1 and 4.2, the presence of the bluff body alter slightly the measurements increasing the discrepancy by no more than 2 %.
Figure 4.12. Spatial evolution of the centerline velocity for Reynolds 137 and 256, comparison with the theoretical solution and the bandwidth of the experimental measurements.

Table 4.2. Comparison of the results obtained via direct and indirect measurement of the incoming flow Reynolds number with bluff body downstream.

<table>
<thead>
<tr>
<th>Flow meter lecture</th>
<th>PIV integration parabolic fitting of $5^\text{th}$ order</th>
<th>Discrep.</th>
<th>PIV integration bi-directional natural neighbor interpolation</th>
<th>Discrep.</th>
</tr>
</thead>
<tbody>
<tr>
<td>137</td>
<td>139.1</td>
<td>1.6 %</td>
<td>139.3</td>
<td>1.7 %</td>
</tr>
<tr>
<td>185</td>
<td>181.7</td>
<td>1.8 %</td>
<td>181.7</td>
<td>1.8 %</td>
</tr>
<tr>
<td>205</td>
<td>203.7</td>
<td>0.6 %</td>
<td>203.7</td>
<td>0.6 %</td>
</tr>
<tr>
<td>256</td>
<td>248.7</td>
<td>2.8 %</td>
<td>250.8</td>
<td>2.0 %</td>
</tr>
</tbody>
</table>
The second aspect to be considered is whether the flow could be assumed to be fully developed before reaching the prism. For the case of the highest Reynolds number (256), which has the longest entrance length, the semi-empirical estimate of the entrance length (equation 4.9) gives a shorter value than the current length of the channel until reaching the bluff body which would ensure the presence of developed flow conditions. Figure 4.13 shows the streamwise PIV “v” velocity profile as a function of the “y” coordinate for the lines that start at a point of coordinates (5, -60, -12.5) (in mm), and reach up to the prism surface for different Reynolds numbers. The maximum and minimum spread of the experimental data for each Reynolds number, of the order of ±5%, is presented in dotted lines in the figure. As it could be observed, the figure shows that the flow velocity does not change significantly for distances larger than 50 mm (five times the square prism section length) in the upstream direction, so the flow could be considered to be developed.

One can see in figure 4.14 the profiles obtained via PIV and their corresponding Poiseuille profile for some Reynolds number of interest. It is clearly...
deduced that the square prism exerts a high influence in the inflow velocity profile.

Finally to conclude the study of the base flow, some power spectral densities (PSD) are shown in figure 4.15. In the left side power spectral densities of the free stream (without buff body) are placed, and on the right side the ones belonging to flow upstream of the bluff body are shown. This way, from Reynolds number 137 to 256, there is any peak on them, only a white noise spectrum that means that the following power spectral densities are not influenced by the incoming flow due to incoming flow has no predominant frequency.

The PSD represents the distribution of the total signal power between the frequencies 0 and $f_s$. The term density is used because power per frequency bandwidth ($f_s/N$) is being considered, where $f_s$ is the sample frequency and
Figure 4.15. Power spectral density at some point upstream the bluff body for different Reynolds numbers. On the left side, is shown the PSD of the flow without bluff body. On the right side the ones obtained upstream of the bluff body.
4.1. CONFINED 3D LAMINAR FLOW

$N$ is the number of samples captured. The PSD is symmetric about $k = N/2$ and has a periodicity every $N$ samples. Therefore the maximum resolvable frequency is half the sampling frequency (the Nyquist frequency) and the resolution is determined by the data set duration. In these experiments the sample frequency was 15 Hz, so the Nyquist frequency or the maximum frequency at which Fourier coefficients can be obtained is 7.5 Hz. The right figures of 4.15 were captured with 100 samples instead of 500 samples that were used in every other test. That is the reason for the higher frequency resolution that can be calculated via $\Delta f = f_s/N$. The power spectral density is given by the square magnitude of the spectral coefficients of the discrete Fourier transform (DFT):

$$X_k = X(f = f_k) = \sum_{n=0}^{N-1} x_n e^{i \frac{2\pi nk}{N}}, \quad k = 0, 1, ..., (N-1), \quad (4.10)$$

$$S_k = S(f = f_k) = \frac{1}{N f_s} ||X_k||^2, \quad k = 0, 1, ..., (N-1). \quad (4.11)$$

In equation 4.10 and 4.11, $X_k$ are the different Fourier coefficients for the DFT, $S_k$ are the corresponding coefficients for the PSD, $f_k$ the equally spaced frequencies given by 4.12 and $x_n$ the discrete values obtained via PIV.

$$f_k = \frac{k f_s}{N}, \quad k = 0, 1, ..., (N-1). \quad (4.12)$$

Since the time between the sample points is not infinitely small, the power in the signal at frequencies above the Nyquist frequency will appear in the PSD at lower frequencies, an effect known as aliasing. Aliasing errors in estimates of PSD are avoided by applying an analog anti-aliasing, low-pass filter with a sharp cut-off at half the sampling frequency.

Besides the periodicity after every $N$ samples of the PSD, the inverse DFT also has a period of $N$ samples. This effectively means that the DFT perceives and acts on an infinite juxtaposition of the input data record and the inverse DFT effectively transforms an infinite juxtaposition of the spectrum. If the beginning and end of the record do not merge smoothly into one another, sudden amplitude jumps are perceived, which give rise to additional frequency components in the spectrum. These end effects are unimportant for records of long duration; however, they deserve attention with short records. These effects are diminished by applying window functions in the time domain. Window functions scale the input data amplitude and force a tapering to zero at the beginning and end of the signal. A further consequence of a finite input record duration is spectral broadening. A spectrum of an infinitely long sine wave is a delta function at the signal frequency. A
finite-length sine wave yields however a broadened peak, in which the peak width is inversely proportional to the input signal duration. An obvious consequence of spectral broadening is that the resolution of distinct signal frequencies in the PSD can be improved by sampling a longer portion of the signal.

In practical implementations of the discrete Fourier transform, equation 4.10 is not directly used but rather a recursive form known as the fast Fourier transform (FFT) is used. There are many realizations of the FFT, but they share one feature in common, namely, that they normally operate on $2^n$ points. The calculation time of the DFT implemented with 4.10 increases with $N^2$. The FFT algorithm reduces the computation time to the order of $N \log N$. A commonly used technique with the FFT is that of zero padding. Without changing the spectral content of the signal, zero padding forces the FFT algorithm to estimate the spectrum at additional frequencies between zero and the maximum frequency, thus improving the resolution. This is easily seen by examining a signal doubled in length by adding zeros (see equation 4.13) with $x_n = 0$ for $n = N, N + 1, \ldots (2N - 1)$.

$$X_k = X(f = f_k) = \sum_{n=0}^{2N-1} x_n e^{i \frac{2\pi nk}{2N}}, \quad k = 0, 1, \ldots, (2N - 1), \quad (4.13)$$

this can be written as in equation 4.14, which is identical to the $N$-point transform for every other $k$ value. However, now $X_k$ is also computed at intermediate values. The spectral content of the signal has in no way been altered, but with the intermediate estimates, interpolation of peak locations can be improved. Zero padding can also be used to extend input data records up to a length of $2^n$ values, in preparation for an FFT.

$$X_k = X(f = f_k) = \sum_{n=0}^{N-1} x_n e^{i \frac{2\pi n(k/2)}{N}}, \quad k = 0, 1, \ldots, (2N - 1). \quad (4.14)$$

### 4.1.3 Experimental results in the downstream region

In a previous numerical study, Martin and Velazquez [56] have pointed out that in highly confined flow (both isothermal and non-isothermal) around a square prism, the transition from a closed recirculation bubble regime to a Karman street type of vortex shedding is not abrupt as in the unconfined case. In particular, they identified an intermediate regime in which the closed recirculation bubble oscillates before entering into the next vortex shedding regime. Specifically, for the geometry used in the present experiment, they identified a steady recirculation bubble for Reynolds < 110, an oscillating
4.1. CONFINED 3D LAMINAR FLOW

Recirculation bubble for $110 < \text{Reynolds} < 170$, and a Karman street for \text{Reynolds} $> 170$. This clearly differs from the 2D (or quasi 2D) unconfined case in which vortex shedding is reported to start in the range of Reynolds numbers from 50 to 60 (depending on the author). The experimental study in the present chapter aims to identify these regimes and their associated parametric range. To do so, the following set of Reynolds numbers was considered: 100, 110, 120, 130, 137, 150, 160, 170, 180, 185, 195, 205, 210, 230 and 256.

The bluff body is a rectangular cylinder, the boundary layer separates from the sides of the body at some point near the maximum width. It has a dramatic reduction of body width and a high adverse pressure gradient leading to boundary layer separation. The subsequent rolling up of the separated vorticity in the free shear layer causes the formation of concentrated local regions of vorticity, known as wake vortices. It is these vortices, and their associate low-pressure centers, in proximity to the rear of the body, which yield very large fluctuating pressures behind a bluff body, but also a surprisingly uniform region of, and the low uniform pressure in the rear region of the body, low pressure behind the body. By integrating such a pressure distribution around the body, we can understand the large drag of a bluff body to be due to the difference between the high-pressure region in the vicinity of the front stagnation point.

The quantitative experimental identification of the different flow topologies was addressed using several means that complement each other. First of all looking at both the frequency and the amplitude of the flow velocity oscillation at three selected points. An impression of the location of these points P1, P2, and P3, whose coordinates were defined in subsection 4.1.1 above, is given in figure 4.7. The frequency was characterized by measuring with the PIV analyst software the power spectral density of the absolute value of the velocity $|V_{xy}| = \sqrt{u^2 + v^2}$ at the selected points. Figure 4.16 shows the results obtained at point P1 for six different Reynolds numbers: 120, 137, 150, 160, 205 and 256. It should be warned that this point was fixed relative to the square prism and not to the end of the recirculating bubble, therefore the point is not the point where maximum deviations from mean occurs at a determined Reynolds number. This will be shown in following sections.

Additionally, two windows have been selected inside the PIV interrogation areas (figure 4.17). Window A is located close to the prism and covers a significant part of the recirculation bubble. Window B is located further downstream and overlaps the region where, when appropriate, the recirculation bubble breaks down. Each window is spanned by a Cartesian network of 100 points where the information related to the time-evolution of the amplitude was summarized by computing the root mean square (rms) of the deviations of the velocity $v$ around its mean value is recorded.

Now, the average value of the rms at these 100 points is presented as a function of the Reynolds number in figure 4.18. The results also include the
Figure 4.16. Power spectral density for Re 120 (top-left), 137 (top-right), 150 (medium-left), 160 (medium-right), 205 (bottom-left) and 256 (bottom-right) at P1 location.
base flow case without prism.

The first thing that could be observed in figure 4.18 is that the rms in the base flow remains nearly constant for all Reynolds numbers. Second, it is to be noted the presence of the prism induces, even at the lowest Reynolds number, an average rms that is nearly double the one associated to the base flow. Looking at the results associated to window A, this rms increases steadily up to Reynolds 180 where a discontinuity in the rms slope can be observed. For Reynolds numbers below 120, flow passing through window B does not recirculate. Beyond that point, the average rms also increases steadily up to Reynolds 180 where a change of slope can be seen again. So, basically, there is a monotonic increase in the rms signal number up to Reynolds number 180 and a change of slope at that precise figure.

Another indication of a possible change of regime is the behaviour of the time-averaged bubble formation length, Williamson [63]. As this author points out, a mean recirculation region in the wake can be defined averaging over large times compared to the typical shedding period. Then, a gradual movement of this formation length towards the prism may indicate a change of regime. In the present study, this time averaging has been performed using 500 frames of each PIV measurements series, the mean flow fields have been obtained, and the average formation length results are presented in figure 4.28. There, a distinct change of behavior of the averaged recirculation region (shortening towards the prism) could be observed around Reynolds 150.

Information regarding the power spectra density PSD associated to the different regimes is shown in figure 4.16. The six selected cases are: a) Reynolds number 120, a case with steady recirculating bubble; b) the case at Reynolds 137, beginning of the flapping bubble; c) and d) cases at Reynolds 150 and 160, located in the middle of the intermittent regime; and e) and f) cases at Reynolds number 205 and 256 in the vortex shedding regime. The PSD of the base flow were shown in figure 4.15. The frequency content at
Figure 4.18. Average value of the stream-wise velocity rms at the 100 points for the case with prism.

Figure 4.19. Frequency of vortex shedding as a function of the Reynolds number.
4.1. CONFINED 3D LAMINAR FLOW

Figure 4.20. Time averaged formation length as a function of the Reynolds number.

Reynolds number 205 is maximum and, beyond that, it starts to diminish and to broaden. This fact suggests that a further change of regime might be expected if the Reynolds number keeps increasing.

The last step of identification consist of the use of a vortex identification method. The issue of vortex identification still remains an open subject, a review of its status can be found, for example, in the work of Haller [37]. In the present work, it has been decided to use the Q-criterion of Hunt et al., [46], because of its robustness and lack of ambiguity although some of its finer details might be open for discussion (see Haller [37]). This method identifies vortices of an incompressible flow as connected fluid regions with positive second invariant of \( \frac{\partial u}{\partial x} \), that is, as the regions where the vorticity magnitude prevails over the strain-rate magnitude.

\[
Q = \frac{1}{2} (||\Omega||^2 - ||S||^2) > 0,
\]

where \( \Omega \) is the vorticity tensor, the antisymmetric part of \( \frac{\partial u}{\partial x} \) and \( S \) is the strain-rate tensor, the symmetric part of \( \frac{\partial u}{\partial x} \). Application of this criterion to the PIV results obtained in this study has led to the following outcome:

- Up to Reynolds 150, the Q maps obtained with prism are qualitatively similar to the Q maps of the base flow without prism.
In the range from Reynolds 150 to 170, Q type structures that could be identified as vortex appear in an intermittent way. That is, they are not present in all 500 frames that make up for a PIV interrogation series, but only in a limited number of them.

Beyond Reynolds 180, the Q-type structures appear in all PIV frames.

These results are consistent with the results presented in figure 4.28 and they also suggest that transition between regimes is not abrupt (the intermittency) and that it spans a certain range of Reynolds numbers. Figure 4.21 shows a summary of the Q-maps for some selected Reynolds numbers (including a base flow case without prism).

In figure 4.19, frequencies of vortex shedding is shown as a function of
4.1. **CONFINED 3D LAMINAR FLOW**

<table>
<thead>
<tr>
<th>Table 4.3. Strouhal numbers as a function of the Reynolds number.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Re</td>
</tr>
<tr>
<td>St</td>
</tr>
<tr>
<td>St*Re</td>
</tr>
</tbody>
</table>

Reynolds number. It has been measured calculating the peak of frequency of variation of velocities at points P1, P2 and P3. It is easy to see that the frequency of vortex shedding remains constant during the variation of Reynolds number. This leads to decreasing linear relation with the Strouhal number calculated as $St = fD/U$, where $f$ is the frequency of vortex shedding, $D$ is the hydraulic diameter of the bluff body and $U$ is the mean velocity of the flow. Table 4.3 shows the relation between Strouhal number and Reynolds number in these experiments. It could be observed that the behavior of the parameter Strouhal times Reynolds is constant with regard to the Reynolds number. This is different from what has been reported in the literature in other situations. In particular, the reader is directed to the article of Rehimi et al, [54], where apart from their own results, the authors present and compare them with results provided by other researchers. In all cases presented in figure 13 of the article by Rehimi et al, [54], the parameter $St*Re$ presents a monotonic increase along with the Reynolds number. In the present work, see table 4.3, this parameter remains constant and this is caused by the fact that the shedding frequency is constant within the span of Reynolds number that has been analyzed. The main conceptual difference between the present experimental campaign and those reported by Rehimi [54], is that in the present case the flow is strongly confined in the span wise direction as well; i.e.: the flow is fully 3D. Then, this suggests that 3D confinement affects the shedding behavior and leads to the somewhat unexpected result of having a constant shedding frequency. As it could be observed in table 4.3, the parameter $St*Re$ stays nearly constant around the value of 71.0 with deviations that amount to a 5% at most.

Summarizing, these experimental results tend to confirm validity of the numerical results presented by Martin and Velazquez [56]. In particular, they seem to confirm the existence of the three regimes characterized by a steady recirculation bubble, an intermediate oscillating recirculation bubble and a vortex shedding respectively. In particular, these three regimes could be characterized as follows:

- Reynolds < 120. The power spectral density measurement does not show any distinguished frequency. The dimensionless rms of the streamwise velocity has a decreasing tendency, same as the one of free of bluff body. The bubble length is increasing and the Q map are like the ones of free flow. This interval is determined as a steady recirculation bubble state.
• $120 < \text{Reynolds} < 150-170$. There is a frequency that stands out in the power spectral density measurement. Its power density is larger than the one associated to other frequencies but the order of magnitude is similar. The rms value of the velocity signal shows a marked monotonic increase at some specific points (P1, for example) in the flow field. The rms has changed its negative slope for a positive one. Q maps show some zones downstream of the bluff bodies with $Q$ positive. This is the intermediate oscillating recirculation bubble state. It finishes between Reynolds numbers 150 and 170 where the bubble length has an unexpected fall and the Q maps show their first structures that can be defined as vortices.

• $170 < \text{Reynolds}$. A single frequency dominates the spectrum and its power density is larger by three orders of magnitude than the one associated to other frequencies. The rms value of the velocity signal grows now exponentially as compared to values at lower Reynolds numbers. The bubble length grows a little but finally remains nearly constant, with a slightly negative tendency. Q maps show greater vortices zones shedding from the bluff body harmonically. This is the vortex shedding state.

3D numerical computations have been carried out to cross-check the experimental results. The numerical solver has been generated using the OpenFOAM C++ libraries. Details of the solver validation under a variety of conditions and a description of a grid sensitivity analysis have been provided by Martin and Velazquez [56]. The main findings of that work are already reported in the introduction section. In the present case, the number of elements in the computational domain was 1,855,000. In particular, at Reynolds 100, the numerical results obtained using the experimental inlet velocity profile do not show vortex shedding, while the same computation with the theoretical inlet velocity profile presents a well developed Karman vortex street. At Reynolds 170, both the experimental data and numerical data obtained with the experimental inlet velocity profile show the presence of vortex shedding. Figures 4.22 and 4.24 show the comparison between instant (dimensionless) experimental and numerical flow fields at Reynolds 100 and 170 respectively in three x-y planes at z stations: $z = -1$, $-6.5$, and $-12.5 \text{ mm}$. To allow for a better comparison, the velocity maps have been made dimensionless dividing by the average (integrated) inlet velocity in each case.

It could be observed in figure 4.22 that the size of the recirculation region decreases significantly as the z-planes used for representation purposes get closer to the duct walls. The reason is the presence of the boundary layer located nearby. However, the rate of decrement is not uniform since, as it could be observed, differences get more accentuated closer to the wall.
Figure 4.22. Numerical (top) and experimental (bottom) results at Reynolds 100 in x-y planes at three z stations: $z = -12.5\ mm$, $-6.5\ mm$, and $-1.0\ mm$. Dimensionless variables on images.
Figure 4.23. Numerical (top) and experimental (bottom) results at Reynolds 170 in x-y planes at three z stations: $z = -12.5$ mm, $-6.5$ mm, and $-1.0$ mm. Dimensionless variables on images.
A similar conclusion could be stated in connection to the results presented in figure 4.24 (Reynolds equal to 170). It is only in the z-plane closer to the wall ($z = -1 \text{ mm}$) where the typical vortex street behavior disappears. This suggests that if the purpose of the prism is to destabilize the flow field, this destabilization occurs over a large portion of the prism span which may have positive implications for some engineering applications related, for example, to thermal control. Focusing now on the experimental results only, the equivalent of figures 4.22 and 4.24 but considering the z-y planes are presented in figure 4.31. There, the differences in flow topology can be appreciated for the cases with and without vortex shedding and, again, it is to be noted the strong flow destabilization that the prism creates near the channel walls.

These results have practical implications regarding the engineering design of micro heat sinks. In particular, to efficiently promote mixing, the prism should destabilize the fluid as much as possible and this, together with the requirement that it should not break down under the hydrodynamic pressure, points towards the use of thick geometries which, in turn, lead to high blockage ratios. Then, in situations of this type, when looking at practical applications, it is important to keep in mind the fact that the flow rate needed to destabilize the fluid is about four times higher than in free stream conditions (Reynolds number of the order of 170 versus Reynolds 50). At the same time, the high blockage ratio implies a high value for the pressure drop, so the pumping power (that scales as the product of flow rate times the pressure drop) needs to be increased accordingly.

One of the aspects that were not addressed in the previous sections and subsections is the influence of wall proximity on flow topology. Doing it experimentally would require to manufacture a test channel where the blockage ratio could easily be varied, which is out of the scope of the present study. Another option would be to keep the channel as it is, and change the prism cross section height. However, if much smaller blockage ratios were desired the ensuing smaller prisms would pose restrictions on the PIV resolution and accuracy. This is why this part of the study has been carried out using Computational Fluid Dynamics (CFD) techniques. The numerical method that has been chosen is the one described by Martin and Velazquez [56].

A series of different computations was carried out with different blockage ratios while keeping a channel square section with the aim of identifying the $Re_c$ at which Karman-type shedding starts and the shedding frequency. The 2D case was also computed for comparison purposes. In all cases, the mesh was structured and it contained 2,615,000 hexahedral elements. If “H” is used to label the channel cross section height and “h” the prism cross section height, the channel total length was 15 H, and the prism was located at a distance of 5 H from the inlet section. The CPU time needed to converge each case was 56 hours running in parallel mode on a three nodes Intel(R) Core (TM) i7-3930k CPU 3.20 GHz. For each blockage ratio a number of
Figure 4.24. PIV velocity profiles at Reynolds 100 (top) and 170 (bottom) in y-z planes at three x stations: $x = 5.0 \, mm$, $-1.0 \, mm$, and $-6.5 \, mm$. Dimensionless variables on images.
4.1. CONFINED 3D LAMINAR FLOW

Table 4.4. Computed critical $Re_c$ and Strouhal number $St_c$ for different blockage ratios.

<table>
<thead>
<tr>
<th>h/H</th>
<th>1/1.25</th>
<th>1/2.5</th>
<th>1/5</th>
<th>1/10</th>
<th>2D</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Re_c$</td>
<td>175</td>
<td>166</td>
<td>66</td>
<td>50</td>
<td>48</td>
</tr>
<tr>
<td>$St_c$</td>
<td>0.42</td>
<td>0.36</td>
<td>0.18</td>
<td>0.13</td>
<td>0.32</td>
</tr>
</tbody>
</table>

cases of the order of 10 was run to identify the $Re_c$. To be consistent, all cases presented in table 4.4 were computed using the Poiseuille type solution as the inlet boundary condition. The summary of the results obtained is presented in table 4.4.

The results presented in table 4.4 show that the influence of wall proximity on $Re_c$ and $St_c$ does not scale linearly on the blockage ratio h/H. In fact, halving the parameter h/H from 1/1.25 to 1/2.5 causes $Re_c$ to change by a mere 5% (175 to 166), while halving h/H from 1/2.5 to 1/5 causes $Re_c$ to change by a factor of 60% (166 to 66). If h/H keeps decreasing, for example h/H = 1/10, the 2D solution is almost recovered. This fact suggests that wall influence can be classified roughly into two limiting regimes: the quasi-2D regime and the regime where transition to vortex shedding is significantly delayed. The boundary between the two regimes appears to be rather narrow in terms of the parameter h/H and the understanding of their intrinsic characteristics may open up a new line of research.

Figure 4.25 shows the time-averaged rms maps of the V velocity in the x-y plane corresponding to Re 120, 137, 160 and 205. These rms maps have been obtained after averaging the 500 frames associated to each PIV sequence. The rms values presented are dimensionless since velocity signals have been divided by their associated bulk velocity. It could be observed that the topology of these rms maps depends strongly on the flow regime. In the cases of Re 160 (intermittent vortex shedding) and 205 (vortex shedding) the regions of high rms extend over a significant portion of the flow field and this is associated to the fact that periodic shedding causes significant oscillations in the stream-wise velocity thereby leading to a higher rms value.

Now, the point where rms is maximum inside window B has been selected for each Reynolds number (note that the position of this specific point depends on Reynolds number) and the time evolution of the dimensionless stream-wise velocity at this particular point has been plotted, see figure 4.26. At Reynolds 120, the velocity signal that yields the maximum rms is, basically, random noise while a distinct stream wise pulsation can be observed at Reynolds 137. According to the results presented in figures 4.28 and 4.21, there is no vortex shedding at the specific Reynolds number. At Reynolds 160, in the intermittent regime, the velocity signal maintains the same frequency but the dimensionless velocity amplitude has nearly doubled from 0.8 to 1.5. At Reynolds 205 the dimensionless amplitude is even larger.
Figure 4.25. Time-averaged dimensionless rms values of stream-wise velocity at the x-y center-plane for Re 120, 137, 160 and 205. Windows A and B are depicted as well for illustration purposes.
while the frequency did not change significantly. The coordinates \((x, y, z)\) in \(mm\) of the points presented in figure 4.26 are as follows:

- Re 120: \((x, y, z) = (2.8, 52.9, -12.5)\)
- Re 137: \((x, y, z) = (2.8, 53.7, -12.5)\)
- Re 160: \((x, y, z) = (1.5, 53.7, -12.5)\)
- Re 205: \((x, y, z) = (0.7, 42.5, -12.5)\)

The time averaged stream-wise dimensionless velocity fields for the selected Reynolds numbers are presented in figure 4.27. It could be observed that the size of the recirculation region grows significantly as a function of Reynolds number for both the \(x - y\) and the \(y - z\) planes. Since velocity vectors are also shown, it could be noted that the average velocity profiles at downstream stations away from the prism (for instance, at \(y = 6\)) tend to be flatter at the highest Reynolds. Out of the information contained in figure 4.27, it is possible to obtain the spatial evolution of the average stream-wise velocity at the channel center-line. This is shown in figure 4.28 that should be compared to the results presented in figures 4a and 4b of the article by Rehimi et al [54]. Three main aspects are different between them, here the flow is confined in the span-wise direction, the obstacle cross-section is square and the blockage ratio is \(2.5/1\). In the work of Rehimi et al [54], the flow is approximately 2D, the obstacle cross-section is circular, and the blockage ratio is \(3/1\). When comparing those two figures (note that the range or Re is practically the same) the following differences could be found:

- Here, the position of zero mean velocity at the wake center-line moves away from the prism up to Re 150; then, it moves back up until 170, and it moves away afterwards. In the work of Rehimi et al [54], this position moves away from the prism and, then, it moves back.

- Here, the position of zero mean velocity at the wake center-line is located further away from the prism at Re 256 than at Re 100 (by a factor of 20 \%). In the work of Rehimi et al [54] the opposite happens.

- In the present case, the average center-line velocity for the higher Reynolds numbers grows asymptotically up to a distance of six prism diameters. In the work of Rehimi et al [54] this velocity reaches a maximum around a distance of five cylinder diameters and decreases afterwards.

Figure 4.28 gives the evolution of the dimensionless longitudinal center-line velocity and the rms of the span-wise velocity with \(y\)-position for different Reynolds numbers. On the right side of this figure one could see the variation
Figure 4.26. Time evolution of the dimensionless velocity signal $V$ at the points where the mean rms is maximum for Reynolds number equals to 120, 137, 160 and 205.
Figure 4.27. Time averaged stream wise velocity profiles for Reynolds numbers 100, 150, 170 and 205 (from left to right) at planes $x - y$ at the top and $z - y$ at the bottom.
of the span-wise velocity at the centerline, and easily deduce that at low 
Reynolds number with recirculating bubble, this variation is higher inside 
the bubble than outside. The opposite occurs at high Reynolds number 
where vortex shedding occurs.

An impression of the time evolution of the velocity field in the intermit-
tency regime, Re 160, is also presented in figure 4.29

Regarding this subject of flow topography, a question arises in connection 
to the mechanism leading to destabilization. In particular, the question is 
whether some 3-D instabilities, somewhat similar to the modes A and B 
described by Williamson [63], are present in the flow. In the present work, 
the PIV time sequences for the different Re have been searched to look 
for inceptions of stream-wise vortex loops (mode A) and finer scale vortex 
pairs (mode B) without success. The opinion of the authors is that the 
strong confinement in the span-wise direction prevents the appearance of 
these modes. For example, in figure 13 of the article by Williamson [63] it 
can be observed that the spatial scale of mode A in the span-wise direction is 
about four times the cylinder diameter. However, the distance between the 
lateral walls in the present study is 2.5 times the prism cross-section height. 
More specifically, the PIV results obtained seem to suggest that, owing to the 
high lateral confinement, transition between the different regimes is smooth. 
An impression of the span-wise structures (time averaged PIV U velocity 
profiles) that can be observed as a function of Re are presented in figure 
4.30.

Vorticity is recognized as a primitive variable of considerable interest in 
fluid dynamics. However, its time-resolved measurement has significantly 
lagged behind the development of velocity and pressure measurement tech-
niques. The necessity to obtain a measure of the curl of the velocity, that 
is, to accurately measure velocity differences over very small distances, has 
been a formidable challenge. Particle image velocimetry techniques can give 
a measure of the vorticity although their spatial resolution does not typically 
allow as fine a resolution of the scales of the vorticity field. The velocity 
field is not well suited for defining and identifying organized structures in 
time-dependent vortical flows also because the streamlines and path-lines are 
completely different in two different inertial frames of reference.

Vorticity characterizes the rotation rate of a fluid particle. In the case 
of constant-density incompressible flows, vorticity is acquired by a pressure 
gradient introduced at a physical surface. The pressure gradient at the sur-
face is balanced by the stress gradient, which is related to the vorticity flux 
entering the flow. Thus the existence of vorticity generally indicates that 
viscous effects are important.

For planar data gradients in the perpendicular to the plane direction 
cannot be calculated, so only rotation around this axis can be determined 
as equation 4.16. Two examples of vorticity maps are plotted in figure 4.31
Figure 4.28. Stream-wise evolution of the mean longitudinal velocity along the center-line in the bluff body wake on the left and rms of the span-wise velocity along the center-line in the bluff body wake on the right.
Figure 4.29. Stream-wise velocity \((V)\) fields for Reynolds number equal to 160 at different times of the shedding cycle.

\[
\omega = \frac{1}{2} \left( \frac{\partial U_2}{\partial x_1} - \frac{\partial U_1}{\partial x_2} \right).
\]  

(4.16)

The calculation of the vorticity or velocity gradient tensor is based on the derivatives of the velocity field deduced from the PIV data. These measurements are generally noisy and by applying the derivatives, high noise frequencies are amplified. For this reason proper orthogonal decomposition (POD) of the flow was applied. POD is used as a filter that takes into consideration the physics of the problem by taking into account the energetic distribution of the flow and giving information about the fluid energy distribution in the measurement domain for the acquisitions duration. Essentially, POD is a linear procedure that takes a given collection of input data and creates an orthogonal basis constituted by functions estimated as the solutions of an integral eigenvalue problem known as a Fredholm equation. These eigenfunctions are by definition characteristic of the most probable realizations of the input data. Moreover, it can be shown that they are optimal in terms of representation of the energy present within the data. Historically, the proper orthogonal decomposition was introduced in the context of turbulence by Lumley as an objective definition of coherent structures. It is a natural idea to replace the usual Fourier decomposition in non-homogeneous directions.

From a mathematical point of view, the proper orthogonal decomposition is just a transformation that diagonalizes a given matrix \(A\) and brings it to
Figure 4.30. Time averaged dimensionless span-wise U velocity for different Re.
Figure 4.31. Flow vorticity and streamlines for Reynolds number equal to 100 and 170, left and right respectively.
a canonical form $A = U \Sigma V^*$, where $\Sigma$ is a diagonal matrix.

The data acquired by PIV methods can be considered as a velocity fields $[U(x, t_1), ..., U(x, t_N)]$. Our aim is to approximate this vector-valued function as a finite sum in the separated-variables form:

$$ U(x, t) \simeq \sum_{k=1}^{N} a^{(k)}(t) \phi^{(k)}(x). \quad (4.17) $$

This approximation becomes exact as $N \to +\infty$. One can use basis functions given a priori, for example Fourier series, Legendre polynomials or Chebyshev polynomials. An alternative approach is to determine the functions $\phi^{(k)}(x)$ that are naturally intrinsic for the approximation of the function $U(x, t)$, and that approach is called proper orthogonal decomposition. These functions must fulfill the orthonormal condition given in 4.18 where $\delta_{k_1,k_2}$ is the Kronecker delta and $\Omega$ refer to the domain considered.

$$ \int_{\Omega} \phi^{(k_1)}(x) \phi^{(k_2)}(x) dx = \delta_{k_1,k_2}, \quad (4.18) $$

then, the coefficients relative to each function (they are going to be named modes) is calculated via equation 4.19:

$$ a^{(k)}(t) = \int_{\Omega} U(x, t) \phi^{(k)}(x) dx. \quad (4.19) $$

An approximation to any desired accuracy can always be obtained if $N$ can be chosen large enough. In problems where the spatial data size is much larger than the number of images, like PIV measurements, snapshot POD is the favorite approach for the POD problem, and for this reason this approach is the one used in this work.

For filtering purposes, avoiding amplify noisy frequencies, data has been post-processed with 91% of the total energy, retaining the largest cumulative eigenvalues. Table 4.5 shows the first six eigenvalues of the most energetic modes for the velocity field obtained from the plane $x - y$ whose $z$ position is $-12.5 \text{ mm}$. Clearly from table 4.5, one can see a difference between the first two modes when the Karman street has developed on the flow (this happens at Reynolds number 150-160), being these modes von Karman harmonics. For Reynolds number 120 and 180, the first 498 modes are shown in figure 4.32. Both curves behave the same from mode number around 10, but only for Reynolds number where Karman street has developed, the first two modes are much larger than the rest of them. Figure 4.33 shows the first six modes from importance for a Reynolds number equal to 185.
CHAPTER 4. LOW REYNOLDS NUMBER VORTEX STUDIES

Table 4.5. Relative contribution (λᵢ/∑ᵢ λᵢ) of the first six eigenvalues for different Reynolds numbers.

<table>
<thead>
<tr>
<th>Re</th>
<th>100</th>
<th>110</th>
<th>120</th>
<th>130</th>
<th>137</th>
<th>150</th>
<th>160</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mode 1</td>
<td>0.0336</td>
<td>0.0532</td>
<td>0.0406</td>
<td>0.0271</td>
<td>0.0691</td>
<td>0.0625</td>
<td>0.1109</td>
</tr>
<tr>
<td>Mode 2</td>
<td>0.0279</td>
<td>0.0293</td>
<td>0.0272</td>
<td>0.0225</td>
<td>0.0218</td>
<td>0.0603</td>
<td>0.1046</td>
</tr>
<tr>
<td>Mode 3</td>
<td>0.0234</td>
<td>0.0186</td>
<td>0.0192</td>
<td>0.0162</td>
<td>0.0198</td>
<td>0.0277</td>
<td>0.0277</td>
</tr>
<tr>
<td>Mode 4</td>
<td>0.0161</td>
<td>0.0132</td>
<td>0.0155</td>
<td>0.0158</td>
<td>0.0160</td>
<td>0.0168</td>
<td>0.0237</td>
</tr>
<tr>
<td>Mode 5</td>
<td>0.0139</td>
<td>0.0127</td>
<td>0.0129</td>
<td>0.0132</td>
<td>0.0150</td>
<td>0.0165</td>
<td>0.0206</td>
</tr>
<tr>
<td>Mode 6</td>
<td>0.0134</td>
<td>0.0118</td>
<td>0.0091</td>
<td>0.0125</td>
<td>0.0139</td>
<td>0.0158</td>
<td>0.0163</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Re</th>
<th>170</th>
<th>180</th>
<th>185</th>
<th>195</th>
<th>205</th>
<th>230</th>
<th>256</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mode 1</td>
<td>0.1152</td>
<td>0.1275</td>
<td>0.1542</td>
<td>0.1482</td>
<td>0.1631</td>
<td>0.1326</td>
<td>0.0784</td>
</tr>
<tr>
<td>Mode 2</td>
<td>0.1134</td>
<td>0.1246</td>
<td>0.1505</td>
<td>0.1476</td>
<td>0.1601</td>
<td>0.1286</td>
<td>0.0768</td>
</tr>
<tr>
<td>Mode 3</td>
<td>0.0338</td>
<td>0.0459</td>
<td>0.0521</td>
<td>0.0512</td>
<td>0.0526</td>
<td>0.0543</td>
<td>0.0536</td>
</tr>
<tr>
<td>Mode 4</td>
<td>0.0323</td>
<td>0.0220</td>
<td>0.0279</td>
<td>0.0505</td>
<td>0.0496</td>
<td>0.0538</td>
<td>0.0523</td>
</tr>
<tr>
<td>Mode 5</td>
<td>0.0229</td>
<td>0.0219</td>
<td>0.0267</td>
<td>0.0226</td>
<td>0.0367</td>
<td>0.0308</td>
<td>0.0388</td>
</tr>
<tr>
<td>Mode 6</td>
<td>0.0191</td>
<td>0.0175</td>
<td>0.0176</td>
<td>0.0184</td>
<td>0.0224</td>
<td>0.0282</td>
<td>0.0384</td>
</tr>
</tbody>
</table>

Figure 4.32. Modes energy fraction and cumulative for Re = 120 on the left (no Karman street) and Re = 185 on the right (Karman street).
Figure 4.33. The first six modes obtained downstream of the bluff body for $Re = 185$ (vector fields and vorticity maps).
It seems that walls exert a dissipating effect over all range of modes, reducing the relative weight of the first harmonics, mainly the ones belonging to von Karman vortex shedding, and increasing the fraction of energy transported by the less important harmonics. These difficult the task of filtering due to the fact that 91% of the total energy is not contained in the first six modes but in the first two hundred.

4.1.4 Conclusions

The experimental work that has been performed shows that 3D confinement effects play a significant role on the behavior of the flow past a square prism in the laminar regime. Differences between the 3D confined and 2D unconfined cases are both qualitative and quantitative. Qualitatively, three different regimes have been identified in the case of a 1/2.5 blockage ratio, as opposed to the 2D unconfined case. These three regimes are: a steady recirculation bubble, a pulsating recirculation bubble, and a Karman-type vortex shedding regime. The transition between the steady and pulsating recirculation bubble regimes appears to be smooth. However, transition from the pulsating bubble to the vortex shedding regime shows indication of a narrow (in terms of the Re) intermittency regime where the length of the recirculation bubble obtained via time averaging of the PIV frames decreases before increasing again as a function of Re. Specifically, the results obtained suggests that the steady recirculation bubble regime lasts up to about Re 120; the pulsating recirculation regime covers the span for Re 120 to 150; the intermittency regime stays up to Re 180, and from then on Karman-type shedding starts. This is markedly different from the 2D unconfined case and, also, from the 2D confined case as reported by other researchers. This suggests that additional confinement in the span wise direction, which tends to be the rule in industrial applications, has a strong effect on flow topology that should be accounted for when designing actual systems and products. Also, it is worth noting that the experimental results that have been presented (in particular, the sequence of regimes as a function of Re) bear a striking resemblance to the sequence of events that characterize a completely different problem (shallow turbulent wake flow) as described by other researchers. Whether this is by chance or it reveals a similitude of mechanisms leading to instability at the fundamental level is something that might be an interesting subject for study in the future.

Another difference with 2D confined cases is that once the shedding regime starts, the shedding frequency remains nearly constant. That is, the parameter $St \times Re$ stays nearly constant for a span of Re ranging from 170 to 256 and this is, again, in contrast, to the results reported by other researchers in the 2D confined case where a steady increase in $St \times Re$ as a function of Re is the norm. This fact also has some implications for practical engineering designs. The reason is that if the intention is to promote mixing...
in the laminar regime using this configuration, increasing the flow rate (and, thereby, increasing the pressure drop and the pumping power as well) does not guarantee a significantly larger unsteadiness in the flow field that may contribute to increase, for instance, the transport of a passive scalar.

Finally, regarding the effect of wall proximity, the results obtained point in the direction of the existence of two different regimes as a function of the prism blockage ratio. These two regimes are a quasi-2D regime and a regime where transition to shedding is significantly delayed. Surprisingly, transition between these two regimes appears to occur within a narrow span of the blockage ratio parameter $h/H$, instead of being of a smooth nature; which suggests that this specific matter deserves some detailed study in the future.

4.2 Confined 3D flow induced vibrations of a tethered prism at a high blockage ratio

The present section deals with an experimental study on the problem of 3D channel flow past a buoyant tethered prism in the low Reynolds number regime (Reynolds number in the range from 100 to 700 based on the prism cross section height) at a blockage ratio of 1:2.5. The objective of the work is to study the effects of confinement associated to the 3D character of the flow (the channel has a square section) and to the high blockage ratio. In particular, different flow regimes are studied and characterized using a PIV system and an optical camera. Apart from describing and understanding the flow regimes involved, the work being presented might have some connections with practical applications related to the issue of disorganized mixing at low Reynolds numbers. Anticipating some results to be described in the next sections, it has been found that there is, among others, a regime where the prism oscillation creates what appears to be a disorganize wake. This could be of interest in the field of mixing enhancement because it has been found, Lee et al [65] and Olayiwola and Walzel [66], that forced pulsation of laminar flow may actually increase disorganized mixing and/or cross flow transport. In the case presented in this chapter, the disorganize regime is self sustained and no external system to generate flow pulses is needed, thereby contributing to the engineering simplicity of a potential working device.

The organization of the section is as follows: the experimental setup and the methodology are described first; then, results are presented and discussed and, finally, conclusions are presented.

4.2.1 Experimental Details

Experimental set-up

A closed-loop circuit, with water as the working fluid, was used to perform the experiments. The circuit components were: a) a primary closed tank,
b) a square section channel inserted into the tank that contained the test section with the tethered prism, c) a pump, d) a flow meter, e) a valve to control the mass flow rate, and f) a secondary open tank. A scheme of the circuit is shown in figure 4.1 with the exception of the square prism that now is a tethered square prism.

The same experimental set-up than as one described at the beginning of this chapter was used to perform the experiments, so flow meter, pump and tanks details were already explained in the experimental set-up section 4.1.1. In figure 4.2 one could see the tank and channel dimensions, as well as prism dimensions, that are 10 mm x 10 mm x 25 mm, so the blockage ratio was (as the previous experiment) 1:2.5. In this case the square prism was not fixed to walls but linked by two tethers to them. Hence the prism has one degree of freedom in its motion perpendicular to the fluid flow. Three different prisms were manufactured having different solid to fluid mass ratios: 0.56, 0.70 and 0.91 respectively. The prisms were manufactured in methacrylate as well. They were hollow and the size of their empty internal cavities determined their mass ratio. Two tethers, located at the two end span wise sections of the prism were used to allow for the desired motion. The tether’s length was 60 mm and the maximum measured peak to peak prism oscillation amplitude in the experiments was around 3 mm. This means that the maximum tether deflection angle away from the vertical direction was around 1.4 degrees, so the prism motion could be considered horizontal. The tethers were cut out of a nylon fish line so they are considered inextensible.

To quantify flow uniformity promoted by the honeycomb section, PIV measurements were also performed upstream of the prism. The room temperature of the experiments, that influences water viscosity, was 19 °C and was kept constant during the tests. The volume flow rate was varied in the range from 0.387 to 2.709 liters/min that yields to a Reynolds numbers defined based on the square prism cross section height and averaged inlet velocity from 100 to 700.

4.2.2 PIV measurements

The PIV set was the Dantec system that was described in subsection 4.1.1 therefore here they are only listed and not explained in detail. Flow illumination was provided by a pulsed Nd:YAG 800 mJ laser. Each laser pulse lasted for 10 μs. Images were taken using a Dantec Dynamics Flow Sense 2ME camera with a resolution of 1600 x 1200 pixels. The camera lens was a Zeiss Makro-Planar T* 2/50 mm ZF. The flow was seeded with Polyamid Seeding Particles of 5 μm (PSP-5). In any case, the particle response time was of the order of μs, as computed using the method described in previous chapters, which is well below the typical threshold characteristic time of the tests. Synchronization between image capturing and flow illumination and the analysis was carried out using the Dynamic Studio Dantec software.
PIV measurements were carried out in six different interrogation 2D areas located downstream of the tethered prism. A front view (looking from the “y” direction) of upstream areas is presented in figure 4.5. They were placed in the following planes: \( z = -12.5 \text{ mm} \), \( z = -6.5 \text{ mm} \), \( z = -1 \text{ mm} \), \( x = 12.5 \text{ mm} \), \( x = 6.5 \text{ mm} \) and \( x = 1 \text{ mm} \) respectively. It should be mentioned that the coordinate axes have changed from the last experiment since the position of the bottom left corner of the square prism is not fixed and therefore the axis has moved to a wall. All interrogation areas had the same physical dimensions: 80 mm x 25 mm, and each of them contained 1600 x 500 pixels. In all cases, each experiment was repeated three times to assess the repeatability of the results. Sampling of the flow field was carried out at a frequency of 15 Hz. Each sample was generated by processing the information associated to two laser pulses separated 5 milliseconds in time. Then, in the case of the highest Re (700), where the flow velocity was around 0.07 m/s, a particle would travel at the order of 0.35 mm between consecutive pulses in the stream-wise direction that is a distance much smaller (by two orders of magnitude) than the characteristic length of the problem (the prism cross section height of 10 mm). Each PIV area was divided into smaller sub-interrogation areas of 6 mm x 6 mm containing 128 pixels x 128 pixels each one that corresponds to 21 pixel per millimeter. The post-processing software allowed for a parallel self-consistent re-computation of the flow field in successive interrogation windows of 64 pixels x 64 pixels (3 mm x 3 mm), 32 pixels x 32 pixels (1.5 mm x 1.5 mm) and 16 pixels x 16 pixels (0.8 mm x 0.8 mm). That is, a velocity vector was computed for each window of 0.8 mm x 0.8 mm and this computation was consistent with the computations performed at the larger windows. The processing software also allowed for the possibility of performing flow computations at smaller windows of 8 pixel x 8 pixel (0.4 mm x 0.4 mm). However, these finer computations were not carried out because, in this case, the typical distance traveled by flow particles at the highest Re (0.35 mm) was of the same order of magnitude of the window size. Then, the spatial flow resolution of the experimental tests was of the order of 1 mm that is considered to be sufficient to describe the phenomena being analyzed.

Regarding the time resolution, it will be shown in a later section that the typical frequency of the large scale vortical structures being observed was in the range from 0.5 Hz to 3 Hz that corresponds to a characteristic time of about 0.3 s to 2 s. In this regard, each recording sequence consisted of 500 frames separated 0.067 s in time from each other (15 Hz) and lasting for a total of 33.5 s. Then, the characteristic time of the observed phenomena was of the order of at least 5 times larger than the sampling time and, also, 15 times smaller than the total recording time.
4.2.3 Results and discussion of the PIV measurements

Laminar duct flow in the clean configuration without the prism

First of all, the flow in the duct has been characterized without the prism as it is shown in subsection 4.1.2. So the reader is directed to that section in order not to repeat the analysis.

The main results are summarized here for clarifying purposes:

- The measurements done by the flow meter directly and the measurements done via post-processing the PIV velocity fields differs by less than 1%.

- Velocity profiles are fully developed as they are shown in figure 4.11 and follow almost perfectly the theoretic profiles described by Constantinescu [77] until Reynolds number 300. Thereafter the distance from the honeycomb to the square prism is not sufficient to fully develop the incoming flow. From equation 4.9 it is easy to see that the entrance length to see a Poiseuille velocity profile at Reynolds number equal to 700 (1750 based on channel hydraulic diameter) is more than 2 meters, which is difficult to reproduce with this experimental set-up. The PIV obtained velocity profile and the theoretical one (the one provided by Constantinescu [77]) is shown in figure 4.35.

- From figure 4.15 it is easily deduced that the square prism is not inducing upstream oscillations that could re-feed the oscillations downstream.
Figure 4.35. Comparison between Poiseuille velocity profile defined as in 4.8 and the PIV obtained profile for different Reynolds numbers.
Experimental results in the downstream region

The experimental study in the present chapter aims to identify different regimes of the system and the influence that parameters as Reynolds number or ratio between square prisms mass and displaced fluid mass have on it. In order to gather enough information, three square prism were manufactured. The ratio of densities is 0.56, 0.7 and 0.91. All of them tighten the tether due to buoyancy forces. A map of the different forces applied on the square prism is shown in figure 4.36. In this figure, $B$ is the buoyancy forces, $F_D$ and $F_L$ are the "y" and "x" component of the fluid forces applied on the prism and $T$ is the tension in the tether. In this case, the fluid force component in the x-direction is the lift force and the fluid force component in the y-direction is the drag force.

The equations for these forces are written in the next lines:

\[
B = (1 - m^*) m_d g,
\]
\[
F_D = \frac{1}{2} \rho V_b^2 c_D A,
\]
\[
F_L = \frac{1}{2} \rho V_b^2 c_L A,
\]
\[
T = (B + F_D) \cos \theta + F_L \sin \theta.
\]

(4.20)

where $m^*$ is the solid to fluid mass ratio, $m_d$ is the displaced fluid mass, $g$ is the gravitational acceleration, $V_b$ is the bulk velocity (mean stream-wise
4.2. **CONFINED 3D FLOW-INDUCED VIBRATIONS**

<table>
<thead>
<tr>
<th>(m^*)</th>
<th>B</th>
<th>((F_D/c_D)_{Re=137})</th>
<th>((F_D/c_D)_{Re=256})</th>
<th>((F_D/c_D)_{Re=700})</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.57</td>
<td>1.05E-2</td>
<td>2.3E-5</td>
<td>8.2E-5</td>
<td>6.1E-4</td>
</tr>
<tr>
<td>0.70</td>
<td>7.4E-3</td>
<td>2.3E-5</td>
<td>8.2E-5</td>
<td>6.1E-4</td>
</tr>
<tr>
<td>0.91</td>
<td>2.2E-3</td>
<td>2.3E-5</td>
<td>8.2E-5</td>
<td>6.1E-4</td>
</tr>
</tbody>
</table>

In this equation \(L\) is the tether length. Tether tension is changing as the prism is moving due to a little deflection caused by the finite length of the tether. An usual approximation assumes a net zero lift force that yields to a mean tension tether only influenced by the mean drag forces. Even so, as Reynolds number is varying from 100 to 700, bulk velocity is changing, so the mean drag forces change from one Reynolds number to another and therefore natural frequency of the system also changes. Table 4.6 shows values of these forces in order to get an idea of orders of magnitude.

In table 4.6, drag forces have been divided by the drag coefficient due to a lack of reference data on which this experiment should be based on. Drag coefficient may be markedly influenced by wall proximity and common values may not work. In any event a common value of drag coefficient of 2-3 turns drag forces into a force that cannot be neglected at high Reynolds number. The natural frequency without fluid flow, just buoyancy effects as the unique forces that contribute to the tether tension, has a value of 2.035 Hz.

Data in this experiment has been divided into two sections, one regarding fluid topology and the other regarding movement of the prism.

Fluid topology is studied first. As it has been said in the previous experiments, PIV data has been used to get information about the velocity fields in the surrounded areas of the square prism. Three different states could be observed in the flow topology while Reynolds number increase from 100 to 700 independently of the densities ratio. First a steady recirculation bubble appears for very low Reynolds number, then a sudden transition into a vortex-shedding regime appears on Reynolds number depending on solid to fluid mass ratio and finally at high Reynolds number the clean vortex-shedding gives way to a "dirty" vortex-shedding regime where there is not a unique frequency and this shedding is irregular from the qualitative point of view.
Figure 4.37. Vortex-shedding frequency as a function of Reynolds number for three different solid to fluid mass ratio.

These results have been deduced via different ways of analysis. Figure 4.37 shows the frequency of this Karman-style vortex shedding as a function of Reynolds number. The frequency has been obtained by the same manner as it was obtained in section 4.1.3.

As opposed to the previous case where the square prism was fixed to the channel walls, this time, frequency grows linearly as Reynolds number does. That suggests a constant Strouhal number as table 4.7 indicates. The mean value of the Strouhal number for fluid to mass density ratio equals to 0.57 is 0.2149 with an error less than 5% (first value has been omitted since not dominant frequency appears on the spectrum of Reynolds number 100. For solid to fluid mass ratios equal to 0.70 and 0.91, the mean Strouhal numbers are 0.2136 and 0.2160 respectively. It is also clear that Strouhal number is independent on not only Reynolds number but solid to fluid mass ratio.

The power spectral density of some points are shown in figure 4.38 for clarification. The top-left figure shows a white noise PSD. Bottom-right PSD has a peak at some frequency but the spectrum is not as clear as the other power spectral density where a clean Karman-street like vortex shedding is developing.

As it happened with the prism fixed at walls, the time-averaged of the stream-wise velocity in the center-line gives a nice view in order to find dif-
Figure 4.38. From top to bottom and left to right, power spectral density at some point upstream the bluff body for Reynolds number and densities ratio pair of (100, 0.56), (300, 0.56), (205, 0.70), (400, 0.70), (500, 0.91), (700, 0.91).
Table 4.7. Strouhal numbers versus Reynolds numbers for different solid to fluid mass ratios.

<table>
<thead>
<tr>
<th>Re</th>
<th>(St_{m^*=0.56})</th>
<th>(St_{m^*=0.70})</th>
<th>(St_{m^*=0.91})</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>0.99</td>
<td>0.32</td>
<td>0.19</td>
</tr>
<tr>
<td>137</td>
<td>0.21</td>
<td>0.19</td>
<td>0.23</td>
</tr>
<tr>
<td>150</td>
<td>0.16</td>
<td>0.24</td>
<td>0.24</td>
</tr>
<tr>
<td>185</td>
<td>0.25</td>
<td>0.25</td>
<td>0.24</td>
</tr>
<tr>
<td>195</td>
<td>0.26</td>
<td>0.27</td>
<td>0.24</td>
</tr>
<tr>
<td>205</td>
<td>0.30</td>
<td>0.25</td>
<td>0.24</td>
</tr>
<tr>
<td>256</td>
<td>0.23</td>
<td>0.24</td>
<td>0.21</td>
</tr>
<tr>
<td>300</td>
<td>0.22</td>
<td>0.22</td>
<td>0.21</td>
</tr>
<tr>
<td>350</td>
<td>0.22</td>
<td>0.11</td>
<td>0.22</td>
</tr>
<tr>
<td>400</td>
<td>0.22</td>
<td>0.21</td>
<td>0.20</td>
</tr>
<tr>
<td>450</td>
<td>0.20</td>
<td>0.20</td>
<td>0.16</td>
</tr>
<tr>
<td>500</td>
<td>0.17</td>
<td>0.27</td>
<td>0.19</td>
</tr>
<tr>
<td>550</td>
<td>0.22</td>
<td>0.18</td>
<td>0.21</td>
</tr>
<tr>
<td>600</td>
<td>0.15</td>
<td>0.21</td>
<td>0.22</td>
</tr>
<tr>
<td>650</td>
<td>0.15</td>
<td>0.16</td>
<td>0.20</td>
</tr>
<tr>
<td>700</td>
<td>0.25</td>
<td>0.19</td>
<td>0.22</td>
</tr>
</tbody>
</table>

Different regimes. Figure 4.39 shows this velocity as a function of "y" direction and the bubble length for this Reynolds number obtained from them when this time-averaged velocity cross the zero value.

All three solid to fluid mass ratios have the same behavior regarding averaged bubble length. The beginning has a positive slope until reaching Reynolds number 150 when the bubble length is considerably reduced in a sudden dramatic fall. In this stage the fluid flow has just started to shed vortices, what on average increases the stream-wise velocity at the final stages of bubble and obviously reduces the bubble length. While Reynolds number keeps on increasing, this averaged bubble length recovers an estimated value of half length it has at the beginning and then finally decreases slightly with a low negative gradient. It is worth pointing out the different ways of recovering depending on the densities ratio. A slow and gradual recovery is obtained for \(m^* = 0.56\) while the other two are more abrupt and faster in the sense of getting them at lower Reynolds numbers. All the aforementioned related to bubble length is corroborated in stream-wise velocity y-profile in adjacent figures. The recovering effect of the bubble length is related to the square prism motion. A preview of the results that will come later is that the prism, at very low Reynolds numbers, remains motionless, generating a whole bubble behind it. When lift forces are high enough, this prism acquires a harmonic motion and the bubble breaks. System behavior changes when the prism gives up in its oscillatory movement. That occurs at some
4.2. CONFINED 3D FLOW-INDUCED VIBRATIONS

Figure 4.39. On the top-left, the averaged bubble length is plotted against Reynolds numbers. Top-right, and bottom figures show show the time-averaged stream-wise velocity in the center-line for different Reynolds number with $m^* = 0.56$, $m^* = 0.70$ and $m^* = 0.91$ respectively (top to bottom and left to right).
Reynolds number dependent on the density ratio.

Maps of the velocity, vorticity, Q-maps and kinetic energy (with the mean velocity reference, il est, root mean square (rms) of the variation of the velocities) have been plotted in figure 4.40 for some interesting regimes. This gives a qualitative idea of the different regimes.

Figures 4.40 and 4.41 show the three different fluid regimes that have been detected. At the top of the first figure (4.40), a steady recirculation bubble appears. There is not time-variation of velocities and Q-maps only show little red zones that lead to the conclusion that no vortex is shed. The four middle figures show the classical vortex shedding together with a prism oscillating motion. It is easily seen how bigger red zones are dropping from the prism indicating vortex shedding. This red zone, mainly the widest ones, are in agreement with both velocities directions. The kinetic energy source now appears as two large red areas projecting from the prism sides. Finally, in the last four figures, fluid flow shed vortices as it dodges the prism but not as regularly as the last case. Vortex origins are also more difficult to place on Q-maps. Regarding figure 4.41, a cycle of a vortex shedding has been represented. Each column belongs to a time-equidistant snap. It should be emphasized how fast vortex shedding disappear caused by the vicinity of channel walls. Q-maps show that no more than four prism sides are enough to bring the vortex to an imperceptible plane.

Windows A and B, as shown in figure 4.17 have been taken as a reference to measure the mean rms of the stream-wise velocity for different Reynolds number and solid to fluid mass ratio. The results are in figure 4.42. As expected, window closer to the prism obtains higher rms as sensed from figures 4.40 and 4.41 since the main part of velocities variation are in the prism nearness. The sudden rise at low Reynolds numbers coincides with the beginning of vortex shedding. Window A suffers a higher jump, resulting on a higher rms until the prism stops oscillating and becomes "vibrating" at the same position, then drops to a value close to the nearly constant value of the rms at window B. Regarding solid to fluid mass ratio, highest $m^*$ gets the largest rms followed by $m^* = 0.56$ and $m^* = 0.70$. At any rate differences are not expected to be considerable enough to draw a trend.

Shedding frequency ($f_s$) and mean rms of the stream-wise velocity in window A ($V_{rms_A}$) have been transcribed into tables from 4.8 to 4.10 together with other parameters.

Regarding the square prism motion, videos of the prism have also been recorded in parallel for completeness purposes. An image post-processing software has been developed to get the amplitude and the frequency of the prism motion. This has been based on monitoring the position of the prism corner frame by frame (a post-processed frame is shown in figure 4.43).

There are basically three different prism motions depending on Reynolds numbers. For Reynolds numbers low enough, the prism remains quiet, then from a certain value of Reynolds number (depending on densities ratio) the
Figure 4.40. From left to right, instantaneous map of stream-wise velocity, span-wise velocity and Q-criterion and on the right border the root mean square of stream-wise velocity in the whole range of time.
Figure 4.41. Time-snaps on a oscillation cycle for $m^* = 0.91$ and Reynolds number equal to 300. From top to bottom, stream-wise velocity ($V$), span-wise velocity ($U$) and Q-maps. Black border square shows prism position in previous snap.
Figure 4.42. Mean rms on windows A and B as a function of Reynolds number for the three densities ratio.

Figure 4.43. Middle step of the video post-processing stage.
prism starts moving sinusoidally with an amplitude and a frequency clearly defined. This regime lasts a short interval on Reynolds number, and thereafter the prism motion becomes rotatory about the prism span-direction with not translational motion of the center of the square prism cross-section (referred to x-y 2D view).

This analysis yields to the result shown in tables from 4.8 to 4.10. $A_p$ is the peak to peak motion amplitude, $f_p$ is the frequency of the movement, $error$ is an error indicator calculated via $error = 100(|std_1 - std_2|/((std_1 + std_2)/2))$ being $std_1$ the standard deviation over all frames contained in the video recorded of the upper left corner x-position and $std_2$ the relative to the upper right corner x-position. The difference from the side length of the square prism (10 mm) and the time instantaneous subtraction of the intersections x-position is at any moment less than 5%.

If we join these two analysis we reach a states map on Reynolds numbers and solid to fluid mass ratios. Each state is defined by a conjunction of the

---

**Figure 4.44.** X-position of the upper left and upper right corner have been monitored versus time.
different regimes of the fluid flow and the prism motion. The main states are:

- **State 1**: characterized by the no motion of the prism (it remains in the same position during the tests) and by the steady recirculating bubble downwards the prism. In the velocity field there is noisy power spectral density, with more or less the same rms of the stream-wise velocity in windows A and B and a close to free flow Q-map.

- **State 2**: in this case while there is no prism movement, fluid topology has changed from State 1, and now a Karman-street like vortex-shedding. It is characterized by its shedding frequency and the rms of the stream-wise velocity. This state seems to be a transient one, since the Reynolds number interval is too narrow.

- **State 3**: this state is an interesting one distinguished by a Karman-street like vortex shedding in the flow field and a harmonic motion of the prism. It is in this state where vortex-induced vibration exists. It is possible to measure the frequency and amplitude of the prism motion and also the shedding frequency and rms of the velocities downstream of the prism and correlate with each other. Frequency (both shedding frequency and prism motion frequency) grows as Reynolds number does while remaining this state, but the amplitude of the prism motion has a maximum value on a certain Reynolds number, from where it starts decreasing.

- **State 4**: The frequency of the vortex shedding goes on increasing as Reynolds number grows, but the amplitude decay at some Reynolds number and the next prism motion regime starts. This is the one that instead of translation, the prism rotates around it. In this case, the power spectral densities are not as clean as the ones in State 3, but a high peak keeps dominating the spectrum. As Reynolds numbers go on growing, more frequencies start appearing in the spectrum and it begins to resemble a noise spectrum but still with some peaks. Regarding the motion of the prism, although it is mainly rotatory, some random translational movement appears and that yields to a pseudo-amplitude that is indicative of the number of these random movements.

- **State 5**: The last main state is characterized by completely irregular rotatory movement of the prism that still has some sudden translations and regarding to the flow topology, it is impossible to distinguish a peak in the spectrum, but the rms of the velocities remain high, which means that there is some kind of an irregular vortex shedding. Q-maps and kinetic energy maps show a totally different regime from the one belonging to the State 1 but not so different from the Karman-street like vortex shedding although there is not a dominant frequency.
Figure 4.45. States map.

Figure 4.45 presents the states map for different Reynolds number and solid to fluid mass ratios for a square prism in a 3D channel (square section) with a blockage ratio of 1:2.5.

4.2.4 Conclusions

Using the experiments, the existence of different states of a tethered square prism in a high blockage ratio 3D channel has been demonstrated. In these experiments, square prism movement is not only affected by VIV and possibly by the new concept developed by Semin et al. [75], confinement-induced vibration (CIV), but the walls perpendicular to the prism motion play an important role.

Five main regimes, considering both prism motion and fluid topology, have been identified varying the Reynolds number of the incoming flow (distilled water was the working fluid) between 100 and 700. Furthermore, three
different densities ratios were studied, 0.57, 0.7 and 0.91. Regarding flow topology, Karman street like appears at Reynolds numbers lower than the threshold for its fixed state (100-137 instead of 160-170). This regular vortex shedding goes on until reaching Reynolds number around 500 where the frequency spectrum of vortex shedding starts blurring and other peaks become relevant. Prism motion follows a similar progression, it starts motionless and begins its oscillatory movement somewhat after the onset of vortex shedding. This is followed by a transition to a bouncing motion around its span-wise axis at Reynolds number between 350 and 500.

The highest values of fluid rms are reached at state 2 (regular vortex shedding and an oscillating prism) with a linear increase in the frequency of the vortex shedding. This yields to a Strouhal number that is independent of Reynolds number and densities ratio.

The study of the influence of some parameters remains open and it will be the subject of future work. For example, the influence of the blockage ratio on both the states map and the amplitude and Strouhal numbers of the prism motion; or the effect that the perpendicular to prism walls exert over the flow topology varying its distance between them. An interesting analysis is the calculus of the fluid forces that perform on the prism over time. This will bring the instantaneous values of lift and drag coefficients and also an estimation of the natural frequency for different Reynolds numbers. This is possible as the velocity fields are known all over the channel but due to experimentally obtained values their derivatives should be filtered, smoothed or processed in any way in order to avoid noise amplification.
4.2.5 Summary of the results on VIV experiments.

Table 4.8. Summary results for $m^* = 0.57$

<table>
<thead>
<tr>
<th>$m^*$</th>
<th>Re</th>
<th>$V_{rms}$</th>
<th>$A_p$</th>
<th>$f_s$</th>
<th>$f_p$</th>
<th>$\text{error}_p$</th>
<th>State</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.57</td>
<td>100</td>
<td>0.241</td>
<td>0.027</td>
<td>0.0</td>
<td>0.5</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td></td>
<td>137</td>
<td>0.252</td>
<td>0.019</td>
<td>0.67</td>
<td>0.1</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td></td>
<td>150</td>
<td>0.235</td>
<td>0.040</td>
<td>0.64</td>
<td>2.5</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td></td>
<td>185</td>
<td>0.341</td>
<td>0.594</td>
<td>0.95</td>
<td>0.95</td>
<td>1.8</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>195</td>
<td>0.378</td>
<td>0.785</td>
<td>0.95</td>
<td>0.98</td>
<td>2.8</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>205</td>
<td>0.501</td>
<td>0.812</td>
<td>1.03</td>
<td>1.01</td>
<td>3.3</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>256</td>
<td>0.524</td>
<td>0.813</td>
<td>1.20</td>
<td>1.25</td>
<td>1.8</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>300</td>
<td>0.522</td>
<td>0.894</td>
<td>1.36</td>
<td>1.39</td>
<td>1.7</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>350</td>
<td>0.542</td>
<td>0.966</td>
<td>1.60</td>
<td>1.55</td>
<td>4.2</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>400</td>
<td>0.545</td>
<td>0.951</td>
<td>1.77</td>
<td>1.77</td>
<td>5.1</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>450</td>
<td>0.545</td>
<td>0.842</td>
<td>1.95</td>
<td>1.99</td>
<td>5.6</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>500</td>
<td>0.566</td>
<td>0.217</td>
<td>2.18</td>
<td>-</td>
<td>0.9</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>550</td>
<td>0.499</td>
<td>0.216</td>
<td>2.75</td>
<td>-</td>
<td>0.4</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>600</td>
<td>0.455</td>
<td>0.276</td>
<td>2.93</td>
<td>-</td>
<td>0.3</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>650</td>
<td>0.453</td>
<td>0.408</td>
<td>-</td>
<td>0.1</td>
<td>5</td>
<td></td>
</tr>
<tr>
<td></td>
<td>700</td>
<td>0.463</td>
<td>0.540</td>
<td>-</td>
<td>0.5</td>
<td>5</td>
<td></td>
</tr>
</tbody>
</table>
### Table 4.9. Summary results for $m^* = 0.70$

<table>
<thead>
<tr>
<th>$m^*$</th>
<th>Re</th>
<th>$V_{rms_A}$</th>
<th>$A_p$ [m]</th>
<th>$f_s$ [Hz]</th>
<th>$f_p$ [Hz]</th>
<th>$\text{error}_p$ [%]</th>
<th>State</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.70</td>
<td>100</td>
<td>0.198</td>
<td>0.017</td>
<td>0.7</td>
<td>0</td>
<td>0.1</td>
<td>2</td>
</tr>
<tr>
<td></td>
<td>137</td>
<td>0.189</td>
<td>0.239</td>
<td>0.66</td>
<td>0.85</td>
<td>2.6</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>150</td>
<td>0.264</td>
<td>0.608</td>
<td>0.73</td>
<td>0.73</td>
<td>2.1</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>185</td>
<td>0.341</td>
<td>0.814</td>
<td>0.86</td>
<td>0.88</td>
<td>2.7</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>195</td>
<td>0.378</td>
<td>0.883</td>
<td>0.89</td>
<td>0.94</td>
<td>2.8</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>205</td>
<td>0.501</td>
<td>0.827</td>
<td>0.95</td>
<td>0.95</td>
<td>3.1</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>256</td>
<td>0.524</td>
<td>0.863</td>
<td>1.16</td>
<td>1.16</td>
<td>3.1</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>300</td>
<td>0.520</td>
<td>0.884</td>
<td>1.32</td>
<td>1.35</td>
<td>3.3</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>350</td>
<td>0.542</td>
<td>0.131</td>
<td>1.80</td>
<td>-</td>
<td>0.8</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>400</td>
<td>0.545</td>
<td>0.144</td>
<td>2.06</td>
<td>-</td>
<td>1.1</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>450</td>
<td>0.545</td>
<td>0.185</td>
<td>-</td>
<td>-</td>
<td>0.3</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>500</td>
<td>0.557</td>
<td>0.283</td>
<td>-</td>
<td>-</td>
<td>0.2</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>550</td>
<td>0.499</td>
<td>0.492</td>
<td>-</td>
<td>-</td>
<td>2.0</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>600</td>
<td>0.455</td>
<td>0.493</td>
<td>-</td>
<td>-</td>
<td>2.7</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>650</td>
<td>0.453</td>
<td>0.256</td>
<td>-</td>
<td>-</td>
<td>0.5</td>
<td>5</td>
</tr>
<tr>
<td></td>
<td>700</td>
<td>0.463</td>
<td>0.340</td>
<td>-</td>
<td>-</td>
<td>0.7</td>
<td>5</td>
</tr>
</tbody>
</table>

### Table 4.10. Summary results for $m^* = 0.91$

<table>
<thead>
<tr>
<th>$m^*$</th>
<th>Re</th>
<th>$V_{rms_A}$</th>
<th>$A_p$ [m]</th>
<th>$f_s$ [Hz]</th>
<th>$f_p$ [Hz]</th>
<th>$\text{error}_p$ [%]</th>
<th>State</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.91</td>
<td>100</td>
<td>0.150</td>
<td>0.013</td>
<td>0</td>
<td>0</td>
<td>3.1</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>137</td>
<td>0.129</td>
<td>0.013</td>
<td>0.66</td>
<td>0</td>
<td>4.2</td>
<td>2</td>
</tr>
<tr>
<td></td>
<td>150</td>
<td>0.560</td>
<td>0.022</td>
<td>0.67</td>
<td>0</td>
<td>4.1</td>
<td>2</td>
</tr>
<tr>
<td></td>
<td>185</td>
<td>0.570</td>
<td>0.396</td>
<td>0.81</td>
<td>0.88</td>
<td>3.8</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>195</td>
<td>0.577</td>
<td>0.450</td>
<td>0.86</td>
<td>0.91</td>
<td>2.7</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>205</td>
<td>0.579</td>
<td>0.469</td>
<td>0.91</td>
<td>0.95</td>
<td>3.4</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>256</td>
<td>0.584</td>
<td>0.619</td>
<td>1.10</td>
<td>1.16</td>
<td>3.5</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>300</td>
<td>0.633</td>
<td>0.726</td>
<td>1.27</td>
<td>1.33</td>
<td>5.2</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>350</td>
<td>0.542</td>
<td>0.637</td>
<td>1.61</td>
<td>1.58</td>
<td>5.5</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>400</td>
<td>0.600</td>
<td>0.640</td>
<td>1.66</td>
<td>1.73</td>
<td>3.7</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>450</td>
<td>0.436</td>
<td>0.759</td>
<td>2.88</td>
<td>1.86</td>
<td>4.2</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>500</td>
<td>0.500</td>
<td>0.490</td>
<td>2.42</td>
<td>-</td>
<td>4.9</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>550</td>
<td>0.455</td>
<td>0.334</td>
<td>3.26</td>
<td>-</td>
<td>3.4</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>600</td>
<td>0.455</td>
<td>0.362</td>
<td>3.46</td>
<td>-</td>
<td>5.6</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>650</td>
<td>0.471</td>
<td>0.419</td>
<td>-</td>
<td>-</td>
<td>5.0</td>
<td>5</td>
</tr>
<tr>
<td></td>
<td>700</td>
<td>0.476</td>
<td>0.397</td>
<td>-</td>
<td>-</td>
<td>3.1</td>
<td>5</td>
</tr>
</tbody>
</table>
Chapter 5

ARTICLES PUBLISHED

During the development of the thesis, two articles have been published in major journals of engineering sector. These are:


Furthermore two more articles are under revision:


- M. Reyes, A. Velazquez, E. Martin & J.R. Arias, Experimental study on the 3D confined flow induced vibrations of a tethered prism at a high blockage ratio. *Journal of Fluid Mechanics*
Chapter 6

CONCLUSIONS

The following conclusions can be summarized from the experiments performed in this thesis:

- The effect of tip clearance was studied on a micro-channel flow based thermal control system. Thermal control system usually has some important restrictions owed to the design of the whole device in which the system is working for. One of these restrictions is that the flow can not be considered as a fully developed flow and that make the fluid dynamic problem difficult to relate with others research works. The present study has accounted for two parameters of practical interest, namely the heat transfer and the pressure drop (which has a direct impact on the pumping power and that affect the total cost of the device). Four configurations involving a tip clearance have been analyzed and compared to a baseline configuration of micro-channel flow without tip clearance. This baseline configuration consist on fifteen parallel micro-channels of fifteen mm of length and separated by a step of one mm. The micro-channel were square cross-section of five hundred microns of side. Tip clearance of half, once and twice times the height of the micro-channel were considered. Another configuration with the micro-channel rotated ninety degrees took side in the study. For each configuration, six different volume flow rates were considered. These flow rates, in the case of the baseline configuration, led to Reynolds numbers in the range from 416 to 2600, containing both laminar and transient regime flows. The main conclusion of the study is that implementation of tip clearance in active micro-channel based thermal control systems is an attractive option from the practical industrial application standpoint owing to two arguments: a) The added manufacturing cost is negligible since most of the manufacturing complexity is associated to the micro-machining of the micro-channels, while the top wall can be easily set at a lower or higher height with no extra cost of maintenance; and b) while saving large quantities of pumping
power due to a decrease on pressure drop, the heat transfer holds a great value relative to the baseline if tip clearance is implemented (in numbers, pressure drop is reduced to twenty percent of the original while the heat transfer rate is only lowered to eighty percent of the baseline.

- The behavior of a vapour chamber heat spreader for avionics thermal control purposes has been analyzed with and experimental and theoretical/numerical studies. It is well known that this kind of systems has a strong restriction on the weight and dimensions due to the high prices per kilogram and cubic meter that avionic devices deal with. So thermal efficiency has to be combined with a low system weight and reduced dimensions. From the thermal dissipation standpoint, it was found that vapour chamber based heat spreaders are more efficient than the equivalent metallic fin plates. However, although the same space has been occupied by both devices, heat spreaders are heavier than metallic fin plates. The benefit of the use of heat spreaders versus fin plates from the thermal efficiency point of view is three times better in natural convection conditions than in forced convection. This could be critical due to the strict regulation laws that this devices must pass, being one of them a standard heat dissipated in natural convection caused by an incident. Higher thermal efficiencies, in forced flow conditions, of the vapour chamber heat spreader can be achieved using higher fin height. However this comes at the unwelcome expense of widening the gap existing in between adjacent electronic boards inside avionics boxes, which translates into placing fewer boards per box. On the contrary, an attractive advantage is its robust off-design behavior, as the box topples ninety degrees either forwards or backwards, performance degradation is much less the its counterpart. The development of a theoretical/numerical model of the heat spreader, coupled to an optimization algorithm showed that it is possible to save weight changing the dimensions of our device while dissipating the same heat rate. The model also showed that the weight reduction rate does not scale linearly with the increase in component temperature (and cost).

- It has been shown that 3D confinement effects significantly change the behavior of the flow past a square prism in the laminar regime as it has been demonstrated in the previous experiments. Differences between the 3D confined and 2D unconfined cases are both qualitative and quantitative. Qualitatively, three different regimes have been identified in the case of a high blockage ratio as opposed to the 2D unconfined case. A steady recirculation bubble, a pulsating recirculation bubble, and a Karman-type vortex shedding regime are these cited regimes. Transition between the steady and pulsating recircula-
tion bubble regimes appears to be smooth, however, transition from the pulsating bubble to the vortex shedding regime shows indication of a narrow (in terms of the Reynolds numbers) intermittent regime where the length of the time averaged recirculation bubble has a sudden decrease and a slow recovery as Reynolds number increase. Specifically, results obtained suggest that the first regime lasts up to about Reynolds number 120, the pulsating recirculation regime covers the span from that Reynolds number to 150, where the intermittent regime stays up to 180, from where the Karman-type shedding starts. This results give an idea of how different is this behavior compared to the 2D unconfined case or also the 2D confined case as reported by other researches. The strong confinement in the span wise direction, which tends to be the rule in industrial applications, has a drastic effect on flow topology that should be accounted for when designing systems or products. Also, it is worth noting that the experimental results that have been presented (in particular, the sequence of regimes as a function of Reynolds numbers) bear a striking resemblance to the sequence of events that characterize a completely different problem (shallow turbulent wake flow) as described by other researchers. Whether this is by chance or it reveals a similitude of mechanisms leading to instability at the fundamental level is something that might be an interesting subject for study in the future. An interesting result of the experiments that were carried out is that once the shedding regime starts, the shedding frequency remains nearly constant, and that yields to an almost invariable product of Reynolds number by Strouhal number for a span of Reynolds numbers from 170 to 256 and this is, again, in contrast to the results reported by other researcher in the 2D confined case where a steady increase of the cited parameter as a function of Reynolds number is the norm. This fact also has some implications for practical engineering designs being that increasing the flow rate (and, thereby, increasing the pressure drop and the pumping power as well) does not guarantee a significantly larger unsteadiness in the flow field that may contribute to increase, for instance, the transport of a passive scalar if the intention is to promote mixing. Finally, regarding the effect of wall proximity, the numerical results which has been borne out with the experimental data, show two clear limit regimes as a function of the prism blockage ratio, a 2D regime or quasi-2D regime and the regime where transition to shedding is significantly delayed. Surprisingly, transition between these two regimes appears to occur within a narrow span of the blockage ratio parameter, instead of being of a smooth nature.

- The existence of different states of a tethered square prism in a high blockage ratio and under a strong influence of three-dimensional effects
has also been demonstrated doing some experiments with PIV (Particle Image Velocimetry) technique. The flow topology and prism motion is highly influence by not only the VIV and maybe CIV (confinement-induced vibration) phenomena but also the closeness of the walls (both those devoted to the confinement and those devoted to the three-dimensional effects). Five interesting regimes were identified while the incoming flow Reynolds number were varied between 100 and 700. This regimes are a conjunction of a flow stage and a prism stage. Different flow states were nearly the same as those obtained with the prism fixed to the walls, these were steady recirculation bubble, Karman street like vortex shedding and a fuzzy vortex shedding. Regarding prism motion, another three different stages were observed, a motionless state, an oscillating movement with a particular frequency and amplitude and a bouncing motion around its span-wise axis. This states line has been widespread to three different square prism to fluid mass ratio, 0.57, 0.7 and 0.91. The highest values of fluid velocities variations (rms) were reached at state 2 (regular vortex shedding and an oscillating prism) with a linear increase of the frequency of the vortex shedding and an amplitude nearly constant (effect of walls prevents from larger amplitudes). This yields to a Strouhal number that in not dependent on Reynolds number neither densities ratio. The study of the influence of some parameters remains open due to is/will be the subject of present/future work. That is for example the study of the influence that the blockage ratio exert on both the states map and the amplitude and Strouhal numbers of the prism motion; or the effect that the perpendicular to prism generates over the flow topology varying the distance between them. An interesting analysis is the calculus of the fluid forces that perform on the prism over time. This will bring the instantaneous values of lift and drag coefficients and also an estimation of the natural frequency for different Reynolds numbers.
Bibliography


BIBLIOGRAPHY


[32] R. Revellin, R. Rullière, F. Lefèvre, J. Bonjour, Experimental validation of an analytical model for predicting the thermal and


