UNIVERSIDAD POLITÉCNICA DE MADRID

ESCUELA TÉCNICA SUPERIOR DE
INGENIERÍA AERONÁUTICA Y DEL
ESPACIO

DOCTORAL THESIS

**Enhancing scalar transport in confined
laminar flows**

FRANCISCO SASTRE MATESANZ

INGENIERO AERONÁUTICO

January, 2019

UNIVERSIDAD
POLITÉCNICA
DE MADRID

DEPARTMENT OF FLUID MECHANICS AND
AEROSPACE PROPULSION

in the

ESCUELA TÉCNICA SUPERIOR DE INGENIERÍA
AERONÁUTICA Y DEL ESPACIO

DOCTORAL THESIS

Enhancing scalar transport in confined laminar flows

Author:

Francisco SASTRE
MATESANZ
INGENIERO
AERONÁUTICO

Supervisor:

Dr. Angel VELAZQUEZ
LOPEZ
DOCTOR INGENIERO
INDUSTRIAL

Universidad Politécnica de Madrid

January, 2019

Tribunal nombrado por el Sr. Rector Magfco. de la Universidad Politécnica de Madrid, el día.....de.....de 20....

Presidente:_____

Vocal: _____

Vocal:_____

Vocal:_____

Secretario:_____

Suplente:_____

Suplente:_____

Realizado el acto de defensa y lectura de la Tesis el día.....de.....de 20 ...
en la E.T.S.I. /Facultad.....

Calificación

EL PRESIDENTE

LOS VOCALES

EL SECRETARIO

“Research is what I’m doing when I don’t know what I’m doing.”

Wernher von Braun

Abstract

Enhancing scalar transport in confined laminar flows

by Francisco SASTRE MATESANZ

Experimental research is carried out to shed light over the kind of some flow topologies and mechanisms that may enhance mixing in confined flows. The purpose is to understand said mechanisms to find suitable ways of optimization and application for the industry. A summary of results has been published in journals that appear in the Journal Citation Reports.

In the first place, enhancement caused by flow induced vibration in a channel with high blockage ratio is studied. Research was carried out both with tethered (moving) prisms and with a fixed one. It is to be noted that passive scalar transport, such as temperature, is a goal of the industry as a whole, especially when dealing with high density fluids or small scales, which leads to low Reynolds' numbers and thus laminar flows. The ability to evacuate heat from warm walls and mix thermal boundary layer is studied at Reynolds' numbers between 80 and 800, based on the prism cross section.

In a second place, a previously tested geometry for a finned heat sink is studied so as to comprehend the causes of its good performance. A lid is placed against square section channels connecting two stagnation chambers. A pump induces enough pressure drop so as to the fluid to circulate along those channels and theoretically evacuate heat from them. Tip clearance effects are studied, which can be summarised in reducing the pressure losses a factor of five times more than the heat extraction losses. All the experimental results are compared to CFD simulations which are also used to provide further knowledge of the problem. Needless to say that the application of more efficient cooling systems is a very broad field.

Acknowledgements

To begin with, I have to express my most sincere gratitude to my Thesis Advisor, Professor Ángel Velázquez. Firstly, along with Professor Jose Luis Montañés, for having placed his trust in me to carry out the research project and this Thesis, but foremost for having been a "*sensei*" for me during these unique years, for guiding me and for his professional, close and cordial advise whenever I needed it. In the same way, I would like to thank Professor Juan Ramón Arias for his advice, help and friendly manner. Also to Professor Antonio Barro, for his knowledge and inspiring ideas. Of course to our colleagues from Vigo, Professor Elena Martín and Alejandro Valeije, for their dedication, valuable collaboration and hard work. Without all of them, this Thesis would have simply not been possible.

I also feel very grateful to laboratory heads Miguel and Ricardo de las Heras, because their always diligent help with the technical aspects of the assemblies has been immeasurable.

In the personal level, I am and always be grateful to my parents, and I would like to express my endless affection towards them, basically for being the best parents a son could expect. Their guidance in life is what has led me to be where and what I am, thus, the accomplishment of this Thesis is just the last item in an infinite list of them.

I also have to address my love and a very special acknowledgement to my girlfriend Kateryna. She has been my companion and inspiration all these years, and a fundamental support in my daily life. The achievement this Thesis means for me, is doubtlessly part of hers also.

Last but not least, I must mention that this path would have been highly untamed without the encouragement and spirit of my laboratory colleagues: Dr. David Vicente (strategy advisor), Dr. Ji Xu (code guru), César Enriquez (political advisor), Sergio Henriquez (human database), David Gutierrez (L^AT_EX consultant), Óscar Santiago (cultural provider), Rafel Clemente (expert in 3D printers), Eloy Millán (Galician sage), and others.

My utmost gratitude for everyone.

Contents

Abstract	v
Acknowledgements	vii
Introduction	1
1.1 Motivation	1
1.2 Basic aspects of mixing	2
1.2.1 Mixing in microdevices	5
1.3 Literature review	5
1.3.1 Industrial aspects	5
1.3.2 Mixing techniques	7
1.3.3 The confined cylinder problem	9
1.3.4 Tip clearance based mixing enhancement	12
Thesis plan	17
2.1 Flow induced vibration mixing enhancement	17
2.2 Tip clearance based mixing enhancement	18
3 Flow induced vibration mixing enhancement	19
3.1 Introduction	19
3.2 Experimental setup	21
3.2.1 The test channel	22
3.2.2 The test section	24
3.2.3 Auxiliary systems	29
3.2.4 The Particle Image Velocimetry (PIV) system	31
3.3 Definition of the experimental campaigns	31
3.4 Experimental results	32
3.4.1 EC #1 campaign	32
3.4.2 EC #2 campaign	34
3.4.3 EC #3 and EC #5 campaigns	40
3.4.4 EC #4 and EC #6 campaigns	44
3.5 Discussion	45
3.6 Conclusions	55
4 Tip clearance based mixing enhancement	59
4.1 Introduction	59

4.2	Experimental setup	62
4.3	Description of the flow solver and computational domain	70
4.4	Comparison between experimental and numerical isothermal results	76
4.5	Numerical results for the thermal cases and discussion	81
4.6	Experimental transitional flow rates	93
4.7	Conclusions	95
5	Conclusions	97
6	Published articles	99
7	Future work	101
	Bibliography	103

List of Figures

3.1	Sketch of the experimental setup.	21
3.2	Assembly of the pieces conforming a standard channel section	22
3.3	View of the test channel and primary (bottom) and secondary (top) tanks. The heated section (90 mm height) can be observed midway along the channel where the origin of coordinates is placed. Distances are expressed in mm.	23
3.4	View of the discharge zone in the secondary tank	24
3.5	Close-up view of the test section (left) and of the centerline plane (right). Distances are expressed in mm. . .	24
3.6	Heating section detail.	25
3.7	Numbered wall temperature control thermocouples. . .	25
3.8	Cartridge sample.	26
3.9	Electrical layout.	26
3.10	Thermocouple sample.	27
3.11	Position of the 9 points (located downstream of the aluminium block) where water temperature is measured. The view is from the top of the channel into the test section. The prism is shaded in a darker grey mesh. Measurements are expressed in mm.	28
3.12	Test section with explanatory thermocouple.	29
3.13	Pictures of the test section (notice that the outside of the aluminium block is isolated with low thermal conductivity methacrylate panels) and two prisms with the attached tethers.	29
3.14	Overview of the experimental setup.	30
3.15	Average value of the 100 measured velocity profiles for Re 80, 160, 240 and 320 (solid lines), their maximum spread (grey bands), and the theoretical profiles (dashed lines).	33
3.16	Histogram of the 100 centreline velocity measurements around the mean for Re 80 (upper row) to 360 (lower row), in isothermal clean channel.	34

3.17	Counterpart of figure 3.15 for the case of heated walls (no theoretical results are available in this case)	35
3.18	Histogram of the 100 centreline velocity measurements around the mean for Re 80 (upper row) to 360 (lower row), downstream of the heated section and clean channel.	36
3.19	Reconstructed temperature field in a plane located 20 mm downstream of the aluminium block for Re 80 and 320.	38
3.20	Sketch showing the PIV interrogation window in the channel.	41
3.21	Forces acting on the prism.	42
3.22	Time averaged "z" velocity profiles for Re 200 and 733. Results are presented in planes $z=115$ (left) and $z=140$ (right) for the fixed prism and, also, for the case with $m^*=0.91$	43
3.23	Power spectra density plots of the downstream velocity in the channel centreline for the isotherm fixed prism at Re 200 and Re 733.	44
3.24	Counterpart of figure 3.22 but considering now the thermal case.	45
3.25	Counterpart of figure 3.25 but considering now the thermal case.	45
3.26	Electric power supplied by the DC unit to the aluminum walls (heat transfer to the fluid) as a function of Re and m^* for constant wall temperature. The case of the fixed prism is also included for comparison purposes.	46
3.27	Nusselt vs Reynolds for constant wall temperature. . .	47
3.28	Temperature increase in degrees with entry as reference. First row Re80, last row Re360. First column $m^*=0.56$, second column $m^*=0.91$	49
3.29	Time-averaged PIV plots of the dimensionless stream-wise (U/U_{mean}) and cross-wise (V/V_{mean}) velocities at Re 200, for EC #4 (the thermal fixed prism) and EC #6 (the thermal moving prism with $m^*=0.91$).	51
3.30	Average aluminium wall temperature as a function of Re and m^* once the threshold level of electric power supply (315 W) has been reached. The case of the fixed prism is also included for comparison purposes.	52
3.31	Nusselt vs Reynolds for constant wall heat flux.	53
3.32	Analogue to 3.28. First row Re533, last row Re800. First column $m^*=0.56$, second column $m^*=0.91$	54
3.33	RMS in time for 30 points in the flow field.	55

4.1	Sketch of the experimental platform for configuration C0.	62
4.2		63
4.3	Photograph of the device for configuration C0 (left), and honeycomb and grid inserted into the upstream stagnation chamber (right).	64
4.4		65
4.5	Whole layout of the experimental setup.	67
4.6	Uncertainty bands and histograms of actual measurements for case Q4, C2 at sections S_{-1} , S_0 and S_1 . Histograms taken at a vertical coordinate 20 mm above the lowest surface of the stagnation chamber. U_∞ is the mean velocity (flow rate divided by the actual cross flow area).	69
4.7		73
4.8	Thermal boundary conditions for computational domains C0 (left) and C2 (right). Constant temperature walls depicted in red colour, adiabatic walls in grey color.	73
4.9	Surface and colour map of section S_{-4}	75
4.10	Comparison experimental (blue) vs normalized (red) vs mapped (green) entries.	76
4.11	Comparison between measured and computed x velocity profiles for section S_{-1} (stagnation chamber) and planes Pl1, Pl2 and Pl3 for configuration C2 at a volume flow rate Q4. Numerical results: red line. Experimental results: blue line with spread bands.	77
4.12	Counterpart of Figure 4.11 but, now, for the flow along a channel centre line. Numerical results: red line. Experimental results: blue line with spread bands.	78
4.13	Counterpart of Figure 4.11 but, now, for the flow along the top of a fin. Numerical results: red line. Experimental results: blue line with spread bands.	79
4.14	Current lines in C2-Q4.	81
4.15	Nu versus flow rate for configurations C0 to C3.	82
4.16	Pressure drop versus flow rate for configurations C0 to C3.	83
4.17	Local Nusselt number as function of Graetz number.	86
4.18	Dimensionless transverse velocity plots for configurations C0 to C3 and flow rate Q4 at section S_0	88
4.19	Counterpart of Figure 4.18 at section S_1	88
4.20	Temperature plots for configurations C0 to C3 and flow rate Q4 at section S_1	89
4.21	Sketch showing the key used to name walls in figures 17 to 20 below.	90

4.22	Local heat flux at the walls as a function of x for the case C0-Q4.	90
4.23	Local heat flux at the walls as a function of x for the case C1-Q4.	91
4.24	Local heat flux at the walls as a function of x for the case C2-Q4.	91
4.25	Local heat flux at the walls as a function of x for the case C3-Q4.	92
4.26	Flow streamlines (for the C2-Q4 thermal simulation) coloured by the transverse velocity defined in relation 4.10.	93
4.27	Device C1.	94
4.28	Device C2.	95

List of Tables

3.1	First three columns: coordinates (in mm) of the inserts where the thermocouples were placed. Other columns: measured time-averaged temperature ($^{\circ}\text{C}$) of the 10 thermocouples inserted into the aluminium block (T1 to T10) for some representative Re. Average temperature, T_{av} , standard deviation, T_{st} , and the maximum deviation, ΔT_{max} , are also included.	28
3.2	Measured electric power supply and Nu as a function of Re for experimental campaign EC #2.	40
3.3	Nu as a function of Re and m^* for the constant wall temperature case.	47
3.4	Detailed temperature measurements ($^{\circ}\text{C}$) of the 10 thermocouples (T1 to T10) inserted into the aluminium walls as a function of Re for the fixed prism and the case with $m^*=0.91$	52
3.5	Nu as function of Re and m^* for the constant wall heat transfer case.	53
3.6	RMS of the dimensionless water temperature for the case of constant wall temperature as a function of Re and m^*	54
3.7	RMS of the dimensionless water temperature for the case of constant wall heat transfer as a function of Re and m^*	55
4.1	Volume flow rates (Q1 to Q4) in litres per hour used in the experiment, and associated Re for configuration C0.	66
4.2	Mesh sensitivity analysis for the thermal case C2-Q4. Three meshes (#1, #2, and #3) were considered. The finer mesh #3 was assumed to be the reference mesh. Table shows mesh data (Nc: number of cells and size of elements), relative discrepancy of the streamwise velocity profile at S_{-1} (ϵ_1), and relative discrepancy (ϵ_2) of the total heat transfer.	74

4.3	Comparison between experimental and numerical pressure drops (inlet to outlet water lines) for the isothermal configuration C0. Discrepancies (ϵ_3) are expressed in percentages.	79
4.4	Cross section area, CS, mean velocity, U_∞ , Re, hydraulic diameter and computed (ΔP_{CFD}) and estimated (ΔP_{est}) pressure drops for the 16 cases (C0 to C3, Q1 to Q4) that have been considered.	84

List of Abbreviations

A_{hw}	Hot wall area [m^2]
A_w	Wall area [m^2]
$\vec{B}$	Buoyancy force (N)
C	Configuration
c	Channel cross section [m]
C_p	Specific heat [J/kgK]
CS	Cross section [cm^2]
D_H	Hydraulic diameter [cm]
EC	Experimental campaign
H	Channel height [mm]
H_p	Electrical heating power [W]
k_{film}	Thermal conductivity at T_{film} [W/mK]
$\vec{F}_{Cd}$	Drag force (N)
$\vec{F}_{Cl}$	Transversal force (N)
h	Heat transfer coefficient [W/Km^2]
k	Thermal conductivity [W/Km]
L	Channel length [m]
L_{Th}	Tether length [m]
m_p	Prism mass [Kg]
m^*	Density ratio
N_c	Number of cells
Nu	Nusselt number
P	Pressure [Pa]
Pr	Prandtl number
PSD	Power Spectral Density [$1/Hz$]
Q	Volume flow rate [l/h]
Q_w	Heat transfer rate [W]
Re	Reynolds number
RMS	Root mean squared deviation
T	Temperature [K]
$\vec{T}$	Tether tension (N)
T_{av}	Average temperature [$^{\circ}C$]
T_{in}	Inlet water temperature [$^{\circ}C$]
T_{film}	Water film temperature [$^{\circ}C$]
T_{out}	Outlet water temperature [$^{\circ}C$]

T_{st}	Temperature standard deviation [$^{\circ}\text{C}$]
T_{wall}	Hot wall temperature [$^{\circ}\text{C}$]
ΔT_{max}	Maximum temperature difference [$^{\circ}\text{C}$]
TC	Tip clearance [mm]
u	Streamwise velocity component [cm/s]
u_m	Section mean velocity [m/s]
v	Velocity vector [cm/s]
W	Wall
x	Spatial coordinate [m]
y	Spatial coordinate [m]
z	Spatial coordinate [m]

List of Symbols

α	Angle [rad]
ε	Deviation
ρ	Flow density [kg/m^3]
μ	Flow viscosity [kg/ms]

Dedicated to my parents.

Introduction

1.1 Motivation

Many different facets has the concept of homogenizing the heterogeneous: mixing, heat transfer, passive scalar transport, etc., being all of them different sides of the same coin. Enhancement of those is a very active field of research, not only because its intrinsic theoretical, academic and scientific interest but also because its countless applications in a myriad of sectors in the industry. As for what to fluid flows concerns, it has special significance the so-called laminar regime, which is characterized by its ordered stream and thus, by its slow characteristic time for mixing processes.

When approaching the problem from a engineering standpoint, we seek two fundamental objectives: enhancement of mass/heat transfer and optimization of the system. Each one of the two main experiments in this thesis is related to those goals.

In which is referred to mixing enhancement in confined laminar flows, methods used by the industry can be arranged into two big groups: active and passive. The first one imply the input of external work to the flow by means, for example, of a whirling propeller. The second one commonly make use of complex geometries, but there is also the possibility of using the interaction between flow and structure to produce a self sustained motion that achieves the objective of enhancing mixing.

The design of a mixing system has usually two main subsystems: the mass transporter/heat exchanger and the pump. As it has been mentioned, one of the main purposes sought by this thesis is the optimization itself of this type of system. It has been studied previously that the presence of small tip clearances in fin-based heat exchangers can decrease the pressure drop dramatically while keeping a reasonable rate of heat evacuation. The goal of the second experiment is

to acquire deeper knowledge of the flow field and three-dimensional structures that may be helping to keep the heat exchange rate while allowing the pressure drop to improve drastically, as well as to optimize the system in base to that knowledge.

1.2 Basic aspects of mixing

"Sometimes I had to spend a whole day mixing a boiling mass with a heavy iron rod nearly as large as myself." (Marie Curie).

Mixing is a process that takes place in the nature itself. It has also been taking a fundamental place in human history as well as in our quotidian life. Indian steel-makers used adequate mixing of several kind of impurities, maybe knowingly or non intendedly, in iron ore which can catalyse carbon nanotube growth, that filled with cementite later in the process of forging created the famous and feared pattern of Damascus steel more than 2000 years ago, M. Reibold and P. Paufler [1]. We make use of mixing "*know-how*" on a daily basis, from the very moment we stir coffee with milk or sugar instead of waiting for it to get mixed by itself.

The most basic definition of *mixing* could be: "*the reduction of inhomogeneity, which can be of concentration, phase or temperature and whose usual objectives are mass transfer, reactions, product properties or refrigeration*", E.L. Paul and Atiemo-Obeng [2]. More technically, we could define mixing as the process whose main objective is to reduce the length scales of two or more species below a certain threshold, Sturman and Ottino [3]. Mixing is rarely a process with a sole intended effect, thus, categorisation of mixing is difficult. However, effectiveness of mixing can be assessed from an engineering standpoint in the context of the quality of the final product [4], and also from a more mathematical point of view: streamline crossing is a central kinematic feature of "*good mixing*". In [3] the phenomenon of mixing is divided into three stages, of which only the first one always takes place:

- Mechanical mixing: stretching and folding motion causes the interfaces between the species to elongate and form filaments.
- Break-up: when the filaments are sufficiently stretched, differences in the interfacial tension on opposite sides can cause it to break. This step is usually absent in fluids with high viscosity.

- Molecular diffusion: if the fluids are miscible, Brownian motion of individual molecules act to homogenize the fluid at molecular scale.

As R.O. Grigoriev states in [5], when analysing the physical nature of mixing the first logical step to begin with is the advection-diffusion equation:

$$\frac{\partial c}{\partial t} = -\vec{v} \cdot \nabla c + D \cdot \nabla^2 c \quad (1.1)$$

Where c is the solute species concentration, v the velocity and D the diffusion factor. D for spherical particles in a fluid medium is defined by the Stokes-Einstein relation:

$$D = \frac{k_B \cdot T}{6\pi\mu r} \quad (1.2)$$

where k_B is the Boltzmann's constant, T the temperature, μ the viscosity of the medium and r the radius of the particles of solute.

For liquids soluble in other liquids it applies that $D \sim 1/\mu$.

When speaking of temperature diffusion the equivalent for 1.1 is the heat transport equation:

$$\frac{\partial T}{\partial t} = -\vec{v} \cdot \nabla T + \alpha \cdot \nabla^2 T \quad (1.3)$$

where temperature T replaces concentration and thermal diffusivity α replaces D :

$$\alpha = \frac{k}{\rho c_p} \quad (1.4)$$

where k is the thermal conductivity, ρ the density and c_p the heat capacity at constant pressure.

When looking at equations 1.1 and 1.3 we observe that the evolution of mixing in time can be governed by momentum, diffusivity or

both. When it comes to compare them, it is defined the Peclet number for mass and thermal diffusion:

$$Pe_m = \frac{UL}{D} \quad ; \quad Pe_T = \frac{UL}{\alpha} \quad (1.5)$$

There are two more relevant numbers to define: Reynolds and Prandtl. Reynolds number compares the relevance of convective to viscous terms in the momentum equations, as Prandtl compares the velocity of propagation of momentum to heat.

$$Re = \frac{UL}{\nu} \quad ; \quad Pr = \frac{\nu}{\alpha} \quad (1.6)$$

Naturally, the three of them relate in the way: $Pe = Re \cdot Pr$. The last relation to introduce is the so-called Schmidt number, that compares the ratio of previous transport numbers: $Sc = Re/Pe = \nu/D$, which can be interpreted as the ratio between the speed of propagation of momentum and of concentration.

For more clarity, let's see a very familiar example given by Sturman and Ottino in [3]: milk in a coffee cup. As characteristic length is chosen the typical diameter of a cup $L \sim 4 \text{ cm}$, stirring 1 turn per second yields a characteristic velocity of $U \sim 5 \text{ cm/s}$. Adding that water has a kinematic viscosity of $\sim 0.01 \text{ cm}^2/\text{s}$, the Reynolds is $Re \sim 2000$, which means that advection terms are dominant over viscous ones. Also, knowing that the diffusion coefficient of small molecules in water is $D \sim 1e-5 \text{ cm}^2/\text{s}$, Peclet number yields $Pe \sim 2e6$. Both indicate that a few turns of the spoon (advection/mechanical mixing) will get the job done. Instead, if we just poured the milk in the coffee cup, assuming we could do this without any kind of input of momentum, the characteristic time for the molecules of milk to cross the characteristic length would be around $t_{dif} \sim L^2/D = 1.6e6 \text{ s} \sim 20 \text{ days}$. Diffusion is still taking place and is the mechanism that refines the mixing at the end, but is the job of the mechanical mixing to reduce the characteristic lengths of the problem to a point in which the characteristic time of diffusion is comparable to the one of advection.

1.2.1 Mixing in microdevices

In a similar way as before, through a rough analysis of an illustrative example we find that in typical microfluidic applications like a small cylindric vessel whose characteristic cross section (x, y) is $h \sim 200 \mu m$ and characteristic axial (z) velocity is around $U \sim 1 \text{ cm/s}$, and assuming we are working with water or a fluid with a similar kinematic viscosity and diffusion coefficient, Re number will be $Re \sim 2$ which means that viscosity plays a central role in this kind of devices. Mass transfer Pe numbers will range around $Pe_m \sim 2000$ ($Pe_T \sim 1400$) which means that advection is still dominant over diffusion when it comes to mixing. If like in the given example before we assumed that the only mixing mechanism is diffusion, typical time scales would be $t_{dif} \sim h^2/D = 40 \text{ s}$, which at the given velocity would scale the device up to $L \sim 40 \text{ cm}$ at least ($D \sim 1e-5 \text{ cm}^2/\text{s}$ is a value for small molecules). In the very first place slenderness ratios $\sim 40/0.02 = 2000$ are definitely not acceptable for typical applications, not to mention that this fact would trigger other phenomenon like Taylor dispersion (shear flow makes the diffusion to be more prone streamwise than transversally) which would stop mixing from being accomplished.

At the light shed by the given example, it is straightforward that mechanical mixing is fundamental in microdevices in order to reduce the scales of diffusion. R. Mezaki states in his book [6] that there is striking shortage of mixing data for laminar high viscosity or non-newtonian fluids.

1.3 Literature review

1.3.1 Industrial aspects

In their book "*Mixing in the process industries*" Nienow and Edwards [7] state that long are gone the days when the process of mixing was left exclusively to the experienced worker who had the "feel" for the product and the academic interest was limited to the power consumption of the paddle mixers as an example of the application of the dimensional analysis.

Tatterson and Calabrese [8] summarize after a industrialist and academics meeting that the cost due to inadequate understanding of mixing for the process industries in the U.S. was of the order of 1000 to 10000 million \$ per annum, which translates typically to around 5 % of the losses in a chemical company. Sometimes it can even represent a cost of opportunity, because poor mixing prevents new products to reach the market, due to low yield costs or unacceptable product quality.

Liquids, solids and gases have to be mixed in all combinations to satisfy very different processes or product quality requirements. Mixing is a central matter in a very large amount of industries which cluster an incommensurable number of companies:

- Chemicals
- Pharmaceutical
- Petrochemical
- Biotechnology
- Polymer processing
- Paints
- Cosmetics
- Food processing
- Paper industry
- Waste water treatment
- Mineral processing

It does not only have an economic relevance. In the food processing industry exist several regulations since 2006 in the E.U. (Regulation (EC) No 1925/2006) and in 2007 in the U.S. the Food and Drug Administration (FDA) issued a final rule for current good manufacturing practices (CGMPs) for dietary supplements, which compel to the companies to meet certain standards of purity, strength or composition in which an adequate mixing plays a core role. In a much stricter manner, it is fundamental in the pharmaceutical industry that the active components are properly distributed in a carrier or excipient.

Industrial mixing is a mature academic field and, thus, there exist several books exclusively focused not only in the industrial applications, such the already mentioned [7] and others like Funt [9], Paul [2], Aamo [10], Cullen [4] and Mezaki [6]; but also in the physical and mathematical nature of mixing: Bockhorn [11], Sturman [3] and Grigoriev [5].

1.3.2 Mixing techniques

There are several techniques of mixing available for the industry. Traditionally, use of the so-called agitators was the most common, normally consisting in a mixing tank with a moving impeller. Lately static mixers have proved not only their comparable efficiency but also the capability to deal with a continuous input, and thus continuous processing, but also the advantages emerging from their lack of moving parts: they are energy efficient, their impermeability and, in the food and pharmaceutical industries, their sanitary design. A review of static mixers in the industry is presented in Thakur [12].

The focus of this thesis is to study mixing phenomena, specifically mixing of passive scalars, in the laminar flow regime. This regime is typical of small length scales and/or high viscosities. It is known that turbulent regime enhances mixing because of its intrinsic chaotic nature, while on the contrary it is typically difficult to oblige streamlines to cross in laminar regime. This field has been thoroughly studied in the last years.

The physics of laminar mixing has been analysed intensely through the last years. The aim of purely theoretical models is to provide with valid new ideas to be tested experimentally or numerically. Some representative studies of these are Brancher and Leprevorst [13], who propose the superimposition of a twisted quadripolar Stokes flow to a Poiseuille flow to create chaotic advection; Hitt and McGarry study in [14] the effects of pulsatile flow, concluding that high frequencies can lead to interfacial instabilities; Bretonnet [15] studies the influence of variable density in the shear layer of laminar mixing, due to different compounds (mass transfer) or different temperature (heat transfer), which can create a transverse velocity profile or "deflection" of the flow; in Chinesta [16] the microstructure evolution during liquid-liquid laminar mixing is studied from the standpoint of the area tensor.

Two large groups can be differentiated among mixers, namely, active and passive methods. Active systems need from an external energy input in order to introduce disorder into the flow.

There are many active methods of mixing. Industrially, they are usually the straightforward choice in absence of constraints and when the main objective is a high mixing rate, at the expense of the system complexity and robustness. Some of the most typical solutions are confined impellers and jets, which have experienced considerable design improvement in the last years. Woziwodzki et. al. has come with some innovative ideas relating to impellers such as the utilization of eccentric assembly (Woziwodzki et. al. [17]), double (Woziwodzki and Jedrzejczak [18]) or unsteady impellers (Woziwodzki [19]). Similarly, Ng investigates numerically the consequences of applying unsteady impeller rotation in vessels, Ng et. al. [20], or the impact in mixing quality of wall baffles and impeller eccentricity, Ng and Ng [21]. Hashimoto et. al. [22] studied in 2012 the effect of alternative pitched impellers in laminar mixing inside vessels and found their effectiveness was much better than that of ordinary ones. In what to jet mixing concerns, a brief sample are the works of Devahastin and Mujumdar [23] and Wang et. al. [24] [25] on laminar confined impinging jets, Woodfield et. al. [26] on mixing enhancement in a multiple jet microcombustor and Rafferty and Kaminski [27] on laminar semi-confined twin jets.

All the mentioned methods above are not so easy to implement in the micro-scale, so alternative methods are more suitable, such as artificial electro-activated micro cilia, Toonder et. al. [28]; actuated vortex generators, Shang et. al. [29]; or pulsatile/oscillatory flows, such as Phelan et. al. [30] and Silva et. al. [31].

Passive mixers usually make use of complex geometries to improve mixing. One example are interdigital micromixers, which try to force the interpenetration of several lamellae alternatively injecting them in a common chamber. An experimental study of this kind of mixers is presented in Hessel et. al. [32] and its numerical analysis counterpart in Hardt and Schonerfeld [33]. Other possibility of the same nature are tree-like microchannel nets, Senn and Poulikakos [34]; screw extruders, from which several modifications are studied in Camesasca et. al. [35], concluding that the addition of ridges and pins to the basic model enhances the mixing; Djenidi and Moghtaderi [36] study several ways of passively improving a microrreactor such as holed or flat plates or square bars; grooved geometries to induce

vorticity are also a very common option, Wang et. al. [37]; SMX mixers are one of the most common option chosen by the industry when it comes to high viscosity or non-newtonian fluids (polymer processing, paints, etc.), there are diverse studies for newtonian fluids, Zalc et. al. [38], as for other kind of non-newtonian fluids such as shear thinning ones, Liu et. al. [39].

Among the examples given of passive mixers for confined laminar flows are also found the blunt object method, which pursue the flow-structure interaction to enhance the mixing process; and the fined/pined channel geometries. Namely, the flow-wall interaction of the wake produced by a squared section prism inside a square section channel with a high blockage ratio, and the influence of tip clearance in a fined channel geometry are the objects of the study carried on in this thesis.

1.3.3 The confined cylinder problem

The problem of the cylinder, whatever its cross section geometry is, inside a flow has been studied extensively in the fluid mechanics literature. In the particular scope of this thesis are found all of those studies which contemplate laminar confined flow, and in special those which consider moving prisms.

One of the most representative studies for this thesis was Pottebaum and Gharib [40] in 2006. In it is investigated the effect of transverse oscillations of a hot circular section prism on heat transfer. Its main conclusion is that low amplitude oscillations at Strouhal frequencies or its harmonics are positive from the heat transfer standpoint.

In 2010, Barrero-Gil and Fernandez-Arroyo [41] studied the fluid excitation force of a circular cylinder during Vortex Induced Vibrations. An experiment was carried out in which a circular cylinder with high aspect ratio (10) was supported by two aluminium beams and an actuator to provide further excitation or damping, in order to study the flow induced forces. The context of the study was a free stream (air) in a transitional Reynolds number (2000), which is slightly contrastive to the framework of this thesis, but conclusions still apply in general terms. They proposed an alternative way to study the problem in which the oscillation was not forced but excited by the flow and emphasize the relevance of the study of the

phenomenon of induced vibrations for civil engineering, specially for prone geometries, usually blunt obstacles such as circular or square section prisms.

Meis et. al. [42] did a 2D numerical research in 2010 concerning to heat transfer enhancement through vortex promoting techniques. They tried diverse obstacle shapes, blockage ratios, obstacle position and orientations. They determined that blockage ratios higher than $1/2$ produced a big penalty in the pumping power compared to the benefit of the heat transfer rate; that the second best obstacle section shape was the rectangular one, just after the triangular one; and, finally, that a slight inclination ($\sim 30^\circ$) could increase greatly the efficiency of the layout (in Nu terms and less pumping power) in a much more notorious way than an offset of the obstacle from the centreline.

Also Meis published in Meis et. al. [43] an experimental and numerical study seeking a better comprehension of the problem addressed in [42], in particular the design with a rectangular cross section prism. They managed to observe in the experiment that there was little gain depending on the tilting angle of the prism, meaning that the effects dominating 2D simulations may not be the same ones that govern realistic boundary conditions or 3D simulations. In fact, they finally determined that the layout tested was rather robust with regard to experimental boundary conditions (flow forced pulsation, geometrical inlet asymmetries, stagnation chamber geometry, unbalancing flow rates, etc.).

In 2011, Martin and Velazquez [44] study the influence of aspect ratio and wall temperature on the onset of the Karman street regime for a square section prism in a square section channel in 3D, which typically occurs at Reynolds number around 50 in the 2D case, and found not only that said critical Re was fairly sensible to the aspect ratio (increased a 600% for aspect ratios of $1/2$) but also that heating the prism walls helped in order to promote the Karman street regime, which is a relevant conclusion from the heat transfer problem standpoint. In their case the vortex promoter was static, but a very remarkable conclusion for the present thesis is that in opposition to the free stream case (ideally 2D) the transition in 3D from a recirculation bubble to a vortex shedding wake is not abrupt in the case of confined flow because of the fluid-wall interaction. Ribeiro et. al. [45] published in 2012 an study in which three dimensional effects of the same problem but with a cylindrical prism where exhaustively examined and was also determined, numerically and experimentally, that the interaction of the side and end walls around the prism and wake are of

critical relevance.

Kumar and Dhiman [46] published in 2015 a review on the effects of different blockage ratios in the numerical problem of a semicircular obstacle submerged in a confined flow at low Re numbers, low enough that every case run was stationary ($Re = 1-40$). They observed that confinement played a fundamental role in the determination of the recirculation bubble length: the higher the blockage ratio, the lower the bubble length. For $Re = 40$, recirculation length was of the order of 1.5 prism diameters. It is relevant to note that in this thesis, although the Re numbers reached up to 800 and blockage ratio used was 40%, the flow could only be observed around 6 prism diameters downstream from the obstacle (the heating section inside which the prism is placed is an opaque body), and thus sharp flow structures are usually dissipated. Also, drag coefficient C_D increases with blockage ratio but decreases with Re number.

In 2016 and 2017, Salcedo et. al. published two studies about the same concept, numerical [47] and experimental [48] respectively. The case being analysed was that of two cylinders in tandem submerged in a channel with blockage ratio 0.3 and heated walls. Their standpoint was to optimize the distance between prisms in order to enhance the heat transfer. One of the conclusions they reach, which is significant for this thesis, is that at the lowest Re experimented (100), buoyancy effects are relevant for the problem.

Highly relevant for the standpoint of a confined square prism and low Re number are the studies conducted by Ortega Casanova between 2016 and 2018. In Ortega-Casanova [49] he analyses numerically and two-dimensionally a conceptually similar problem that one performed experimentally in this thesis, in which he concludes that the optimum heaving is done at $St = 1$ and with amplitudes as high as possible, yielding efficiencies of 30%. Other important outcome of his research is that the most effective heaving is not necessarily the most expensive in energy consumption terms. In Ortega-Casanova [50] he explores the possibility of pitching the prism (spinning as if like an impeller) instead of heaving it, concluding that the overall mixing efficiency was 46%, 42% higher but 155% more costful. He later analyse in Ortega-Casanova [51] the onset for vortex shedding for a rectangular prism with constant blockage ratio (1/5) for different aspect ratios, reaching the conclusion that the lower the aspect ratio, the lower the triggering Re. Finally, in 2018 he published a thorough study, Ortega-Casanova and Lai [52], in which he simulates numerically both heaving and pitching of the square prism with more

than 60 different motion combinations and conclude that efficiencies of 54% can be achieved with less cost than just pitching it.

Because of the importance and similarities of the object of study between the next articles and the experiment carried out for the development of the present thesis, have the next articles been left last to refer to.

First, in Reyes et. al. [53] an experimental study about a square section static prism inside a square section isothermal channel (aspect ratio = 1) with a blockage ratio of $1/2.5$ (0.4) is conducted. The main conclusion is that, in opposition to the 2D case, the three-dimensional effect of the end walls (the ones in which the prism is attached) introduce significant differences in the flow field, namely the Re number of the onset for the Karman street escalates from 50 (free stream) to around 170. Also, new flow regimes are present: the transition from a steady recirculation bubble to the Karman street is not abrupt and in between a pulsating recirculation bubble is present, which reaches an intermittent regime in order to finally become a vortex shedding wake.

Second, in Reyes et. al. [54] the same problem is addressed with a crucial difference: in this case the prism is left to oscillate self self-sustainedly by tethering it with thin cables to the channel walls, but still, inside an isothermal channel. Three different prism to water densities are tested (different buoyancy forces). New complex flow topologies appear because of the interaction between the flow and the structure depending on the Re number, which on the other hand are great interest because of its pseudo-chaotic nature from the mixing standpoint.

1.3.4 Tip clearance based mixing enhancement

Temperature, as a passive scalar of a fluid, can also be the subject of mixing applications, and thus heat exchangers are one of the most studied devices in literature. From the engineering perspective, the concern is not only enhancing the amount of heat that can be transferred between a solid surface and a fluid but also in reducing the cost, in terms of energy input to the system, necessary to make it work. In other words, efficiency is always a matter of motivation for research.

Amid the great variety of heat exchangers that use a working fluid, can be found those which base their operation in geometries that increase the contact surface between the solid body and the fluid, and more specifically those which do so by means of small pillars, fins or ridges. Just like with every other kind of heat exchanger, it is huge the research effort and literature production around these particular ones, however the focus of this thesis is a very specific way of improving the efficiency of this technology: reduce in a significant way the pumping energy while keeping high the heat transfer rate, by means of the introduction of a tip clearance between said fins and its lid or cover. It is in this framework in which the next study summary and the second experiment of this thesis fit.

The main purpose in Mei et. al. [55] in 2013 was to establish, via numerical simulation, new correlations for the Nusselt number and friction factor that account for the tip clearance in the tested case (a chemical reactor consisting in a micro-pinned channel). It is concluded that when working at low Reynolds numbers (400), the friction factor and Nu show a strong dependence with tip clearance, and it is also mentioned an optimum for said tip clearance may be obtained as function of the pins' height.

Some of the studies that consider a pin array explore the possibility of placing the pins staggered instead of in-line to the flow. This way, Roth et. al. [56] find that not only the correlations for the friction factor and Nu are still strongly dependent to the tip clearance but also that staggered assemblies perform better than in-line ones. Other way to modify the typical geometries employed is to make oblique pins, such as those in 2014 Far's study [57], in which the main objective was to innovate by using a different working fluid (NEPCM slurry), which was found to require much more pumping power than water and the only possibility to make it work while keeping reasonable the pump size or power was to introduce a tip clearance, which besides can enhance the cooling performance.

In 2014, Pandit et. al. [58] review several pin shapes in order to maximize the heat exchange, finally establishing that the best one was tilted squares / diamonds. Also, it is optimized the pin-to-channel height, determining that it was near 50%, or in other words, the optimum tip clearance was 100% of the pin height.

A similar study was conducted by Zhao et. al. 2015 [59], but in this case they tried to minimize the friction factor, resulting the best shape the elliptical one; and also Yang et. al. [60], that concluded the

circular ones to be the best. Instead of pin/pillar geometries, Park [61] in 2015 estimated numerically and experimentally the optimum tip clearance for a groove/fin array. In their case it happened to be near 70% of the fin height.

Shi and Dong [62] try to optimize several design parameters of a staggered pin array from the standpoint of entropy generation. They concluded that optimum tip clearance to pin height fell above their maximum considered of 25% in the majority of cases.

It has also been studied the case when the pin to channel height is much greater than one, or in other words, pin height is insignificant in comparison to tip clearance. In this cases literature usually speaks of pin roughened walls of heat exchangers. It is this the framework of Yeom et. al. [63], in which it is found the benefits are promising.

In 2011 Liu et. al. [64] publish an experimental study on a micro pin fin heat sink, with Re numbers ranging from 60 to 800 and water as working fluid. They obtain and compare friction factor and Nusselt number correlations for the given geometry. Their hot surface temperature was around 70 °C and the flow rate corresponding to Re 800 is 60 litres per hour. At these given conditions they determine this kind of device is capable of heat transfer values of around $2.8 \times 10^6 \text{ W/m}^2$. They did not study the effect on tip clearance, but their boundary conditions are very similar to those of Liu et. al. [65], who study a very similar layout and determine that the friction factor for Re greater than 50 grows much slower for bigger tip clearances, meaning that pressure losses can be reduced by a 50% for tip clearances of around 0.2 of the height of the pins, while the increase of wet area because of the additional surface at the top of them and the new flow patterns make the Nusselt number not to drop proportionally, increasing that way the efficiency.

It is this same phenomenon the one observed by Reyes et. al. [66] in 2010. In this case the device tested was a grooved block of aluminium with 15 square section fins (hydraulic diameter = 500 μm) connecting two stagnation chambers separated 15 mm. The working fluid is water at 35°C, while the hot aluminium block is at a temperature of 70°C. They compare three different tip clearances (50%, 100% and 200%) with the no clearance case, as well as to force the flow perpendicularly to the channels with a fixed tip clearance of 100% (500 μm), everything with Reynolds numbers ranging 416 - 2600. They found the tip clearance can reduce de pumping power by around 70% while deteriorating the thermal efficiency only 15%. It was also observed

that perpendicular ridges outperformed in-line channels, requiring 30% less pumping power for the same heat evacuation target. One of the biggest question that remains unanswered is which is the mechanism causing the observed behaviour: ¿does the tip clearance promote 3-D structures that are impossible to achieve without clearance, or is it a different phenomenon taking place?

Thesis plan

This thesis is divided into two clearly differentiable parts, that will conform chapters 3 and 4.

2.1 Flow induced vibration mixing enhancement

The main objective of this experiment was to determine if any of the possible flow topologies resulting from flow induced vibrations found in [54] were suitable to enhance mixing. To find it out, temperature was chosen as scalar to be traced. The concept is to create a thermal boundary layer with the use of a short channel section with hot walls, inside of which a prism is tethered and buoyant in the fluid (water). The whole assembly will be placed vertically, so the problem is still axisymmetric with regard to the buoyancy effects. Had the prism been placed immediately upstream from the heated section, the main focus of the study should be the amount of energy the resulting flow topology can extract from the hot walls. On the contrary, the prism could have been placed immediately downstream the hot section, in which case the main target should have been the ability of the prism wake to mix the generated thermal boundary layer. It was finally determined the most interesting setup was to place it right in the middle of the hot section. In order to study the mixing capability, the temperature profile in the output section will be measured. Besides, so as to assess the "refrigeration" capacity of the prism wake, wall temperature and power input to the hot section will be controlled.

2.2 Tip clearance based mixing enhancement

Here the main motivation is to find the mechanism that causes the flow topology inside the geometry tested by Reyes et. al. [66] to be able to keep high heat transfer rate in spite of the presence of a significant tip clearance, which other way does make the pressure losses to drop. The device to be tested consists in two stagnation chambers connected by four square section channels with different tip clearances. The focus is to resolve experimentally the velocity fields in several representative or critical sections in order to validate robustly a CFD model of the problem, in which more complex analysis can be carried out.

Chapter 3

Flow induced vibration mixing enhancement

3.1 Introduction

A very illustrative example of the type of work that is normally carried out in this field can be found in the article published by Celik et al. [67]. In their 2D numerical study, the authors considered a straight channel with a circular cylinder placed on its centreline. The blockage ratio of the configuration was 1:3. A uniform heat flux was applied to the channel walls while the cylinder itself was assumed to remain adiabatic. The flow Reynolds number (Re), based on the cylinder diameter was 100 and the Prandtl number (Pr) was varied in the range between 0.1 and 100. First, the authors addressed the case of the fixed (unmoving) cylinder so as to compute in this case the frequency of the downstream shed vortices (the reference frequency). Next, they prescribed a harmonic cross flow wise motion of the cylinder and computed the downstream Nusselt number (Nu). The frequencies that they prescribed were close to the reference frequency. Their findings for $Pr = 5$ (that is the case closest to considering water as the working fluid) were as follows: a) the limiting Nu in the case of the clean channel with no cylinder present was 14, b) the limiting Nu in the case of the cylinder moving with a frequency of 0.75 times the reference frequency was 17.5 (a 25% improvement), and c) prescribing frequencies above the reference frequency did not improve heat transfer or even deteriorated it. All this was regarding the limiting Nu in the region far downstream of the cylinder. In the region close to the cylinder, local heat transfer enhancement was larger (close to a 50% of improvement).

Another numerical 2D study has been reported by Fu and Tong [68]. In this case, the authors considered three different blockage ratios, 1:8, 1:4, and 1:2 for different combinations of frequency and amplitude of the cylinder motion at Re 500 based on the channel height (the working fluid was air). In yet another study, Fu and Tong [69], performed comparisons in terms of the average Nu . In a short channel section downstream of the cylinder, heat transfer associated to the moving cylinder improved by a factor of 31% the heat transfer rate of the clean channel (no cylinder present). When averaging the Nu over the whole channel, the improvement was 23%. So these conclusions are qualitatively and quantitatively similar to those already mentioned of Celik et. al. [67].

Yang and Chen [70] have presented another 2D numerical study in which four heated blocks were attached to walls of a channel in which a cylinder was moving with prescribed frequencies and amplitudes. The aspect ratio of the block was 1:4 and Re was higher than in previous references (800 to 8000 based on the cylinder diameter). The working fluid was also air. Depending on the selected frequencies and amplitudes of the cylinder motion, heat transfer rate improvements (defined as the space-time averaged Nu in the case with cylinder versus the case without cylinder) ranged from 8% to 17%. The case in which the cylinder itself is the heated element and its motion is used to enhance heat transfer from the cylinder to the surrounding fluid has also been studied extensively. For some examples, the interested reader is referred to the works of Ghazanfarian and Nobari [71], Nobari and Ghazanfarian [72] and Pottebaum and Gharib [40].

In all the references mentioned previously, the cylinder motion was prescribed (an active system). It means that in an actual engineering situation some kind of motor with an associated mechanism has to be used. This, by itself, might be difficult, in particular when dealing with mini/micro systems. It might also be prone to suffer from functioning problems such as leakages, motor failures, etc. In this context, the objective of the present thesis is to study whether a very simple passive system in which the prism motion is self-generated and self-sustained might provide some acceptable enhancement of the heat transfer rate from the channel walls into the fluid. Specifically, this self-sustained motion occurs because of the interaction between the incoming flow and the cylinder (the phenomenon called flow induced vibration). In this way, the system design and actuation is much simpler (it is, for instance, water tight) which may facilitate its actual use in industrial applications.

Regarding the basic fluid mechanics aspects of this problem, the reader is referred to the isothermal experimental studies published by Reyes et. al. [53] and Reyes et. al. [54]. In these two studies, detailed Particle Image Velocimetry (PIV) based analysis were presented on the flow topology past a fixed square section prism and a free prism allowed to move unhindered in the cross-flow direction inside a square section channel. The different flow regimes were analysed and it was found that, depending on the Re and on the body to fluid density ratio (the body was buoyant), both periodic and chaotic flow conditions could be observed. Now, in the present thesis, the objective is to move one step ahead by way of including thermal effects so as to find out whether these flow regimes might enhance the mixing of a passive scalar such as temperature.

3.2 Experimental setup

The experimental setup had the form of a closed loop circuit, see figure 3.1. The main parts of the setup were: the test channel, the test section, the power supply unit, three tanks, a flow meter, a pump, and a radiator to prevent the recirculating water from overheating (before entering the main tank, water temperature was monitored to be constant at the level of 25 °C).

FIGURE 3.1: Sketch of the experimental setup.

3.2.1 The test channel

Every element in the test channel was manufactured from transparent methacrylate plates 10 mm thick, cut with the saw, joined together with Betaplex methacrylate glue and sealed with silicone when necessary. The most critical elements were also reinforced with screws. The optimal assembly strategy found for the test channel is shown in figure 3.2

FIGURE 3.2: Assembly of the pieces conforming a standard channel section

Placed vertically, its square cross section was $25 \text{ mm} \times 25 \text{ mm}$. Its total height was 1490 mm. The channel was inserted into a large primary tank where the flow was homogenized. To achieve further homogenization, so as to shorten as much as possible the hydrodynamic entrance length, a honeycomb section was inserted at the entrance of the test channel (the inflow section). The outflow section discharged into a secondary tank where the water level remained constant; the idea being to avoid pressure fluctuations in the test section of the channel. A view of the channel together with information on the location of the origin of coordinates and all associated distances is presented in figure 3.3.

The inlet to the main stagnation chamber (primary tank) was done

FIGURE 3.3: View of the test channel and primary (bottom) and secondary (top) tanks. The heated section (90 mm height) can be observed midway along the channel where the origin of coordinates is placed. Distances are expressed in mm.

through eight hoses, situated two per side, in order to prevent asymmetries in the entrance section. Similarly, the outflow from the secondary tank was done through two hoses situated in one of the four sides but shielded from the discharge zone by a plastic wall, the assembly shown in the figure 3.4.

From the secondary tank, the outflow hose connected directly to the auxiliary tank, which was placed at ground level, and thus it is hermetic, so the level of the water was stationary and the pressurized air inside acted as dumper for any kind of pressure disturbance.

FIGURE 3.4: View of the discharge zone in the secondary tank

3.2.2 The test section

The heated test section was 90 mm long and it was located midway along the channel (see figure 3.3). It consisted of a hollow aluminium block, where the tethered prism was inserted, located a distance 950 mm away from the channel inflow section (see figure 3.3). A close-up view of this test section is shown in figure 3.5.

FIGURE 3.5: Close-up view of the test section (left) and of the centerline plane (right). Distances are expressed in mm.

The thickness of the aluminium walls was 15 mm. These walls were externally insulated with 10 mm thick methacrylate panels to minimize thermal losses. A simple thermal model was developed that showed that, because of the methacrylate very low thermal conductivity and its thickness, losses to the outside ambient air were negligible compared to the thermal power transferred to the fluid. 16 electrical cartridges (4 per side) were inserted into the aluminium

block to provide controlled heating. Cartridges were "20 W 40x7 mm RS-Pro", close-up on figure 3.8. The disassembled heating section is shown in picture 3.6. Also, a sketch of the position of the inserted thermocouples is provided in figure 3.7

(A) Heating section without elements. Big holes for the cartridges, small ones for the thermocouples.

(B) Disassembled heating section. Big holes for the cartridges, medium ones for the screws.

FIGURE 3.6: Heating section detail.

FIGURE 3.7: Numbered wall temperature control thermocouples.

10 T-type 1 mm width thermocouples, shown in figure 3.10, were inserted into the aluminium to monitor its temperature spatial distribution. The objective being to keep a constant wall temperature of

FIGURE 3.8: Cartridge sample.

about 60 °C. All the 16 cartridges were fed with a DC power supply, whose output voltage was controlled via a computerised control system.

Because of the buoyancy effects, an ascending thermal boundary layer was present in the heated-wall experimental campaigns, independently of which Reynolds case was being tested. Thus, the two upper rows of cartridges were powered with a square wave, controlling its duty cycle using a relay, so as to maintain a uniform wall temperature. A sketch of the electrical assembly and sensors is shown in figure 3.9.

FIGURE 3.9: Electrical layout.

Table 3.1 shows the time-averaged temperatures measured by the 10 thermocouples (T1 to T10), and both the standard and maximum temperature deviations for the case of the fixed (unmoving) prism for some representative Re . The coordinates of the inserts where the thermocouples were placed are also given in table 1 (notice that some

coordinates are negative and some other larger than 25 mm because these thermocouples were placed inside the aluminium walls and they were not in contact with the fluid). The thermocouples were of the Tc-direct brand and they measured with an uncertainty of ± 0.5 °C. They were connected to a NI9214 data acquisition system with a sensibility of ± 0.02 °C. At the lowest Re, the maximum temperature deviation between thermocouples located in the aluminium block was of the order of 1 °C. This deviation was of the order of 4 °C to 5 °C at the highest Reynolds number.

FIGURE 3.10: Thermocouple sample.

Nine additional temperature measurement points were placed in a plane located 20 mm downstream of the aluminium block (60 mm downstream of the prism), see figure 3.5, to measure the spatial distribution of water temperature inside the channel. To minimize flow disturbances, these temperatures were acquired one by one using a sequential procedure. Specifically: the same thermocouple was inserted through the channel wall into the flow field, was allowed to measure and, then, retrieved. The inserts were sealed with plastic material to prevent leaking. The process was sequentially repeated for all 9 measurement locations. Figure 3.11 shows the actual spatial locations where the thermocouples were inserted. The view is from the top of the channel down towards the test section; this why the prism (shaded in grey colour) is also seen in the figure. A picture of the disassembled test section with an inserted thermocouple is shown in figure 3.12. Finally, an actual photograph of test section together with some of the prisms used in the experiments are presented in figure 3.13.

				Re					
				80	133	260	360	466	733
	x	y	z	T (°C)					
T1	12.5	-1.0	70.0	58.8	57.8	58.6	59.3	58.9	53.1
T2	12.5	26.0	70.0	58.7	57.8	58.1	58.8	58.4	52.7
T3	12.5	-1.0	25.0	58.9	58.2	58.9	59.4	59.8	55.0
T4	12.5	26.0	25.0	58.7	58.0	58.5	59.2	59.4	54.7
T5	26.0	12.5	70.0	58.4	57.3	57.1	57.5	57.0	50.8
T6	-1.0	12.5	70.0	58.5	57.4	57.5	57.4	56.9	50.7
T7	26.0	12.5	45.0	59.5	58.8	59.0	59.3	59.1	53.2
T8	-1.0	12.5	45.0	59.6	58.7	59.2	59.5	59.0	53.4
T9	26.0	12.5	20.0	58.9	58.2	58.7	59.4	59.8	55.1
T10	-1.0	12.5	20.0	59.5	58.5	59.1	59.8	60.0	55.4
T_{av}				59.0	58.1	58.5	59.0	58.8	53.4
T_{st}				0.4	0.5	0.7	0.8	1.1	1.7
ΔT_{max}				1.2	1.5	2.1	2.3	3.2	4.7

TABLE 3.1: First three columns: coordinates (in mm) of the inserts where the thermocouples were placed. Other columns: measured time-averaged temperature (°C) of the 10 thermocouples inserted into the aluminium block (T1 to T10) for some representative Re. Average temperature, T_{av} , standard deviation, T_{st} , and the maximum deviation, ΔT_{max} , are also included.

FIGURE 3.11: Position of the 9 points (located downstream of the aluminium block) where water temperature is measured. The view is from the top of the channel into the test section. The prism is shaded in a darker grey mesh. Measurements are expressed in mm.

FIGURE 3.12: Test section with explanatory thermocouple.

FIGURE 3.13: Pictures of the test section (notice that the outside of the aluminium block is isolated with low thermal conductivity methacrylate panels) and two prisms with the attached tethers.

Regarding the prisms, they were hollowed out when manufactured so as to achieve different density ratios, m^* . Specifically, three prisms with the following m^* were manufactured: 0.56, 0.70 and 0.91.

3.2.3 Auxiliary systems

The flow meter was an electromagnetic Siemens Sitrans FM MAG1100 equipped with a Sitrans FM MAG5000 transmitter. The

measurement range was from 0 to 5 l/min and the uncertainty was $\pm 5\%$. The pump was an ITT Totton centrifugal with a magnetic drive DC15/5. An overview picture of the whole experimental setup is presented in figure 3.14.

FIGURE 3.14: Overview of the experimental setup.

3.2.4 The Particle Image Velocimetry (PIV) system

This PIV system was from Dantec Dynamics. Flow illumination was achieved with a pulsed Nd:YAG 800 mJ Laser. The time lapse between two laser pulses (needed to collect a single sample) was adapted for each Reynolds number so as to achieve constant travel distances of around 4 pixels ($1/4$ of the 16 pixels of the smallest sub-interrogation windows used). This time τ ranged between 1 and 20 ms, and imply that physical travel distances of the seeding particles were around 0.2 mm between pulses, much smaller than the 10 mm characteristic length of the problem (the prism cross section length).

The camera was a Dantec Dynamics Flow Sense 2ME with a resolution of 1600×1200 pixels. The lens was a Zeiss Makro-Planar T* 2/50 mm ZF. Flow seeding was performed with 10 μm hollow glass spheres (HGS-10). Dantec Dynamics Studio software was used to synchronize image capturing and flow illumination, and to perform the analysis. The size of the interrogation areas was $50 \text{ mm} \times 25 \text{ mm}$. These areas were subdivided into smaller areas by the analysis software to achieve convergence in the re-computation process of the flow field. It was estimated that the spatial resolution of the flow field was of the order of 1 mm that should be enough to capture the large flow structures whose size was of the order of the prism cross-section length (10 mm).

The sampling frequency was 15 Hz. Afterwards, it was found that the relevant flow frequencies were of the order of 1 Hz, so the temporal resolution of the experimental setup was adequate to capture the relevant flow features.

3.3 Definition of the experimental campaigns

Six different experimental campaigns (EC) were carried out in the present study. They were labeled EC #1 to EC #6. Their definition is as follows:

- EC #1. Clean channel with no prism present. Isothermal walls. Re , based on a characteristic length of 1 cm (the prism cross section length in the cases where the prism is present) equal to: 80, 160, 200, 240, 320, 400, 600, and 800. The lowest volume flow rate

corresponding to Re 80 was 0.268 l/min (average inlet velocity equal to 0.007 m/s). The largest volume flow rate corresponding to Re 800 was 2.68 l/min (average inlet velocity equal to 0.070 m/s).

- EC #2. Clean channel with no prism present. Walls heated to the temperature level specified in table 1. The Re range equal to that of EC #1.
- EC #3. Channel with prism anchored to the walls (no prism motion allowed). Isothermal walls. Re (based on the prism cross section length of 1 cm) equal to: 80, 106, 133, 160, 200, 240, 260, 280, 300, 320, 360, 400, 466, 533, 600, 666, 733, and 800.
- EC #4. Channel with prism anchored to the walls (no prism motion allowed). Walls heated to the temperature levels specified in table 1. The range of Re equal to that of EC #3.
- EC #5. Channel with tethered prism placed inside and allowed to move freely. Isothermal walls. The range of Re equal to that of EC #3. Three different prisms to water density ratios were considered for each case: 0.56, 0.70 and 0.91.
- EC #6. Channel with tethered prism allowed to move freely. Walls heated to the temperature levels specified in table 1. The range of Re equal to that of EC #3. Three different prisms to water density ratios were considered for each case: 0.56, 0.70 and 0.91.

3.4 Experimental results

3.4.1 EC #1 campaign

This campaign could be considered as the setup calibration campaign because results were compared to those obtained theoretically by [73] for laminar flow in a square section channel. This is observed in figure 3.15 where the average value of the 100 measured velocity profiles, their maximum spread, and the theoretical profiles are presented. Out of the whole set of results, four Reynolds numbers (80, 160, 240, and 320) were selected for illustration purposes in figure 3.15. In these cases, the flow is hydrodynamically developed before reaching the measurement section so that comparison with the theoretical results of [73] can be consistently performed.

FIGURE 3.15: Average value of the 100 measured velocity profiles for Re 80, 160, 240 and 320 (solid lines), their maximum spread (grey bands), and the theoretical profiles (dashed lines).

Regarding the spread of the measurements, see the grey shaded bands in figure 3.15, the histogram presented in figure 3.16 shows a typical distribution of the 100 velocity measurements around the mean. Points A, B and C are in $y/c = 0.5, 0.75$ and 0.95 respectively.

FIGURE 3.16: Histogram of the 100 centreline velocity measurements around the mean for Re 80 (upper row) to 360 (lower row), in isothermal clean channel.

3.4.2 EC #2 campaign

This case is like the previous one but for the fact that the aluminium block is heated up to a constant temperature of 60 °C. The counterpart

of figure 3.15 for this case is presented in figure 3.17. There, it could be observed that at the lower Re , buoyancy dominates the velocity profiles (notice the high velocity plumes ascending close to the walls at $y/c = 0$ and $y/c = 1$). As the Re increases, this effect losses its importance until the convection from the main body of fluid dominates again the velocity profile (except in a small region close to the walls) at Re 320. Also, the equivalent to figure 3.16 is presented in figure 3.18.

FIGURE 3.17: Counterpart of figure 3.15 for the case of heated walls (no theoretical results are available in this case)

FIGURE 3.18: Histogram of the 100 centreline velocity measurements around the mean for Re 80 (upper row) to 360 (lower row), downstream of the heated section and clean channel.

A quick approach to the relevance of buoyancy in the problem can

be performed through dimensionless analysis. Grashof number compares buoyancy to viscous forces:

$$Gr = \frac{g\beta(T_s - T_\infty)L^3}{\nu^2} \quad (3.1)$$

with

- $g = 9.81 \frac{m}{s^2}$
- $\beta = 2.5 \times 10^{-4} K^{-1}$ for water.
- $T_s - T_\infty = 35 K$
- $L = 9 \times 10^{-2} m$
- $\nu = 0.893 \times 10^{-6} \frac{m^2}{s}$ for water at 25 °C.

yields $Gr \sim 8 \times 10^7$. Laminar thermal boundary layers are typical for Grashof numbers of $10^3 - 10^6$, and transition to turbulent thermal boundary layers takes place for $Gr \sim 10^8 - 10^9$. Also, Richardson number can provide us with useful information to compare the relevance of free to forced convection:

$$Ri \sim \frac{g\beta(T_s - T_\infty)L}{U^2} = \frac{Gr}{Re^2} \quad (3.2)$$

yield values from 2000 to 10 with Re from 200 to 2000 respectively, based on the channel cross section. For values of Ri greater than 10, forced convection could be neglected, while for values smaller than 10 and greater than 0.1, both forced and natural convection are comparable. Thus, it can be said that, in general terms, in experimental campaign EC #2 buoyancy forces are significant and heat transfer is governed by natural convection, except for the higher Re numbers, in which mixed convection is the reigning mechanism.

The temperature information obtained from the thermocouples inserted in the flow field in the transversal plane located 20 mm downstream of the aluminium block was processed, using a nearest neighbour algorithm [74] together with symmetry considerations, to reconstruct the temperature field in that plane. An example of this reconstruction for Re 80 and Re 320 is presented in figure 3.19. These temperature planes will be used in the next sections to assess the degree of mixing in the flow field.

FIGURE 3.19: Reconstructed temperature field in a plane located 20 mm downstream of the aluminium block for Re 80 and 320.

The Nusselt number, Nu , in this thermal case is defined as: $Nu = hL/k_{film}$ where h , L , and k_{film} are the convection heat transfer coefficient, the characteristic length, and the water thermal conductivity at the film temperature. The characteristic length for Nu was taken to be the hydraulic diameter of the channel (0.025 m). The film temperature

was defined as:

$$T_{film} = \frac{T_{out} + T_{in}}{2} = \frac{\frac{T_{in} + T_{out}}{2} + T_{wall}}{2} \quad (3.3)$$

Where T_{in} , T_{out} , and T_{wall} were inlet, outlet, and wall temperature respectively. T_{in} , and T_{wall} were measured directly. The fluid temperature downstream of the heated section, T_{out} , was computed by means of an energy balance: $\dot{Q}_{in} + H_P = \dot{Q}_{out}$, where $\dot{Q}_{in}$ and $\dot{Q}_{out}$ where the thermal energy per unit time entering and leaving the test section respectively, and H_P was the measured electrical heating power. Then, h was computed by performing the following balance:

$$H_P = hA_{hw} (T_{wall} - T_{bulk}) = hA_{hw} \left(T_{wall} - \frac{T_{in} + T_{out}}{2} \right) \quad (3.4)$$

Where A_{hw} was the heated wall area. Once h and T_{film} were obtained in this way, the Nusselt number was computed.

H_P (i.e.: the thermal power supplied to the flow of water to keep the aluminium block at a constant temperature of 60 °C) and Nu as a function of Re are presented in table 3.2. There, it could be observed that this power supply was nearly constant in the range from Re 80 to 600 (Re 200 to 1500 if based on the channel hydraulic diameter) which suggests that Nu is constant (as it could be observed in table 3.2). This is in line with the well-known theoretical result [75] that states that the Nu in a heated square/rectangular channel in the laminar regime does not depend on the Re .

It is to be noted that the Nu as given in reference [75] for a square section channel in the laminar regime is 3.6. However, that result is for a hydrodynamically and thermally developed flow, and with heat transfer being applied along all channel walls, which is not the case addressed in the present study, because although the flow in at the beginning of the hot section is hydrodynamically developed, it is not thermally developed; thereby the differences in Nu values. Given the uncertainty associated to the measurements of the thermocouples and the power unit, it was estimated that the typical uncertainty in the computed Nu was of the order of $\pm 5 \%$.

There is an alternative way to calculate Nu , based on the water temperature profiles obtained downstream the test section, like the

Re	H_P (W)	Nu
80	214	32.0
160	215	32.0
240	214	31.9
320	214	31.8
400	219	31.8
600	220	31.7

TABLE 3.2: Measured electric power supply and Nu as a function of Re for experimental campaign EC #2.

ones presented on figure 3.19. An average of the outlet temperature profile must be done in order to calculate our alternative bulk temperature:

$$T_{bulk} = \frac{\bar{T}_{out} + T_{in}}{2} \quad (3.5)$$

and use this information to calculate T_{film} like 3.3 and h like 3.4. This way, the method is completely independent from the DC power supply information.

3.4.3 EC #3 and EC #5 campaigns

These two campaigns correspond to the isothermal case with prism present. In particular, in EC #3 the prism is anchored to the channel walls, while in EC #5 it is tethered and allowed to move. Flow characterization was carried out using the PIV system described in sub-section 3.2.4. The sketch of the actual PIV interrogation window integrated into the experimental result is presented in figure 3.20. It could be observed that this window is located 60 mm (6 diameters) downstream of the prism. This is so because the prism is enclosed in between solid aluminium walls that do not allow for PIV sensing. This means that the flow structures that can be actually sensed are, at least, at a distance of six diameters away from the prism. Furthermore, given the fact that the flow is highly confined, these structures are likely to be smeared out as a consequence of the strong interactions taking place and the viscous effects enhanced by the nearby walls.

FIGURE 3.20: Sketch showing the PIV interrogation window in the channel.

The solid to fluid mass ratio m^* influences the prism motion and, as a consequence, the flow field topology downstream of the prism and the associated mixing. This is apparent when the prism equations of motion are written, see figure 3.21.

The two equations of motion in the limit $\alpha \ll 1$ ($\cos(\alpha) \approx 1$, $\sin(\alpha) \approx \alpha$) are:

$$B + F_{Cd} = T \quad (3.6)$$

$$F_{Cl} - T\alpha = m_p \frac{d^2\alpha}{dt^2} L_{Th} \quad (3.7)$$

Where B , F_{Cd} , T , F_{Cl} , α , m_p , t , L_{Th} are buoyancy force, drag force, tether tension, lift force, angle of motion, prism mass, time and tether length. Introducing T from equation 3.6 into equation 3.7 yields:

$$F_{Cl} - (B + F_{Cd})\alpha = m_p \frac{d^2\alpha}{dt^2} L_{Th} \quad (3.8)$$

That is: the buoyancy force B , that depends directly on the solid to fluid mass ratio m^* , enters the prism equation of motion in such a way that the lower m^* the larger B , and, therefore, the larger the restoring

force $T \sin(\alpha) \approx T \alpha$. For the cases considered in the experiments, the maximum values of α obtained were of the order of $0.05 \text{ rad} \ll 1 \text{ rad}$, so the approximate equations (3.6-3.8) hold to a reasonable degree of accuracy. In practice, different values of m^* could be obtained using different materials for the manufacturing of the prism although, as described in section 3.2.2, the method chosen is the present study was to use appropriately hollowed out cylinders of the same material.

FIGURE 3.21: Forces acting on the prism.

The two horizontal dashed lines A and B in figure 3.20 above indicate the two specific locations that have been used for analysis purposes. In particular the downstream velocity has been time averaged and it is presented in figure 13 along these two lines for Re 200 and Re 733 for both the fixed prism and the moving prism with $m^* = 0.91$. According to the results presented in reference [54] the case of Re 200 corresponds to the flow regime in which the prism undertakes low frequency and moderately high amplitude oscillations with a well-defined Karman-type wake, while Re 733 corresponds to a chaotic regime.

When looking at the right hand side plot of figure 3.22 (section A of figure 3.20), it is apparent that at Re 200 and at a distance of 9 prism diameters downstream of the prism itself, the flow is close to attaining its limiting Poiseuille-type limiting behavior (the peak dimensionless velocity at the channel centreline should be 2.2 in fully developed flow). However, interestingly enough, the flow is closer to its limiting state in the case of the moving prism. In this case, the peak velocity is 1.96 while in the case of the fixed prism the peak velocity

FIGURE 3.22: Time averaged "z" velocity profiles for Re 200 and 733. Results are presented in planes $z = 115$ (left) and $z = 140$ (right) for the fixed prism and, also, for the case with $m^* = 0.91$.

is 1.78. This suggests that the flow topology induced by the moving prism contributes to homogenize the flow field somewhat faster than the flow topology induced by the fixed prism. In a sense, this result appears to contradict intuition because one would think that the large moving structures generated by the oscillating prism would require a longer channel length to attain their limiting state. However, this question is somewhat out of the main focus of the present article and it will be addressed by the authors in a future work.

In the chaotic regime (Re 733) the behaviour is just the opposite: the flow generated by the fixed prism is closer to its limiting Poiseuille type limiting state. The results of section B (located closer to the prism at a distance of 6.5 diameters in the downstream direction) are consistent with the results obtained for section A.

The time averaging process was carried out at a frequency of 15 Hz during 20 seconds. That is: 300 time frames were used to generate each of the curves presented in figure 3.22. The power spectral density plots of the points of each curve located at the channel centreline ($y/c = 0.5$) are presented in figure 3.23. It could be observed that at Re 200 the significant frequencies for the fixed prism were below 1 Hz. For the case with $m^* = 0.91$, a distinct frequency appears at 2 Hz. This also supports the idea that the sampling carried out at 15 Hz was consistent with the characteristic frequencies of the phenomena itself. In the case of Re 733, the PSD plots show the typical flat appearance of the chaotic flow.

FIGURE 3.23: Power spectra density plots of the downstream velocity in the channel centreline for the isotherm fixed prism at Re 200 and Re 733.

3.4.4 EC #4 and EC #6 campaigns

The counterpart of figure 3.22 for the thermal cases is presented in figure 3.24. It could be observed that, in this case, the flow profile at Re 200 is still dominated by buoyancy since the velocity in the region close to the walls is markedly higher than in the channel centerline. At Re 733 the opposite happens: velocity at the centerline is higher than in the near wall region. The PSD plots of these cases are presented in figure 3.25.

FIGURE 3.24: Counterpart of figure 3.22 but considering now the thermal case.

FIGURE 3.25: Counterpart of figure 3.25 but considering now the thermal case.

3.5 Discussion

The present work focuses on the question of whether a passive system whose motion is caused by fluid-body interaction is able to improve heat transfer significantly in practical engineering applications. In this view, figure 3.26 shows the summary of results regarding the electric power supply to the aluminium walls as a function of Re and m^* (including the fixed prism case) for constant wall temperature.

For any given m^* , the supplied power was increased along with Re to keep a constant aluminium wall temperature. The maximum available power delivered by the DC unit was 315 W. This threshold level was achieved at Re 320 for $m^* = 0.91$ and at Re 520 for $m^* = 0.56$. For the case of the fixed prism the threshold level was also reached at Re 520.

FIGURE 3.26: Electric power supplied by the DC unit to the aluminum walls (heat transfer to the fluid) as a function of Re and m^* for constant wall temperature. The case of the fixed prism is also included for comparison purposes.

To quantify the different heat transfer rates, Nu as a function of Re and m^* has been computed and the results are presented in table 3.3. It could be observed that Nu grows along with Re and that the configuration that yields the higher Nu is the one with $m^* = 0.91$. These results correlate with the following expression: $Nu = 23.31(Re^{0.09}m^{*0.22})$.

As compared to the results presented in table 3.2 (the clean channel) the maximum improvements are of the order of 36 %. It could be argued that this improvement comes at the expense of a higher pressure drop and, therefore, of a higher pumping power. This is true; however, it has been estimated that the pressure drop caused by the prism is negligible compared to the hydrostatic pressure difference in the vertical channel. In particular, the numerical simulations of Martín and Velázquez [44] show that for a fixed prism inside a

channel with the same geometry and blockage ratio the total drag coefficient is 4.8 at $Re = 100$. This yields a dimensional pressure drop of about 0.25 Pa that is not feasible to measure with standard laboratory equipment (measurement uncertainties of ± 50 Pa are typical for this type of pressure sensors).

Re	$m^* = 0.56$	$m^* = 0.70$	$m^* = 0.91$	Fixed prism
80	30.7	32.6	33.3	31.9
133	31.7	33.2	32.0	31.7
160	31.1	33.3	32.8	32.3
200	30.8	32.6	34.7	33.0
240	31.1	33.7	36.6	34.6
300	33.6	38.3	40.0	37.3
360	36.5	39.6	43.2	40.1

TABLE 3.3: Nu as a function of Re and m^* for the constant wall temperature case.

For illustrative purposes, Nu vs Re is presented in figure 3.27. Also, a comparison between both methods described at the end of section 3.4.2 to calculate Nu.

FIGURE 3.27: Nusselt vs Reynolds for constant wall temperature.

As already mentioned, the results presented in figure 3.26 and table 3.3 show that the most efficient configuration from the heat transfer

point of view is the one with moving prism at $m^* = 0.91$. For instance, the electric power needed to keep the aluminium block at a surface temperature 60°C ($\text{Re } 320$) is 307 W for $m^* = 0.91$, 280 W for $m^* = 0.70$, and 265 W for $m^* = 0.56$. That is, the heat transfer difference between the cases with highest and lowest m^* was 42 W (a 16 % difference). Interestingly enough, the thermal power associated to the case with the fixed prism was 278 W , which is in between the limits set up by the highest and lowest m^* .

As it can be observed in figure 16, this behaviour is consistently observed for Re up to 400. This suggests that not all types of prism motion improve heat transfer as compared to the case of the fixed prism. In fact, the case with $m^* = 0.56$ yields a heat transfer consistently smaller than the case with the fixed prism all over the discussed Re range. The authors think that this is related with the flow topologies generated by the prism motion and their interaction with the buoyant plumes that originate at the solid aluminium walls. In figure 3.28 it is shown the thermal difference, measured in degrees, between the entrance and the outflow section for the Re being studied as the rows and m^* of 0.56 and 0.91 as the columns respectively. As may be seen, the thermal distribution in the case of $m^*=0.91$ is more homogeneous: boundary layer with lower temperature and hotter fluid core, specially in the higher Re numbers, meaning better mixing.

Ideally, the best study option for this effect would be to perform a PIV based analysis of these structures, but this appears to be impossible owing to the fact that the aluminium walls are opaque. Then, the only feasible approach seems to be a CFD based one that is out of the scope of this work (it will be addressed by the authors in a future work).

These experimental results are, nevertheless, somewhat unexpected because, in a sense, it would be tempting to regard a fixed prism as a system characterized by an infinite buoyant force that leads to an infinite restoring force that suppresses all possible motion. Within this perspective, the fixed prism could be considered as the asymptotic limit of a sequence of prisms in which the buoyant force keeps growing. However, in view of the observed experimental results this does not appear to be the correct approach. Possibly, what happens is that the presence of an additional equation of motion for the cases in which the prism moves changes the system dynamics completely.

Figure 3.29 presents the time average of 300 PIV frames (20 seconds

FIGURE 3.28: Temperature increase in degrees with entry as reference. First row Re_{80} , last row Re_{360} . First column $m^*=0.56$, second column $m^*=0.91$.

of experiment) for Re_{200} . Both the dimensionless stream-wise U velocity and the cross-wise velocity V are plotted for EC #4 (the thermal fixed prism) and EC #6 (the thermal moving prism with $m^*=0.91$). The two upper subplots (stream-wise velocity) show a similar topology: the actual values of the time-averaged velocity might be slightly different but the global behaviour is basically the same. However, the two lower subplots that depict the cross-wise velocity are different.

In particular, the left one (fixed prism) shows that the average cross-wise velocity near the channel walls is small: it has different shades of green colour that means that this average velocity value is around zero (see the colour bar to the right of the plot).

On the other hand, see the lower right subplot, the case with the moving prism shows that the time-averaged cross wise velocity is positive the vicinity of the channel left wall (it has shades of yellow-orange colour) that means that its direction is from the left wall towards the channel centreline. At the same time, it could be observed that the time-averaged cross-section velocity is negative in the vicinity of the channel right wall (it has shades of blue colour) that means that, again, its direction is from the (right) wall towards the channel centreline. That is, the presence of the moving prism modifies the topology of the time-averaged cross wise velocity in the sense that it promotes motion from the walls towards the channel centreline, thereby promoting the transport of heat from the walls into the bulk flow. The fact that this cross-wise velocity is small when compared to the stream-wise velocity imposed a limit on the additional amount of heat that can be actually transferred from the channel walls into the main body of fluid.

Once the maximum power level was reached, further increments of Re led to a decrease in the wall temperature because of the growing heat transfer rate. This can be observed in figure 3.30 where the average wall temperature is presented as a function of Re and m^* (including, also, the fixed prism case). The actual measurements of the 10 thermocouples as function of Re for the cases of $m^* = 0.91$ and fixed prism are presented in table 3.4 for four representative Re (106, 200, 260, and 733). At the highest Re , the largest temperature difference in the aluminium walls was 4 °C. It is also worth noting that the larger the Re , the more similar are the results of $m^* = 0.70$, $m^* = 0.56$ and fixed prism, while the wall temperature for $m^* = 0.91$ keeps consistently lower. Again, this suggests that the dynamics of the system influences significantly the heat transfer rate. In this regard, the behaviour of the cross-wise time averaged velocity in the vicinity of the walls is qualitatively similar to the one shown in figure 3.29.

Nu as a function of Re and m^* has also been computed for the cases presented in figure 3.30 and the results are shown in table 3.5. Also, a counterpart of figure 3.27 is shown in figure. These results correlate with the following expression: $Nu = 4.07(Re^{0.40}m^{*0.37})$. As compared to the clean channel case (see table 3.2) the improvement on Nu is about 100 %. As compared to the fixed prism case (see table

FIGURE 3.29: Time-averaged PIV plots of the dimensionless stream-wise (U/U_{mean}) and cross-wise (V/V_{mean}) velocities at Re 200, for EC #4 (the thermal fixed prism) and EC #6 (the thermal moving prism with $m^*=0.91$).

3.5) the improvement on Nu is about 20 %.

The analogue figure of temperature increase for the studied Re numbers is presented in figure 3.32. It is visually even clearer that mixing for $m^*=0.91$ is much more homogeneous than that of $m^*=0.56$.

Another aspect to be analysed is the spatial variation of temperature on the plane where the thermocouples were inserted into the fluid. To quantify this variation, the RMS of the reconstructed 2D dimensionless temperature distribution downstream of the prism has been computed (following the procedure illustrated in figure 3.19) and the results are presented in tables 3.6 and 3.7 for the cases of constant wall temperature and constant wall heat transfer cases respectively. This information is also shown as a graph in figure FIG. When looking at the results of table 3.6, it could be observed that from

FIGURE 3.30: Average aluminium wall temperature as a function of Re and m^* once the threshold level of electric power supply (315 W) has been reached. The case of the fixed prism is also included for comparison purposes.

	Re 106		Re 200		Re 260		Re 733	
	Fix.	$m^*=0.91$	Fix.	$m^*=0.91$	Fix.	$m^*=0.91$	Fix.	$m^*=0.91$
T1	57.9	60.4	60.3	59.0	58.6	60.5	53.1	51.6
T2	57.8	60.0	59.9	58.5	58.1	59.5	52.7	51.0
T3	58.3	59.4	59.6	58.8	58.9	60.0	55.0	53.0
T4	58.1	59.2	59.3	58.5	58.5	59.7	54.7	52.8
T5	57.4	59.4	59.3	57.9	57.1	58.8	50.8	49.1
T6	57.5	59.9	59.5	58.1	57.5	59.1	50.7	49.4
T7	58.8	60.6	60.5	59.4	58.9	60.1	53.2	50.4
T8	58.8	60.8	60.5	59.3	59.2	60.2	53.4	50.3
T9	58.3	59.3	59.5	58.6	58.7	59.8	55.1	53.0
T10	58.7	59.9	59.8	59.3	59.1	60.5	55.4	53.7

TABLE 3.4: Detailed temperature measurements (°C) of the 10 thermocouples (T1 to T10) inserted into the aluminium walls as a function of Re for the fixed prism and the case with $m^* = 0.91$.

Re 200 onwards the RMS of the temperature signal for $m^* = 0.91$ case is consistently lower than in the other cases. This suggests that local mixing is achieved more efficiently in this particular case of $m^* = 0.91$ which leads to a better global mixing and higher Nu , as it is indeed

Re	$m^*=0.56$	$m^*=0.70$	$m^*=0.91$	Fixed prism
533	41.3	44.0	47.4	44.0
600	42.1	47.0	50.8	46.1
666	44.3	48.5	51.8	47.7
733	44.7	50.6	54.2	49.1
800	46.9	53.0	59.4	50.9

TABLE 3.5: Nu as function of Re and m^* for the constant wall heat transfer case.

FIGURE 3.31: Nusselt vs Reynolds for constant wall heat flux.

the case. That is, the results presented in table 3.6, that were obtained processing local temperature data, support the results of figure 3.26 that were that were of a global nature. For the constant wall heat transfer case, see table 3.7, the trend is the same and the RMS of the temperature signal for $m^*= 0.91$ is the lowest of all.

FIGURE 3.32: Analogue to 3.28. First row $Re=533$, last row $Re=800$. First column $m^*=0.56$, second column $m^*=0.91$.

Re	$m^*=0.56$	$m^*=0.70$	$m^*=0.91$	Fixed prism
80	3.51	3.82	3.67	2.97
133	4.22	4.60	4.33	4.24
160	4.11	4.56	3.96	3.53
200	3.89	3.11	2.51	3.36
240	3.37	2.65	2.63	2.59
300	1.39	1.60	1.41	1.98
360	1.19	1.25	0.94	1.80

TABLE 3.6: RMS of the dimensionless water temperature for the case of constant wall temperature as a function of Re and m^* .

Re	$m^*=0.56$	$m^*=0.70$	$m^*=0.91$	Fixed prism
Re 533	0.74	0.72	0.47	1.30
Re 600	0.76	0.70	0.52	0.72
Re 666	0.55	0.51	0.46	0.60
Re 733	0.48	0.47	0.46	0.56
Re 800	0.60	0.49	0.31	0.52

TABLE 3.7: RMS of the dimensionless water temperature for the case of constant wall heat transfer as a function of Re and m^* .

To conclude, it is presented in figure the mean of the RMS of the x-velocity of a reticle of 30 points in the bosom of the PIV window. Theoretically, the RMS in time for the velocity gives us an idea of the non-stationariness of the resulting flow. It is difficult to draw a conclusion if we study the differences between prisms, but it is clearly visible that the flow topology is qualitatively different between the isothermal case and the heated wall one.

FIGURE 3.33: RMS in time for 30 points in the flow field.

3.6 Conclusions

An experimental study has been performed on the heat transfer rate in a passive system in which the flow is disturbed (so as to promote mixing) due to fluid-body interaction effects. The test setup consisted of a square section channel ($25 \text{ mm} \times 25 \text{ mm}$) in which a buoyant

tethered prism was placed. All around the prism, heated aluminium walls transferred thermal energy to the fluid. The Reynolds number based on the prism cross section length (10 mm) was varied between 80 and 800. Three prism-to-water density ratio were considered: $m^* = 0.56$, 0.70 and 0.91. The clean channel case and the case with a fixed prism were also addressed for comparison purposes. The conclusions of the study could be stated as follows.

- In the configuration that has been studied, the presence of either a fixed or a moving buoyant prism always improved the heat transfer rate as compared to the clean channel case. In the case of the moving prism, the improvement on Nu was as high as 100 %. In the case of the fixed prism, the maximum improvement was 66 %.
- The reason for the improvement could be associated to the fact that both the fixed and the moving prisms modify substantially the flow topology in a way so as to enhance mixing. It should be noted that the length of the wall that was heated was 90 mm while the prism cross section length was 10 mm. If the heated portion of the wall is longer, it is expected that the flow tends to attain its limiting Poiseuille type behaviour and the enhancement effect should die away further downstream of the prism.
- Not all moving prisms generated heat transfer rates higher than the case of the fixed prism. In fact, the case with $m^* = 0.56$ (the lowest one) consistently yielded the lowest Nu values. This suggests that not all types of prism motions improve heat transfer in the same way. The reason, again, could be ascribed to the different flow topologies that appear associated to the different values of m^* .
- For the configuration and parametric range presented in this study, the best configuration was the one with $m^* = 0.91$.
- The 2D temperature profiles in the channel sections downstream of the moving prism have been analysed and it has been found that the lowest RMS correspond to the case with $m^* = 0.91$. This, again, supports the idea that this specific configuration promotes mixing in a way that is more efficient than others with lower m^* , and also more efficient than the fixed prism case.
- From the fundamental Fluid Mechanics point of view, the present study poses some questions that will need further analysis in the future. One of these is why depending on the value

of m^* , the prism in self sustained motion may either improve or deteriorate the heat transfer as compared to the fixed prism.

- From the engineering applications context, all that has been presented suggests that when devising practical applications, these types of passive configurations, that are conceptually very simple, might have some advantages over active systems when dealing with mixing at low Reynolds numbers. Specifically, an active system needs the presence of a motor with independent power supply that moves impeller type (or other) devices. This, on one hand, leads to a more complex system and, on the other, requires some mechanical connection between the external motor and the internal propeller-like device which might cause liquid leakage outside the channel (pipe). On the other hand, the external motor can control the amount of mechanical work that can be put into the fluid so the mixing can be more efficient. Thereby, the final selection of the system, either active or passive, might be the outcome of a compromise between system complexity and actual efficiency. In any case, the present study shows that the passive option could be, in some cases, an acceptable solution in terms of simplicity and efficiency.

Chapter 4

Tip clearance based mixing enhancement

4.1 Introduction

A recurrent subject in R&D for thermal engineering application purposes is the potential advantage of implementing tip clearances in heat sink configurations with fins, pin fins, etc., to reduce pressure drop (directly related to pumping power) without imposing a large penalty on heat transfer. This is important in applications in which the pumping power for the cooling fluid cannot be increased at will but, instead, it is prescribed by some global specifications, as it happens, for example, in avionics systems. In these environments, reducing the pressure drop is critical even if a small penalty is to be paid in terms of heat transfer performance. The reason is that the global nature of the system specifications does not account for local hot spots and, thereby, the thermal engineer needs to implement local solutions that allow for an appropriate cooling performance. This happens, also, when addressing the cooling of electronics equipment of computing centres, big data centres, etc.

From an engineering standpoint, this tip clearance approach has the advantage of being passive (no moving parts or external power sources are needed) and robust. The drawback is that a positive outcome, reduced pressure drop, is counterbalanced by a negative one, smaller heat transfer rates. Then, the critical design aspect in actual engineering applications is to find out a reasonable compromise between these two contradicting effects. In this context, reaching a detailed understanding of the flow topology involved may help to approach this goal that is the objective of the present study.

To the knowledge of the author, the first journal article published in this field was the one of Sparrow et al. [76] back in 1978. They considered a shrouded fin array with and without tip clearance. Later, Sparrow and Kaddle [77] addressed the same geometry from an experimental perspective. Other articles published up to 2010 are those of Wirtz et al. [78], Coetzer and Visser [79], Min et al. [80], Dogruoz et al. [81], Jeng [82], Rozati et al. [83], and Moores et al. [84].

More recently, Reyes et al. [66] carried out an experimental study on the effects of tip clearance on a micro channel based heat sink. The authors considered three different values for the clearance height and compared it to a reference case with no clearance present. The Reynolds number, Re , based on the hydraulic diameter of the flow passages was varied between 400 and 2600. The working fluid was water and the heat sink wall temperature was kept constant and equal to 70 °C. At the lowest Re , the authors found that pressure drop and heat transfer were very sensitive to the tip clearance. The optimum configuration was the one in which the tip clearance (500 μm) was equal to the fin height. In this case, the ensuing Nusselt number, Nu , was 0.83 times the reference Nu , while the pressure drop, ΔP , was 0.20 times the reference ΔP . However, for the highest Re , it was observed that both heat transfer and pressure drop showed a weak dependence on the tip clearance. Specifically, heat transfer degradation and pressure drop improvement rates at the higher Re were similar regardless of the tip clearance height.

Tullius et al. [85] addressed the problem of optimization of micro pin fins in mini channels. Their study was of a numerical nature and several clearance ratios were considered. They reported the contradicting effect of pressure losses versus heat transfer mentioned earlier. In particular, they showed that pressure drop had a greater sensitivity with regard to the problem governing parameters than heat transfer. Interestingly enough, they were able to generate a correlation for Nu that accounted for the tip clearance height as well.

An experimental study on the influence of the tip clearance in a group of micro-cylinders placed on a heat sink has been described by Liu et al. [39]. The authors concentrated on the low Re regime and for Re 400 and the optimum clearance they reported a 15% deterioration of heat transfer and a 50% reduction of pressure losses as compared to their reference case. Liang et al. [86] have extended these studies to the case in which the working fluid is non-Newtonian (power-law polymeric in their case) and found similar qualitative trends. Mei et al. [55] studied the effect of tip clearance in micro-pin type geometries

and reported heat transfer degradation rates of the order of 50% with reduction of pressure losses by a factor of about 4.

The effect of tip and lateral clearance on a new heat sink concept has been addressed by Ho et al. [87]. In this article, besides an experimental study, the authors proposed a semi-analytical model that could be used for designing purposes. An experimental study that accounts for, among other aspects, the influence that tip clearances have on the thermal performance of pin-fin arrays has been published by Li [88]. In this study, the authors were able to find linear correlations between thermal performance, Re and tip to outer surface distance. Jadhav and Balaji [89] have presented yet another experimental study and it is worth to note that they report in their conclusions that the effect of tip clearance on thermal performance is more pronounced at lower flow velocities. Finally, it is worth mentioning the computational study of Giri and Das [90]. These authors considered a shrouded rectangular fin array and they varied clearance spacing, fin spacing and Re . In their conclusions they suggested the presence of potentially optimum values for the clearance.

Summarizing, there is a common qualitative tendency in all the above mentioned references regardless of the type of geometry under consideration (square section pin fin, circular section pin fin, conventional straight fin, etc.). Namely: the main effect of tip clearance is to degrade heat transfer somewhat while alleviating pressure losses significantly. However, actual quantification of these effects appears to be very much problem dependent. So, in this context, the objective of the present study is to focus on a single, and very general, geometry, heat sink with straight fins and tip clearance, and clarify its actual flow topology. To do it, the study being presented is of a dual nature: experimental and numerical.

Obviously, it would have been much better to perform a fully experimental work. However, the geometry being considered was three dimensional and complex, and the most reliable information provided by the Particle Image Velocimetry technique, PIV, was limited to some specific planes that were not fully representative of the actual 3D topology. Then, the strategy was to perform a series of isothermal experiments on the test model and use the results to validate the 3D flow solver via comparison with local PIV data. Then, thus validated, the flow solver was applied to the thermal cases.

4.2 Experimental setup

The experimental model was manufactured out of methacrylate. It basically consisted of four square section channels connecting two stagnation chambers. The cross section dimensions of the channels were $10 \text{ mm} \times 10 \text{ mm}$ and their length was 250 mm . The upstream stagnation chamber had the following dimensions: length 200 mm , span 90 mm and depth 20 mm . The dimensions of the downstream stagnation chamber were: length 50 mm , span 90 mm and depth 20 mm . It could be observed that the depth of the stagnation chambers (20 mm) was twice the channel cross section height (10 mm). This means that the flow had to go through a forward step like structure before entering the channels. A sketch of the basic platform is shown in figure 4.1.

(A) View of the upstream and downstream stagnation chambers and channels.

(B) View of the system closed with the two inlet and two outlet lines. Only the fluid domain is shown.

FIGURE 4.1: Sketch of the experimental platform for configuration C0.

As starting point for the manufacturing we had a $540 \times 130 \times h$ mm methacrylate block, with h varying from 25 to 35 mm depending on the configuration. After milling the desired geometry with a 5 mm cutter it was necessary to correct surface imperfections in order to obtain a mirror-finish suitable for photography. So as to accomplish that task, a special diamond cutter was used to mill usually between 0.05 to 0.15 mm from the desired surfaces. Before and after pictures along with the said cutter are shown in figure 4.2.

(A) Diamond mill cutter.

(B) Before.

(C) After.

FIGURE 4.2

Three different models were manufactured: the so-called configurations C0, C1 and C2. Configuration C0 was the reference configuration because the top cover was pressed directly over the fins (no tip clearance present). In configurations C1 and C2 the covers were manufactured so as to allow for tip clearances, TC, of 5 mm and 10 mm respectively (50% and 100% of the fin height). These percentages were similar to those used in reference [66].

To homogenize the flow streamwise velocity in the upstream stagnation chamber, a series of five grids framed into a single piece were inserted right downstream of the inlet flow lines. The nets introduce a pressure drop proportional to the normal speed component, thus homogenizing it transversally. Immediately afterwards, a plastic honeycomb (manufactured with a 3D printer) was inserted in order to straighten the flow lines, avoiding velocity components in the YZ

plane. Figure 4.3 shows a photograph of the devices inserted into the model.

FIGURE 4.3: Photograph of the device for configuration C0 (left), and honeycomb and grid inserted into the upstream stagnation chamber (right).

The upstream stagnation chamber was fed with two water lines that discharged into the chamber perpendicular to the top cover. The inner diameter of each line was 8 mm. A sketch (top and side views, a schematics and a photograph) of the basic setup is presented in Figure 4.4. The top subplot of figure 4.4 provides information on the location of the planes (Pl1, Pl2, Pl3, Pl25, and Pl35) where PIV measurements were actually taken. Pl2 and Pl3 covered part of the stagnation chamber and the centre plane of the channels. Pl1, Pl25 and Pl35 covered part of the stagnation chamber and the tip clearance areas. The top subplot of figure 4.4 also provides information on the sections (S_{-3} to S_5) where actual comparison between PIV measurements and CFD results was carried out. Sections S_{-3} to S_{-1} were located in the stagnation chamber. Sections S_0 to S_5 were located on the channels between fins and on top of the fins.

(A) Detail of the experimental setup for configuration C2 and planes (P1, P2, P3, P25, and P35) and sections (S_{-3} to S_5) used for comparison between experimental and numerical results. Measurements are given in mm.

(B) Schematics of the installation.

(C) Close up view of the installation. The two crosses in bottom left figure mark the position of the pressure sensors.

FIGURE 4.4

Water was propelled using an ITT Totton centrifugal pump with a magnetic drive DC15/5". The flowmeter was a Siemens Sitrans FM MAG1100 equipped with a Sitrans FM MAG5000 transmitter. Its measurement uncertainty was $\pm 5\%$. Pressure differences were measured using a Gems Sensor. This sensor measured in the range of -1000 Pa to +1000 Pa with an uncertainty of 1 %. These were the specifications provided by the manufacturer. The authors measured, also, pressure differences up to 1500 Pa whose uncertainty of 1 % cannot be guaranteed by the sensor manufacturer. It is clear that measurement uncertainties should not be validated using numerical results. However, in this case, the numerical computations showed good agreement with the measurements in that range which suggests that, as an indication, not as a validation, the sensor could be reasonably used up to 1500 Pa. The actual pressure drop was measured in between the inlet and outlet lines. Water temperature was measured during the experiments and checked to be 20 °C with maximum deviations of the order of ± 0.5 °C.

Four volume flow rates (Q1 to Q4) were tested for each of the three configurations C0, C1 and C2. The actual volume flow rates figures (in litres per hour) are given in Table 4.1. Re for these volume flow rates and configuration C0 (Re was based on the hydraulic diameter and average velocity at the channels cross-section) are also shown in table 4.1. These Re are similar to those used in Reyes et al. [66]. Q1 to Q3 are in the laminar regime while Q4 could be considered to be close the beginning of the transitional regime.

Case key	Flow rate (l/h)	Re (C0)
Q1	53.5	416
Q2	83.6	650
Q3	110.3	858
Q4	167.2	1300

TABLE 4.1: Volume flow rates (Q1 to Q4) in litres per hour used in the experiment, and associated Re for configuration C0.

A Dantec Dynamics PIV system was used for characterization purposes. The pulsed laser system was a Nd:YAG 800 ms. The time lapse between pulses was adjusted manually for each case so as to achieve constant travel distances of 4-8 pixels ($1/4$ to $1/2$ of the smallest subinterrogation windows used, 16 px) independently of the case being tested. Naturally, this τ between pulses also varied depending on the

zone (chamber or channel). A first approximation of τ was done theoretically for scenario C0 and then adjusted if necessary for the cases C1 and C2. The values ranged from 1 to 20 ms.

The Dantec Dynamics Flow Sense 2ME camera had a resolution of 1600×1200 pixels. Hollow glass spheres HGS-10 whose diameter were $10 \mu\text{m}$ were used for flow seeding purposes. Synchronization between image capturing and flow illumination was achieved using the Dantec Dynamics Studio software. The size of the interrogation areas was $60 \text{ mm} \times 30 \text{ mm}$. These areas were partitioned into smaller areas by the PIV software to converge the re-computation of the flow field. The time span between laser pulses (required to collect a flow snapshot) was of the order of 5 ms. In the worst case scenario of the maximum flow velocity (configuration C0 at the highest flow rate), a particle would travel of the order of 0.6 mm between pulses that is a distance much smaller than the problem characteristic length of 10 mm (the fin height). That is: the problem characteristic length in the most critical case was of the order of 17 times the PIV spatial resolution. In the most favourable case, the ratio of the problem characteristic length to the PIV spatial resolution was 188. On figure FIG the entire setup is displayed.

FIGURE 4.5: Whole layout of the experimental setup.

Regarding the repeatability of the measurements, 300 frames were collected for each capture and 15 captures were taken for each of the 12 cases (four volume flow rates times three configurations). However, for a few cases, believed to be critical, 600 frames for each capture were taken instead. It was assumed that the worst scenario was the one with highest flow rate (Q4) and largest tip clearance (C2).

The reason is that the combination of highest flow rate and largest tip clearance lead to the stronger 3D effects. A very small Re (think, for instance, of the Stokes limit) or the situation with no tip clearance (configuration C0) would present nearly a 2D flow. In this case, the most critical measurement sections were S_{-1} , S_0 and S_1 (see Figure 4.4). Figure 4.6 shows the uncertainty bands for the horizontal velocity in this case (Q4/C2 in sections S_{-1} , S_0 and S_1) and the actual histogram of the measurements at a height of 20 mm above the lowest surface of the stagnation chamber. Afterwards it was checked that these bands represented the worst case as compared to the others.

FIGURE 4.6: Uncertainty bands and histograms of actual measurements for case Q4, C2 at sections S_{-1} , S_0 and S_1 . Histograms taken at a vertical coordinate 20 mm above the lowest surface of the stagnation chamber. U_∞ is the mean velocity (flow rate divided by the actual cross flow area).

4.3 Description of the flow solver and computational domain

The standard OpenFOAM® C++ library simpleFoam [91] has been used to simulate the steady laminar isothermal problem in the device described in the previous section. The set of continuity and momentum dimensionless equations solved was:

$$\tilde{\nabla} \cdot \tilde{\mathbf{v}} = 0 \quad (4.1)$$

$$\tilde{\mathbf{v}} \cdot \tilde{\nabla} \tilde{\mathbf{v}} = -\tilde{\nabla} \tilde{P} + \tilde{\nabla} \cdot \left(\frac{1}{Re} \tilde{\mu} \tilde{\nabla} \tilde{\mathbf{v}} \right) \quad (4.2)$$

where $\tilde{\mathbf{v}} = \tilde{u} \vec{i} + \tilde{v} \vec{j} + \tilde{w} \vec{k}$, stands for the dimensionless velocity in the x, y, and z coordinates (see Figure 4.1) while $\tilde{P}$ represents the dimensionless pressure. Additionally, for the non-isothermal cases, the following heat transfer equation was implemented for the temperature $\tilde{T}$:

$$\tilde{\mathbf{v}} \cdot \tilde{\nabla} \tilde{T} = \tilde{\nabla} \cdot \left(\frac{1}{RePr} \tilde{k} \tilde{\nabla} \tilde{T} \right) \quad (4.3)$$

The independent and dependent variables were made dimensionless as follows:

$$\tilde{x} = \frac{x}{D_H}, \quad \tilde{y} = \frac{y}{D_H}, \quad \tilde{z} = \frac{z}{D_H} \quad (4.4)$$

$$\tilde{u} = \frac{u}{U_\infty}, \quad \tilde{v} = \frac{v}{U_\infty}, \quad \tilde{w} = \frac{w}{U_\infty}, \quad \tilde{P} = \frac{P}{\rho_\infty U_\infty^2}, \quad \tilde{T} = \frac{T}{T_\infty} \quad (4.5)$$

where D_H is the reference length (the hydraulic diameter of the device cross-section in the region where the channels are located), U_∞ is the mean fluid velocity at that cross section, and ρ_∞ and T_∞ are the

water density and temperature at the inlet. The Reynolds and Prandtl numbers were defined as:

$$Re = \frac{\rho_{\infty} U_{\infty} D_H}{\mu_{\infty}}, \quad Pr = \frac{\mu_{\infty} C p_{\infty}}{k_{\infty}} \quad (4.6)$$

where μ_{∞} , k_{∞} and Cp_{∞} stand for the fluid viscosity, conductivity and specific heat evaluated at the inlet temperature. Finally, $\tilde{\mu}$ and $\tilde{k}$ stand for the fluid dimensionless viscosity and conductivity that are temperature dependent and characteristic of the working fluid. Expressions for water in eq. 4.7 and 4.8 have been obtained from Martin and Velázquez [44]:

$$\tilde{\mu}(\tilde{T}) = \frac{\mu(T)}{\mu_{\infty}} = 1 - 5.646(\tilde{T} - 1) + 12.259(\tilde{T} - 1)^2 \quad (4.7)$$

$$\tilde{k}(\tilde{T}) = \frac{k(T)}{k_{\infty}} = 1 + 0.786(\tilde{T} - 1) + 1.176(\tilde{T} - 1)^2 \quad (4.8)$$

The adaptation of the solver simpleFoam, which is based on a finite volume formulation (Ferziger and Milovan [92], OpenFOAM User Guide [93]) for a laminar steady flow was carried out, adding, in a very simple way, the heat flow equation 4.3 and creating additional functions for the transport dimensionless variables $\tilde{\mu}(\tilde{T})$ and $\tilde{k}(\tilde{T})$ that are evaluated for each iteration. The solver simpleFoam uses the pressure-velocity coupling SIMPLE (Semi-Implicit Pressure Linked Equations) algorithm proposed by Caretto et al. [94] to generate a pressure field that fulfils the continuity equation 4.1. This algorithm is organized into three steps: a) a guessed velocity field is obtained from the last iteration pressure field, b) a pressure correction equation is solved, and c) the pressure thus obtained is used to correct the velocity field in a way that the mass conservation equation is fulfilled. The under-relaxation factor used in this work for the simpleFoam solver (Jasak [95]) was 0.3. Other solution variables were under-relaxed by a factor of 0.7 to accelerate convergence.

The discretization schemes chosen for each term of equations 4.2 and 4.3 are the following: second order Gaussian spatial integration schemes for the diffusive terms and gradient terms, and linear upwind differencing interpolation schemes for the convective terms were chosen. The segregated linear systems obtained upon discretization of the problem were solved with different iterative methods

(Saad [96]): equations for the velocity were solved using a Gauss-Seidel smoothed solver, while a GAMG (Geometric Algebraic Multi Grid) algorithm was applied for the pressure and temperature equations. This algorithm solves the equation, first, on a coarse grid and then, step by step, increases the grid resolution. This mapping of the coarse solution onto the finer grid leads to good convergence rates. Gauss-Seidel schemes were chosen as smoothers (Moukalled et al. [97]). Absolute tolerances of 10^{-6} were selected for the problem.

To validate the implementations introduced in the standard simpleFoam solver to include the thermal equation 4.3 and its related boundary conditions, numerical tests on a squared channel of 10 mm height and 500 mm long were carried out imposing a dynamically developed flow of inlet Reynolds number 100 (based on the channel hydraulic diameter D_H) with a uniform inlet temperature of 35 °C. The channel walls were at a constant uniform temperature of 70 °C. The fluid was water with constant transport properties evaluated at the inlet temperature. For this simple geometry, the global Nusselt number of 2.98 for the thermally fully developed flow is well known (Shah and London [98]). The numerical results led to a constant Nusselt value of 2.79 reached at a dimensionless distance from the inlet ($x/D_H Re_{D_H} Pr$) of 0.07, that are in very good agreement with the results of Lyczkowsky et al. [99].

The description of the four computational domains tested numerically (see figure 4.7), both for the isothermal and thermal problem, is as follows. The inlet of the computational domains coincides with the entrance to the grids while the outlet matches S_5 (see figure 4.4) for all configurations C0, C1 and C2. The other boundaries of the computational domain match the walls of the device for each configuration. Additionally, a fourth computational domain, corresponding to a device with a tip clearance of 20 mm (200% of the fin height) also analysed in [66], named con configuration C3, was simulated. Neither the grid nor the honeycomb were modelled in the computational domains that made up the matrix of cases to be analysed. In a single case (described at the end of this section) the honeycomb was also modelled for comparison purposes. However, the computational cost associated to this particular case was so high that it was decided not to use it for the computation of the matrix of cases. Uniform boundaries conditions for the normal dimensionless velocity (corresponding to each of the four volumetric flows Q1, Q2, Q3 and Q4) and temperature for the thermal tests (of 35 °C) were imposed at the inlet, while standard boundary conditions of $\tilde{P} = 0$ and zero normal gradients for the other variables were used at the outlet. At the walls, a no

slip condition was imposed while for the thermal cases all walls were insulated except for the section of platform wall along the channel region (from S_0 to S_5), that was considered to be at a constant temperature of 70 °C ($\tilde{T} = 1.114$). Fluid properties for the isothermal test were calculated for a temperature of 20 °C. For clarification purposes, figure 4.8 shows in red colour the constant temperature walls, and in grey colour the adiabatic walls for configuration C0 and C2. Uniform structured meshes with cell sizes of $\Delta\tilde{x} = 0.05$, $\Delta\tilde{y} = \Delta\tilde{z} = 0.02$ for each domain were generated with the snappyHexMesh utility, leading to meshes of 17.5, 23.5, 26.4 and 40.5 millions of cells for configurations C0, C1, C2 and C3, respectively. Figure 4.7, shows a partial view of the C2 computational domain and half the mesh at the cross section, where the former view is stopped.

(A) Partial view of the computational domain for configuration C2.

(B) Half the mesh at the cross section where the left view of the domain is stopped.

FIGURE 4.7

(A)

(B)

FIGURE 4.8: Thermal boundary conditions for computational domains C0 (left) and C2 (right). Constant temperature walls depicted in red colour, adiabatic walls in grey color.

A sensitivity analysis was performed in two coarser grids for the thermal problem of configuration C2 domain with volumetric flow Q4 (C2-Q4). Comparison of global errors of the streamwise velocity profile at S_{-1} and of the total heat transfer results is shown in Table 4.2 taking as reference the results obtained with the final mesh (Mesh 3).

Mesh	Nc	$\Delta\tilde{x}$	$\Delta\tilde{y}$	$\Delta\tilde{z}$	ϵ_1	ϵ_2
#1	3.4×10^6	0.10	0.04	0.04	6.2 %	7.7 %
#2	7.7×10^6	0.075	0.03	0.03	2.3 %	1.4 %
#3	26.4×10^6	0.05	0.02	0.20	Ref.	Ref.

TABLE 4.2: Mesh sensitivity analysis for the thermal case C2-Q4. Three meshes (#1, #2, and #3) were considered. The finer mesh #3 was assumed to be the reference mesh. Table shows mesh data (Nc: number of cells and size of elements), relative discrepancy of the streamwise velocity profile at S_{-1} (ϵ_1), and relative discrepancy (ϵ_2) of the total heat transfer.

Computational times in excess of 60 hours were necessary to solve the thermal case C2-Q4 with the reference mesh on an Intel® Xeon E5-2620v3 processor of 2.4 GHz in a 128 Gb RAM workstation.

In addition to the computational domains described before, a more detailed domain for configuration C0 that included the complete device from the inlet to the outlet water line was developed. As the aim of this more extensive domain was to calculate numerically the pressure drop, the honeycomb structure was included. The final unstructured mesh consisted of 56 million of cells formed mostly by hexahedrons. Isothermal tests were carried out for this problem for Q1, Q2 and Q3 with similar boundary conditions at the inlets, outlets and walls as the ones explained before. Fluid properties were calculated for a temperature of 20 °C. Results did not differ significantly from those obtained with the mesh used in the matrix of computational cases.

(A)

(B)

FIGURE 4.9: Surface and colour map of section S_{-4} .

Also, the velocity profile of a transversal plane 10 mm downstream from the honeycomb, called S_{-4} , was built up from the information obtained via PIV in the mentioned streamwise planes (Pl_1 to Pl_{35}) and an auxiliary one from side to side of the stagnation chamber at half the height of it. All of these velocity vectors serve as support to generate a velocity surface in the whole section S_{-4} , which will be used as entry condition in the CFD solver. An example of the generation of the mentioned surface along with the vectors used (in red) is shown in figure 4.9. The idea being to compare the relevance of the entry condition in the final flow obtained inside the channels. A comparison between the resulting flow inside the channels (sections S_0 to S_4)

experimentally, numerically with the mapped entry and the numerically with the normalized entry is given in figure 4.10 for the cases C0-Q1 and C0-Q4.

FIGURE 4.10: Comparison experimental (blue) vs normalized (red) vs mapped (green) entries.

4.4 Comparison between experimental and numerical isothermal results

This comparison was carried out for all configurations and volume flow rates (a total of 12 cases) under isothermal conditions. However, in this section, and for the sake of brevity, only the most critical comparison case is presented. This case, that is the worst case scenario from de numerical computation point of view, is the one with the largest tip clearance C2 and highest volume flow rate Q4. Figures 4.11, 4.12, and 4.13 show the comparison between measured and computed profiles in the stagnation chamber (section S_{-1} and planes Pl1, Pl2 and Pl3), along the centreline of a channel (sections S_0 , S_1 , S_2 and S_3 for the plane Pl2), and on top of a fin (sections S_0 , S_1 , S_2 and S_3 for the plane Pl35). As it could be observed, there is a reasonably good agreement between computed and measured profiles that suggest that the computational tool thus formulated and developed can be used with confidence.

FIGURE 4.11: Comparison between measured and computed x velocity profiles for section S_{-1} (stagnation chamber) and planes PI1, PI2 and PI3 for configuration C2 at a volume flow rate Q4. Numerical results: red line. Experimental results: blue line with spread bands.

FIGURE 4.12: Counterpart of Figure 4.11 but, now, for the flow along a channel centre line. Numerical results: red line. Experimental results: blue line with spread bands.

FIGURE 4.13: Counterpart of Figure 4.11 but, now, for the flow along the top of a fin. Numerical results: red line. Experimental results: blue line with spread bands.

Also, the comparison has been performed at a global level in terms of the pressure drop. Discrepancies (ϵ) between the experiment and the numerical values obtained when using the domain with the largest number of cells described in section 4.3 are summarized in Table 4.3.

Flow rate	Re	ΔP_{exp} (Pa)	ΔP_{num} (Pa)	ϵ_3
Q1	416	57	55	3.6 %
Q2	650	130	123	5.4 %
Q3	858	195	180	7.7 %

TABLE 4.3: Comparison between experimental and numerical pressure drops (inlet to outlet water lines) for the isothermal configuration C0. Discrepancies (ϵ_3) are expressed in percentages.

It could be observed in figures 4.11, 4.12, and 4.13 that the larger discrepancies occur in section $S_{.1}$ that is located in the stagnation chamber upstream of the entrance of the channels. This larger discrepancy could be caused by the local flow topology in that region. Specifically, in a very short distance, streamlines have to change direction abruptly twice: horizontal streamlines parallel to the bottom floor of the stagnation chamber became vertical when climbing the vertical wall leading to the entrance of the channels; and once they get there they have to become horizontal again to align themselves to the channels. On top of that, the 3D geometry imposed a spanwise velocity that further complicated the flow field and the structure of the streamlines. At the end of section 4.3, it was stated that a 56 million cells mesh was used for some comparison purposes. Then, it was checked that, in this case, the comparison between experimental and numerical results led to smaller discrepancies than those observed in Figure 4.11 for section $S_{.1}$ in the stagnation chamber. However, the improvement was marginal for the sections located in the channels region. Then, because of the very large computational cost associated to the 56 million cells mesh and, also, because of the fact that the region of interest for this study was the channel region, it was decided to proceed with the planned computational domains.

As an example of the power of this analysis method, the behaviour of the current lines for the most restrictive case (C2-Q4) is shown in figure 4.14.

FIGURE 4.14: Current lines in C2-Q4.

4.5 Numerical results for the thermal cases and discussion

First of all, as it was indicated previously in section 4.3, one additional configuration (C3) has been added to the previous ones. This new configuration, which was analysed via CFD only, had a tip clearance of 20 mm. This value is twice the tip clearance of configuration C2. Global values of the Nusselt number and pressure drop per unit length are given in figures 4.15 and 4.16 respectively. The Nusselt

number, Nu , is defined as:

$$Nu = \frac{Q_W}{\frac{k_\infty}{H}(T_W - T_\infty)A_W} \quad (4.9)$$

where Q_W stands for the heat transfer rate (in W) across the heated bottom wall of the device, H is the height of the channels (10 mm) and A_W represents the total surface of the heated wall for each configuration, that is, 0.03 m^2 for C0 and 0.0425 m^2 for C1, C2 and C3.

FIGURE 4.15: Nu versus flow rate for configurations C0 to C3.

FIGURE 4.16: Pressure drop versus flow rate for configurations C0 to C3.

As expected, Nu grows along with the volume flow rate and decreases when the tip clearance increases. At the same time, the pressure drop also grows along with the volume flow rate and decreases when the tip clearance increases. However, from configuration C0 to C3 Nu decreases, typically, by factor of about 3, while the pressure drop decreases by a factor of about 30 (ten times as much).

The first question, now, is to understand whether the simultaneous decrease in both Nu and pressure drop is caused, only, by the fact that the flow is neither hydrodynamically nor thermally developed, or tip clearances play some specific role. To assess this question, some data are presented, first, in table 4.4. This includes, for each configuration and flow rate: cross section area, mean velocity, hydraulic diameter and Reynolds number based on the hydraulic diameter with properties evaluated at the inlet temperature of 35 °C. Furthermore, two columns are added containing the actual computed pressure drop and the estimated pressure drop in isothermal conditions following the method described by Shah and London [98] and summarized later by White [100].

C	Q	CS (cm ²)	U _∞ (cm/s)	Re	DH (cm)	ΔP _{CFD} (Pa)	ΔP _{est} (Pa)
C0	Q1	4.0	3.72	560	1.00	1.70	0.80
C0	Q2	4.0	5.81	874	1.00	2.98	1.36
C0	Q3	4.0	7.66	1154	1.00	4.23	1.91
C0	Q4	4.0	11.61	1748	1.00	6.70	3.30
C1	Q1	8.5	1.75	332	1.26	0.63	0.23
C1	Q2	8.5	2.73	518	1.26	1.10	0.39
C1	Q3	8.5	3.60	684	1.26	1.45	0.54
C1	Q4	8.5	5.46	1036	1.26	2.40	0.90
C2	Q1	13.0	1.14	320	1.86	0.20	0.07
C2	Q2	13.0	1.79	500	1.86	0.33	0.12
C2	Q3	13.0	2.36	660	1.86	0.48	0.17
C2	Q4	13.0	3.57	1000	1.86	0.80	0.30
C3	Q1	22.0	0.68	300	2.93	0.05	0.02
C3	Q2	22.0	1.06	466	2.93	0.08	0.03
C3	Q3	22.0	1.39	616	2.93	0.80	0.05
C3	Q4	22.0	2.11	932	2.93	0.21	0.08

TABLE 4.4: Cross section area, CS, mean velocity, U_∞, Re, hydraulic diameter and computed (ΔP_{CFD}) and estimated (ΔP_{est}) pressure drops for the 16 cases (C0 to C3, Q1 to Q4) that have been considered.

The method described in references [98] and [100] provides an estimate for the pressure drop in an isothermal channel in which the flow is developing but has not reached hydrodynamically developed conditions. Details of the method can be found in section 4.9 in reference [100]. The cross section geometries addressed in [98] are, among many others, rectangular. Unfortunately, no information is provided on a cross section shape as the one that is being addressed in this article. Then, to carry out the comparison, an equivalent cross section rectangular geometry was generated for each case presented in Table 4.4. For example, for the case C3-Q4 the cross section area is 22 cm² and the hydraulic diameter is 2.93 cm. Then, the equivalent rectangle that has the same area and hydraulic diameter is the one whose dimensions are 13.35 cm x 1.65 cm. In this case, the computed pressure drop with the actual geometry is 0.21 Pa, while the one estimated, using the equivalent geometry, with the method of White [100] and Shah and London [98] is 0.08 Pa. That is, the computed pressure drop doubles the estimated (approximate) value. Furthermore, the comparison presented in table 4.4 considers isothermal conditions for the pressure drop estimate ([91]-[44]) and thermal conditions for the pressure drop CFD computation. This means that in the case of

an isothermal CFD computation with larger viscosity and smaller Re , the pressure drop should be even higher (in laminar flows the pressure drop scales with the inverse of Re); thereby enlarging the difference between computed and estimated results. To further justify this statement, this particular C3-Q4 case has been computed in isothermal conditions and it was found that the actual pressure drop was 0.27 Pa. Then, this seems to suggest that the tip clearance modifies the flow topology in a way that causes the pressure drop to be significantly higher than it should be. It could be counter argued that this is so because of the presence of the four channels in the test geometry. However, it is to be noted that in this C3 configuration the total cross section area was 22 cm² while the cross section area of the channels was 4 cm² only.

The second way to evaluate the global results is connected to the heat transfer rates. To do it, the local Nusselt number (hD_H/k) along the x coordinate (see Figure 4.1) has been plotted as function of the Graetz number ($x/D_H Re_{D_H} Pr$). The properties needed to obtain k , Re_{D_H} and Pr were evaluated at the film temperature. This film temperature was the average between the bulk temperature (the average cross section temperature at coordinate x) and the wall temperature. The results obtained for all configurations and flow rates are presented in Figure 4.17.

FIGURE 4.17: Local Nusselt number as function of Graetz number.

The main aspect to be noted in the results resented in Figure 4.17 is that the local Nu vs Graetz curves tend to collapse into four different families that correspond, precisely, to the four configurations being considered. Then, it could be stated that the geometry is one of the leading governing parameters of the problem since no single Nu vs Graetz curve has been found that collapses all 16 cases that have been computed. That is, the previous results seem to suggest that the specific details of the geometry exert some influence upon the hydrodynamics and thermal behaviour of the problem and that the state of either hydrodynamics or thermal development is not the only explanation for what has been observed.

To illustrate the details of the flow field, figure 4.18 shows, first, the dimensionless modulus of the transverse velocity defined as:

$$\left| \tilde{u}_{yz} \right| = \frac{\left(u_y^2 + u_z^2 \right)^{1/2}}{u_\infty} \quad (4.10)$$

for the four configurations C0 to C4 at the same flow rate Q_4 in section S_0 (the inlet section of the channels, see figure 13, In each subplot of figure 4.18, the value of U_∞ used to render the transverse velocity dimensionless is the mean velocity corresponding to each selected configuration and flow rate (the flow rate divided by the cross-section area). The following aspects could be noted in the results presented in figure 4.18:

- The larger the tip clearance is, the larger the typical transverse velocities at section S_0 (referred to their own mean velocity) are. This occurs not only at the far left and far right sides of the cross section geometry but, also, on top of the fins.
- The above mentioned presence of larger transverse velocities means that flow accommodation induces a 3D flow topology that has the practical effect of transporting heat from the heated walls into the main bulk flow. Or to put it in another words: this 3D flow topology departs from the classical quasi-one dimensional flow structure of conventional channels.
- Stream-wise velocities (z direction) in configuration C0 are larger than, for example, in configuration C3 and this translates into higher heat transfer rates (higher Nu). This is obvious, but the important point is that the 3D structures (made apparent by the modulus of the transverse velocities) that appear in the configurations with tip clearances have the practical effect of alleviating Nu degradation.
- The larger the tip clearance, the larger the flow region (close to top wall) that is not affected by the local details of the flow around the fin tips. Transverse velocities in this region are very small and heat transport is expected to be small too. Then, there is a balance between of Nu alleviation in the area close to the fins and flow that goes nearly undisturbed close to the top wall. That is: there should be an optimum tip clearance for each flow rate that minimizes Nu losses and minimizes pressure drop as well (that are related to the tip clearance itself).

The next question is to ascertain for how long along the x stream-wise coordinate the distinct behaviour shown in Figure 4.18 is present. The answer is provided in Figure 4.19 where the counterpart of Figure 4.18 is shown at section S_1 . There, it could be observed that the transverse velocity structures get weaker very soon and that typical transverse velocities at this section S_1 are of the order of 6 times smaller than at section S_0 .

FIGURE 4.18: Dimensionless transverse velocity plots for configurations C0 to C3 and flow rate Q4 at section S₀.

FIGURE 4.19: Counterpart of Figure 4.18 at section S₁.

The cross-section temperature plots associated to the transverse velocity profiles of Figure 4.19 are presented in Figure 4.20. It is of interest to note the plumes of hot fluid that can be noticed at the bottom of the channels and on top of the fins (especially for configurations C1

and C2). Again, these plumes that transport heat from the walls into the main bulk of fluid could be explained because of the transverse velocity structures that were discussed before. Plumes can be observed, also, in configuration C0 but they are smaller than in C1 and C2. The area of heated flow at the bottom of the channels is largest in configuration C3 but the biggest portion of undisturbed cold flow is responsible for the overall smaller Nu.

FIGURE 4.20: Temperature plots for configurations C0 to C3 and flow rate Q4 at section S_1 .

The results presented in Figure 4.20 suggest that it is of interest to plot the evolution of the heat flux (W/m^2) at the bottom lines, side lines, and top lines of the fins as a function of the x coordinate along the channels. This is done for cases, C0-Q4, C1-Q4, C2-Q4, and C3-Q4 in figure 17 to 20. Figure 4.21 presents a sketch to show the key that has been used to name walls in figures 17 to 20.

The following aspects could be noted in these figures:

- Three different heat transfer regions could be discerned along the length of the channels for the four configurations: region A (from $x = 0.20$ m, the channel's entrance, up to approximately $x = 0.23$ m), region B ($x = 0.23$ m up to $x = 0.35$ m), and region C ($x = 0.35$ m up to $x = 0.45$ m, the channel's exit).
- The largest heat fluxes can be found in region A, and this applies to all heated walls W1 to W9. Unfortunately, this region A is

FIGURE 4.21: Sketch showing the key used to name walls in figures 17 to 20 below.

FIGURE 4.22: Local heat flux at the walls as a function of x for the case C0-Q4.

quite narrow (3 mm in the present case) which prevents from obtaining larger values of the global Nu . The smaller the gap is, the higher the heat fluxes are in this region, being a consequence of the larger average velocities.

- The Nu curves present a distinct behaviour in region B that is about 12 mm long (half the device length). For all configurations, the higher values of Nu are obtained for walls W9, W8, W6, W4, and W2. These are the side walls of the fins and the top wall of the fin located furthest away from the device symmetry

FIGURE 4.23: Local heat flux at the walls as a function of x for the case C1-Q4.

FIGURE 4.24: Local heat flux at the walls as a function of x for the case C2-Q4.

axis. The lower values of Nu are obtained for walls W7 and W3. These are the bottom walls of the channels. However, distinct behaviours are obtained for the top walls of the fins W5 and W1.

FIGURE 4.25: Local heat flux at the walls as a function of x for the case C3-Q4.

While for configuration C3 Nu values at these walls are high and equivalent to those of the side walls, for configurations C1 and C2, intermediate values of Nu are obtained. All this suggests that the sides of the fins are the most favourable places from the heat transfer point of view in these types of configurations with a small gap present.

- In region C, the other half of the device, Nusselt numbers tend to homogenize for all walls. There are still differences but they are much smaller than in region B.
- From the point of view of a practical thermal engineering design, region B is interesting because: a) it is large (it occupies half the device), b) its Nu, on the average, nearly doubles its limiting value on the second half of the device, and c) specific locations where Nu is distinctly higher (the fin's side walls) can be clearly identified so that, if needed, heat sources can be placed close to these high heat transfer regions. In this context, it is to be noted that the bottom channel walls quickly reach their limiting heat transfer behaviour, while the fin's side walls retain at higher Nu (three times as large) for nearly half of the device length.

Finally, visualization of the flow topology can be improved by

means of plotting 3D streamlines. This, again in done for case C2-Q4 and the results are presented in Figure 21. The view is from the device outlet looking towards the inlet (the viewer sees the incoming flow). The streamlines have been coloured according to the value of $|\tilde{U}_{yz}|$ defined in relation 4.10.

FIGURE 4.26: Flow streamlines (for the C2-Q4 thermal simulation) coloured by the transverse velocity defined in relation 4.10.

It could be observed that the streamlines fold over the top of the fins, and that transverse velocities in the regions close to the fin's side walls are large, thereby lending support to the observed higher Nu in these regions. The folding of the streamlines is also interesting because it contributes to increase transport in the bulk region of the flow.

4.6 Experimental transitional flow rates

During the experimental campaign, two additional flow rates were tested: Q5 and Q6. They correspond to Re numbers of 1959 and 2600 respectively, based on the mean velocity in the channels of device C0. The purpose was to complete the experimental matrix of cases tested by Reyes et. al. [66].

It is theoretically foreseeable that these Re numbers are in the transitional regime between laminar and turbulent flow, and as it was observed, the velocity fields these flow rates produced were also non stationary. It is because of this that it was not feasible, in process time terms, to simulate these cases numerically.

Nonetheless, there are some effects that occur at this flow rates that are worth mentioning. First of all, a recirculation bubble is formed at the beginning of the "floor" of the channels (walls W1, W3, W5, W7 and W9), which gets bigger along with the tip clearance. In second place, this recirculation bubble oscillates and tend to break spontaneously at the highest Re number and highest tip clearance (C2 Re2600).

For comparison purposes, two figures (Figure 4.27 and Figure 4.28) are being displayed, corresponding respectively to C1 and C2 devices. Each one has two columns associated to Re1959 and Re2600; and four rows, associated with planes Pl2, Pl25, Pl3 and Pl35.

FIGURE 4.27: Device C1.

FIGURE 4.28: Device C2.

In Figure 4.28, in the case of Pl_{25} and Re_{2600} (element (2,2) in the chart), it is better perceived the mentioned instability of the recirculation bubble, shedding vortex downstream.

4.7 Conclusions

- Even though the geometry that has been addressed is complex, the flow solver that has been generated for analysis purposes is reasonably accurate. Its accuracy has been assessed by comparing its results with a series of PIV visualizations obtained from dedicated experimental tests.
- It appears that the behaviour of both heat transfer and pressure drop on this device cannot be explained in view, only, of flow development and thermal development aspects. This is so because the Nusselt versus Graetz curves that have been generated do not collapse into a single fit; instead, they collapse into several families that are governed by the tip clearance parameter. This suggest that the specific geometries of typical tip clearance based configurations play a distinct role on the flow topology and, accordingly, on heat transfer and pressure drop.
- It has been found that, when looking at the local Nu , three distinct regions are present in the device. These are: a narrow (around 10% of the length) high Nu region located at the entrance of the channels, a homogenized Nu region located at the far end of the device ($\sim 45\%$), and an intermediate region of

moderately high Nu that occupies half of the device length. In it, Nu changes significantly depending of the location: bottom of the channels, sides of the fins, or top of the fins, having the side of the fins the best heat transfer rate by a factor of 3 when compared to the bottom of the channels. This makes the region B to have as much as twice the Nu of the region C. From a thermal engineering point of view, this region is interesting because it is large and because it shows a variety of heat transfer behaviours. Therefore, it is amenable for optimization (outside of the scope of the present article).

- Visualization of 3D streamlines shows that transverse velocities are high close to side of the fins, thereby lending support to the observed high Nu in these regions. Furthermore, downstream folding of these streamlines (not observed in a conventional quasi-one dimensional straight channel based heat sink) suggests that mixing could be enhanced by promoting this folding of the flow topology.

Chapter 5

Conclusions

The main objective of this brief chapter is, in a certain way, to summarize the overall conclusions of what has been said throughout the present thesis.

- In a square section channel, a fixed prism with a blockage ratio of 1:2.5 used as passive enhancing mixing system can improve Nu by around 66% in the laminar regime. Besides, if the prism is left "free", tethered to the walls upstream, the Nu will be affected. Depending on the m^* (density ratio with respect to water) of the prism, the Nu can be deteriorated (e.g. $m^* = 0.56$) or increased if high enough, even by a factor of 100% as compared with the clean channel in the case of $m^* = 0.91$.
- The effects of the interaction between the wall confinement, the prism and the fluid flow have been tested and thoroughly analysed from the standpoint of mixing enhancement, although the fluid mechanics basics governing the prism motion remain blurry. These principles are the ones in charge for the difference in Nu variation as function of the m^* , and thus the ones must be known in order to determine the optimum density ratio.
- It is clear that the introduction of a tip clearance is positive to the compromise of heat flux and pressure drop in the laminar regime. With case C0 as a reference, C1, C2 and C3 have Nu drops of around 20%, 35% and 65% respectively. Meanwhile, pressure drops are around 70%, 90% and 95%.
- Tip clearance, depending on its size, modifies the flow field not only quantitatively but topologically, the main outcome of which is that an optimum gap exists for each particular flow.

- There are up to three distinct channel regions, namely A, B and C, in which to heat transfer refers. Region A is very short ($\sim 10\%$) and in C the Nu has stabilized in a low asymptote, but region B, occupying around 45% of the channel length, may be the most important from a design standpoint because of its ability to evacuate heat from every wall conforming the channels, in special the vertical/lateral channel walls.
- Passive systems have been proved to be a suitable mixing enhancement method when dealing with low Re numbers in confined flows. Indeed, they have some advantages over active ones such as water/air tightness and design simplicity, to the detriment of a better control over the process and, thus, the effectiveness that active systems possess. Whether the final choice is an active or a passive system will be dictated by the proximity of the demand to a higher design simplicity or a better process control in each specific case of industrial application.

Chapter 6

Published articles

During the time this thesis was being written, two articles were published:

- F. Sastre & A. Velazquez, Experimental study on heat transfer enhancement caused by the flow-induced vibration of a prism inside a channel. *Applied Thermal Engineering* **125** (2017) 412–424. Q1 of the JCR.
- F. Sastre, A. Valeije, E. Martin & A. Velazquez, Experimental and numerical study on the flow topology of finned heat sinks with tip clearance. *International Journal of Thermal Sciences* **132** (2018) 146–160. Q1 of the JCR.

The results gathered from experiment described in chapter 3 were presented on EuroMech's European Fluid Mechanics Conference 11 in Sevilla:

- F. Sastre & A. Velazquez, Experimental study on the flow heat transfer past a self-oscilating prism, *EFMC11*, 12-16 September 2016.

Also, a mathematical tool for the better analysis of experimental data is being developed in collaboration with Soledad LeClainche and Jose Manuel Vega. The results of applying it to the data from the experiment described in chapter 3 are reflected in the conference paper:

- S. LeClainche, F. Sastre, J.M. Vega & A. Velazquez, Higher order dynamic mode decomposition applied to post-process a limited

amount of noisy PIV data, *47th AIAA Fluid Dynamic Conference*, 2017.

Chapter 7

Future work

During the elaboration of this thesis, numerous ideas arose from the experimental results and setups. Specially one of them is worth mentioning because it has led to other research project.

It has to do with the better comprehension of the effects of the prism motion in the first experiment. A first step would be to know beforehand the exact motion law of the prism. For this purpose, a forcing system was manufactured, which changed the nature of the fluid-structure interaction of the problem, but allowed the researchers to impose different movement laws to said prism. In this context, the resulting flow field could be studied in order to judge which configuration gave the best results in mixing effectiveness terms. This concept is being researched in the context of other PhD. thesis and one article is being sent to Experimental Thermal and Fluid Science: R. Clemente, F. Sastre, A. Barrero-Gil and A. Velazquez, "Mixing downstream of a moving square cylinder in the confined laminar regime: influence of different motion laws".

Bibliography

- [1] M. Reibold and P. Paufler. "Carbon nanotubes in an ancient Damascus sabre". In: *Nature* 444.7117 (2006), p. 286.
- [2] E.L. Paul, V.A. Atiemo-Obeng, and S.M. Kresta. *Handbook of industrial mixing: science and practice*. John Wiley & Sons, 2004.
- [3] R. Sturman, J.M. Ottino, and S. Wiggins. *The mathematical foundations of mixing: the linked twist map as a paradigm in applications: micro to macro, fluids to solids*. Vol. 22. Cambridge University Press, 2006.
- [4] P.J. Cullen. *Food mixing: Principles and applications*. John Wiley & Sons, 2009.
- [5] R.O. Grigoriev. "Mixing in laminar fluid flows: from microfluidics to oceanic currents". In: *Transport and Mixing in Laminar Flows: From Microfluidics to Oceanic Currents* (2011).
- [6] R. Mezaki, M. Mochizuki, and K. Ogawa. *Engineering data on mixing*. Elsevier, 1999.
- [7] A.W. Nienow, M.F. Edwards, and N. Harnby. *Mixing in the Process Industries*. Ed. by Butterworth-Heinemann. 1997.
- [8] G. Tatterson and R. Calabrese. "Industrial mixing research needs". In: *AIChE* (1989).
- [9] J.M. Funt. *Mixing of Rubber*. Smithers Rapra Perspectives Series. Smithers Rapra Technology, 2009.
- [10] O.M. Aamo and M. Krstic. *Flow Control by Feedback: Stabilization and Mixing*. Communications and Control Engineering. Springer London, 2013.
- [11] H. Bockhorn et al. *Micro and Macro Mixing: Analysis, Simulation and Numerical Calculation*. Heat and Mass Transfer. Springer Berlin Heidelberg, 2009.
- [12] R.K. Thakur et al. "Static mixers in the process industries—a review". In: *Chemical Engineering Research and Design* 81.7 (2003), pp. 787–826.
- [13] J.P. Brancher and J.C. Leprevost. "Laminar mixing induced by a twisted quadripolar Stokes flow". In: *International journal of heat and mass transfer* 47.6-7 (2004), pp. 1561–1567.

- [14] D.L. Hitt and M. McGarry. "Numerical simulations of laminar mixing surfaces in pulsatile microchannel flows". In: *Mathematics and Computers in Simulation* 65.4-5 (2004), pp. 399–416.
- [15] L. Bretonnet et al. "Deflection, drift, and advective growth in variable-density, laminar mixing layers". In: *Physics of Fluids* 19.10 (2007), p. 103601.
- [16] F. Chinesta and M.R. Mackley. "Microstructure evolution during liquid–liquid laminar mixing: a kinetic theory approach". In: *International Journal of Material Forming* 1.1 (2008), pp. 47–55.
- [17] S. Woziwodzki, L. Broniarz-Press, and M. Ochowiak. "Effect of eccentricity on transitional mixing in vessel equipped with turbine impellers". In: *Chemical Engineering Research and Design* 88.12 (2010), pp. 1607–1614.
- [18] S. Woziwodzki and L. Jedrzejczak. "Effect of eccentricity on laminar mixing in vessel stirred by double turbine impellers". In: *Chemical Engineering Research and Design* 89.11 (2011), pp. 2268–2278.
- [19] S. Woziwodzki. "Mixing of viscous Newtonian fluids in a vessel equipped with steady and unsteady rotating dual-turbine impellers". In: *Chemical Engineering Research and Design* 92.3 (2014), pp. 435–446.
- [20] K.C. Ng, E.Y.K. Ng, and W.H. Lam. "Lagrangian simulation of steady and unsteady laminar mixing by plate impeller in a cylindrical vessel". In: *Industrial Engineering Chemistry Research* 52.29 (2013), pp. 10004–10014.
- [21] K.C. Ng and E.Y.K. Ng. "Laminar mixing performances of baffling, shaft eccentricity and unsteady mixing in a cylindrical vessel". In: *Chemical Engineering Science* 104 (2013), pp. 960–974.
- [22] S. Hashimoto, R. Osaka, and M. Kawamata. "Analysis on laminar chaotic mixing based on configuration of streak lobes in an impeller-agitated vessel". In: *Industrial Engineering Chemistry Research* 51.19 (2012), pp. 6939–6947.
- [23] S. Devahastin and A.S. Mujumdar. "A numerical study of flow and mixing characteristics of laminar confined impinging streams". In: *Chemical Engineering Journal* 85.2-3 (2002), pp. 215–223.
- [24] S.J. Wang, S. Devahastin, and A.S. Mujumdar. "A numerical investigation of some approaches to improve mixing in laminar confined impinging streams". In: *Applied thermal engineering* 25.2-3 (2005), pp. 253–269.

- [25] S.J. Wang, S. Devahastin, and A.S. Mujumdar. "Effect of temperature difference on flow and mixing characteristics of laminar confined opposing jets". In: *Applied Thermal Engineering* 26.5-6 (2006), pp. 519–529.
- [26] P.L. Woodfield, K. Nakabe, and K. Suzuki. "Numerical study for enhancement of laminar flow mixing using multiple confined jets in a micro-can combustor". In: *International journal of heat and mass transfer* 46.14 (2003), pp. 2655–2663.
- [27] I. Rafferty and D. Kaminski. "Non-dimensionalization and mixing quantification of laminar twin semi-confined jets". In: *International Journal of Heat and Fluid Flow* 48 (2014), pp. 15–23.
- [28] J. Toonder et al. "Artificial cilia for active micro-fluidic mixing". In: *Lab on a Chip* 8.4 (2008), pp. 533–541.
- [29] X. Shang, X. Huang, and C. Yang. "Mixing enhancement by the vortex in a microfluidic mixer with actuation". In: *Experimental Thermal and Fluid Science* 67 (2015), pp. 57–61.
- [30] F.R. Phelan Jr, N.R. Hughes, and J.A. Pathak. "Chaotic mixing in microfluidic devices driven by oscillatory cross flow". In: *Physics of Fluids* 20.2 (2008), p. 023101.
- [31] J.P. Silva, A. Santos, and V. Semiao. "Experimental characterization of pulsed Newtonian fluid flows inside T-shaped micromixers with variable inlets widths". In: *Experimental Thermal and Fluid Science* 89 (2017), pp. 249–258.
- [32] V. Hessel et al. "Laminar mixing in different interdigital micromixers: I. Experimental characterization". In: *AIChE Journal* 49.3 (2003), pp. 566–577.
- [33] S. Hardt and F. Schonfeld. "Laminar mixing in different interdigital micromixers: II. Numerical simulations". In: *AIChE journal* 49.3 (2003), pp. 578–584.
- [34] S.M. Senn and D. Poulikakos. "Laminar mixing, heat transfer and pressure drop in tree-like microchannel nets and their application for thermal management in polymer electrolyte fuel cells". In: *Journal of Power Sources* 130.1-2 (2004), pp. 178–191.
- [35] M. Camesasca, I. Manas-Zloczower, and M. Kaufman. "Influence of extruder geometry on laminar mixing: entropic analysis". In: *Plastics, Rubber and Composites* 33.9-10 (2004), pp. 372–376.
- [36] L. Djenidi and B. Moghtaderi. "Numerical investigation of laminar mixing in a coaxial microreactor". In: *Journal of Fluid Mechanics* 568 (2006), pp. 223–242.
- [37] L. Wang et al. "Mixing enhancement of novel passive microfluidic mixers with cylindrical grooves". In: *Chemical engineering science* 81 (2012), pp. 157–163.

- [38] J.M. Zalc et al. "Characterization of flow and mixing in an SMX static mixer". In: *AIChE Journal* 48.3 (2002), pp. 427–436.
- [39] S. Liu, A.N. Hrymak, and P.E. Wood. "Laminar mixing of shear thinning fluids in a SMX static mixer". In: *Chemical Engineering Science* 61.6 (2006), pp. 1753–1759.
- [40] T. S. Pottebaum and M. Gharib. "Using oscillations to enhance heat transfer for a circular cylinder". In: *International Journal of Heat and Mass Transfer* 49.17-18 (2006), pp. 3190–3210.
- [41] A. Barrero-Gil and P. Fernandez-Arroyo. "Fluid excitation of an oscillating circular cylinder in cross-flow". In: *European Journal of Mechanics-B/Fluids* 29.5 (2010), pp. 364–368.
- [42] M. Meis et al. "Heat transfer enhancement in micro-channels caused by vortex promoters". In: *International Journal of Heat and Mass Transfer* 53.1-3 (2010), pp. 29–40.
- [43] M. Meis et al. "Experimental and numerical study on the heat transfer downstream of a confined rectangular cylinder in the laminar regime". In: *International Journal of Heat and Mass Transfer* 128 (2019), pp. 161–170.
- [44] E. Martin and A. Velazquez. "Effect of span length and temperature on the 3-D confined flow around a vortex promoter". In: *International Journal of Heat and Fluid Flow* 32.6 (2011), pp. 1173–1185.
- [45] V.M. Ribeiro et al. "Three-dimensional effects in laminar flow past a confined cylinder". In: *Chemical engineering science* 84 (2012), pp. 155–169.
- [46] A. Kumar and A. Dhiman. "Laminar flow and heat transfer phenomena across a confined semicircular bluff body at low Reynolds numbers". In: *Heat Transfer Engineering* 36.18 (2015), pp. 1540–1551.
- [47] E. Salcedo et al. "Unsteady mixed convection heat transfer from two confined isothermal circular cylinders in tandem: buoyancy and tube spacing effects". In: *International Journal of Heat and Fluid Flow* 60 (2016), pp. 12–30.
- [48] E. Salcedo et al. "Experimental study on laminar flow over two confined isothermal cylinders in tandem during mixed convection". In: *International Journal of Thermal Sciences* 115 (2017), pp. 176–196.
- [49] J. Ortega-Casanova. "Enhancing mixing at a very low Reynolds number by a heaving square cylinder". In: *Journal of Fluids and Structures* 65 (2016), pp. 1–20.
- [50] J. Ortega-Casanova. "CFD study on mixing enhancement in a channel at a low Reynolds number by pitching a square cylinder". In: *Computers & Fluids* 145 (2017), pp. 141–152.

- [51] J. Ortega-Casanova. "On the onset of vortex shedding from 2D confined rectangular cylinders having different aspect ratios: Application to promote mixing fluids". In: *Chemical Engineering and Processing: Process Intensification* 120 (2017), pp. 81–92.
- [52] J. Ortega-Casanova and C.H. Lai. "CFD study on laminar mixing at a very low Reynolds number by pitching and heaving a square cylinder". In: *Computers & Fluids* 168 (2018), pp. 318–327.
- [53] M. Reyes et al. "Experimental study on the confined 3D laminar flow past a square prism with a high blockage ratio". In: *International Journal of Heat and Fluid Flow* 44 (2013), pp. 444–457.
- [54] M. Reyes et al. "Experimental study on the flow regimes past a confined prism undergoing self-sustained oscillations". In: *International Journal of Heat and Fluid Flow* 54 (2015), pp. 65–76.
- [55] D. Mei et al. "Effect of tip clearance on the heat transfer and pressure drop performance in the micro-reactor with micro-pin-fin arrays at low Reynolds number". In: *International Journal of Heat and Mass Transfer* 70 (2014), pp. 709–718.
- [56] R. Roth et al. "Heat transfer in freestanding microchannels with in-line and staggered pin fin structures with clearance". In: *International Journal of Heat and Mass Transfer* 67 (2013), pp. 1–15.
- [57] B.R. Far et al. "Effects of pin tip-clearance on the performance of an enhanced microchannel heat sink with oblique fins and phase change material slurry". In: *International Journal of Heat and Mass Transfer* 83 (2015), pp. 136–145.
- [58] J. Pandit et al. "Effect of pin fin to channel height ratio and pin fin geometry on heat transfer performance for flow in rectangular channels". In: *International Journal of heat and mass transfer* 77 (2014), pp. 359–368.
- [59] H. Zhao et al. "Pressure drop and friction factor of a rectangular channel with staggered mini pin fins of different shapes". In: *Experimental Thermal and fluid science* 71 (2016), pp. 57–69.
- [60] D. Yang et al. "Numerical and experimental analysis of cooling performance of single-phase array microchannel heat sinks with different pin-fin configurations". In: *Applied Thermal Engineering* 112 (2017), pp. 1547–1556.
- [61] H.K. Park and B.J. Chung. "Optimal tip clearance in the laminar forced convection heat transfer of a finned plate in a square duct". In: *International Communications in Heat and Mass Transfer* 63 (2015), pp. 73–81.

- [62] Z. Shi and T. Dong. "Entropy generation and optimization of laminar convective heat transfer and fluid flow in a microchannel with staggered arrays of pin fin structure with tip clearance". In: *Energy Conversion and Management* 94 (2015), pp. 493–504.
- [63] T. Yeom et al. "Enhanced heat transfer of heat sink channels with micro pin fin roughened walls". In: *International Journal of Heat and Mass Transfer* 92 (2016), pp. 617–627.
- [64] M. Liu et al. "Experimental study on liquid flow and heat transfer in micro square pin fin heat sink". In: *International Journal of Heat and Mass Transfer* 54.25-26 (2011), pp. 5602–5611.
- [65] Z. Liu, N. Guan, and C. Zhang. "Influence of tip clearance on heat transfer efficiency in micro-cylinders-group heat sink". In: *Experimental Thermal and Fluid Science* 46 (2013), pp. 64–73.
- [66] M. Reyes et al. "Experimental study of heat transfer and pressure drop in micro-channel based heat sinks with tip clearance". In: *Applied Thermal Engineering* 31 (2011), pp. 887–893.
- [67] B. Celik and A. Beskok. "Mixing induced by a transversely oscillating circular cylinder in a straight channel". In: *Physics of Fluids* 21.7 (2009), p. 073601.
- [68] W. S. Fu and B. H. Tong. "Effects of eccentricity of cylinder and blockage ratio on heat transfer by an oscillating cylinder in a channel flow". In: *Int. Comm. Heat Mass Transfer* 30 (2003), pp. 401–412.
- [69] W. S. Fu and B. H. Tong. "Numerical investigation of heat transfer of a heated channel with an oscillating cylinder". In: *Numerical Heat Transfer* 43 (2003), pp. 639–658.
- [70] Y. T. Yang and C. H. Chen. "Numerical simulation of turbulent fluid flow and heat transfer characteristics of heated blocks in the channel with an oscillating cylinder". In: *International Journal of Heat and Mass Transfer* 51 (2008), pp. 1603–1612.
- [71] J. Ghazanfarian and M. R. H. Nobari. "A numerical study of convective heat transfer from a rotating cylinder with cross-flow oscillation". In: *International Journal of Heat and Mass Transfer* 52 (2009), pp. 5402–5411.
- [72] M. R. H. Nobari and J. Ghazanfarian. "Convective heat transfer from a rotating cylinder with inline oscillation". In: *International Journal of Thermal Sciences* 49 (2010), pp. 2026–2036.
- [73] M. Schafer and M. S. Turek. "Flow Simulation with High Performance Computers II". In: ed. by E.H. Hirshel. Vol. 52. Vieweg, 1996. Chap. Benchmark computations of laminar flow around a cylinder, pp. 547–566.

- [74] D. T. Sandwell. "Biharmonic spline interpolation of GEOS-3 and SEASAT altimeter data". In: *Geophysical Research Letters* 2 (1987), pp. 139–142.
- [75] F. P. Incropera and D. P. DeWitt. *Introduction to heat transfer*. Ed. by John Wiley & Sons. 1996.
- [76] E. M. Sparrow, B. R. Baliga, and S. V. Patankar. "Forced convection heat transfer from a shrouded fin array with and without tip clearance". In: *Journal of Heat Transfer* 100 (1978), pp. 572–579.
- [77] E. M. Sparrow and D. S. Kadle. "Effect of tip to shroud clearance on turbulent heat transfer from a shrouded, longitudinal fin array". In: *Journal of Heat Transfer* 108 (1986), pp. 519–524.
- [78] R. A. Wirtz, W. Chen, and R. Zhou. "Effect of flow bypass on the performance of longitudinal fin heat sinks". In: *Journal of Electronic Packaging* 116 (1994), pp. 206–211.
- [79] C. B. Coetzer and J. A. Visser. "Compact modelling of forced flow in longitudinal fin". In: *Journal of Electronic Packaging* 125 (2003), pp. 319–324.
- [80] J. Y. Min, S. P. Jang, and S. J. Kim. "Effect of tip clearance on the cooling performance of a micro-channel heat sink". In: *International Journal of Heat and Mass Transfer* 47 (2004), pp. 1099–1103.
- [81] M. B. Dogruoz, M. Urdaneta, and A. Ortega. "Experiments and modeling of the hydraulic resistance and heat transfer of in-line square pin heat sinks with top by-pass flow". In: *International Journal of Heat and Mass Transfer* 48 (2005), pp. 5058–5071.
- [82] T. M. Jeng. "A porous model for the square pin-fin heat sink situated in a rectangular channel with laminar side-bypass flow". In: *International Journal of Heat and Mass Transfer* 51 (2008), pp. 2214–2226.
- [83] A. Rozati, D. K. Tafti, and N. E. Blackwell. "Effect of pin tip clearance on flow and heat transfer at low Reynolds numbers". In: *Journal of Heat Transfer* 130 (2008), pp. 1–10.
- [84] K. A. Moores, J. Kim, and Y. K. Joshi. "Heat transfer and fluid flow in shrouded pin fin arrays with and without tip clearance". In: *International Journal of Heat and Mass Transfer* 52 (2009), pp. 5978–5989.
- [85] J. F. Tullius, T. K. Tullius, and Y. Bayazitoglu. "Optimization of short micro pin fins in minichannels". In: *International Journal of Heat and Mass Transfer* 55 (2012), pp. 3921–3932.
- [86] C. Liang, J. R. Arias, and A. Velazquez. "Tip Clearance Effects on Microchannel-Based Heat Sink with Polymeric Fluid". In: *Journal of Thermophysics and Heat Transfer* 30.2 (2016).

- [87] J. Y. Ho et al. "Convective heat transfer performance of airfoil heat sinks fabricated by selective laser melting". In: *International Journal of Thermal Sciences* 114 (2017), pp. 213–228.
- [88] X. J. Li, J. Z. Zhang, and X. M. Tan. "Effects of piezoelectric fan on overall performance of air-based micro pin-fin heat sink". In: *International Journal of Thermal Sciences* 126 (2018), pp. 1–12.
- [89] R. S. Jadhav and C. Balaji. "Fluid flow and heat transfer characteristics of a vertical channel with detached pin-fin arrays arranged in staggered manner on two opposite endwalls". In: *International Journal of Thermal Sciences* 105 (2016), pp. 57–74.
- [90] A. Giri and B. Das. "A numerical study of entry region laminar mixed convection over shrouded vertical fin arrays". In: *International Journal of Thermal Sciences* 60 (2012), pp. 212–224.
- [91] www.openfoam.com.
- [92] J. H. Ferziger and M. Peric. *Computational Methods for Fluid Dynamics*, 2nd Ed. Springer, 1999.
- [93] OpenFOAM Foundation. *OpenFOAM User Guide. Version 2.3.1, 3rd Ed.* 2014.
- [94] L. S. Caretto et al. "Two Calculation Procedures for Steady Three-Dimensional Flows with Recirculation". In: *International Conference of Numerical Methods in Fluid Dynamics* (1972).
- [95] H. Jasak. "Error Analysis and Estimation for the Finite Volume Method with Applications to Fluid Flows". PhD thesis. 1996.
- [96] Y. Saad. *Iterative Methods for Sparse Linear Systems*, 2nd Ed. Ed. by Society for Industrial and Applied Mathematics. 2003.
- [97] F. Moukalled, L. Mangani, and M. Darwish. *The Finite Volume Method in Computational Fluid Dynamics: An Advanced Introduction with OpenFOAM® and Matlab*. Springer, 2016.
- [98] R. K. Shah, A. L. London, and F. M. White. *Laminar Flow Forced Convection in Ducts*. New York: Academic Press, 1978.
- [99] R. W. Lyckowski, C. W. Solbrig, and D. Gidaspow. "Forced convective heat transfer in rectangular ducts - General case of wall resistances and peripheral conduction". In: Institute of Gas Technology, Tech. Info. Center, 1969. Chap. File 3229.
- [100] F. M. White and I. Corfield. *Viscous Fluid Flow*. New York: McGraw-Hill, 1991.