

École polytechnique de Louvain

Cooling electronic components using electrohydrodynamically induced convection

Authors : **Javier SALGADO GONZÁLEZ**
Supervisors : **Valérie GELBGRAS, Miltiadis V. PAPALEXANDRIS**
Readers : **Philippe CHATELAIN**
Academic year 2018–2019
Master [120] in Mechanical Engineering

ABSTRACT

The rapid growth of the development of electronic components makes them more and more powerful and smaller. Despite being very reliable components due to the absence of moving parts, its reliability and lifespan can be reduced owing to failures caused by overheating. Thus, new efficient cooling techniques should be studied to guarantee the correct operation and lifespan of this type of components.

An electrohydrodynamic system is studied in the following Master Thesis based on the experiment of Moghanlou, F. S. et al. (2014): *Experimental study on electrohydrodynamically induced heat transfer enhancement in a minichannel*. A CFD analysis is performed in order to determine the effect of electric field on heat transfer enhancement and pressure drop for different Reynolds numbers for laminar flow through a square minichannel. The device is composed of three different parts: inlet, outlet and test section. The fluid passes through the inlet section in order to become hydrodynamically fully developed. The test section is composed of a copper wire located at the top simulating the high voltage electrode, and a heated plate located at the bottom as the electronic component to be cooled down. Besides, the bottom of the test section (the heated plate) is considered as ground. The walls of the minichannel are thermally and electrically insulated. The high voltage electrode injects charge through the fluid, producing a secondary flow towards the ground (heated plate). The neutral molecules of the fluid are pushed by this secondary flow, thus the velocity profile of the flow is modified.

Some different numerical simulations with the ANSYS Fluent software are performed in order to study the electrohydrodynamic physics and determine the effects of the injection of charge in the flow. Some different parameters are studied with the purpose of analysing the heat transfer enhancement and the pressure drop and explain the behaviour of the electrodynamic device.

By analysing the results, it can be concluded that the effects of the electric field increase when the Reynolds number decreases. In fact, for the scenarios with the lower Reynolds number, the modification of the fluid is more significant. A recirculation phenomenon is observed at the outlet of the test section. The electric field contributes negatively to the momentum equation at this part of the minichannel, reintroducing fluid towards the test section and not contributing to the heat transfer enhancement. An increase of pressure drop is observed with the appliance of the voltage and this augmentation rises with the voltage applied, and thus with the strength of the electric field.

PEC values performed shows that the 2D configuration is efficient for scenarios with 5 kV and 10 kV applied at the wire. PEC values increase with the voltage. The recirculation problem becomes more noticeable when the voltage applied is greater and thus, with the strength of the electric field. Regarding the 3D model, none of the scenarios shows a PEC value greater than unity so the overall efficiency of the cooling technique studied is lower. However, by performing a segmented temperature analysis of the heated plate, the temperature increase due to the recirculation phenomenon is localised at the end part of the plate, and the maximum temperatures reached for the first part of the plate are lower. This temperature reduction is perfectly noticed in scenarios with the lower Reynolds number and less visible in the scenarios with the higher Reynolds number.

ACKNOWLEDGEMENTS

I would like to thank my thesis advisor Mr. Papalexandris and Mrs. Gelbgras for the guidance in the realisation of this work. Their tips and advice are been fundamental to accomplish the objectives defined. I would also thanks Mr. Chatelain to be the third jury member of the committee and to read this Master Thesis.

In addition, I would like to acknowledge my family and my girlfriend for all the support offered during these months, and during all the degree and master studies.

Finally, I would like to thank my colleagues for the comfortable working environment in which I have developed this Master Thesis.

Javier Salgado González

CONTENTS

Abstract	i
Acknowledgements	iii
List of figures	ix
List of tables	xiii
List of acronyms	xv
1. Introduction.....	1
1.1 Context and motivation.....	1
1.2 Objectives	2
1.3 Thesis structure	2
2. State of the art	5
2.1 General heat transfer concepts.....	5
2.2 Introduction and description of electronics components: overheating problems	8
2.3 Current electronic components cooling methods	12
2.4 Applications related to Electrohydrodynamics: Meso/micropumps: EHD pumps	16
2.4.1 Mechanical micropumps	16
2.4.2 Non mechanical or dynamic micropumps.....	16
2.4.3 EHD micropumps.....	17
2.5 Dielectric fluids for cooling: mineral oils	19
2.6 Social impacts	21
2.7 Economic impacts.....	22
2.8 Technological impacts	23
3. Governing equations	25
3.1 Hydrodynamics.....	25
3.2 Electrostatics	26
3.3 Electrohydrodynamics.....	28
4. Simulation software: ANSYS Fluent.....	29
4.1 ANSYS Fluent	29

4.2	Models.....	29
4.3	User-Defined Scalar transport equations and User-Defined Functions.....	30
4.4	Discretization of the domain: mesh generation.....	30
4.5	Pressure-Based Solver. Coupled Algorithm.....	31
4.6	Convergence criteria	32
4.7	Boundary conditions	32
4.8	Computational resources	33
5.	Methodology	35
5.1	General overview	35
5.2	Analysis parameters	35
5.3	Geometry.....	36
5.3.1	2D simulations.....	36
5.3.2	3D simulations.....	37
5.4	Meshing.....	38
5.4.1	Mesh parameters for 2D simulations.....	38
5.4.2	Mesh parameters for 3D simulations.....	39
5.5	Solver Settings	40
5.6	Assumptions	41
5.7	Input data	43
5.7.1	2D simulations: scenarios.....	43
5.7.2	3D simulations: scenarios.....	43
5.8	UDF and UDS implementation	43
5.9	Limitations	45
6.	Results and discussion.....	47
6.1	Parametric study: general comments	47
6.2	2D simulations.....	48
6.2.1	2D Parametric study: General comments	48
6.2.2	2D Parametric study: scenarios 1.a to 1.d.....	50

6.3	3D simulations.....	58
6.3.1	3D Parametric study: General comments	58
6.3.2	3D Parametric study: Mesh study	58
6.3.3	3D Parametric study: scenarios 3.a to 3.d.....	61
6.3.4	3D Parametric study: scenarios 3.e to 3.h	71
6.4	Weak points.....	79
7.	Conclusions.....	81
8.	Bibliography.....	85

LIST OF FIGURES

Figure 1 — Conduction heat transfer [26]	5
Figure 2 — Natural and forced convection [9]	6
Figure 3 — Relation between ambient temperature of electrolytic condenser and life [11]	9
Figure 4 — Relation between ambient temperature of semiconductor and failure rate [11]	9
Figure 5 — Example of chip carrier [20]	10
Figure 6 — Common PCB [22]	10
Figure 7 — Cooling load equal to power consumption [17]	11
Figure 8 — Temperature change of an electronic component with time [17]	12
Figure 9 — Heat fluxes that can be removed at specified temperature with certain heat transfer mechanisms [17]	13
Figure 10 — Heat extraction of a chip carrier [17]	13
Figure 11 — Natural convection mechanism [17]	14
Figure 12 — Indirect liquid cooling system [14]	15
Figure 13 — Classification of meso/micro pumps [27]	16
Figure 14 — Example: Debiotech's Insulin Nanopump [57]	16
Figure 15 — Injection pump configuration [36]	17
Figure 16 — Travelling wave induction pump configuration [24]	18
Figure 17 — Induction pump configuration [27]	18
Figure 18 — Heterocharge layers and different types of electrode configuration [27]	19
Figure 19 — Hydrocarbons in the mineral oil [37]	21
Figure 20 — Energy sources [53]	22
Figure 21 — Secondary flow velocity profile [71]	27
Figure 22 — Overview of Pressure-Based Solution Methods [66]	32
Figure 23 — Model 2D: general view: Plane XY	36
Figure 24 — Model 3D: general view and bottom view	37
Figure 25 — Model 3D: mesh-cross sectional view	39
Figure 26 — Mesh: test zone	40

Figure 27 — Scenario 1.d: Electric current vectors coloured by charge density (C/kg) injected from the wire and collected by the heated plate ($t = 10$ s). Plane XY, $x = 0.070$ m. Wire and heated plate right edge.....	48
Figure 28 — y-component of electric field (E_y): (a) $t = 0.005$ s. (b) $t = 10$ s. (c) $t = 20$ s. (d) $t = 26$ s. Plane XY, $x = 0.070$ m. Wire and heated plate right edge.....	49
Figure 29 — Scenario 1.d: Induced charges near the electrode. Plane XY, $x = 0.070$ m, $t = 26$ s	50
Figure 30 — Scenario 1.a: u-velocity profile at the middle plane. Plane XY, $t = 22$ s.....	50
Figure 31 — Scenario 1.a: v-velocity profile at the middle plane. Plane XY, $t = 5$ s	51
Figure 32 — Scenario 1.a: Temperature contour at the middle plane (XY plane). Plane XY, $t = 22$ s... 51	51
Figure 33 — Scenario 1.a: Pressure contour at the middle plane. Plane XY, $t = 22$ s.	51
Figure 34 — Scenarios 1.b to 1.d: Average temperature and maximum temperature of the heated plate.....	52
Figure 35 — Scenarios 1.b to 1.d: (a) Temperature measured at point T2 (22, -2.5, 0) mm. (b) Temperature measured at point T3 (45, -2.5, 0) mm. (c) Temperature measured at point T4 (68, -2.5, 0) mm.	53
Figure 36 — Scenario 1.d: x- component of electric field (V/m) along the minichannel. Plane XY	53
Figure 37 — Scenario 1.d: Different contours at $t = 16$ s along de minichannel: (a) x- component of electric field (V/m). (b) Charge density (C/kg). (c) Temperature (K). (d) u-velocity component (m/s). Plane XY, $t = 16$ s.	54
Figure 38 — Scenario 1.d: Different contours at $t = 26.5$ s along de minichannel: (a) x- component of electric field (V/m). (b) Charge density (C/kg). (c) Temperature (K). (d) u-velocity component (m/s). Plane XY, $t = 26.5$ s.	55
Figure 39 — Scenario 1.d: Different contours at $t = 16$ s along de minichannel: (a) y- component of electric field (V/m). (b) v-velocity component (m/s). Plane XY, $t = 16$ s.....	55
Figure 40 — Ratio of Nu/Nu_0 vs. applied voltage (kV) for scenarios 1.a to 1.d	56
Figure 41 — Ratio of $\Delta P/\Delta P_0$ vs. applied voltage (kV) for scenarios 1.a to 1.d	57
Figure 42 — PEC vs. applied voltage (kV) for scenarios 1.a to 1.d	57
Figure 43 — Mesh study: Details of different mesh studied at the outlet section: (a) Case 2.1. (b) Case 2.2. (c) Case 2.3	58
Figure 44 — Mesh study: temperatures measured at different point locations defined in Table 11. (a) Case a. (b) Case b. (c) Case c.	60
Figure 45 — Scenario 3.a: Velocity u profile along the minichannel. XY planes at $x = 0.004$ m, $x = 0.029$ m, $x = 0.054$ m, $t = 22$ s.....	61

Figure 46 — Scenario 3.a: v-velocity profile along the minichannel. XY planes at $x = 0.0165$ m, $x = 0.029$ m, $x = 0.0415$ m, $x = 0.054$ m and $x = 0.064$ m, $t = 22$ s.	61
Figure 47 — Scenario 3.a: v-velocity profile along the minichannel. (a) XY plane, $x = 0.0165$ m, (b) XY plane, $x = 0.029$ m, (c) XY plane, $x = 0.0415$ m, (d) XY plane, $x = 0.054$ m, (e) XY plane, $x = 0.064$ m, (f) XY plane, $x = 0.074$ m, $t = 22$ s.	62
Figure 48 — Scenario 3.a: Temperature contour of the heated plate reached the steady state. Top view, plane XZ, $t = 21$ s.	63
Figure 49 — Scenario 3.a: Temperature contour along the minichannel. YZ planes at $x = 0.004$ m, $x = 0.0165$ m, $x = 0.029$ m, $x = 0.0415$ m, $x = 0.054$ m and $x = 0.064$ m, $t = 22$ s.	64
Figure 50 — Scenario 3.a: Pressure contour along the minichannel. YZ planes at $x = 0.004$ m, $x = 0.0165$ m, $x = 0.029$ m, $x = 0.0415$ m, $x = 0.054$ m and $x = 0.064$ m, $t = 22$ s.	64
Figure 51 — Scenarios 3.b to 3.d: (a) Average temperature of the heated plate and (b) maximum temperature of the heated plate	65
Figure 52 — Scenarios 3.b to 3.d: (a) Temperature measured at point T2 (4, -2.5, 0) mm. (b) Temperature measured at point T3 (29, -2.5, 0) mm. (c) Temperature measured at point T4 (54, -2.5, 0) mm.	65
Figure 53 — Scenario 3.d: Electric field x - component. $t = 0.1$ s.	66
Figure 54 — Scenario 3.d: Electric field y - component. $t = 0.1$ s.	66
Figure 55 — Scenario 3.d: Electric field z - component. $t = 0.1$ s.	66
Figure 56 — Scenario 3.d: Charge density. $t = 0.1$ s.	67
Figure 57 — Scenario 3.d: Charge density. $t = 26$ s.	67
Figure 58 — Scenario 3.b: Recirculation problems. YZ plane at the outlet of the test section $x = 0.054$ m. $t = 26$ s.	68
Figure 59 — Scenario 3.d: Recirculation problems. YZ plane at the outlet of the test section $x = 0.054$ m. $t = 26$ s.	68
Figure 60 — Scenario 3.d: velocity modification at the middle plane of the minichannel. Velocity vectors coloured by temperature Plane XY, $x = 0.070$ m. $t = 26$ s.	69
Figure 61 — Scenario 3.b: u-velocity modification at the middle plane of the minichannel. Plane XY, $t = 26$ s.	69
Figure 62 — Scenario 3.b: Temperature contour of the heated plate. Top view, plane XZ, $t = 26$ s.	69
Figure 63 — Scenario 3.b: Joule Heat source at the heated plate. Plane XY, $t = 26$ s.	70
Figure 64 — Scenario 3.b: Joule heat source at the middle plane. XY planes at $x = 0.004$ m, $x = 0.029$ m, $x = 0.054$ m, $t = 26$ s.	71

Figure 65 — Scenario 3.e: v_{velocity} profile along the minichannel. XY planes at $x = 0.0165 \text{ m}$, $x = 0.029 \text{ m}$, $x = 0.0415 \text{ m}$, $x = 0.054 \text{ m}$ and $x = 0.064 \text{ m}$, $t = 11 \text{ s}$.	72
Figure 66 — Scenario 3.e: Temperature contour along the minichannel. YZ planes at $x = 0.004 \text{ m}$, $x = 0.0165 \text{ m}$, $x = 0.029 \text{ m}$, $x = 0.0415 \text{ m}$, $x = 0.054 \text{ m}$ and $x = 0.064 \text{ m}$, $t = 13 \text{ s}$.	72
Figure 67 — Scenario 3.e: Pressure contour along the minichannel. YZ planes at $x = 0.004 \text{ m}$, $x = 0.0165 \text{ m}$, $x = 0.029 \text{ m}$, $x = 0.0415 \text{ m}$, $x = 0.054 \text{ m}$ and $x = 0.064 \text{ m}$.	72
Figure 68 — Scenarios 3.f to 3.h: Average temperature of the heated plate and maximum temperature of the heated plate	73
Figure 69 — Scenarios 3.f to 3.h: (a) Temperature measured at point T2 (4, -2.5, 0) mm. (b) Temperature measured at point T3 (29, -2.5, 0) mm. (c) Temperature measured at point T4 (54, -2.5, 0) mm.	73
Figure 70 — Scenario 3.h: u_{velocity} modification at the outlet of the test section. $x = 0.054 \text{ m}$, $t = 19.5 \text{ s}$	74
Figure 71 — Scenario 3.h: u_{velocity} modification at the middle plane of the minichannel. Plane XY, $t = 19.5 \text{ s}$	74
Figure 72 — Scenario 3.h: v_{velocity} modification at the middle plane of the minichannel. Vectors coloured by temperature Plane XY, $t = 19.5 \text{ s}$.	75
Figure 73 — Scenario 3.f to 3.h: v_{velocity} and charge density at the middle plane of the minichannel: (a) Scenario 3.f (5 kV). (b) Scenario 3.g (10 kV). (c) Scenario 3.h (15 kV). Plane YZ, $t = 19.5 \text{ s}$	76
Figure 74 — Scenario 3.h: Temperature and charge density at the heated plate. Plane XY, $t = 19.5 \text{ s}$	77
Figure 75 — Ratio of Nu/Nu_0 vs. applied voltage (kV) for scenarios 3.a to 3.h	78
Figure 76 — Ratio of $\Delta P/\Delta P_0$ vs. applied voltage (kV) for scenarios 3.a to 3.h	78
Figure 77 — PEC vs. applied voltage (kV) for scenarios 3.a to 3.h	79

LIST OF TABLES

Table 1 — Typical values of the convective heat transfer [10]	6
Table 2 — Model 2D: geometry and measurement points.....	37
Table 3 — Geometry and measurement planes/points.....	38
Table 4 — Thermophysical properties of oil at 293.15K [23].....	42
Table 5 — Thermophysical properties of copper.....	42
Table 6 — Thermophysical properties of wood	42
Table 7 — Boundary conditions	42
Table 8 — 2D: Simulation scenarios.....	43
Table 9 — 3D: Simulation scenarios.....	43
Table 10 — Scenario 1.a: Dimensionless numbers	51
Table 11 — Mesh study: geometry and measurement planes/points	58
Table 12 — Mesh study: parameters	58
Table 13 — Mesh study: friction factors	59
Table 14 — Scenario 3.a: Dimensionless numbers	63
Table 15 — Scenarios 3.a to 3.c: Temperatures of the heated plate.....	70
Table 16 — Scenario 3.e: Dimensionless numbers	71
Table 17 — Scenarios 3.f to 3.h: Temperatures of the heated plate.....	75

LIST OF ACRONYMS

$\nabla \cdot$	Divergence $\nabla \cdot \mathbf{v} = \frac{\partial v}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial v}{\partial z}$
∇	Gradient $\nabla v = \left(\frac{\partial v}{\partial x}, \frac{\partial v}{\partial y}, \frac{\partial v}{\partial z} \right)$
∇^2	Laplacian $\nabla^2 v = \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}$
A	Heat transfer area (m^2)
c_p	Heat capacity ($J/Kg\ K$)
D_h	Hydraulic diameter (m)
E	Electric field (V/m)
f	Friction factor ($-$)
f_s	Darcy friction factor ($-$)
F_e	Electrical body force (N/m^3)
h	Convection heat transfer coefficient (W/m^2K)
L	Characteristic length (m)
$\dot{m}$	Mass flow rate (Kg/s)
Re	Reynolds number ($-$)
T	Temperature (K)
T_s	Surface temperature (K)
T_f	Fluid temperature (K)
T_w	Surface average temperature (K)
T_b	Bulk temperature (K)
$\dot{Q}$	Heat transfer rate (W)
q''	Heat flux (W/m^2)
ρ	Density (Kg/m^3)
v_s	Characteristic velocity (m/s)
u	Velocity in x-direction (m/s)
v	Velocity in y-direction (m/s)
w	Velocity in z-direction (m/s)
g	Gravity acceleration (m/s^2)
ΔP	Pressure drop (Pa)
α	Thermal diffusivity (m^2/s)

k	Thermal conductivity (W/mK)
K	Electrical conductivity (S/m)
ε	Electrical permittivity (F/m)
η	Performance Evaluation Criterion (PEC) (—)
μ	Dynamic viscosity ($Pa\ s$)
ρ	Density (Kg/m^3)
ρ_e	Space charge density (C/m^3)
z	Charge density (C/kg)
Pr	Prandtl number (—)
Gr	Grashof number (—)
Re	Reynolds number (—)
Ra	Rayleigh number (—)
Po	Poiseuille number (—)
St	Staton number (—)
E	Electric field (V/m)
E_x	Electric field in x-direction (V/m)
E_y	Electric field in y-direction (V/m)
E_z	Electric field in z-direction (V/m)
J	Current density (A/m^2)
ϕ	Electric potential (V)
ϕ_k	User Defined Scalar
S_{ϕ_k}	Source term of a User Defined Scalar
Γ_k	Tensor diffusion coefficient (m^2/s)
CFD	Computational Fluid Dynamics
PCB	Printed Circuit Board
EHD	Electrohydrodynamic
$\dot{W}_e$	Electric power consumption (W)
R	Electrical resistance (Ω)
I	Electric current (A)

1. INTRODUCTION

1.1 Context and motivation

Aviation, automotive or medical and healthcare industry are only some examples of the wide range of electronic components applications in our society. Some devices can produce a relevant amount of heat during their normal operation and must be cooled down to guarantee their working life. In addition, the current trend is to develop smaller and more powerful electronics components, which means an increase in heat generation within the system. This overheating can potentially produce negative consequences and reduce the working life and the reliability of the electronic components.

Consumer devices such as laptops and smartphones tend to suffer overheating due to the reduction of their physical dimensions. The electronics components are placed in small areas that do not contribute to heat extraction [1]. A good practice is to design the electronic components with a large area and thicker physical dimension to increase the heat extraction efficiency [2]. Other problems related to the design are the risks of faulty contact or contact wiring. The possible sparks created could generate fire [3].

Environmental factors or ambient working conditions have also an important role. Hot ambient temperature contributes directly to overheating. In addition, cyclic or big temperature differences can induce different stresses to the material components and produce a certain level of damage. Humidity or different composition of the working atmosphere can erode metal components and contribute to the deterioration of the devices. This material degradation results in overheating.

The overheating of a component affects not only the component, but it can also affect other components of the system and produce important failures. Material degradation, cracks, combustions and even explosions are the consequences of poor heat extraction. So, efficient cooling methods should be developed to maintain a correct working temperature and guarantee electronic devices safety and reliability [4].

The rapid development of electronic devices requires the research of new cooling methods and the enhancement of the heat transfer techniques in order to remove a high heat flux in a limited space and reduce the working temperature of these components. These new methods could be classified into two main groups: active and passive methods, which main difference lies in the application or not of external energy. Electric field, magnetic field or a vibrating wall are used in active methods, while extended or rough surfaces, mini and microchannels or the dispersion of nanoparticles in the fluid are the passive techniques. A great variety of micropumps have been developed to pump fluids through minichannels [23][23].

In conclusion, new advanced cooling techniques are in development to meet the thermal management demands. Spray, cryogenic or microchannels cooling are some examples. An electrohydrodynamic system that pumps dielectric liquid through a microchannel is a new cooling technique studied of this document.

1.2 Objectives

The goal of this Master Thesis is to evaluate the heat transfer enhancement of an electrohydrodynamic system to cooling electronics components, by pumping a dielectric fluid through a microchannel for laminar flow. The simulations are based on the experiment performed by Moghanlou, F. S. et al. [23]. The heat transfer enhancement is studied by the analysis of the effect of different parameters. Consequently, the main steps which will be followed to reach the objectives are:

- To perform a study of current cooling methods of electronic components.
- To conduct a depth study of the mechanism of an electrohydrodynamic system and their theoretical principles.
- To perform an optimal model in 2D and 3D in ANSYS Fluent software for the required analysis and to simulate different scenarios.
- To evaluate different parameters in electronics components cooling efficiency.

1.3 Thesis structure

This thesis is arranged in 7 chapters.

Chapter 2 presents the state of the art. It presents an introduction to the general heat transfer concepts and a short description of different electronics components. After that, it includes a further explanation of the different electronic cooling techniques used currently and an explanation of the use of dielectric fluids for this application. In addition, an overall explanation of the EHD micropumps is presented, which are based on the same theoretical principles that the device studied. It concludes with an explanation of the different impacts that these new cooling techniques involve.

Chapter 3 is focused on the theoretical and physical concepts. Both, hydrodynamic and electrical main concepts are explained to understand the EHD phenomena. The electrohydrodynamics governing equations needed to explain the physics of the device are presented and detailed.

The following chapter 4 presents the software used for the Computational Fluid Dynamics: ANSYS Fluent. It introduces a detailed explanation of the solver and different models used, the relevance of the discretization of the domain and a review of the different boundary conditions and convergence criteria. It is also mentioned the limitation that we have found in order to accomplish the work proposed.

Chapter 5 presents the methodology followed in this thesis. First, a description of the device is done followed by a description of the different analysis parameters studied, in order to analyse the heat transfer enhancement. Two different models in 2D and 3D are performed, and a description of the geometry and the mesh generation is exposed. This chapter furthermore presents all the assumptions done and the parameters selected in the configuration of the simulations setup. Finally, it concludes with a recap of different scenarios to be studied.

Chapter 6 covers the discussion of the results obtained for the 2D and 3D simulations performed. Scenarios are compared and the results are highlighted in order to determine the heat transfer enhancement. Chapter 7 summarises the most relevant findings encountered in the analysis and presents the conclusions.

2. STATE OF THE ART

2.1 General heat transfer concepts

Heat is the energy transferred between substances or systems due to a temperature difference between them. According to the first law of thermodynamics, if two bodies at different temperatures are placed together in absence of work, heat transfer occurs immediately and spontaneously from the hotter one to the colder. Heat has the unit Joule (J) in the International System of Units [5].

There are three modes of heat transfer: conduction, convection and radiation. Only conduction and convection modes are presented because radiation is not considered in the study performed in this thesis.

- Conduction heat transfer: it is an internally way of heat transfer caused by vibrations or rapidly moves of atoms and molecules. It is the most important heat transfer mechanism in solids, especially in metals because free electrons can move around and easily transfer energy from one part of the metal to another [6].

Heat conduction is governed by Fourier's Law. It states that the heat flux transferred is proportional to the magnitude of the temperature gradient with the opposite sign [7].

$$\dot{Q} = -k A \nabla T \quad (1)$$

Where:

- $\dot{Q}$ (W) is the heat transfer rate
- k (W/mK) is the thermal conductivity
- A (m²) is the cross-sectional area
- ∇T (K/m) is the temperature gradient

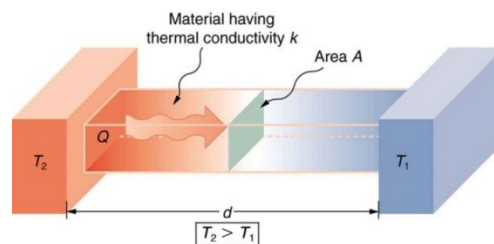


Figure 1 —Conduction heat transfer [26]

Heat conduction transfer rate depends on the properties of the medium. It is relevant to introduce the concept of thermal conductivity. It is the ability of a material to conduct heat. For insulating application, it is recommended to use materials with low thermal conductivities while materials with higher values are used for applications where good heat conduction is needed [8].

- Convection heat transfer: it occurs between a solid surface and a fluid that moves over it. It is usually the main transfer method in liquids and gases. Heat is transferred by the combination of diffusion (conduction) and by bulk fluid motion (advection). Natural or free convection

occurs when the flow is only caused by buoyancy forces. Density variations appear due to the temperature differences within the fluid. If the temperature rises, the density normally decreases and causes the upward movement of the fluid. On the other hand, forced convection occurs when the fluid is pumped or pushed over the surface. Convection with phase change (boiling) can occur too [6].

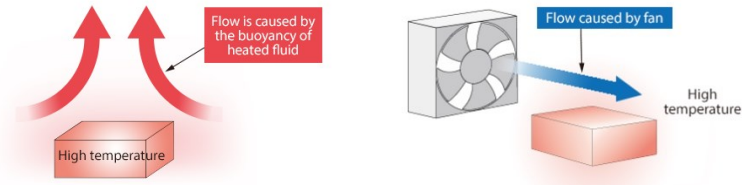


Figure 2 — Natural and forced convection [9]

The heat transfer expression for convection mechanism is:

$$\dot{Q} = hA(T_s - T_f) \quad (2)$$

Where:

- $\dot{Q}$ (W) is the heat transfer rate
- h (W/m²K) is the convection heat transfer coefficient
- A (m²) is the heat transfer area
- T_s (K) is the surface temperature
- T_f (K) is the fluid temperature

The convection heat transfer is affected for the convection mode:

Process		h (W/m ² K)
Free convection	Gases	2 – 20
	Liquids	50 - 100
Forced convection	Gases	25 – 30
	Liquids	100 – 40 000

Table 1 — Typical values of the convective heat transfer [10]

Some dimensionless numbers characterise the heat transfer [12] [13]:

- Reynolds Number: it is the ratio of the inertial forces (fluid and flow properties) and the viscous forces (only fluid properties). It is used to determine the flow regime (laminar or turbulent). The expression is [42]:

$$Re = \frac{\rho v_s L}{\mu} = \frac{v_s L}{\nu} \quad (3)$$

Where:

- Re (—) is the Reynolds number
- ρ (Kg/m³) is the density of the fluid

- v_s (m/s) is the characteristic velocity of the fluid
- L (m) is the characteristic length
- μ (Pa s) is the dynamic viscosity of the fluid
- ν (m² / s) is the kinematic viscosity of the fluid

For internal flows through a pipeline, the term L is expressed as the hydraulic diameter D_h as follow:

$$D_h = \frac{4A}{P} \quad (4)$$

Where:

- D_h (m) is the hydraulic diameter
- A (m²) is the cross-sectional area
- P (m) is the wetted perimeter
- Nusselt Number: it is the ratio of convective to conductive heat transfer. It is the dimensionless parameter that characterizes convective heat transfer.

$$Nu = \frac{hL}{k} \quad (5)$$

Where:

- Nu (—) is the Nusselt number
- h (W/m²K) is the convection heat transfer coefficient
- L (m) is the characteristic length
- k (W/mK) is the thermal conductivity of the fluid
- Prandtl Number: it is the ratio of momentum diffusivity to thermal diffusivity of a fluid. It depends on the fluid properties.

$$Pr = \frac{c_p \mu}{k} \quad (6)$$

Where:

- Pr (—) is the Prandtl Number
- c_p (J/Kg K) is the specific heat
- μ (Pa s) is the dynamic viscosity
- k (W/mK) is the thermal conductivity of the fluid
- Grashof Number: it is the ratio between the buoyancy forces and viscous forces acting on a fluid. It is useful to quantify the opposing forces in convection heat transfer.

$$Gr = \frac{g\beta(T_s - T_f)D_h^3}{\nu^2} \quad (7)$$

Where:

- Gr (—) is the Grashof Number
 - g (m/s^2) is the gravity acceleration
 - β (K^{-1}) is the coefficient of thermal expansion. For gases, it can be calculated following the expression $\beta = 1/T$; and for liquids, it can be calculated if it is known the variation of density with the temperature at constant pressure.
 - T_s (K) is the surface temperature
 - T_f (K) is the bulk temperature
 - D_h (m) is the characteristic length
 - ν (m^2/s) is the kinematic viscosity
- Rayleigh Number: it measures the importance between the effects of the buoyancy forces and the effects of the viscosity forces and thermal conduction.

$$Ra = GrPr \quad (8)$$

Where:

- Ra (—) is the Rayleigh Number
- Gr (—) is the Grashof Number
- Pr (—) is the Prandtl Number

The critical Rayleigh number for the case of infinite parallel plates heated from below is approximately 1700. When the Rayleigh number is below the critical value for a given fluid, the heat transfer mechanism dominant is conduction. When this critical value is exceeded, convection heat transfer is dominant. [44]

2.2 Introduction and description of electronics components: overheating problems

Electronic equipment is widely applied in nearly every aspect of our lives. Under a mechanical point of view, they are extremely reliable as results of no having moving parts. Concerning the thermal environment, they can operate for many years without any problem in case of working at room temperature. Nevertheless, they become potential devices to suffer overheating problems due to the heat that generates the flow of electric current through a resistance. As a result, they normally fail after prolonged use at these temperatures.

First of all, it is important to explain the effect of heat on electronics devices. Electronic devices follow the “twice law”: life increases twice when the temperature reduces 10 °C. Figure 3 shows the relationship between the temperature of electrolytic condenser and life [11].

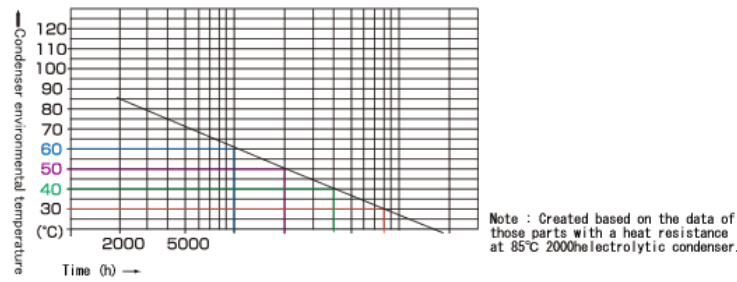


Figure 3 — Relation between ambient temperature of electrolytic condenser and life [11]

With an ambient temperature increase of 10 °C (from 30°C to 40°C), the lifetime of the condenser is reduced in half.

The failure rate of electronic devices due to temperature can be estimated with Arrhenius Law. If the ambient temperature is less than 30°C, the failure rate is lower than unity, but highly increases with the temperature growth. Figure 3 shows the relation between the failure rate of a semiconductor and ambient temperature [11].

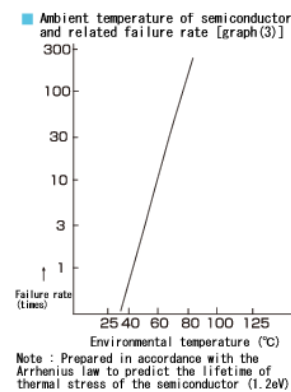


Figure 4 — Relation between ambient temperature of semiconductor and failure rate [11]

Furthermore, it is important to introduce some concepts. The junctions of an electronic component are the circuits through which the electric current flows. These junctions are the potential sites of heat generation and normally their temperature is limited to ensure safe operation [17].

Thermal resistance is a measurement of the temperature difference between two defined surfaces of a material by which the material resists a heat flow. For devices, thermal resistance is the temperature difference between the device and the surrounding ambient when it dissipates a Watt of heat. It is measured in °C/W. Thus, a low value of this parameter is recommended [18].

There are different electronic components:

- Chip + chip carrier: a ceramic, plastic or glass package that protects and contains integrated circuits and provides the connexion to the circuit board. The silicon chip is normally placed in a copper alloy plate at the bottom surface of the carrier. The copper thermal expansion is similar to that of silicon, thus the possible thermal stresses problems of using plastic is avoided. It is important to guarantee a good watertight to avoid problems of moisture.

The chip carrier is the first thermal controller and some thermal considerations are taken in its design. The heat generated is transferred to the package by conduction, convection and radiation. Ceramic, plastic and glass are poor heat conductor and the interior cavity is filled with a poor conductor gas. Consequently, the temperature difference between both parts is large and the junction-to-case resistance vary from 10 °C/W to 100°C/W [17].



Figure 5 — Example of chip carrier [20]

- Printed Circuit Boards (PCB): also called cards, a PCB is a wired plane board used as the mechanical support of different electronic components such as diodes, transistors or resistors. They are made by combining different non-conductive materials (fiberglass, plastic or epoxy) and electrically connect the electronic components using tracks or thin strips of a conducting material such as copper [21]. It is important to pay attention to the thermal design of a PCB, because a simple failure of one of the electronic components could cause the failure of all the electronic system.

Materials should be efficient insulators to prevent electrical breakdown and good heat conductors to dissipate the heat generated. Thermal stresses should be avoided using high strength and good thermal expansion coefficients materials.

Typically, an electronic system is formed with a few PCBs. The power dissipated varies from 5W to 30 W, and the cooling method most commonly used is the direct contact with a fluid, normally air. Only if the electronic system is located in a tight enclosure, cooling is made with a heat exchanger in contact with edge of the PCBs. The device-to-board to edge thermal resistance could reach values from 20 to 60 °C/W, due to the low conductivity of the non-conductive board materials [17].

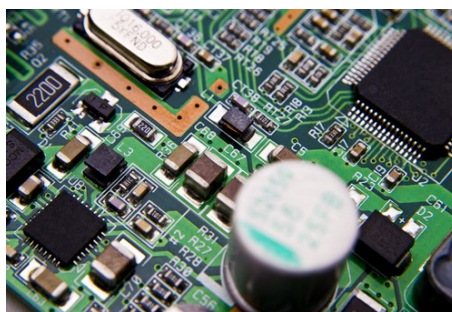


Figure 6 — Common PCB [22]

All the electronic systems need an enclosure to place the circuit boards and the different required connections and peripheral. The enclosure has two different important functions: guarantee the protection of all the components and provide a cooling method. It is designed to easily access for

maintenance or replacement service and includes external indicators, such as lights or a screen to give information and a keypad for user interface.

The materials are usually thin plates of aluminium or steel and have to face some mechanical issues such as vibrations, shocks or moisture problems [17].

The first step to select or design a cooling method is determining the amount of heat to be removed: the cooling load. The heat generated follows the Joule's first law. Joule or Ohmic heating is the physical process by which the pass of electric current through an electrical conductor produces thermal energy [19].

$$\dot{W}_e = \phi I = I^2 R \quad (9)$$

Where:

- $\dot{W}_e$ (W) is the electric power consumption of the electronic device
- R (Ω) is the electrical resistance
- I (A) is the electric current
- ϕ (V) is the electric potential

The first law of thermodynamics states the conservation of energy. In consequence, in the absence of other energy source or interaction, the heat produced by an electronic device in steady operation is equal to its power consumption.

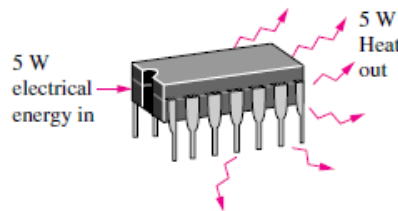


Figure 7 — Cooling load equal to power consumption [17]

Nevertheless, this ideal condition is perturbed by the interaction with different equipment that outputs other forms of energy. The cooling load can be calculated as the power consumption minus all these energy interactions. Another way to perform it is to determine and add up all the individual heat produced by all the components.

Normally, after determining the cooling load of a system, it is usually to add a safety margin to ensure the reliability and the safety of the components. This safety extra cooling load normally raises the cost, the size, the weight and the consumption of the system. So, it is important to well adjust the safety margin to not considerably oversize the drawbacks [17].

The thermal state of an electronic device could be divided into two different operations: transient and steady operation. When a device is turned on, the device components start to absorb the heat generated, so the temperature starts to rise progressively. When the heat generated is equal to the heat removed for the cooling method, the device temperature stabilises at some point and starts the steady operation. For the devices that operate for long periods, the cooling methods are designed for

the steady operation. However, some devices do not reach the steady operation and the cooling method can be designed only for a limited use during the transient state.

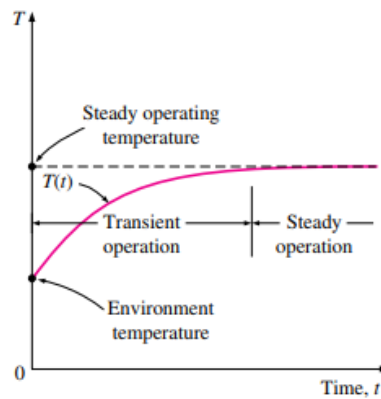


Figure 8 — Temperature change of an electronic component with time [17]

The thermal environment is also important to design the cooling method. The cooling requirements needed for a device placed in a spacecraft differ extremely with those of a common TV. If the ambient conditions are extremes or can deteriorate an electronic device, it is common to use a conditioned fluid as heat sink intermediary between the device and the environment. Water, dielectric fluid and especially air due to its availability, are the most common fluids used.

For example, electronic equipment designed for aircraft applications must meet several requirements. They must be adequately located in odd-shaped and curved spaces and provide efficient paths for the fluid and heat extraction. Normally, forced convection methods are used for such applications, using the ambient air as cooling fluid. But, as the ambient temperature is high, the fluid must be expanded and cooled before entering in the electronic cooling system. However, in vehicles is typically used a liquid cooling method. The liquid passes through the components and after is cooled down in a radiator.

2.3 Current electronic components cooling methods

Overheating is one of the most important issues that electronics industry has to face. Devices trend to be more and more powerful and smaller, so they need innovative thermal management methods in order to improve reliability and system performance.

The thermal management market is valued at USD 8.99 Billion in 2016 and is estimated to reach USD 14.24 Billion by 2022. The compound annual growth rate (CAGR), that describes a constant rate growth over the years of the period considered, is equal to 7.91% during the forecast period (2017 – 2022) [15].

The heat generated varies depending on the electronic device from $5\text{W}/\text{cm}^2$ on a Printed Wiring Board (PWB) to $20\text{ kW}/\text{cm}^2$ for a semiconductor laser [16]. The working temperature must be below the maximum temperature allowed specified by manufacturers.

The main cooling methods existing are:

- Conduction
- Air cooling: natural and forced convection
- Liquid cooling
- Immersion cooling
- Advanced cooling techniques

Manufacturers usually provide the rate of heat dissipation and the maximum allowable temperature for reliable and safe operation. Figure 9 shows the different cooling mechanisms for a certain surface heat flux and temperature difference.

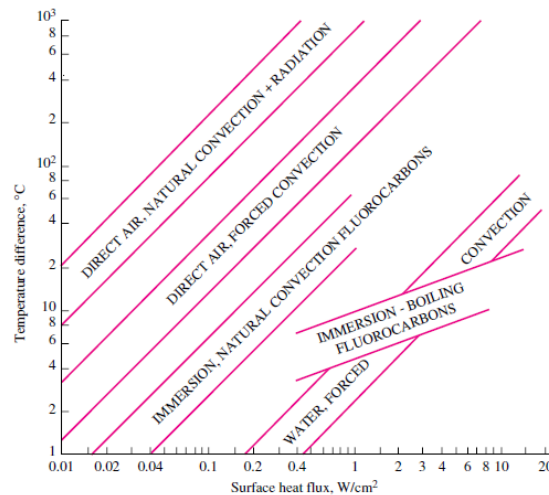


Figure 9 — Heat fluxes that can be removed at specified temperature with certain heat transfer mechanisms [17]

First of all, conduction cooling is based on diffusion heat transfer. It is Important to establish effective heat transfer paths to correctly extract the heat generated and transfer it to the heat sink.

Chip carriers are designed with several leads that remove the heat generated in the chip. The leads are made with a highly conductive material. The heat is transferred to these leads with some bond wires and the heat produced in the lead frame is transferred through the case material [17].

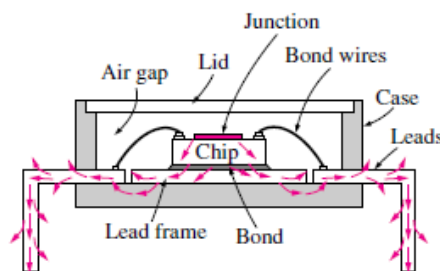


Figure 10 — Heat extraction of a chip carrier [17]

Air cooling is the simplest means of heat removal. It is based on natural or forced convection. For low-power electronic systems, natural convection is widely used. This cooling method is highly reliable because it does not use fans to pump the fluid, so mechanical problems are avoided.

Normally, it is desirable when there are no big obstacles in the fluid path. The heat transfer rate is directly related to the flow rate of the fluid: if the flow rate rises, the heat transfer rate increases.

Electronic components cooled with natural convection need a correct design of the enclosure. They need sufficient vents to ensure the correct fluid flow through the components and guarantee the easy fluid flow exit. They should be as large as possible and the inlet vents must be located at the bottom of the case to allow the upward natural flow to the outflow upper vents. The flow regime normally starts as laminar but it can turn to turbulent if the temperature difference between the air and the heated surface is considerable. However, for temperature differences less than 100 °C and body characteristic length less than 0.5 m, the flow can be considered as laminar. Radiation heat occurs but it is only relevant if the temperatures reached are high [17].

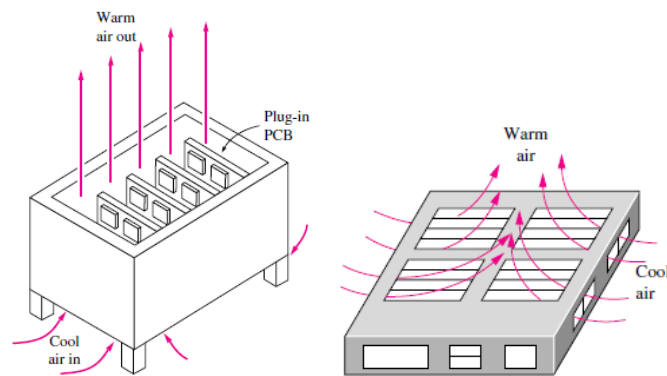


Figure 11 — Natural convection mechanism [17]

When natural convection does not extract sufficient heat, it is common to use a fan to pump the air (forced convection). Following the principle mentioned before (higher fluid flow rate, higher heat transfer), the fan blows the air to the cooling system at a higher velocity and provide a higher air flow rate. The mass air flow rate depends on the environmental condition. For harsh environment with high temperatures, the air flow rate must be sufficient to avoid overheating and higher than the mass flow needed for low temperatures environments. The contribution of radiation heat in this type of cooling method is negligible because all the heat is removed throughout the air injected.

A good practice is to maintain the inlet-outlet temperature difference of 10 °C or K and a maximum exit temperature of 70 °C. These conditions guarantee that the maximum surface temperature of the components does not reach values higher than 100 °C.

The air flow can be internal or external, depending on the body geometry; and laminar or turbulent. Turbulent flows guarantee a better heat transfer coefficient but it needs specific requirements for the fan and the cooling system design. There are different types of fans. For the fan selection, there are two important parameters to take into account: static pressure head and volume flow rate. Axial fans are recommended for low static pressure head and they are cheap and small. For high static pressure head, centrifugal fans are commonly used but they are more complex, bigger and expensive [17].

Liquids are more effective than air. They have much higher thermal conductivity than gases, so higher heat transfer coefficients. Liquid cooling is recommended for applications with high cooling loads that air cooling systems cannot cope with. However, liquid cooling implies some drawbacks such as leakage, corrosion and condensation risks and problems.

There are two different liquid cooling techniques: direct and indirect systems. Liquid removes the heat from the heated surface in direct cooling systems, while there is no "liquid-component" contact in indirect cooling systems. The heat is first transferred to another external medium such as a plate, and the liquid extracts the heat from this plate. Another classification can be done regarding the liquid cycle: if the liquid is recirculated, it is a closed-loop system and if the liquid is discharged after the heat extraction, it is an open-loop system [17].

Typically, electronic equipment is immersed in direct cooling systems and the heat transfer can be natural or forced convection or boiling. The fluids used are the dielectric fluids whose electrical properties are extremely suitable for this application. In section 2.5, a detailed explanation of dielectric liquids is done.

A closed-loop cycle can use water as cooling liquid. Typically, electronic components are mounted on a plate made of very conductive material with some tubes and the fluid passes through the tubes. After the heat extraction, the fluid is recirculated to a heat exchanger to reduce its temperature and returns to the cooling system. An expansion tank absorbs the expansion and contractions of the fluid in order to maintain a correct pressure and volume flow rate of the cooling fluid.

The liquids should meet some special requirements: high thermal conductivity, high specific heat, low viscosity, high dielectric strength and chemical inertness and stability [17].

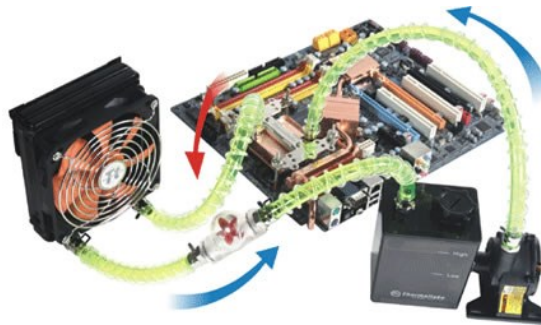


Figure 12 — Indirect liquid cooling system [14]

Boiling guarantee the highest heat transfer coefficients. Immersion cooling consists in submerge high-power electronic component in a dielectric liquid. The temperature reaches very high values so the liquid boils and provide a very high heat transfer rate. Regarding the thermodynamic properties, a liquid boils at the saturation temperature for a given pressure, so the temperature of the bath is constant [17].

2.4 Applications related to Electrohydrodynamics: Meso/micropumps: EHD pumps

The limited space is one of the most difficult challenges to manufacture a reliable and efficient pump. The length scales of the micropumps have the order of 100 nm to 0.1 mm. Micro/meso scale pumps are classified into two different categories: mechanical and non-mechanical (dynamic) pumps, considering if moving parts are used or not [24].

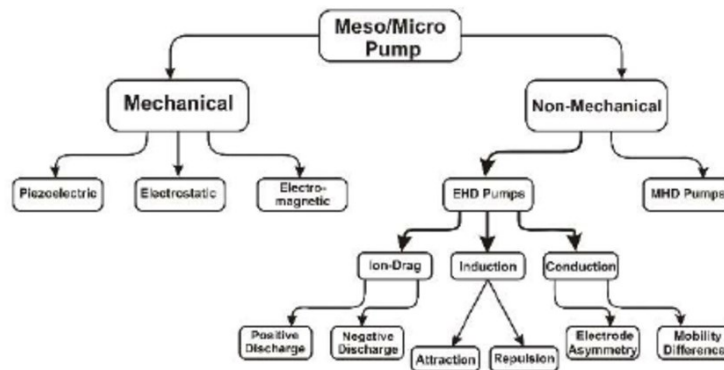


Figure 13 — Classification of meso/micro pumps [27]

2.4.1 Mechanical micropumps

The research in this new type of pumps starts in the early 1980 and it was the base to the development of the microelectromechanical systems (MEMS) around 1990. The main applications of MEMS focus on biomedical and biological fields. They allowed creating new devices to be applied in insulin delivery or injection of glucose and electronic cooling systems, for example in micro integrated circuits, currently still in development [25].

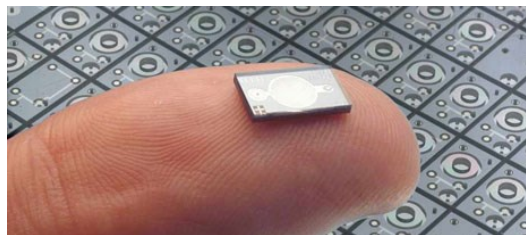


Figure 14 — Example: Debiotech's Insulin Nanopump [57]

2.4.2 Non mechanical or dynamic micropumps

The main principle of dynamic micropumps is adding momentum to the fluid. Generally, these micropumps provide higher flow rate and a much steadier discharge than mechanical pumps nevertheless high-viscosity liquids cannot be pumped. Magnetohydrodynamic (MHD), electroosmotic, bubble type or electrohydrodynamic (EHD) are different types of this technology [25].

The EHD pumps consist in the interaction of electric fields and free charges in a dielectric fluid medium. The fluid is pumped when electric fields push or pull charges through the fluid in a given direction. Compared to other dynamic micropumps, the main advantages of EHD micropumps are the low manufacturing costs and minimal power consumption, the absence of vibration (no moving parts) and very low noise [27]. In addition, the control is very quickly by varying the electric field applied and they are recommended for special environments. They could use different fluids, depends on the application and in single and multiphase flow, with high reliability and high efficiency. Usually, fluid flows will be laminar and with low Reynolds numbers. [24]

2.4.3 EHD micropumps

The EHD micropumps could be classified into three different types: ion-drag, induction and conduction pumps [27].

- Ion-drag pumps: ions are injected from a sharp-edged electrode (emitter) into a dielectric fluid. The charged particles move towards the collector throughout the fluid due to a high voltage electric field applied. This motion causes collisions with neutral molecules of the fluid which exerts a drag force along the channel.

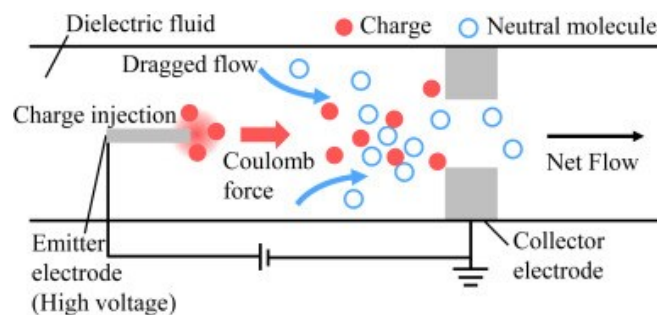


Figure 15 — Injection pump configuration [36]

Ion-drag pump could have two different configurations: positive or negative discharge. It depends on which terminal of the power supply is connected to the high voltage electrode. Positive discharge configuration (emitter connected to the positive terminal of the power supply) produces the positively charge of the nearest molecules of the electrode, being repelled them from the emitter towards the collector. In contrast, the negative terminal of power supply is connected to the high voltage electrode in negative discharge configuration. Negative ions appear in this case, and they are pulled by the collector. In both types, the motion in the fluid is always from the high voltage electrode (emitter) to the ground electrode (collector). [27]

Nevertheless, these pumps often could deteriorate the electrical properties of the working fluid and they could be not easily operated.

- Induction pumps: inductions pumps reduce the deterioration fluid problem of ion-drag [28]. Low conductive fluids contain particles charged positive and negatively in equilibrium that could be disturbed if a gradient in electric conductivity of the fluid is applied. This contributes

to the formation of positive and negative charges depending on the polarity of the applied voltage. The free charges are pushed or pulled to the adjacent electrode by applying electrical waves. This motion creates a drag force and it moves the fluid. [27]

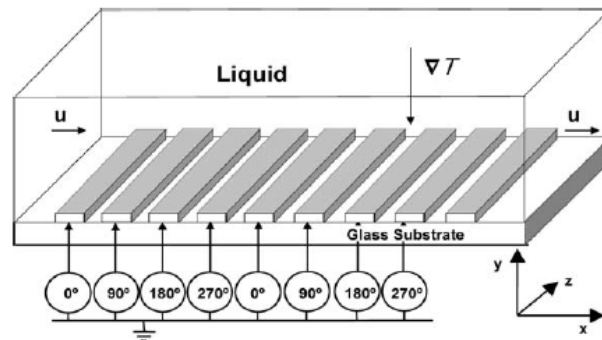


Figure 16 — Travelling wave induction pump configuration [24]

A temperature gradient is used to generate this movement because mainly of two reasons: the dissociation rate of neutral molecules depends on temperature and ionic mobility is inversely proportional to viscosity so liquid conductivity depends on temperature too. The way to induce charges is to apply a travelling wave perpendicular to the conductivity gradient and the sign of this temperature gradient define the direction of the flow. Normally, this temperature gradient is obtained by the Joule heating generated by the electric field but it could be applied an external temperature gradient too [29].

Figure 17 shows the two different induction pump configuration. Figure 17 (a), the flow direction and the travelling wave is the same while in Figure 17 (b) the direction is opposite.

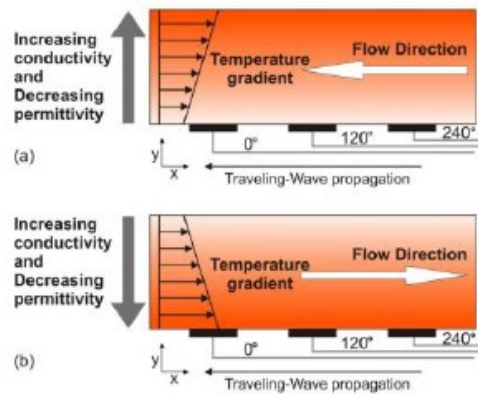


Figure 17 — Induction pump configuration [27]

- **Conduction pumps:** EHD conduction pumps are based on the dissociation and recombination of neutral particles of the electric field [30]. In a neutral medium, the rate of dissociation and recombination is in equilibrium. However, the intense electric field near the electrodes creates a layer in which particles are not in equilibrium: the rate of dissociation is higher than that of recombination. This layer is charged with the opposite sign from that of the adjacent

electrode and it is called heterocharge layer [31]. The thickness of the layer increases with the electric field and the Coulomb force acting in this layer produce the drag force. [27]

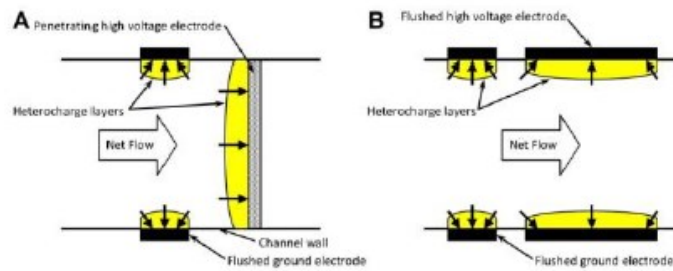


Figure 18 — Heterocharge layers and different types of electrode configuration [27]

Figure 18 shows the heterocharge layers and different types of electrode configuration [27]:

- a) Porous type electrodes
- b) Flushed-type electrodes

2.5 Dielectric fluids for cooling: mineral oils

The use of minerals oils is a promising technique to reduce energy consumption for cooling electronic components. Liquid refrigeration has multiples advantages over traditional air techniques, due to the higher heat capacities of fluids [32]. In addition, the economic savings is not only in terms of energy consumption, but also the design and manufacturing costs of the cooling systems are also reduced. A correct choice of the heat transfer fluid can optimize the cooling procedure and the service life of the electronic components to be cooled. [33] Different solutions and case studies using mineral oil ([34] and [35]) proved the effectiveness and the economic savings of this immersion technique.

The benefits of oil immersion cooling technology versus air could be summarized in the reduction of the common operational issues and the main causes of failures. Typically, air cooling systems have high fluctuations in temperature and relative humidity profile. However, the oil systems operating conditions are smoothed (low temperatures and no sensitivity to humidity or suspended particles), corrosion problems and electrochemical migration are reduced. In addition, moving parts like fans disappear and exposure to electrostatic discharge is avoided [33].

There are five different dielectric fluids used to cool electronic components [48]:

- Mineral oil
- White oil
- Fluorinated oil
- Vegetable oil
- Synthetic isoparaffin

It is important to consider the following key parameters for each type of dielectric fluid [48] [49] [50] [51].

- Heat Transfer Effectiveness: it is one of the most important characteristics of the cooling fluid. Many physical characteristics of the fluid are temperature dependent as the viscosity (dynamic viscosity has the largest effect) or density, so the operational temperature has an important effect on cooling efficiency.
- Electrical characteristics: all these dielectric fluid are good electrical insulators as result of their non-polarity and their high dielectric strength.
- Oxidation stability: this characteristic has a direct impact on the life cycle of oil because hydrocarbons oils are prone to oxidize under heat and oxygen exposition. Sludge accelerates the oxidation process and reduces the performance of the cooling system: it accumulates in the corners and across the boundaries of pipes, reducing the speed of the cooling fluid causing an overheat [37].
- Material compatibility: the dielectric fluid does not influence, interference or modify the physical characteristics of the equipment materials. The compatibility between all circuit board and other materials of the system and the dielectric fluid must be guaranteed. One of the ways that fluids are tested for potential incompatibilities is to determine the fluid's solubility toward rubbers and similar compounds.
- Worker health and safety: safety conditions to the workers and the environment must be guaranteed. Workers exposure can be via inhalation, via ingestion or by skin exposure to dielectric fluids. All these fluids are considered non-toxic and non-hazardous and whether skin contact happens, washing the zone affected with soap and water should be enough. The production of an electric arc as an ignition source is very small due to the low voltages used in electronics systems. The fire point must be as high as possible.
- Biodegradation and environmental fate: the toxicity to marine and soil organism and biodegradation rate are important in the use of these cooling fluids. Biodegradation is the process by which organic substances are broken down into smaller compounds by enzymes produced by living microbial organisms [52]. The speed and the rate of the biodegradability depend on the chemical composition of the fluid, temperature and type of soil or water.
- Cost: it is important to consider spills, waste, leakage and waste to calculate the fluid cost.

In summary, these key parameters allow to choose the best dielectric fluid to use in cooling systems for electronic components.

The mineral oils could be classified into three different groups, depending of its composition [38]:

- Paraffinic oil
- Naphtenic oil
- Aromatic oil

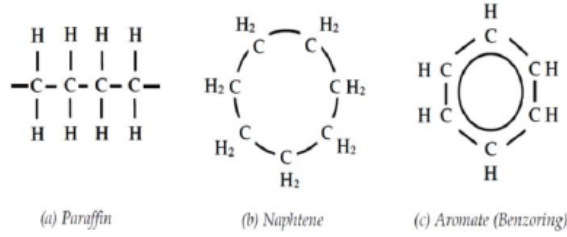


Figure 19 — Hydrocarbons in the mineral oil [37]

Naphtenes do not contain paraffin waxes and are the most widely used in cooling application because they form a smaller quantity of waxes and they have a very good dielectric behaviour [56].

The main properties of minerals oils are low viscosity, good electrical properties and low relative permittivity. It can easily be refined from petroleum. For the coolant function, oils must have a low pour point (measure of oil flow at a relatively low temperature) to ensure the correct flow of the fluid at any temperature. Oil temperature in service must be controlled and must be lower than its flash point (lowest temperature at which it can be flammable) [37].

The correct circulation at higher temperature is guaranteed because the viscosity decreases if the temperature rises. Viscosity has a direct relationship with the heat rate dissipation and follows the equation:

$$\mu = C_1 * \exp\left(\frac{2797.3}{T + 273.15}\right) \quad (10)$$

Where:

- μ (cP) is the dynamic viscosity
- C_1 is a coefficient for scaling
- T ($^{\circ}\text{C}$) is the temperature

Oil has also a safety function actuating as an insulator between different parts at different electrical potential, so it should be not a hazardous material.

2.6 Social impacts

Nowadays, electronic components are present in almost all the common devices and equipment such as TV's, computers or household appliances. The rapid development of electronic equipment forces the society to keep update to the last technologies, discarding a huge quantity of devices. By 2016, the amount of e-waste grew to 44.7 million metric tons, and only in EU, the waste of electrical and electronic equipment (WEEE) is expected to reach 20 tons by 2020. This e-waste, mostly inadequately treated, represents a healthy and environmental hazard [43] [46].

In addition, society trends to be more and more connected. Smart City initiatives need the communication between the citizens, the city and all the elements located within the city. So, intelligent electronic devices (IEDs) will play an important role in our future society. To allow this interaction, it is needed the implementation of thousands of intelligent electronic devices (IEDs), that

must operate without interruption. Thus, it will be critical to ensure the correct working conditions for all this technology and new efficient cooling methods must be developed. To guarantee a safe, healthy and reliable society based on the internet of things (IOT), failure rate of all the components that create this network must be almost totally reduced.

2.7 Economic impacts

The use of mineral oil as a cooling fluid for electronic components is a very attractive technique to be widely applied because offers an opportunity for important energy savings. The low production costs and the wide availability are two factors that justify the choice of mineral oils as cooling fluid. In addition, its good thermal properties place it above the water as cooling fluid [37].

The primary sources of mineral oils are fossil fuels: oil, natural gas and coal. Nowadays, the contribution to the global energy production of fossil fuels is really high, but tends to be reduced in the future (from the actual 85% to 74% in 2040) [53]. Therefore, the availability and the price of mineral oil will should not highly vary next years.

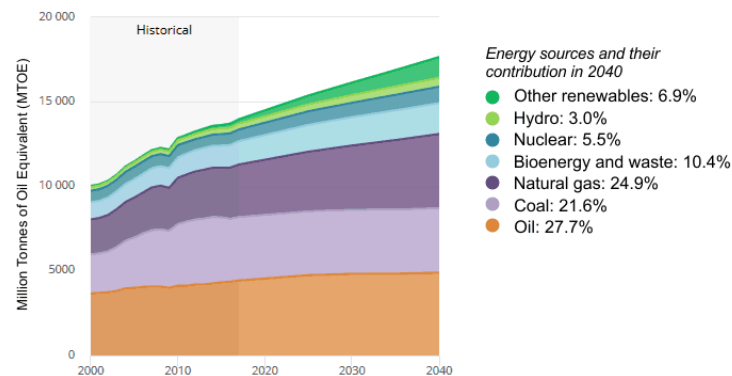


Figure 20 — Energy sources [53]

Normally, the costs of a high-reliability cooling system for a data centre could reach the 50% of the total costs. The investment in the infrastructure needed to build a liquid cooling system, typically an immersion cooling system, is considerably lower compared to air systems due to the reduction of the use of specialized machinery as chillers or temperature and humidity controls [54] [55]. Oil takes away approximately 1200 times more heat by volume when compared to air [37].

At present, about 500 million dollars are invested in cooling infrastructure per data centre. Oil has higher performance and availability and can reduce the cooling systems cost in half because the energy used to cool and circulate the air is avoided. Approximately fan power needs 0.13W of power to move 1W of waste heat into chilled water loop and the technical load to power fan at 100% for fan-powered air is 10.5% more than oil immersion [54] [55].

Consequently, immersion cooling systems with mineral oil are increasingly implemented.

2.8 Technological impacts

An in-depth explanation of negative the effects of overheating in electronics components is explained in chapter 1.1. Typically, manufacturers recommend maintaining the working temperature below a defined value that easily can be overcome if a correct cooling technique is not applied. As it explained before in chapter 2.2, electronic devices follow the “twice law”: life increases twice when the temperature reduces 10°C [11].

One of the most important applications in which it can be needed these new cooling techniques is in data centres. Immersion cooling systems are implemented in different data centres over the world such as the NSA (National Security Agency), which has implemented an immersion cooling system in its Physics Lab. In Barcelona, the PIC (Port d’Informació Científica) infrastructures have four tanks that cool down a scientific computing cluster.

Some positive aspects have been noticed by using these new techniques with mineral oils as cooling methods in data centres. The Power Usage Effectiveness (PUE) is reduced to values of 1.05 in the PIC of Barcelona. PUE is a benchmarking standard to determine how energy efficient data centres are. It is computed as the ratio between the total facility power and the IT equipment power. Values less than 1.2 mean a high efficiency of the system [78].

The mineral oil thermal properties are better than air and water ones. In addition, it avoids dust particles, guarantees proper cleanliness of the system and reduces the noise due to the absence of fans. This and stable temperature in all the equipment helps to reduce the mechanical failures too [79].

Consequently, efficient cooling methods help to increase the durability and the lifetime of electronic components. Thus, the development of efficient cooling methods is needed to increase the lifespan of all the electronic devices and try to reduce the e-waste rate produced every year.

3. GOVERNING EQUATIONS

Electrohydrodynamics is a science that relies on the study of the interaction of fluids with electrical fields. It is an interdisciplinary science between the study of the movement of a fluid and the study on how an electric field could produce this movement [41]. This chapter introduces the governing equations and all the physics principles of EHD.

3.1 Hydrodynamics

First, in hydrodynamic, the Navier-Stokes equations describe the motion of viscous fluids. In this study, it is only considered incompressible flows in which density variations are not linked to the pressure and the mass conservation is a constraint on the velocity field. So, for an incompressible flow the divergence of the flow velocity is zero that implies constant density value [42].

The Boussinesq approximation is a way to solve no isothermal flow, such as natural convection problems, without solving the full compressible formulation of the Navier-Stokes equations. This approximation considers density as constant value unless where it appears in terms multiplied by the gravity acceleration. This approximation is accurate for many kinds of flows, when the temperatures differences and thus, the density differences are smalls, with an easy physical and mathematical formulation [39]. It is used to solve problems where the fluid temperature varies from one place to another, driving a flow of fluid and heat transfer [40].

In this approximation, only the variation of the density when it appears multiplied by the gravity acceleration is considered. The rest of fluid properties are considered as constants. The formulation is as follow [40] [41]:

The continuity equation for conservation of mass is:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (11)$$

Where $\mathbf{u}$ (m / s) is the local velocity of a parcel of fluid.

If density is considered as constant

$$\nabla \cdot \mathbf{u} = 0 \quad (12)$$

The expression of the density is dependent on the temperature:

$$\rho = \rho_0 - \beta \rho_0 \Delta T \quad (13)$$

Where β is the thermal expansion coefficient.

If $\mathbf{F}$ (N) is the gravitational force:

$$\mathbf{F} = \rho \mathbf{g} \quad (14)$$

Then, the conservation of momentum equation:

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} - g \beta \Delta T \quad (15)$$

Where:

- ρ (Kg/m^3) is the density
- p (Pa) is the pressure
- ν (m^2/s) is the kinematic viscosity
- β (K^{-1}) is the thermal expansion coefficient
- First term: temporal variation of velocity
- Second term: convective terms
- Third term: pressure gradient
- Fourth term: viscosity of the fluid
- Fifth term: sum of all the forces involved

The expression for conservation of energy in hydrodynamics can be written as follows [41]:

$$\rho C_p \left[\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right] = \nabla \cdot (\kappa \nabla T) + J \quad (16)$$

Where:

- T (K) is the temperature
- J (J/m^3) is the rate per unit volume of internal heat production
- κ (W/mK) is the thermal conductivity
- C_p ($J/kg K$) is the heat capacity

3.2 Electrostatics

Electrostatics describes the physics of charge in motion and at rest in absence of significant magnetic field [72]. The electrical force will be applied in a certain region of the microfluidic system and it is equivalent to the generator of an electrical circuit. The electrostatic force due to space charge of polarization of dielectric medium is the main reason for heat transfer enhancement in EHD systems [47]. The electric body force has the expression [59]:

$$\mathbf{F}_e = \rho_e \mathbf{E} - \frac{1}{2} E^2 \nabla \epsilon + \nabla \left(\frac{1}{2} E^2 \rho \left(\frac{\partial \epsilon}{\partial \rho} \right)_T \right) \quad (17)$$

Where:

- First term: electrophoretic or Coulomb force
- Second term: dielectrophoretic or dielectric force
- Third term: electrostriction force

The first term is the most relevant in the case of EHD micropumps. It is the force per unit volume on a medium containing free electric charge, responsible for the movement of the fluid. The second term is the force due to the force exerted on a non-homogeneous dielectric liquid by an electric field.

Normally, it is weaker than Coulomb force and it is also relevant when a temperature gradient appears or when an AC-EHD single phase system is considered. Finally, the third term shows the permittivity changes to density to the applied electric field [23]

The electric force produces a secondary flow that pushes the neutral molecules of the dielectric fluid towards the collector, and modifies the velocity profile.

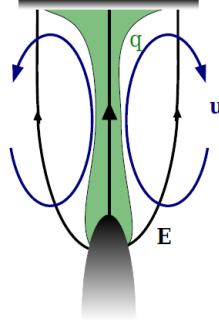


Figure 21 — Secondary flow velocity profile [71]

To obtain the electric force, Maxwell's equations should be considered. In electrohydrodynamic flows, the magnetic effect could be ignored because the characteristic time for the magnetic phenomena ($t_m \sim \mu_M K l^2$) is several orders of magnitude smaller than the characteristic time for electric phenomena ($t_e \sim \varepsilon/K$) [59] [60].

$$\nabla \cdot (\varepsilon \mathbf{E}) = \rho_e \quad (18)$$

$$\nabla \times \mathbf{E} = 0 \quad (19)$$

Where:

- ε is the electric permittivity
- $\mathbf{E}$ (V/m) is the electric field
- ρ_e (C/m³) is the volumetric charge density

In terms of the electrical potential, ϕ , the electrostatic limit follows the Poisson equation:

$$\mathbf{E} = -\nabla \phi \quad (20)$$

$$\nabla \cdot (\varepsilon \nabla \phi) = -\rho_e \quad (21)$$

The volumetric charge density could be expressed as:

$$\rho_e = \rho z \quad (22)$$

Where:

- z (C/Kg) is the charge per unit mass

The charge conservation equation is:

$$\frac{\partial \rho_e}{\partial t} + \nabla \cdot \mathbf{J} = 0 \quad (23)$$

Where:

- $\mathbf{J}$ (A/m^2) is the current density

$$\mathbf{J} = K\mathbf{E} + \rho_e\mathbf{u} \quad (24)$$

Where:

- K (S/m) is the conductivity
- First term: ohmic charge conduction
- Second term: convection of charges

Considering the electrostatic relationship (18) and (19), the conservation equation of the charge (23) can be written:

$$\frac{\partial \rho_z}{\partial t} + \nabla \cdot (\rho_z \mathbf{u}) = -\frac{K}{\varepsilon} \rho_z + \mathbf{E} \cdot \left(\frac{K}{\varepsilon} \nabla \varepsilon - \nabla K \right) \quad (25)$$

If the electrical properties of the fluid K and ε are constant, eq. (25) reduces to:

$$\frac{\partial \rho_z}{\partial t} + \nabla \cdot (\rho_z \mathbf{u}) = -\frac{K}{\varepsilon} \rho_z \quad (26)$$

3.3 Electrohydrodynamics

Electrohydrodynamics describes the effects of electrostatics in liquid media. The Coulomb force, $\mathbf{F}_e = \rho_e \mathbf{E}$, is considered and added to the conservation of momentum equation (15). Consequently, considering constant value for K and the Joule Heating for the energy equation, the three governing equations of our EHD system are:

$$\nabla \cdot \mathbf{u} = 0 \quad (12)$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} - \mathbf{g} \beta \Delta T + \rho_e \mathbf{E} \quad (27)$$

$$\rho C_p \left[\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right] = \kappa \nabla^2 T + \varepsilon |\nabla \phi|^2 \quad (28)$$

4. SIMULATION SOFTWARE: ANSYS FLUENT

4.1 ANSYS Fluent

Create a computer multiphysics model represents a truly complex and laborious task, consequently there are some different computers tools in the market to perform such simulations. ANSYS is a company focused on the development of engineering simulation for more than 45 years [61]. ANSYS Fluent, one of its products, is a CFD software with the physical modelling capabilities needed to accomplish the simulations defined in chapter 5.

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that provides a qualitative and quantitative prediction of fluids flows thanks to mathematical modelling and numerical methods [62]. Solve these problems analytically are in many cases extremely difficult due to the non-linear inertial terms. By discretizing the domain in small grids (mesh), it is possible to obtain an accurate solution.

ANSYS Fluent solves the governing equations and generates the flow field data at each mesh elements (nodes, faces, cells). The data generated are exported to a data processor called CFD-Post, provided by ANSYS, to easily analyse the results. In addition, the data can be exported to Excel as a .csv file, to allow the comparisons between simulations.

4.2 Models

ANSYS Fluent provides a wide list of models for different steady-state or transient, incompressible or compressible, laminar or turbulent fluid flow problems.

For the case study of this document, viscous, energy and potential models are used. Viscous model allows setting and defining the characteristics of the fluid flow. Energy model solves equation (28) and potential model solves equation (18) and adds joule heating to the energy equation.

ANSYS Fluent Potential model solves the equation [83]:

$$\nabla \cdot (\varepsilon \nabla \phi) + S = 0 \quad (29)$$

Where:

- ϕ (V) is the electric potential
- ε (F/m) is the electric permittivity
- S is the source term

When solving this equation, ANSYS Fluent adds the Joule heating (W/m^3) generated to the energy equation.

$$S_{h1} = \varepsilon |\nabla \phi|^2 \quad (30)$$

Thus, the Joule heating is added for solid and fluid regions in which a flow of current appears [83] [84].

4.3 User-Defined Scalar transport equations and User-Defined Functions

To simulate the electrohydrodynamic pump, it is necessary to add the contribution of the Coulomb force to the momentum equation. Fluent allows implementing extra scalar transport equations called User-Defined Scalar (UDS).

For an arbitrary scalar ϕ_k , ANSYS Fluent solves the equation [63]:

$$\frac{\partial \rho \phi_k}{\partial t} + \nabla \cdot (\rho \mathbf{u} \phi_k - \Gamma_k \nabla \phi_k) = S_{\phi_k} \quad (31)$$

Where:

- $(\partial \rho \phi_k) / \partial t$ is the unsteady term
- $\nabla \cdot (\rho \mathbf{u} \phi_k)$ is the convection term
- Γ_k is the tensor diffusion coefficient
- $\nabla \cdot (\Gamma_k \nabla \phi_k)$ is the diffusion term
- S_{ϕ_k} source term

Charge density equation (26) is solved with an UDS equation.

To introduce the Coulomb force into the momentum equation, it is needed a User-Defined Function (UDF). A UDF is a C function that can be loaded with the ANSYS Fluent solver and allow to, for example, customize boundary conditions and material properties, add source terms in ANSYS Fluent transport equations or in user-defined scalar transport equations and enhance ANSYS Fluent models. It allows customizing the simulator to cover all the particular and specific requirements of different simulations. UDF codes use special macros provided by ANSYS Fluent to access to solver data and domain variables [73].

4.4 Discretization of the domain: mesh generation

The aim of discretization the entire domain into small high quality cells is to obtain the domain geometry and make sequent calculation to obtain an accurate solution. ANSYS Fluent allows generating tetrahedral, hex-core or hybrid volume mesh from an existing boundary mesh or a CAD file.

While a good and fine mesh helps the CFD solver to converge to an accurate solution minimizing the resources employed, a coarse mesh can be an important source of errors in a simulation. Consequently, it is important to find a correct balance between the fineness of the mesh and the computational cost needed to solve the equations.

ANSYS Fluent provides an indicator to check the quality of the mesh called orthogonal quality. For each face, two different quantities are calculated:

- The normalized dot product of the area vector of a face and a vector from the centroid of the cell to the centroid of that face.
- The normalized dot product of the area vector of a face and a vector from the centroid of the cell to the centroid of the adjacent cell that shares that face.

The minimum value that results from calculating these two parameters is defined as the orthogonal quality. The values go from 0, that indicates bad quality, to 1, excellent quality. The minimum orthogonal quality should be more than 0.01, with an average value significantly higher.

Another important indicator is the aspect ratio that measures the stretching of a cell. It is computed as the ratio of the maximum value to the minimum value of any of the following distances:

- Normal distances between the cell centroid and face centroids.
- Distances between the cell centroid and nodes.

For an unit cube, the aspect ratio is 1.732. It is recommended avoiding sudden and large changes in cell aspect ratios in areas where the flows suffer large changes or strong gradients [64].

4.5 Pressure-Based Solver. Coupled Algorithm

ANSYS Fluent offers two different numerical methods: pressure-based and density-based solver. Normally, pressure-based solver is recommended for low-speed incompressible flows and density-based solver is more suitable for high-speed compressible flows problems. For the simulation required for the case study of this project, pressure based solver is selected.

The governing integral equations are solved following a control-volume-based technique. First, the domain is divided into discrete control volumes (mesh). The governing equations are integrated on these individual control volumes and the discretized equations are linearized. Finally, the resultant linear equation system is solved [65].

Pressure-based solver can work with two different algorithms: segregated or coupled. The Segregated Algorithm solves the governing equations sequentially; each equation is decoupled from the other equations. The memory requirements needed are low because the discretized equations need only be stored one at time. On the other hand, Coupled Algorithm solves simultaneously the system of momentum and pressure-based continuity equations, increasing the rate of solution convergence. Nevertheless, the memory cost increases by 1.5-2 times [66].

The following graph shows the different steps for the calculations:

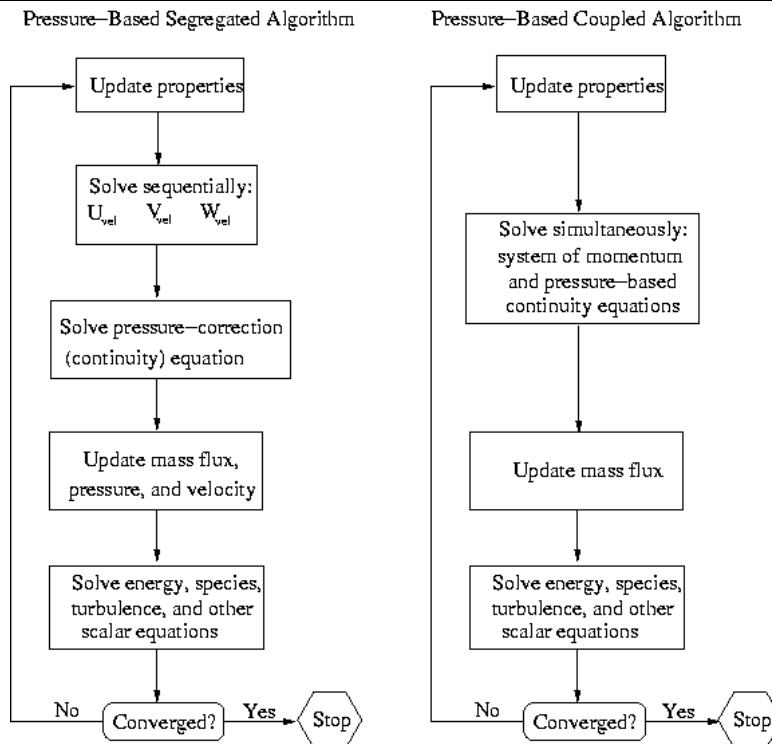


Figure 22 — Overview of Pressure-Based Solution Methods [66]

4.6 Convergence criteria

Typically, the fluid flow problems are not linear and must be solved by an iteratively calculation with CFD solutions. Residuals measure as the local imbalances of a conserved variable in each control volume, and ANSYS Fluent use them as convergence criteria. To obtain a numerically accurate solution, they must be as lower as possible. The default convergence criterion of ANSYS Fluent requires that the residuals drop 3 orders of magnitude for continuity and momentum equation and 6 orders for energy equation [67].

Nevertheless, for complicated problems, it is not always a target reachable. Monitoring some variables like force, drag or average temperature can help the analysis to determine when a simulation is converged. If these variables do not change with more iterations, the simulations can be considered as converged. The final solution must guarantee the mass, momentum and energy conservation [68].

4.7 Boundary conditions

In a CDF analysis, it is relevant to define how the system operates. Boundary conditions are the set of constraints to boundary value required to solve the mathematical model. A wide list of different boundary conditions is available in ANSYS Fluent, and allows defining and setting the boundary values and flowing behaviour.

In the simulations performed, “velocity inlet”, “pressure outlet” and “walls” are the boundary conditions used [69].

- “Velocity inlet” boundary condition defines the flow velocity and all scalar properties of the flow at the domain inlet. The total pressure is not fixed but will rise to the appropriate value to provide the velocity defined.
- When “velocity inlet” is used, ANSYS Fluent recommends using “pressure outlet” as boundary condition outlet. For subsonic flows, it requires the static pressure at the domain outlet. If backflow problems occur, it is possible to define “backflow conditions” to avoid convergence issues.
- “Wall” boundary conditions are used to define solid zones and confine the fluid. For viscous flows, the no-slip boundary condition is applied: tangential fluid velocity equal to wall velocity, and null normal velocity. To define an adiabatic wall, null heat flux must be set.

4.8 Computational resources

ANSYS Fluent v18.1 academic version is used for the 3D simulations of this document. The 2D simulations are done in ANSYS Fluent v19.2. The computers used to perform the simulations are two Intel® Core™ i5-6500, 3.20 GHz, 8Gb RAM for the 3D simulations and an Intel® Core™ i7-4500U, 2.4GHz, 4 Gb RAM for the 2D simulations.

5. METHODOLOGY

This chapter introduces the definition of the model to be simulated. It is based on the experiment performed by Moghanlou, F. S. et al in 2014 [23]. The different considerations and assumptions taken into account to accomplish with the required analysis are also presented.

5.1 General overview

The case study of this document consists in to simulate an electrohydrodynamic system for laminar flow through a square minichannel. The aim is to investigate how the electric field contributes to the cooling process and pressure drop for different Reynolds numbers. The coupled of electrostatics and hydrodynamics produces a modification in the momentum equations, which implies the modification of the velocity profile. The heat transfer enhancement is measured by the analysis of different parameters.

The combination of two different methods of heat transfer enhancement is studied in this document. An active method based on the appliance of electric field to a dielectric liquid flow, and a minichannel as a passive method. The contact between the surface to be cooled and the fluid will be increased due to the secondary flow produced by the electric field. Regarding the passive method, the low hydraulic diameter of minichannels enhances the heat transfer coefficient and increases the pressure drop.

A copper wire is the high voltage electrode placed at the top of the minichannel that injects electrical charge through the liquid (mineral oil), producing an electrical force added to the momentum equation. This induced secondary flow affects the primary flow: the velocity profile is modified near the electrode. The electronic component to be cooled is a copper plate located at the bottom of the minichannel and the dielectric liquid pumped is mineral oil. This heated plate is also considered as ground [23].

The mechanism of the secondary flow is as follows: electrostatics and hydrodynamics are coupled in the momentum equation, modifying the velocity profile. The charge injected from the high electrode push the fluid neutral molecules towards the heated plate that is grounded. This vertical movement of the fluid contributes to increasing the contact between the fluid and the plate to be cooled down.

In order to study the behaviour of the device, 2D and 3D simulations are performed.

5.2 Analysis parameters

This subsection will present the parameters to analyse the heat transfer enhancement and pressure drop.

$$\text{Convection Heat Transfer} \quad h = \frac{\dot{Q}}{A (T_w - T_b)} \quad (32)$$

Where:

- $\dot{Q}/A = q''$ (W/m^2) is the heat flux of the heated plate

- $A \text{ (m}^2\text{)}$ is the area of cross-section of the channel
- $T_w \text{ (K)}$ is the surface average temperature
- $T_b \text{ (K)}$ is the bulk temperature

$$\text{Nusselt number} \quad Nu = \frac{hD_h}{k} \quad (5)$$

Where:

- $Nu \text{ (—)}$ is the Nusselt number
- $h \text{ (W/m}^2\text{K)}$ is the convection heat transfer coefficient
- $L \text{ (m)}$ is the representative dimension
- $k \text{ (W/mK)}$ is the thermal conductivity of the fluid

$$\frac{\text{Performance}}{\text{Criterion (PEC)}} \quad \eta = \frac{j/j_s}{(f/f_s)^{1/3}} \quad (33)$$

Where:

- $j = St * Pr^{2/3} \quad (34)$
- $f = \frac{\Delta P}{(L/D)*((\rho u^2)/2)}$ is the friction factor. (35)
- $f_s = 56.8/Re$ is the friction factor for a square ducts. $f_s * Re = Po$ where Po is the Poiseuille number. [45] (36)
- $St = Nu/(Re * Pr)$ is the Staton number. (37)
- Pr is the Prandtl number (6)
- The suffix “s” refers to smooth surface or the condition without enhancement

5.3 Geometry

The geometry of the simulation is done in SpaceClaim. This chapter presents the geometrical configuration for the 2D and 3D simulations.

5.3.1 2D simulations

The geometry for the 2D simulations (plane XY) is:

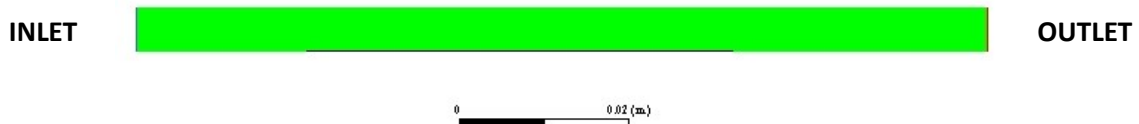


Figure 23 — Model 2D: general view: Plane XY

1. Inlet: flow inlet.
2. Outlet: flow outlet.
3. Walls: The walls confine the fluid. They are thermally and electrically insulated.
4. Heated plate: the heated plate to be cooled. The heat flux considered is 10000 (W/m^2)

5. Wire: high voltage electrode that produces the injection of the electric charge in the oil.

The origin of the coordinates is located at the centre of the inlet boundary.

The wire is located at the top of the channel and the heated plate at the bottom from $x = 0.020$ m to $x = 0.070$ m. Wire and heated plate are aligned and have the same length.

The square duct is 5 mm height and 100 mm length. The dimensions of the different parts and some points/planes located along the x-axis to measure the temperature and the pressure are presented in the next table.

Zone	Length (mm)	Measurement point	Position of measurement plane/point (mm)
Inlet	20	P1	$x = 18$
Test	50	T2	(22, -2.5)
		T3	(55, -2.5)
		T4	(68, -2.5)
Outlet	30	P2	$x = 72$

Table 2 — Model 2D: geometry and measurement points

5.3.2 3D simulations

The geometrical configuration of the simulation is:

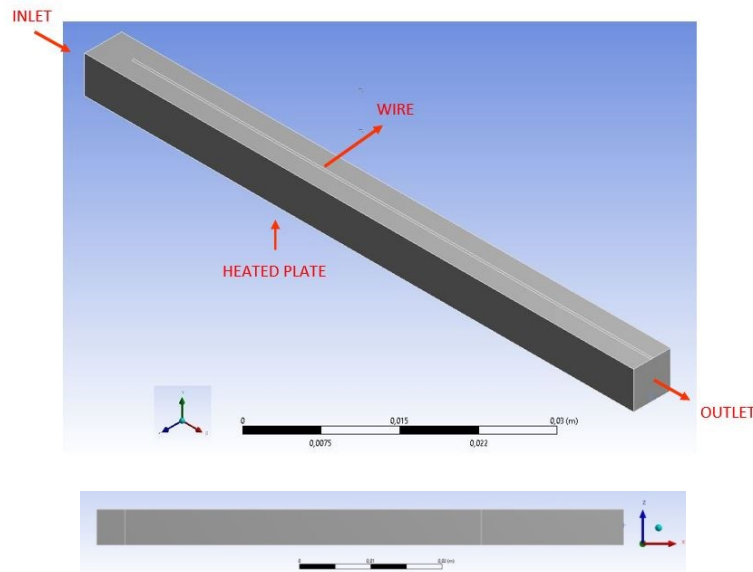


Figure 24 — Model 3D: general view and bottom view

Where:

1. Inlet: flow inlet.
2. Outlet: flow outlet.
3. Walls: The walls confine the fluid. They are thermally and electrically insulated.

4. Heated plate: the heated plate to be cooled. The heat flux considered is 10000 (W/m^2)
5. Wire: high voltage electrode that produces the injection of the electric charge in the oil. It is located on top of the test section and has the same length (50 mm). The diameter of the wire is 0.3 mm. In the model, the wire is simulated as a plate of 50 mm long and 0.3 mm wide.

The origin of the coordinates is located at the centre of the cross-sectional area of the inlet.

The model is divided into three different parts: inlet, test and outlet section. The inlet section adapts and hydrodynamically develops the fluid. The wire injects the charge through the fluid in the test section, where the changes in the velocity profile are produced. In this part of the channel, the charge injected and the electric field contributes to the momentum equation and modifies the velocity profile. Finally, the fluid leaves the microchannel throughout the outlet section.

The square duct is 5x5 mm size and 74 mm length. Along the x-axis of the geometry, some planes/points are located to measure different variables of the fluid.

Zone	Length (mm)	Measurement point	Position of measurement plane/point (mm)
Inlet	4	P1	$x = 2$
Test	50	T2	(4, -2.5, 0)
		T3	(29, -2.5, 0)
		T4	(52, -2.5, 0)
Outlet	20	P2	$x = 56$

Table 3 — Geometry and measurement planes/points

5.4 Meshing

First, in order to determine and see the impact of the mesh in the final results, a mesh study is done for three different cases. As it is explained before in chapter 4.4, the mesh plays a meaningful role in a CFD simulation. Before defining the final mesh of our case study, an in depth analysis is performed to see the impact of the discretization of the domain in the final results. The injection of charge is not considered for this study. The results of this analysis are presented in chapter 6.3.2.

The geometry is modelled in SpaceClaim and meshed with Mesh Fluent. The academic version has a total limit of cells of 512k for the mesh generation.

5.4.1 Mesh parameters for 2D simulations

Some considerations for the different parts of the domain are taken for the mesh generation:

- Mesh type: hexahedral
- Max element size = 0.1 mm in order to have 50 cells along the y-axis.

The statistics of the mesh are:

- Mesh total elements: 50000

- Mesh total nodes: 51051
- Orthogonal quality: 0.999237
- Maximum aspect ratio: 1.45265

5.4.2 *Mesh parameters for 3D simulations*

Some considerations for the different parts of the domain are taken for the mesh generation:

- Mesh type: hexahedral
- Max element size = 0.19 mm
- Wire: maximum element size = 0.1 mm in order to have 3 elements along the z-axis in the wire.

The statistics of the mesh are:

- Mesh total elements: 508 680
- Mesh total nodes: 545 972
- Orthogonal quality: 0.99999
- Maximum aspect ratio: 2.95733

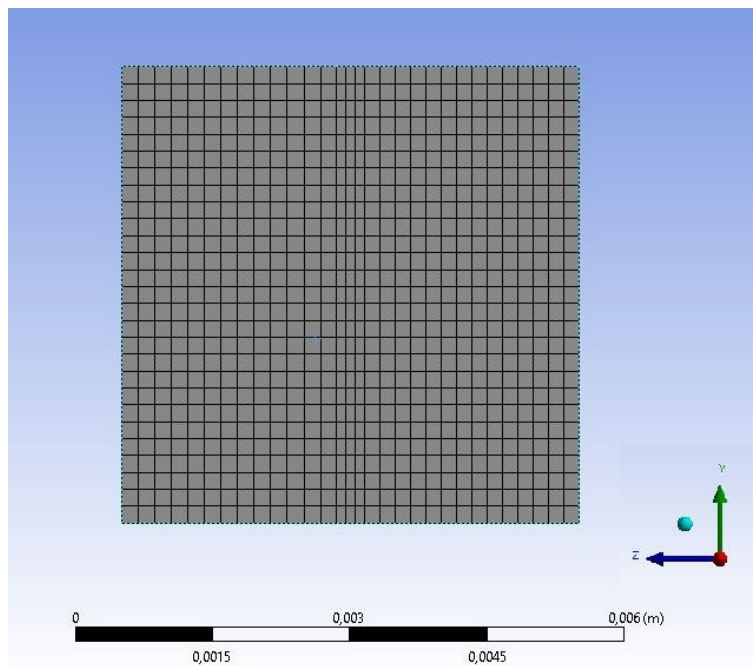


Figure 25 — Model 3D: mesh-cross sectional view

Figure 26 shows the mesh transition between the test and the outlet sections. A refinement of the mesh is needed in the test section to ensure the correct solution of the equations because the injection of the charge will happen in this fluid part. In addition, a refinement at the wire is also implemented.

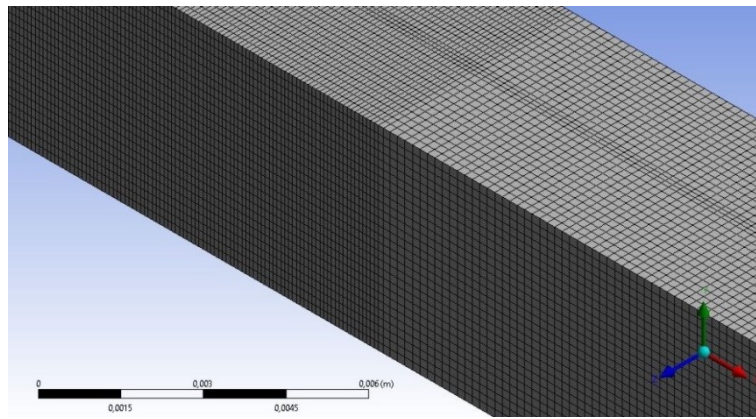


Figure 26 — Mesh: test zone

5.5 Solver Settings

The choice of solver settings is based on the previous training material done before the preparation of these simulations and in the recommendations of the ANSYS User's Guide.

The following settings were used for all the simulation:

- Double Precision
- Pressure-Based Solver
- Solutions Methods:
 - Pressure-velocity coupling: COUPLED
- Spatial Discretization
 - Pressure: Least Squares Cell Based
 - Pressure: Body Force Weighted
 - Momentum: Second Order Upwind
 - Energy: Second Order Upwind
 - Potential: Second Order Upwind
 - UDS-1 (charge density): First Order Upwind
- Convergence criteria:
 - Continuity: 10^{-3}
 - Momentum: 10^{-3}
 - Energy: 10^{-9}
 - UDS-1 (charge density): 10^{-3}
 - Potential: 10^{-9}
- Under-Relaxation Factors:
 - Flow Courant number: 5
 - Momentum: 0.75
 - Pressure: 0.75
 - Density: 0.8
 - Body forces: 1
 - Energy: 0.9

- Potential: 1
- UDS-1 (charge density): 0.7

The Flow Courant Number (CFL) is a mathematical convergence condition for the stability when solving convection or wave phenomena problems. It is used for coupled pressure-velocity scheme and relates the velocity with the time step and the length of the elements of the mesh ($CFL = u\Delta t/\Delta x$). In our case, to improve the stability of the solution, the CFL number is set to 5 [81].

5.6 Assumptions

The following points are considered for all the simulations:

- Transient simulations.
- Laminar fully developed flow.
- Boussinesq approximation for density. The density varies only with temperature in the buoyancy term in the y-momentum equation, as it explained in chapter 3.
- Gravitational acceleration acting in negative y-direction at a rate of 9.81 m/s^2 .
- The inlet temperature equal to 293.15 K.
- Pressure gauge outlet equal to 0 Pa.
- Heat flux of the heated plate equal to $10\,000 \text{ W/m}^2$. This heat flux corresponds to 2.5 W.
- Adiabatic walls. No slip boundary condition: tangential fluid velocity equal to wall velocity, and null normal velocity.
- The temperature differences expected between inlet and outlet are lower, so in case of having backflow at the outlet of the minichannel, the “temperature backflow” is set equal to 296 K.
- Three different materials are used: mineral oil as fluid, copper for the wire and the heated plate and an insulator for the walls of the pipe.

An overall description of the properties of mineral oils is presented in chapter 2.5. Summarizing, mineral oil is adequate as refrigerant due to its low viscosity, good electrical properties and low electrical permittivity. It guarantees a low operation temperature, avoids the problems of oxidation and corrosion and reduces environmental contamination like dust.

Conductor material is used for the wire and for the heated plate. Copper is a conductor material with a high conductivity and low resistivity value. It allows the flow of charge freely on its surface. An in depth explanation of electrical conductor is made in chapter 6.1, in order to explain the phenomena observed in the simulations.

Finally, an insulator material is used for the walls of the pipe. Wood is selected due to its good insulator properties. An insulator does not allow the free flow of charge and does not

permit the electrical conduction. They are characterised by a high value of electrical resistivity and a very low value of electrical conductivity [74][75].

- The properties of the materials are:

OIL	Property	Nomenclature	Units	Value
	Density	ρ	Kg/m ³	856
	Viscosity	μ	Pa s	0.03
	Heat capacity	c_p	J/kg K	1850
	Thermal conductivity	k	W/m K	0.14
	Electric permittivity	ϵ	F/m	1.95×10^{-11}
	Electrical conductivity	K	S/m	3.3×10^{-12}

Table 4 — Thermophysical properties of oil at 293.15K [23]

As the fluid temperature differences are low, the thermos-physical properties of mineral oil are considered as constant. The data of copper and wood are obtained from the Fluent database.

COPPER	Property	Nomenclature	Units	Value
	Density	ρ	Kg/m ³	8978
	Heat capacity	c_p	J/kg K	381
	Thermal conductivity	k	W/m K	387.6
	Electrical conductivity	K	S/m	5.8×10^7
	Electrical resistivity	ρ	$\Omega \cdot m$	1.7×10^{-8}

Table 5 — Thermophysical properties of copper

WOOD	Property	Nomenclature	Units	Value
	Density	ρ	Kg/m ³	700
	Heat capacity	c_p	J/kg K	2300
	Thermal conductivity	k	W/m K	0.173
	Electrical conductivity	K	S/m	1×10^{-30}
	Electrical resistivity	ρ	$\Omega \cdot m$	1×10^{30}

Table 6 — Thermophysical properties of wood

- The boundary conditions are:

Boundary location	Boundary condition	Equations				
Inlet	Velocity inlet	$u = u_{in}$	$v = 0$	$w = 0$	$\frac{\partial z}{\partial x} = 0$	$\frac{\partial \phi}{\partial x} = 0$
Outlet	Pressure outlet	$\frac{\partial u}{\partial x} = 0$	$\frac{\partial v}{\partial x} = 0$	$\frac{\partial w}{\partial x} = 0$	$\frac{\partial z}{\partial x} = 0$	$\frac{\partial \phi}{\partial x} = 0$
Heated plate	Wall	$u = 0$	$v = 0$	$w = 0$	$\frac{\partial z}{\partial n} = 0$	$\phi = 0$
Wire	Wall	$u = 0$	$v = 0$	$w = 0$	$z = z_0$	$\phi = V_0$
Walls	Wall	$u = 0$	$v = 0$	$w = 0$	$\frac{\partial z}{\partial n} = 0$	$\frac{\partial \phi}{\partial n} = 0$

Table 7 — Boundary conditions

- The selection of the time step can be a tricky process. It is necessary to find a balance between the duration of the simulation and meet all the convergences criteria in each time step. For the 2D simulations, the time step selected is 0.001 s and for the 3D simulations, the time step selected is 0.001 s for the firsts 0.1 s of simulation to ensure a correct initialization of the flow and after a time step equal to 0.005 s are used. In some moments of the 2D simulations, the time step should be reduced to 5×10^{-4} s and 2×10^{-4} s to reach the convergences criteria (cases 1.c and 1.d). In conclusion, all the convergences criteria are met for each time step so the validity and the accuracy of the results are guaranteed.

5.7 Input data

5.7.1 2D simulations: scenarios

The next 4 different scenarios are simulated in order to carry out a parametric analysis:

Case	u_{in} (m/s) initial	Re	z_0 (C/kg)	ϕ_0 (kV)
1.a	0.01	1.43	6×10^{-3}	0
1.b				5
1.c				10
1.d				15

Table 8 — 2D: Simulation scenarios

5.7.2 3D simulations: scenarios

The following input data are considered for the simulations:

Case	u_{in} (m/s) initial	Re	z_0 (C/kg)	ϕ_0 (kV)
3.a	0.01	1.43	6×10^{-3}	0
3.b				5
3.c				10
3.d				15
3.e	0.05	7.13	6×10^{-3}	0
3.f				5
3.g				10
3.h				15

Table 9 — 3D: Simulation scenarios

5.8 UDF and UDS implementation

This chapter explains the steps necessities to calculate the electrical equations and how they are coupled with hydrodynamics equations.

Regarding the potential model presented in chapter 4.2, ANSYS Fluent solves the electric potential equation [83]:

$$\nabla \cdot (\epsilon \nabla \phi) + S = 0 \quad (29)$$

Remembering the expression defined in section 3.2:

$$\mathbf{E} = -\nabla\phi \quad (20)$$

$$\nabla \cdot (\varepsilon \nabla \phi) = -\rho_e \quad (21)$$

The volumetric charge density could be expressed as:

$$\rho_e = \rho z \quad (22)$$

Where:

- z (C/Kg) is the charge per unit mass

This potential model solves the equation (21). A *DEFINE_SOURCE* UDF is coded to add the source term ($S = \rho_e = \rho z$) to the Poisson equation.

After solving the potential equation, it is needed to compute the electric field as the minus gradient of electric potential (20).

In order to introduce the space charge equation, a User-Defined Scalar transport equation is used.

$$\frac{\partial \rho \phi_k}{\partial t} + \nabla \cdot (\rho \mathbf{u} \phi_k - \Gamma_k \nabla \phi_k) = S_{\phi_k} \quad (31)$$

Remembering the exposed expressions exposed in chapter 3.2:

The charge conservation equation is:

$$\frac{\partial \rho_e}{\partial t} + \nabla \cdot \mathbf{J} = 0 \quad (23)$$

Where:

- $\mathbf{J}$ (A/m²) is the current density

$$\mathbf{J} = K\mathbf{E} + \rho_e \mathbf{u} \quad (24)$$

Considering the electrostatic relationship (18) and (19), the conservation equation of the charge (23) can be written:

$$\frac{\partial \rho z}{\partial t} + \nabla \cdot (\rho z \mathbf{u}) = -\frac{K}{\varepsilon} \rho z + \mathbf{E} \cdot \left(\frac{K}{\varepsilon} \nabla \varepsilon - \nabla K \right) \quad (25)$$

If the electrical properties of the fluid K and ε are constant, eq. (25) reduces to:

$$\frac{\partial \rho z}{\partial t} + \nabla \cdot (\rho z \mathbf{u}) = -\frac{K}{\varepsilon} \rho z \quad (26)$$

For the UDS equation, the scalar ϕ_k is z (C/kg) and only the unsteady and convection terms are considered. The term $-\frac{K}{\varepsilon} \rho z$ is the source term S_{ϕ_k} of the UDS.

A *DEFINE_SOURCE* UDF is coded to introduce the source term of the UDS equation:

$$S_{\phi_k} = -\frac{K}{\varepsilon} \rho z = \frac{K \nabla \cdot (\varepsilon \nabla \phi)}{\varepsilon} = \frac{\int -\nabla \cdot (K\mathbf{E}) dV}{\int dV} = \frac{\int -\nabla \cdot K\mathbf{E} n dV}{V} = -\frac{\Sigma K\mathbf{E}\mathbf{A}}{V} \quad (39)$$

When equations (20), (21) and (26) are solved, it is necessary to introduce the Coulomb force in the momentum equation. Three different *DEFINE_SOURCE* UDFs are coded to couple the electrostatics and hydrodynamics for the three momentum conservation equations.

The use of a *DEFINE_ADJUST* Fluent macro is needed to calculate the electric field and the source term for charge conservation equation at each iteration. This type of ANSYS Fluent macro is called at the beginning of each iteration before transport equations are solved.

A UDF need to be interpreted or compiled before makes use of them. In our case, for Windows operating systems, Visual Studio is used to compile the codes.

5.9 Limitations

First, it is important to remark that the student version of ANSYS Fluent is used for these simulations. This version has an important limitation in the discretization of the domain: it is not allowed to create a mesh with more than 512K cells.

Another important limitation is the computational resources. For these kinds of simulations, the computational cost usually tends to be very high, so specialized workstations are typically used. For this master thesis, the computational resources are limited and for further work related to this case study, it is recommended to use a proper professional workstation for CFD simulations and the professional ANSYS Fluent version to avoid the limitation of the mesh generation, if a refinement is needed.

The computational resources available are:

- Intel® Core™ i7-4500U, 2.4Ghz, 4 Gb RAM
- 2 x Intel® Core™ i5-6500, 3.20 GHz, 8Gb RAM

The computational cost of the simulations is high. Even if some considerations are taken in order to improve the simulation time, the cases 3.a to 3.d took more than 85-95 hours, the cases 3.e to 3.h took about 60-70 hours. The 2D cases took about 30-40 hours and the steady state cases for the mesh study are quicker and took about 10 minutes.

6. RESULTS AND DISCUSSION

6.1 Parametric study: general comments

First of all, before analysing the different simulations and scenarios proposed, it is important to remark in this parametric study some comments about the passive heat transfer cooling method: the use of a minichannel whereby the dielectric liquid flows. This parameter is not been numerically analysed in the simulations but some qualitative comments are presented.

The expression of equation (5) shows that the heat transfer coefficient is a function of Nusselt number, fluid thermal conductivity and hydraulic diameter. For fully developed laminar flows, Nusselt number is considered as a constant value under a theoretical point of view. For a square duct with a uniform heat flux, the Nusselt number is 3.61 [80].

Considering the fully developed region, the heat transfer coefficient depends on the duct geometry and fluid thermal properties. For a given flow with a constant thermal conductivity, heat transfer coefficient grows if the characteristic length decreases. For non-circular pipes, the hydraulic diameter is used as characteristic length. Thus, the use of a channel with a reduced hydraulic diameter guarantees the improvement of the heat transfer coefficient for a given fluid for a laminar fully developed flow. It is also important to mention that the hydraulic diameter depends on the geometry of the cross-sectional area of the minichannel (4).

$$h \propto \frac{1}{D_h}$$

Concerning the pressure drop, following the friction factor expression for laminar flow (35) and square ducts, pressure drop varies inversely with Reynolds number. As the Reynolds number is directly proportional to hydraulic diameter, the pressure drop increases when the hydraulic diameter is reduced.

$$\Delta P \propto \frac{1}{Re} \rightarrow \Delta P \propto \frac{1}{D_h}$$

So it is important to find a balance between these two phenomena to ensure an enhancement of the heat transfer coefficient without penalizing in excess the pressure drop to drive the fluid through the channel [77].

After this qualitative analysis of the heat transfer enhancement using a minichannel, this document is focused in carry out an in-depth analysis of the active cooling method: the use of electrohydrodynamically induced convection.

6.2 2D simulations

6.2.1 *2D Parametric study: General comments*

In order to perform a parametric study of the device behaviour, some different scenarios are proposed. Before start analysing some special facts and particularities for the different cases studies, some general comments about the behaviour observed are explained in the following chapter.

First of all, it is important to remind the mechanism of the device: a high voltage electrode located on the top of the minichannel injects charge through the dielectric fluid. This charge injected creates a Coulomb force added to the momentum equation. Consequently, the velocity profile is modified: the charged particles push the neutral molecules of the fluid towards the heated plate that is grounded.

This general analysis starts explaining the electrical part of the device. In order to show and explain the concepts exposed, some contours of the scenario with 15 kV applied are provided.

The charge is injected from the electrode through the fluid. Figure 27 shows the electric current coloured by the electric charge (the charge at the wire is not plotted). It can be observed how the electric charge moves towards the heated plate that becomes electrically charged. The heated plate is grounded and as it is made of copper, an electric conductor material, the charge is distributed across its entire surface. Electric conductor materials allow the charges to move about freely, and the charge is uniformly distributed across its entire surface [74].

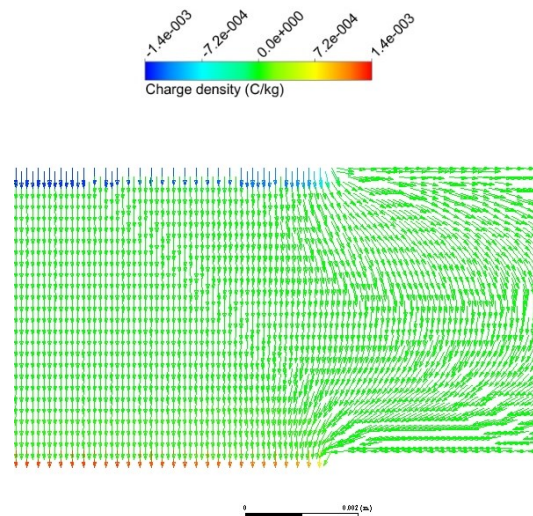


Figure 27 — Scenario 1.d: Electric current vectors coloured by charge density (C/kg) injected from the wire and collected by the heated plate ($t = 10$ s). Plane XY, $x = 0.070$ m. Wire and heated plate right edge.

It is necessary to remark some general concepts about electrical conductors. There is no electric field inside a conductor. As it is mentioned before, the electrons move easily within an electric conductor material and if an electric field appears inside the conductor, they will rapidly move and rearrange themselves owing to cancel this electric field and reach the equilibrium state. This quickly rearranged is related to the very low resistivity of conductor materials. Resistivity determines how well the conductor materials resist the flow of current. So, a low value of this parameter guarantees good

electricity conduction, demonstrating the rapid rearranged of the charges inside a conductor and, thus, that there is no electric field inside a conductor [74] [75].

After explaining this principle, it can be demonstrated that the charge will remain in the surface of the material conductor. This phenomenon is explained thanks to Gauss's Law:

$$\int_A \mathbf{E} \cdot d\mathbf{S} = \frac{1}{\epsilon_0} \int_V \rho dV \quad (40)$$

If there is no electric field inside a conductor, the charge inside will be always zero and consequently will reside entirely on its surface. In our case, part of the charge injected from the electrode is collected by the heated plate and remain in its surface, as Figure 27.

Figure 28 shows that the electric field is perpendicular to the wire and the heated plate. The Gauss Law also explains this phenomenon: if a tangential electric field appears near the surface of a charged conductor material, the charges will rearrange them in order to reach the equilibrium state and cancel this tangential electric field. So, for conductor materials the electric field is always normal to the surface of the conductor. In addition, in this figure we can observe how the electric field increases with time due to the charge injected [74] [75].

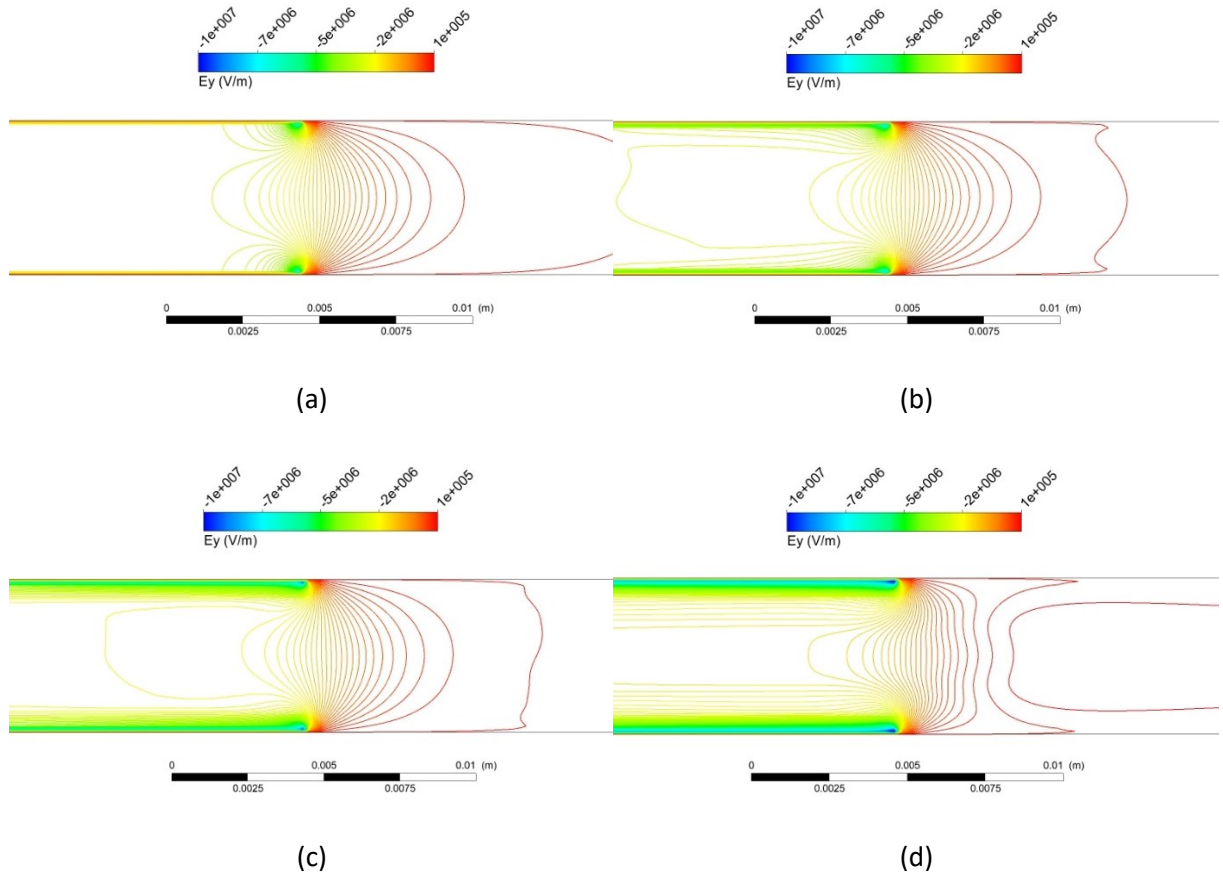


Figure 28 — y-component of electric field (E_y): (a) $t = 0.005$ s. (b) $t = 10$ s. (c) $t = 20$ s. (d) $t = 26$ s. Plane XY, $x = 0.070$ m. Wire and heated plate right edge.

RESULTS AND DISCUSSION

Figure 29 shows the appearance of induced charges near the electrode. The charge is injected gradually and part of it is absorbed by the heated plate because it is made of conductor material (Figure 27). In order to meet the charge conservation equation (26), induced negative charges appear near the electrode. Under a theoretical point of view, when a dielectric fluid is located between a high voltage electrode and a surface grounded, the molecules of the fluid are polarized. These polarized molecules are aligned with the electric field applied and all the molecules far enough the electrode and the plate are neutralized between them. Nevertheless, charges induced near the electrode remains. The dielectric fluid remains electrically neutral but suffers charge redistribution. In this case, mineral oil is a liquid composed of carbon atoms bonded to hydrogen atoms. These molecules create a very low polarity so in the absence of an electric field, they are considered as neutral molecules. However, the appliance of a high electric field causes a sufficient separation of the positive and negative charges of the molecules to make them slightly polar. Thus, induced charges appear near the wire [76].

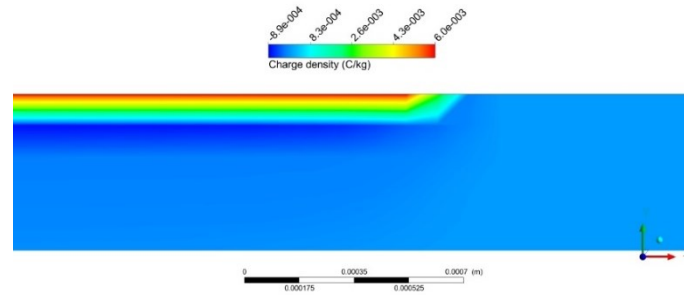


Figure 29 — Scenario 1.d: Induced charges near the electrode. Plane XY, $x = 0.070$ m, $t = 26$ s

6.2.2 2D Parametric study: scenarios 1.a to 1.d

Scenario 1.a

First of all, it is necessary to explain the scenario without the appliance of any voltage. The steady state is reached after 22 s. The velocity profile corresponds to a fully developed laminar flow in 2D. Next contour shows the velocity profile along the minichannel.

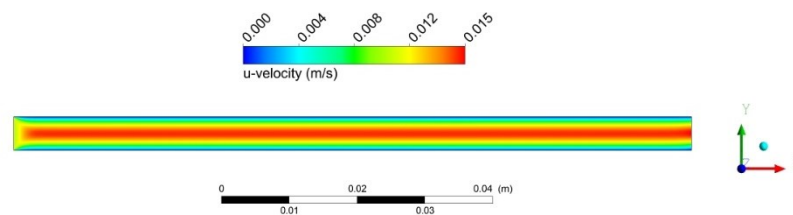
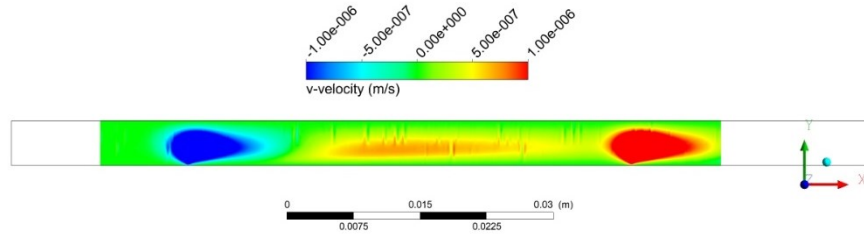


Figure 30 — Scenario 1.a: u -velocity profile at the middle plane. Plane XY, $t = 22$ s

Figure 30 shows how the flow is developed. The u -velocity of a laminar flow has a parabolic profile reaching its maximum value at the middle of the duct. Considering only 2 dimensions, the u -velocity is a function of the y -coordinate.


 Figure 31 — Scenario 1.a: v -velocity profile at the middle plane. Plane XY, $t = 5$ s

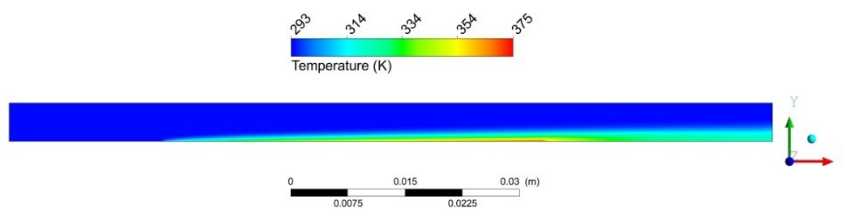
According to the velocity profile of laminar flow, the u -velocity is dominant over the v -velocity. In these simulations, the Boussinesq approximation is used so the density is considered as constant for all the equations except for the gravity term in the momentum equation. So, the v -velocity is influenced by the density variations where the temperature differences of the fluid occur. Figure 31 perfectly shows the variation of the v -velocity along the test section, where the heat transfer occurs. A further explanation of this phenomenon is done for the 3D cases (section 6.3.2). The hot fluid with lower density tends to go up and the cold fluid goes down.

Prandtl Number	Grashof Number	Rayleigh Number
396.43	38.16	1.51×10^4

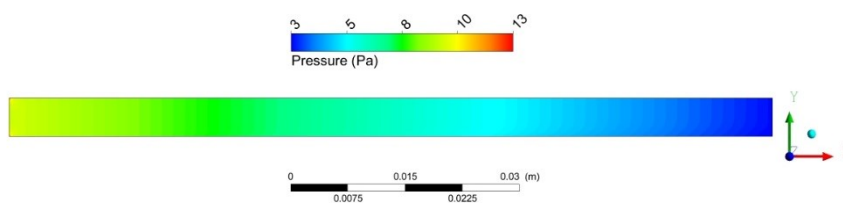
Table 10 — Scenario 1.a: Dimensionless numbers

The dimensionless numbers presented in chapter 2.1 are computed for this scenario. The Grashof number represents the ratio between the buoyancy forces and the viscous forces, and the Rayleigh number quantifies the importance between the effects of the buoyancy forces and the effects of the viscosity and thermal conduction. Due to the values obtained, we can conclude that convection occurs.

Regarding the temperature profile, Figure 32 shows how the temperature of the fluid increases since it passes through the test section. The fluid extracts heat from the heated plate and drives it through the outlet of the duct.


 Figure 32 — Scenario 1.a: Temperature contour at the middle plane (XY plane). Plane XY, $t = 22$ s.

Finally, Figure 33 shows the pressure drop needed to drive the flow through the channel.


 Figure 33 — Scenario 1.a: Pressure contour at the middle plane. Plane XY, $t = 22$ s.

Scenarios 1.b to 1.d

After presenting the overall case study, scenarios 1.b to 1.d, in which the active cooling method is applied, are going to be studied. The voltage applied at the wire is 5, 10 and 15 kV (1.b, 1.c and 1.d respectively).

Next graphs show the average and the maximum temperature of the heated plate after 26.5 s.

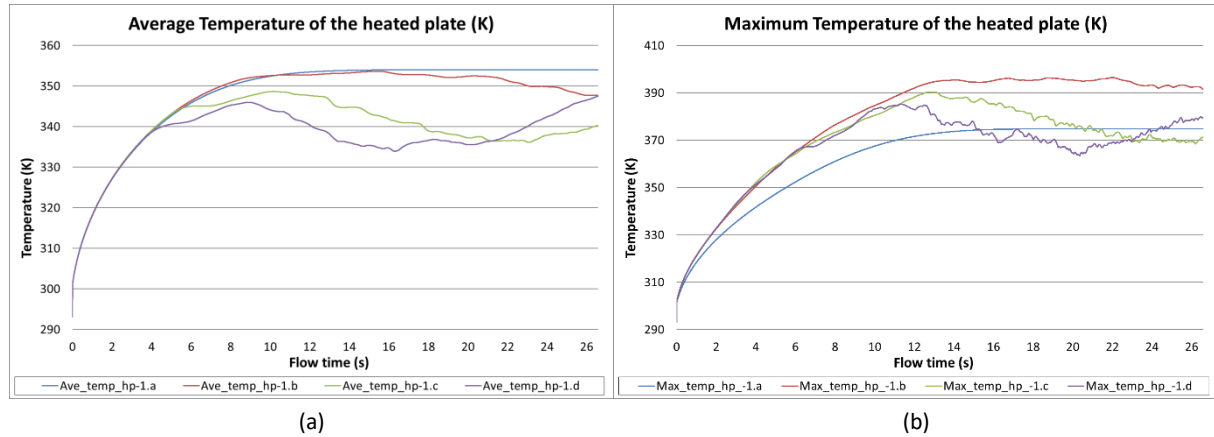
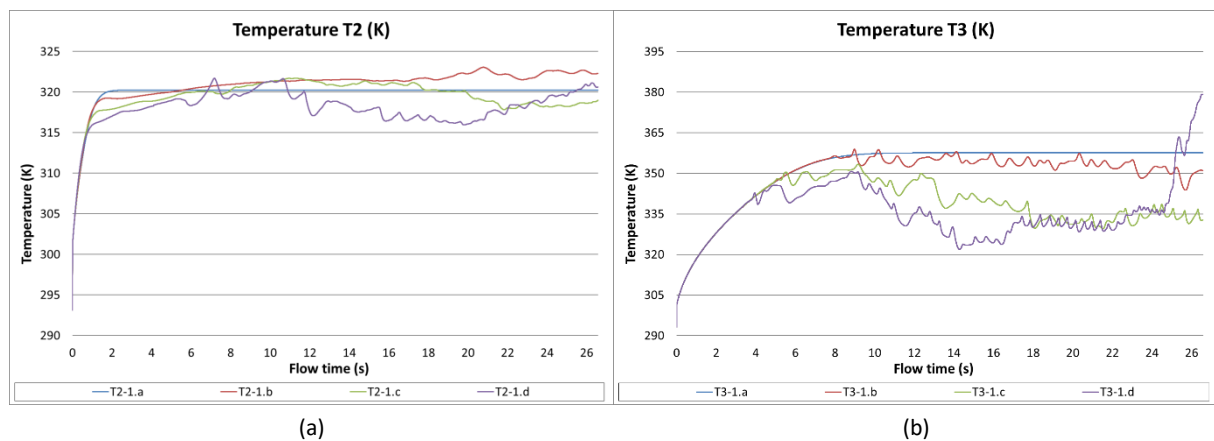
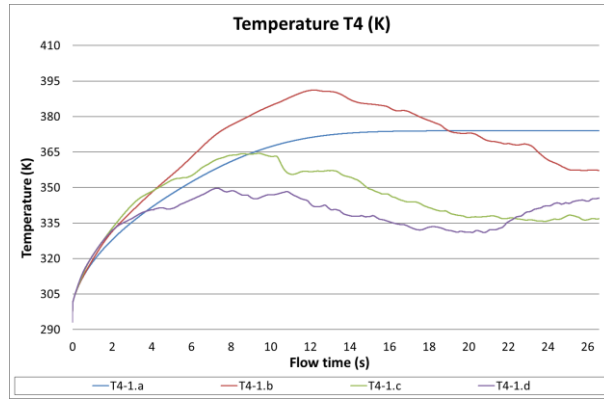


Figure 34 — Scenarios 1.b to 1.d: Average temperature and maximum temperature of the heated plate

First of all, the average temperature of the heated plate for all the scenarios is lower than in case without the appliance of the voltage and the maximum temperature reduction is observed in scenario 1.c. The temperature reduction for all the scenarios is significant, and the best reduction obtained is almost 14 K in scenario 1.c. Regarding the maximum temperatures of the heated plate, only scenario 1.c reaches to decrease the maximum temperature (4 K).

Analysing Figure 34, it can be concluded that the best behaviour of the device is obtained with the configuration 2.c. (10 kV) due to the decrease of the average and the maximum temperature of the heated plate.





(c)

Figure 35 — Scenarios 1.b to 1.d: (a) Temperature measured at point T2 (22, -2.5, 0) mm. (b) Temperature measured at point T3 (45, -2.5, 0) mm. (c) Temperature measured at point T4 (68, -2.5, 0) mm.

Focused in the temperatures measured along the heated plate, oscillations are observed in Figure 35 (a), (b) and (c). This fact is due to the development of the electric field and its contribution to the conservation of momentum equations (27). The interaction between the main flow momentum and the momentum that is applied by the injection of charge is higher when the voltage increases owing to the oscillations observed.

First, the Poisson equation is solved for the electric potential following the equation (21). Then, the electric field is calculated as the gradient of the voltage (20), and finally the equation of conservation of charge density (26) is solved. So, the electric field is gradually increasing with the injection of charge and depends on the voltage applied at the wire.

With the purpose of understanding the evolution of the simulation with time, some different contours at different time steps of scenario 1.d are presented to see the interaction between the electric field and the flow in the test section. This scenario is selected because the modifications in the velocity profile are more noticeable. The same reasoning can be done for scenarios 1.b and 1.c.

At $t = 0.02$ s, a positive value of electric field in x-direction is observed at the right edge of the wire ($x = 0.070$ m) and a negative value at the left edge ($x = 0.020$ m). At the edges of the heated plate, values of electric field in x-direction appear with opposite sign (positive value at $x = 0.020$ m and negative value at $x = 0.070$ m). Since the charge injected is almost negligible, the velocity profile observed is a typical laminar velocity profile without modifications, similar to Figure 30.

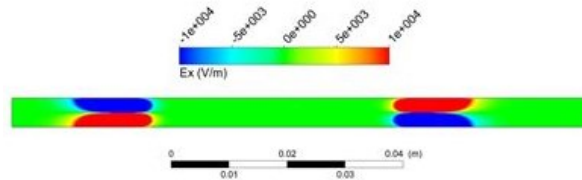


Figure 36 — Scenario 1.d: x- component of electric field (V/m) along the minichannel. Plane XY

At $t = 6$ s, the oscillations are appreciable in the three points of measure, so the value of Coulomb force added to the momentum equation starts to be noticeable. The electric field present at the outlet of the test section is developed towards the entry and the same phenomenon but with

opposite direction happens with the electric field present at the inlet of the test section, in order to meet the conservation of charge density equation. Induced charges appear near the electrode due to the positive charges located at the surface of the heated plate. At the outlet of the test section ($x = 0.070$ m), these negative charges near the electrode create a negative contribution to the x-momentum equation due to its interaction with the positive electric field present in the upper part of the minichannel. At the inlet, the electric field at the upper part is negative, so the fluid is pushed towards the outlet. At the bottom of the minichannel, the same phenomenon is observed: the fluid is pushed towards the inlet section at the outlet of the test zone (positive charge and negative x-component of electric field) and is pushed in x-positive direction at the entry of the test section (positive charge and positive x-component of electric field).

Charge injected starts to be present in the fluid and the flow is modified. The same phenomenon explained before happens along the test section: where the product of the value electric field in x-direction and the charge injected in the fluid is positive, the fluid is pushed towards the outlet (in x-positive direction). But, when the value of this product is negative, the fluid is pushed towards the inlet.

As a result of this interaction, some different “waves” are created along the test section. The fluid pushed by the Coulomb force towards the outlet of the channel (positive x-direction) is modified where the contribution of the Coulomb force to the momentum equation is negative: the fluid that moves in the positive direction of the x-axis is altered by the fluid that is pushed in the negative direction of the x-axis as a result of the negative product of the charge and the electric field. This phenomenon produces these oscillations in the velocity profile. Some contours at $t = 16$ s are presented in order to explain better this fact. They show perfectly the phenomenon explained. The positive contribution of Coulomb force to the momentum equation push the fluid towards the outlet and is interrupted and varied by the fluid with negative velocity value just below, creating a wave (Figure 37). This negative velocity is produced due to the negative value of electric field at the bottom of the minichannel and the positive value of the charge near the plate.

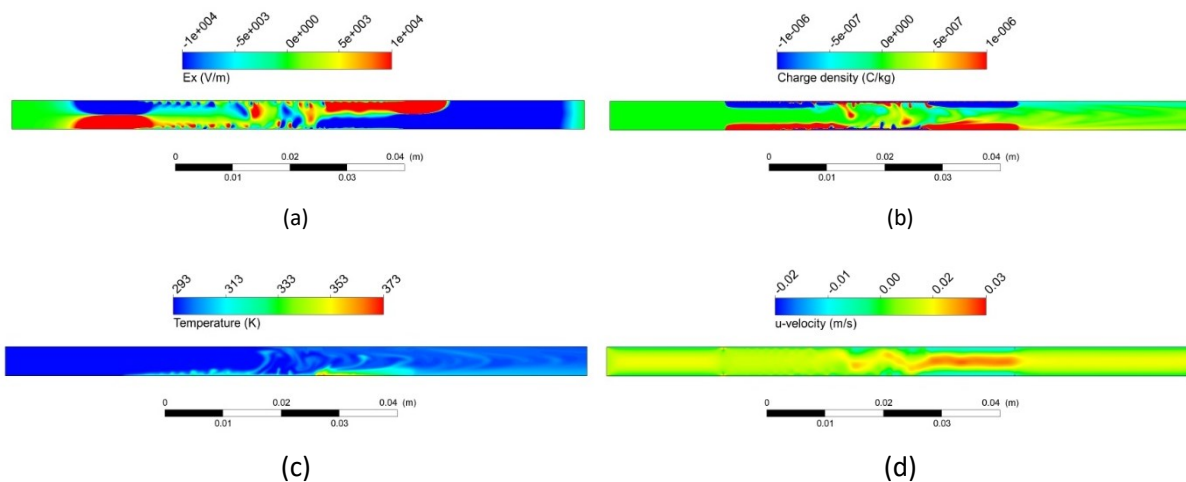


Figure 37 — Scenario 1.d: Different contours at $t = 16$ s along de minichannel: (a) x- component of electric field (V/m). (b) Charge density (C/kg). (c) Temperature (K). (d) u-velocity component (m/s). Plane XY, $t = 16$ s.

This modification of the velocity profile is responsible for the temperature oscillations of the plate. When the fluid is modified by the localized flow that moves in the negative direction of the x-axis, the temperature profile is also altered. The flow with positive velocity manages to extract heat from the plate while the recirculated flow with negative velocity re-introduces hot fluid towards the test zone. This fact increases the maximum temperature measured in the heated plate. Figure 37 and Figure 38 perfectly show this fact at the outlet of the test section ($x = 0.070$ m). The negative velocity at the bottom and top of the minichannel ($x = 0.070$ m, $y = \pm 0.0025$ m) reintroduces hot fluid.

When the electric field is almost totally developed and stable, we can observe that the negative charge is accumulated at the upper part of the channel and the positive charge is located near the plate. The velocity profile of the first part of the test section is more homogeneous and near the electrode and near the plate, the fluid is pushed towards the outlet section. The “waves” of the flow disappear.

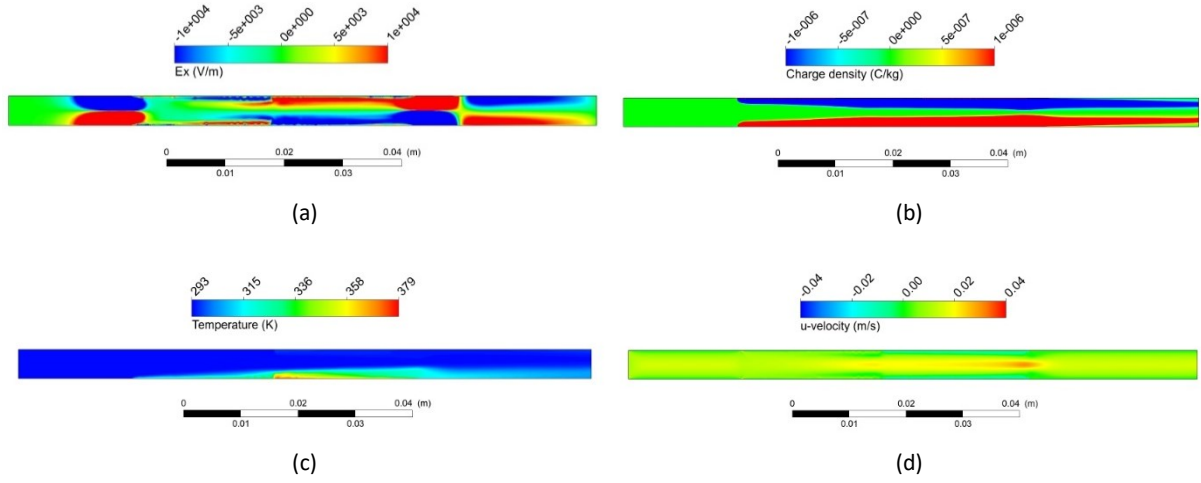


Figure 38 — Scenario 1.d: Different contours at $t = 26.5$ s along de minichannel: (a) x- component of electric field (V/m). (b) Charge density (C/kg). (c) Temperature (K). (d) u-velocity component (m/s). Plane XY, $t = 26.5$ s.

Regarding the v -velocity, the same reasoning can be done for the interaction of the main flow momentum and the momentum applied by the injection of charge. The electric field in y-direction is always negative in the test section so a negative value of v -velocity is observed where the charge is positive, pushing down the fluid. Nevertheless, as it is explained before, the u -velocity is dominant.

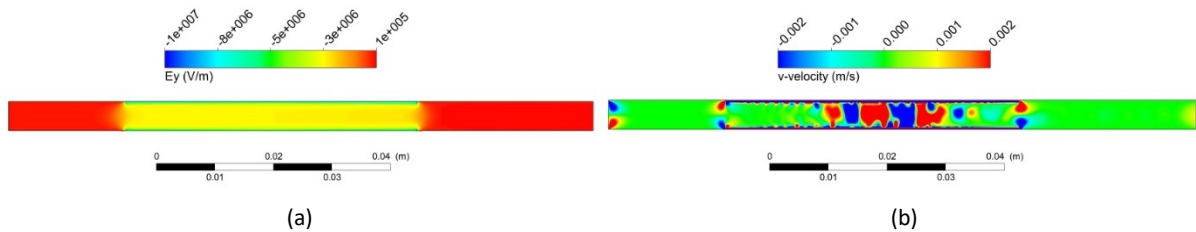


Figure 39 — Scenario 1.d: Different contours at $t = 16$ s along de minichannel: (a) y- component of electric field (V/m). (b) v-velocity component (m/s). Plane XY, $t = 16$ s.

RESULTS AND DISCUSSION

Summarizing, the electric field produces some recirculation issues along the minichannel that are more noteworthy when the voltage applied is greater. The recirculation in scenario 1.d is greater than in scenario 1.b.

It is important to analyse the Joule heating generated. The Joule heating is equal to the electric current that passes through a conductor multiplied by the voltage (9). In the model, the biggest source of Joule heating is the wire. But, since it is treated as an adiabatic wall, it does not introduce heat into the domain. Nevertheless, different behaviour is observed at the heated plate due to the accumulation of charge. As it is explained before in section 6.2.1, the heated plate becomes electrically charged due to the material that is made of (copper). As the heated plate is not considered as an adiabatic wall because it introduces the heat flux into the domain, the Joule heating generated contributes to the increase of the temperature of the plate. However, this contribution to the augmentation of the temperature is lower than the temperature increase that produces the recirculation problems. In the analysis of 3D cases, a deep explanation of this fact is presented.

Analysis of the parameters

The parameters defined in chapter 5.2 are computed. First, the Nusselt number is the ratio of convective to conductive heat transfer. A high value of Nu means that convection is dominant in the heat transfer process. To compute it, the expression (5) defined in chapter 2.1 is used. The ratio Nu/Nu_0 compares the Nusselt number obtained for the 3 scenarios with the appliance of the voltage and the scenario with zero voltage applied. This ratio allows determining if there is a heat transfer enhancement. A value greater than unity means that the heat transfer rate enhances.

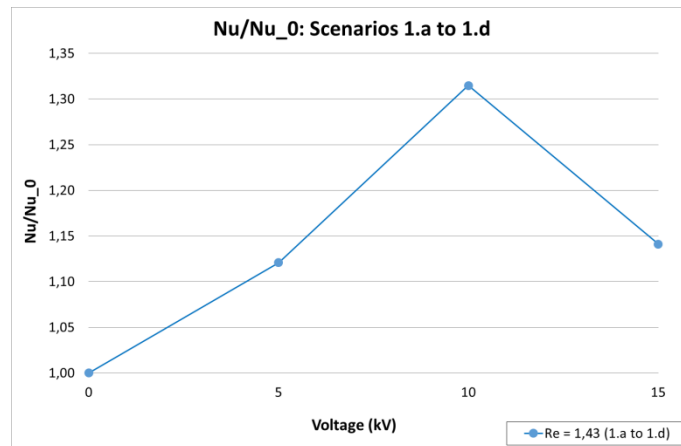


Figure 40 — Ratio of Nu/Nu_0 vs. applied voltage (kV) for scenarios 1.a to 1.d

The average temperature of the heated plate is lower for the 3 scenarios with applied voltage, and thus the heat transfer coefficient is higher than the computed for scenario with zero voltage.

Regarding the pressure drop, Figure 41 that it increases with the voltage applied. This growth is accentuated when the applied voltage is greater because the electric field is also greater, and therefore the contribution of the Coulomb force in the conservation of momentum equations is also higher.

The pressure drop observed is high and is due to the big interaction between the main flow momentum and the momentum applied by the injection of charge. The modification in the velocity profile is important and thus, the pressure needed to drive the fluid considerably increases.

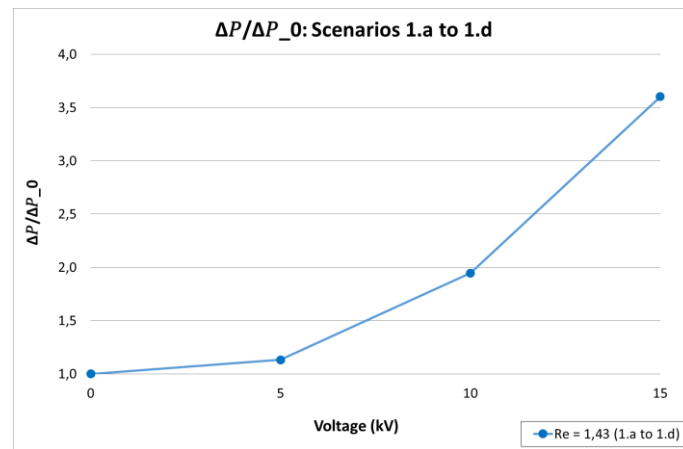


Figure 41 — Ratio of $\Delta P/\Delta P_0$ vs. applied voltage (kV) for scenarios 1.a to 1.d

When analysing a cooling heat transfer method, it is not only important to evaluate the heat transfer enhancement. It is necessary to study the increase in the pumping requirements. Performance Evaluation Criterion (PEC) relates the heat transfer enhancement to the pumping power needed to drive the fluid along the duct. The heat transfer coefficient can be higher but if the increase in the pumping requirements is meaningful, the efficiency of the cooling technique can be lower. So, values greater than unity mean that the rate of heat transfer enhancement is higher than pressure drop needed to pump the fluid.

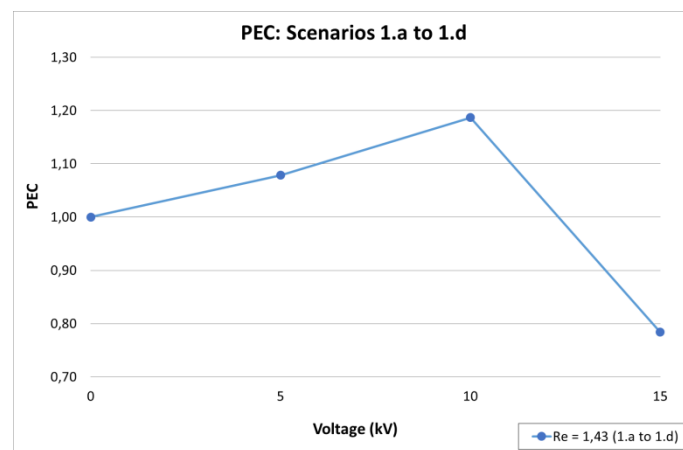


Figure 42 — PEC vs. applied voltage (kV) for scenarios 1.a to 1.d

Scenarios 1.b and 1.c show a better efficiency than the scenario 1.a without voltage applied. The heat transfer enhancement is significant and the pressure drop increase is not too high. However, in scenario 1.d, even if a heat transfer enhancement is noticed due to the value greater than unity of ratio of Nu/Nu_0 , the pressure drop increase is high. The effects of the electric field are noteworthy and the higher recirculation issues produces that the pressure needed to drive the fluid is high, and the value of efficiency is lower than the one computed for the scenario with zero voltage condition.

6.3 3D simulations

6.3.1 3D Parametric study: General comments

After analysing the model in 2D, a three-dimensional model is prepared. First, a previous mesh study is performed in order to determine the best discretization of the domain to guarantee accurate results. It is extremely recommended in CFD calculations find a proper balance between the accuracy of the results desired and the computational cost to achieve them.

6.3.2 3D Parametric study: Mesh study

First of all, nine different scenarios are analysed in order to determine the relevance of the discretization of the domain in the accuracy of a CFD simulation. Three different steady-state scenarios are simulated for three different hexahedral meshes.

The geometry used for this preliminary study is:

Zone	Length (mm)	Measurement point	Position of measurement plane/point (mm)
Inlet	20	Inlet_T1	(10, -2.5, 0)
		P1	x = 18
Test	50	T2	(22, -2.5, 0)
		T3	(50, -2.5, 0)
		T4	(68, -2.5, 0)
Outlet	25	Outlet_T2	(85, -2.5, 0)
		P2	x = 72

Table 11 — Mesh study: geometry and measurement planes/points

Table 12 overviews the different parameters used.

Case	Grid	Max. element face size (mm)	Total elements	Orthogonal quality	Maximum aspect ratio
2.1	10 x 10	0.5	19000	1	1.73205
2.2	20 x 20	0.25	152000	1	1.73205
2.3	29 x 29	0.17	470119	0.999942	1.75831

Table 12 — Mesh study: parameters

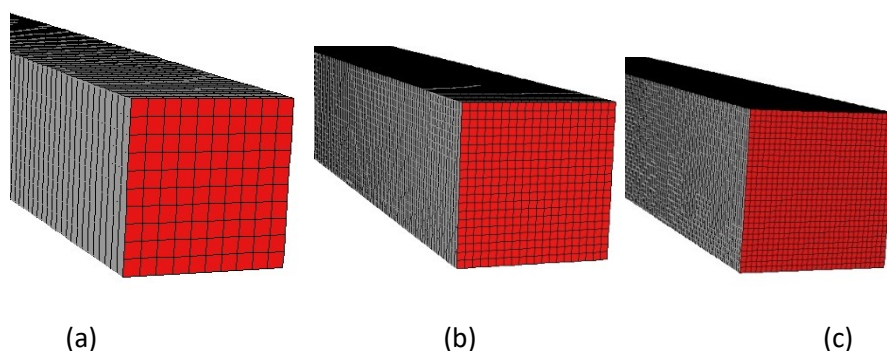


Figure 43 — Mesh study: Details of different mesh studied at the outlet section: (a) Case 2.1. (b) Case 2.2. (c) Case 2.3

For this study, the solver settings used are the mentioned ones in chapter 5.5, with some considerations: the Boussinesq approximation for density is not considered.

First, a hexahedral mesh is selected for these simulations for different reasons. The square geometry of the channel helps to discretise the domain with hexahedral elements over other types of mesh elements. Tetrahedral elements can fix better complex geometries, curves and shapes but can introduce distortion in the elements, for example skewed triangles faces.

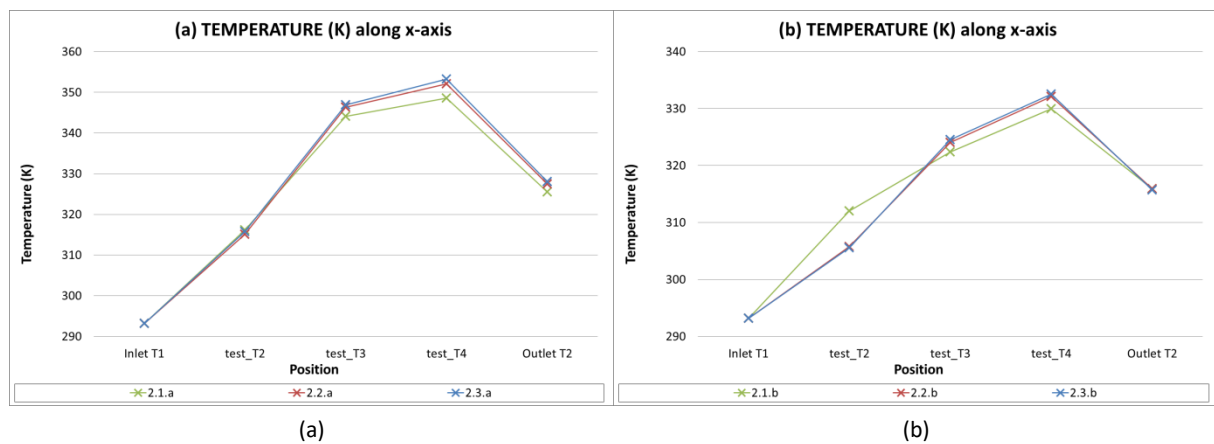
Another point to use a hexahedral mesh is the perfect alignment of the elements with the fluid flow owing to reduce and minimize numerical diffusion. A good mesh is associated with the physics you want to solve, and for the laminar flow of these simulations, hexahedral cells will obtain good accurate results. In addition, the use of hexahedral cells should reduce the computational cost during the simulation and usually need fewer elements to discretise the domain. Thus, due to the limited resources available to accomplish these simulations (limited cells and huge simulation time), a hexahedral mesh is preferred.

Three different velocity inlets are used to analyse the accuracy of the mesh definition. The temperature is measured in 5 different points along the x-axis and the pressure is measured at the inlet and the outlet of the test section. Table 13 and Figure 44 summarize the results obtained:

		1.1			1.2			1.3		
		a	b	c	a	b	c	a	b	c
u	m/s	0.01	0.05	0.1	0.01	0.05	0.1	0.01	0.05	0.1
Re	—	1,43	7,13	14,27	1,43	7,13	14,27	1,43	7,13	14,27
f	—	41,45	8,30	4,15	42,61	8,53	4,27	42,82	8,58	4,29
f_s	—	39,81	7,96	3,98	39,81	7,96	3,98	39,81	7,96	3,98
Error	%	7,03%	7,16%	7,16%	6,57%	6,69%	6,70%	3,94%	4,08%	4,08%

Table 13 — Mesh study: friction factors

Next graphs show the temperatures measured along the minichannel at the points presented in Table 11:



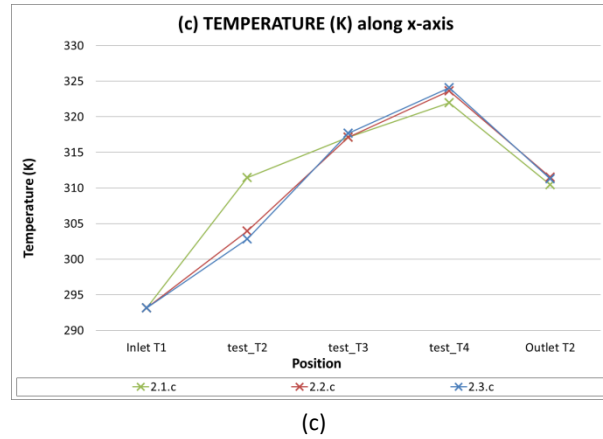


Figure 44 — Mesh study: temperatures measured at different point locations defined in Table 11. (a) Case a. (b) Case b. (c) Case c.

Analysing these graphs, the mesh corresponding to the case 2.1 can be discarded due to the disparity of the results regarding the two other cases, in which the differences are low. It can be observed as well that temperature differences are larger in case c ($Re = 14.27$) than in case a ($Re = 1.43$). In order to determine what mesh grid is more accurate, the friction factor (f) is calculated and compared to friction factor calculated with the classical correlation (f_s). The expressions (35) and (36) presented in chapter 5.2 are used.

As it is expected, errors between the friction factor performed with classical correlation and the obtained in the simulations of case 2.3 are lower than the ones of 2.1 and 2.2, due to the refinement of the mesh.

This mesh study does not pretend to study the velocity profile and the temperatures obtained, but some comments can be done in order to discuss the accuracy of the values as a result of the mesh quality. As a consequence of the velocity profile of fully developed laminar flow, the maximum velocity is reached at the centre of the duct and the velocity near the walls trends to be null. This fact explains why the maximum temperature of the heated plate is reached at the corners of the heated plate. The different maximum temperatures obtained for the heated plate are 390.13 K, 392.06 K and 392.08 K, for the 3 different meshes respectively (2.1, 2.2, 2.3) for the case with a velocity inlet equal to 0.01 m/s. This disparity of the maximum temperature obtained in the mesh with the poorest quality (2.1) confirms that the finest discretization of the domain guarantees the accuracy of the results obtained.

A detailed explanation about the velocity profile and the temperatures obtained is going to be presented in chapters 6.3.3 and 6.3.4.

These steady state simulations not only show the importance of the discretization of the domain in order to have good accurate results, but also guarantee the validity of the model due to the errors obtained for the frictional factor compared to the theoretical ones.

As a general conclusion, it is evident that the most accurate results are obtained with the use of the mesh as finest as possible. However, it is important to find a balance between the discretization of the domain and the computational cost of the simulations.

6.3.3 3D Parametric study: scenarios 3.a to 3.d

Scenario 3.a

At first, the temperatures and the velocity profile are going to be studied for the scenario 3.a without the appliance of the voltage at the wire.

As it is explained before for the 2D cases, the flow studied in the simulations is a fully developed laminar flow. Figure 45 shows how flow is developed.

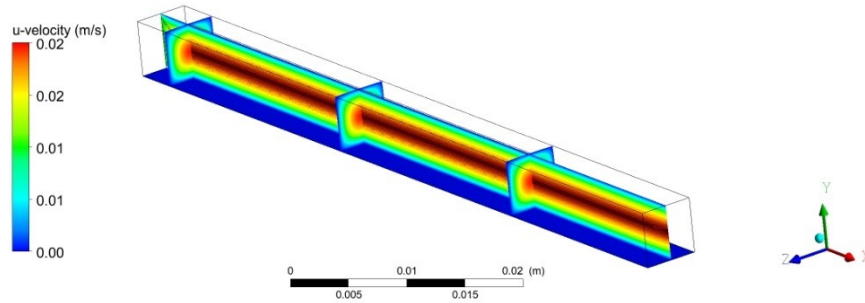


Figure 45 — Scenario 3.a: Velocity u profile along the minichannel. XY planes at $x = 0.004$ m, $x = 0.029$ m, $x = 0.054$ m, $t = 22$ s.

In the previous Figure 45, it can be observed the typical laminar flow velocity profile mentioned before. Due to the no-slip condition at the walls, the u -velocity increases as it moves away from the walls until reach the centre of the minichannel, conforming a parabolic profile. The velocity reaches its maximum value at the middle of the duct owing to meet the conservation of mass principle. The origin of coordinates is considered at the middle point of cross-sectional area of the inlet boundary section. So, the u -velocity decreases for $\pm y$ and z coordinates.

According to the velocity along the y -axis, some contours are plotted in order to explain the effects of the buoyancy forces.

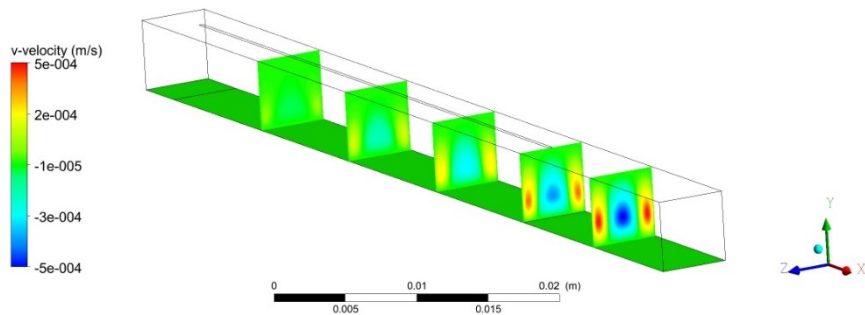


Figure 46 — Scenario 3.a: v -velocity profile along the minichannel. XY planes at $x = 0.0165$ m, $x = 0.029$ m, $x = 0.0415$ m, $x = 0.054$ m and $x = 0.064$ m, $t = 22$ s.

The Boussinesq approximation is used for density. This approximation considers a constant density for all the equations except for the gravity term in the conservation of momentum equation. The gravity force is considered in the y -momentum equation, so the y -axis velocity (v) is modified. As it is

RESULTS AND DISCUSSION

explained in chapter 3.1, this approximation is useful when the temperature differences are low, thus it is desirable for this case study.

Figure 47 shows the evolution of v -velocity along the minichannel. The velocity is increasing along the channel at the same time that the heat transfer between the heated plate and the fluid occurs. Next detailed contours show better this fact:

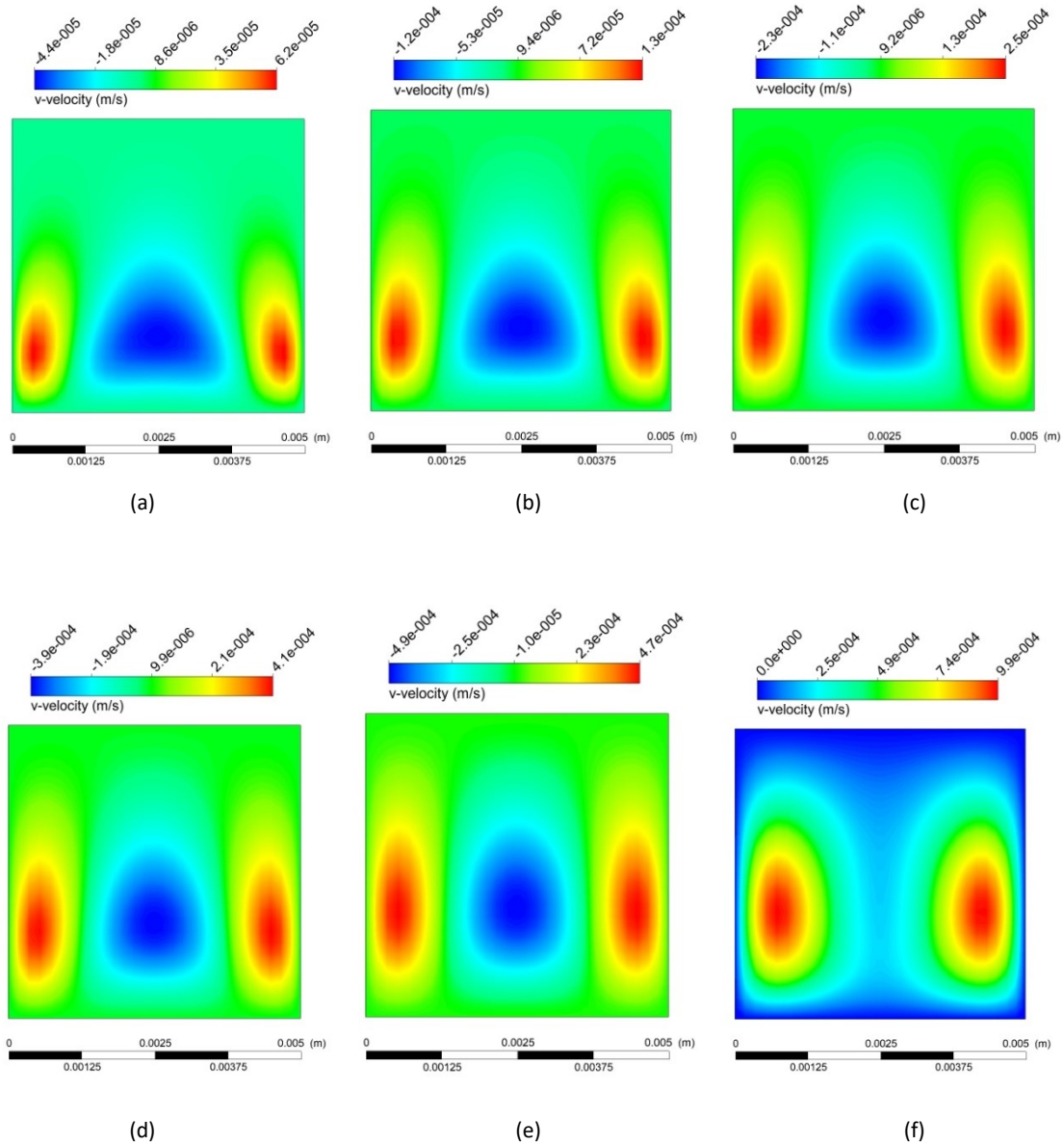


Figure 47 — Scenario 3.a: v -velocity profile along the minichannel. (a) XY plane, $x = 0.0165$ m, (b) XY plane, $x = 0.029$ m, (c) XY plane, $x = 0.0415$ m, (d) XY plane, $x = 0.054$ m, (e) XY plane, $x = 0.064$ m, (f) XY plane, $x = 0.074$ m, $t = 22$ s.

As the fluid passes through the test section in x positive direction (Figure 47 a to f), its temperature increases and the temperature of the heated plate decreases. Due to the laminar velocity profile, the u -velocity at the centre of the channel is considerably greater than the v -velocity. This explains that the maximum velocity values on the y -axis are close to the walls. In fact, as it moves away from the vertical axis in z -direction, the velocity in x decreases and the effects of the buoyancy forces are greater, increasing v -velocity. Therefore, v -velocity of the fluid in the centre of the channel

decreases and near the walls increases. In addition, as the u -velocity is greater in the centre of the duct, the fluid is replaced quicker than in the zones near the walls and the heat extracted with this colder fluid is higher. When the fluid abandons the test section (figures e and f), the v -velocity continues to rise near the walls.

As it is explained in chapter 2.1, the Grashof number computes the ratio between the buoyancy forces and viscous forces acting on a fluid. In this case, the Grashof number is 35.38. It is not a high value but remarks the effect of buoyancy forces over viscous forces. Both values of Grashof number and Rayleigh number computed are very similar to the ones computed for the two dimensional case. So, the same conclusion can be done: the role of convection in heat transfer is significant due to the Rayleigh number is greater than the critical Rayleigh number mentioned in chapter 2.1 (1700).

The dimensionless numbers presented in chapter 2.1 are computed for this scenario.

Prandtl Number	Grashof Number	Rayleigh Number
396.43	35.38	1.40×10^4

Table 14 — Scenario 3.a: Dimensionless numbers

Concerning now the temperature analysis, next contour shows the temperature of the heated plate after reached the steady state. Just to remind, in chapter 2.2, figure 8 shows how the temperature of an electric component rises during a transient period, until its stabilization in the steady state. In this experiment, the heated plate takes 22 seconds to reach this steady state and the temperature contour is as follow:

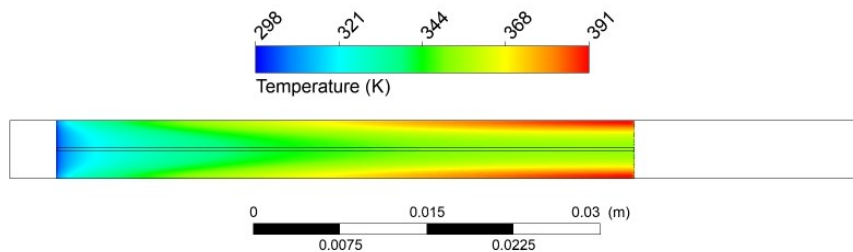


Figure 48 — Scenario 3.a: Temperature contour of the heated plate reached the steady state. Top view, plane XZ, $t = 21$ s.

As it is expected owing to the velocity profile of laminar flow, the u -velocity at the centre of the channel is higher than the u -velocity near the walls, and the v -velocity is greater near the walls as it explained before. At the entrance of the heated plate, the fluid temperature is constant with a value of 293.15 K. The fluid temperature increases along the test section, which explains that the heat removed at the entrance of the test section ($x = 0.004$ m) is greater than at the outlet ($x = 0.054$ m): the temperature measured at the inlet of the test section is lower than the one measured at the outlet.

In concordance with the v -velocity, the temperature of the fluid rises near the walls of the duct where the buoyancy forces are more relevant. The fluid temperature along the middle plane of the minichannel (plane XY, $z = 0$) gradually increases through its pass along the test section.

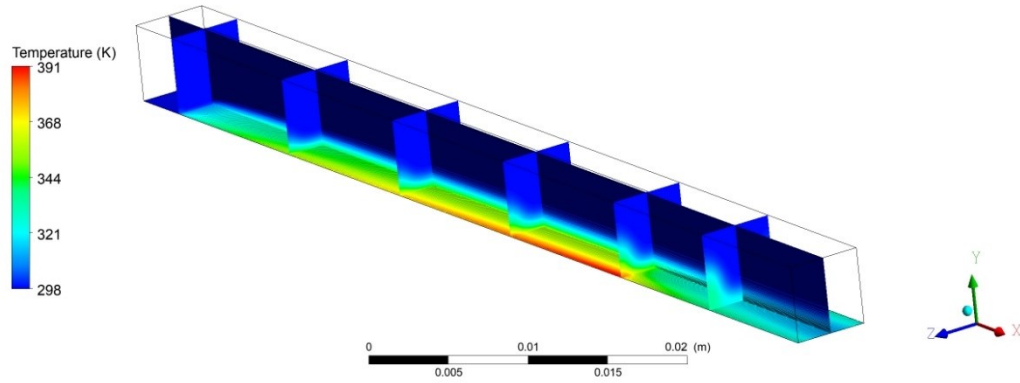


Figure 49 — Scenario 3.a: Temperature contour along the minichannel. YZ planes at $x = 0.004\text{m}$, $x = 0.0165\text{ m}$, $x = 0.029\text{ m}$, $x = 0.0415\text{ m}$, $x = 0.054\text{ m}$ and $x = 0.064\text{m}$, $t = 22\text{ s}$.

The previous contour confirms the explanation done before. The fluid temperature at the middle of the duct is lower than in the walls due to the laminar velocity profile. The fluid is replaced by new colder fluid quicker near the centre of the channel and the fluid temperature increases gradually along the z -axis with the drop of the u -velocity. The v -velocity is lower at the centre of the minichannel due to the higher u -velocity value and gradually increases with the z -coordinate until its maximum near the walls, where the fluid reaches its maximum temperature and the buoyancy forces are more meaningful.

Finally, the pressure contour shows a pressure drop along the minichannel owing to drive the fluid within the duct.

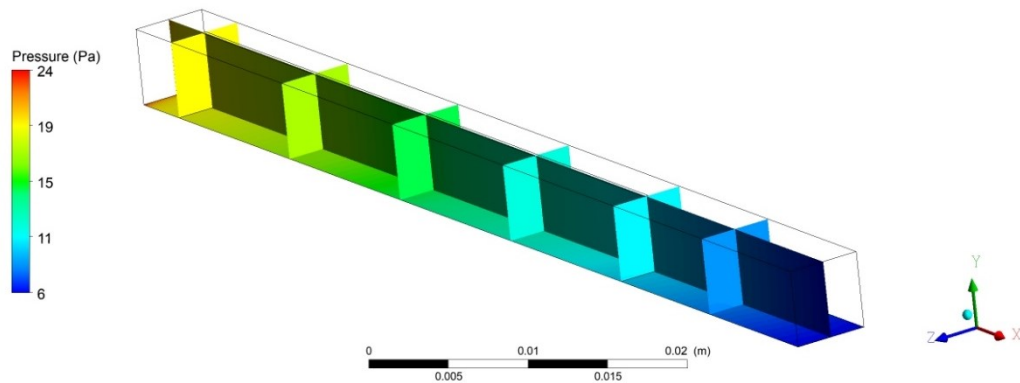


Figure 50 — Scenario 3.a: Pressure contour along the minichannel. YZ planes at $x = 0.004\text{m}$, $x = 0.0165\text{ m}$, $x = 0.029\text{ m}$, $x = 0.0415\text{ m}$, $x = 0.054\text{ m}$ and $x = 0.064\text{m}$, $t = 22\text{ s}$.

Scenarios 3.b to 3.d

After the analysis of the scenario 3.a, the scenarios with the appliance of the voltage at the wire are analysed. Next graphs show the temperatures obtained measured at the different points defined in chapter 5.3.2, and the maximum and average temperature of the heated plate:

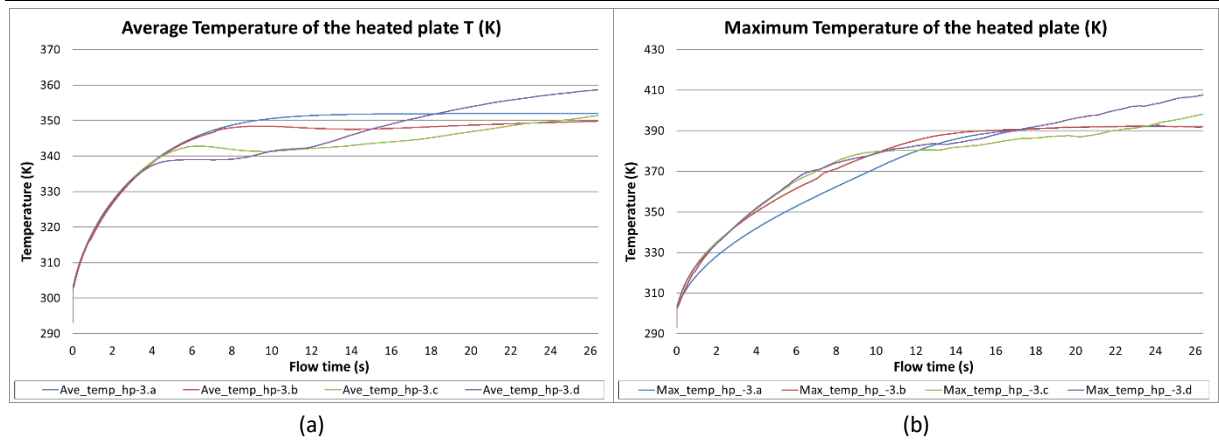


Figure 51 — Scenarios 3.b to 3.d: (a) Average temperature of the heated plate and (b) maximum temperature of the heated plate

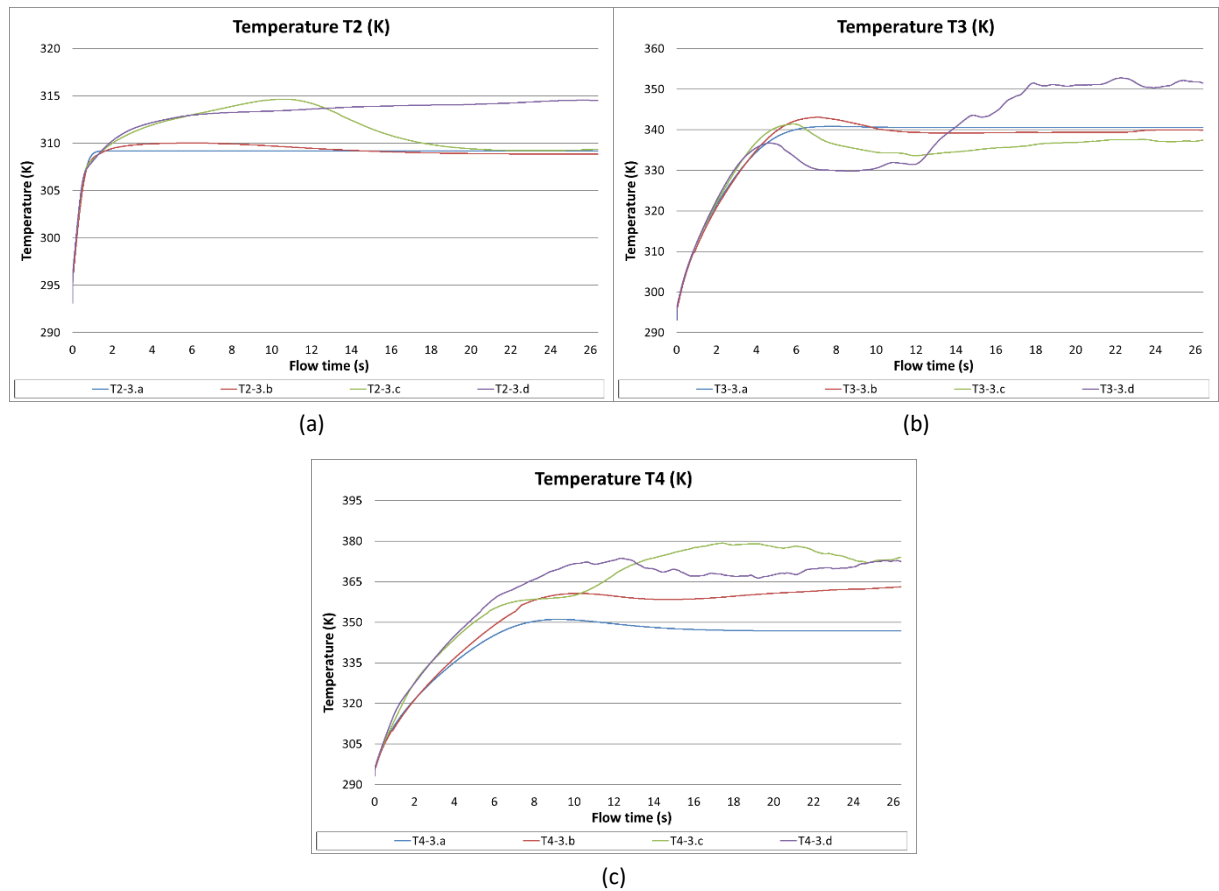


Figure 52 — Scenarios 3.b to 3.d: (a) Temperature measured at point T2 (4, -2.5, 0) mm. (b) Temperature measured at point T3 (29, -2.5, 0) mm. (c) Temperature measured at point T4 (54, -2.5, 0) mm.

First of all, as it can be observed in Figure 52, only the configuration of case 3.b reaches to decrease the temperatures measured unless for the last part of the heated plate (T4). The maximum temperature reached is similar and overall average temperature of the plate is lower.

Before performing an in-depth analysis of the temperatures obtained, some considerations of the electric field created will be discussed. As it is explained before in chapter 6.2.1, the wire injects charge through the fluid and has a defined voltage.

RESULTS AND DISCUSSION

In 3D, a third component of the electric field appears in z-axis. Some contours are plotted in order to see the electric field produced at the initial moments of the simulations. The electric field develops with time as the voltage and the injection of charge do.

Regarding the electric field in y-axis, a similar behaviour as in 2 dimensions is observed. Perpendicular negative electric fields lines are observed from the wire to the heated plate. This negative electric field in y-axis creates a negative value of v -velocity where the charge injected is positive. Near the electrode, where the charge introduced to the fluid is maximum, the v -velocity reaches its maximum negative value and the fluid is pushed in the negative y-direction. Near the heated plate, the buoyancy forces appears and the v -velocity is positive

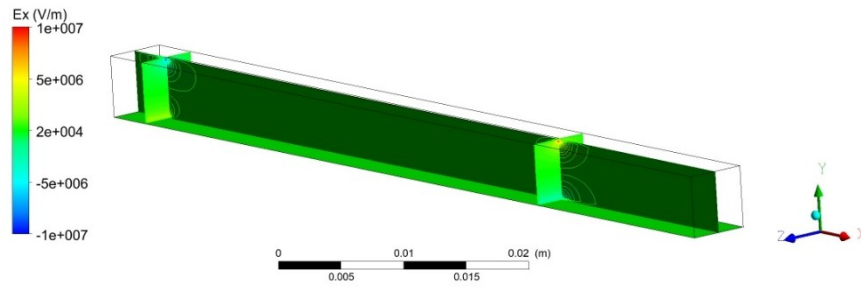


Figure 53 — Scenario 3.d: Electric field x - component. $t = 0.1$ s

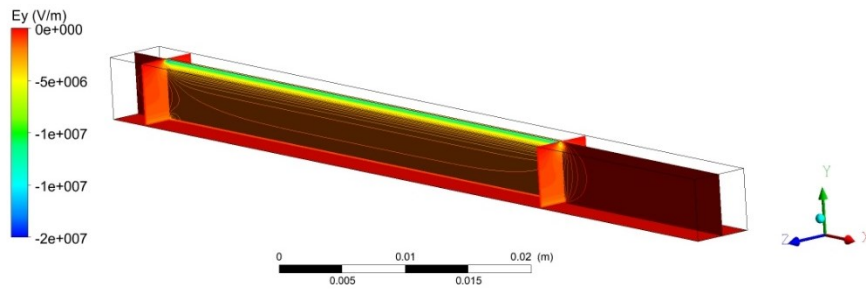


Figure 54 — Scenario 3.d: Electric field y - component. $t = 0.1$ s

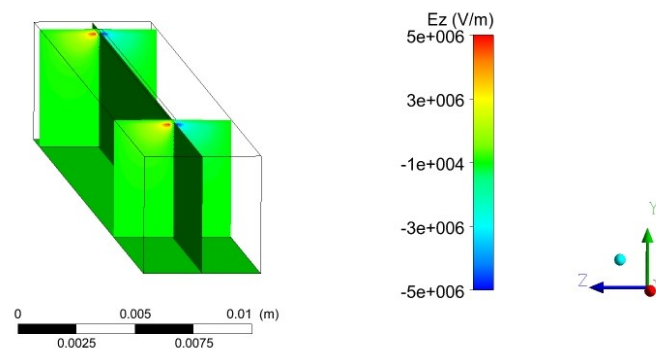
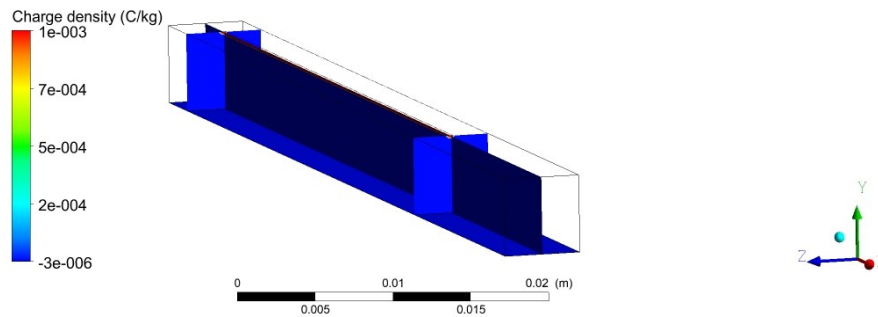
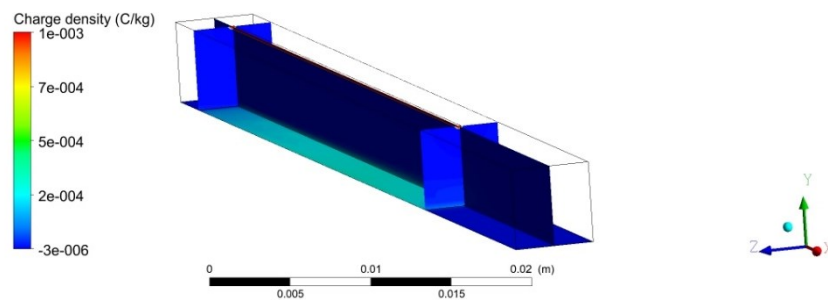


Figure 55 — Scenario 3.d: Electric field z - component. $t = 0.1$ s

The electric field modifies the velocity profile owing to the Coulomb force introduced in the conservation of momentum equation (27). The first evident conclusion is that this velocity profile modification will increase with the voltage applied since the electric field is computed as the gradient of the voltage (20). Thus, the contribution of the electric field to the momentum equations in case 3.d is higher than the one of case 3.b. Regarding the x-direction, which velocity component is dominant in laminar flows, it can be observed in Figure 53 to Figure 55 how the electric field appears in the device: a positive value at the right edge of the wire ($x = 0.054$ m) and a very similar negative value at the entry of the test section ($x = 0.004$ m). As the heated plate becomes electrically charged with the injection of charge due to the material that is made of, an electric field with opposite sign appears at both edges of the plate (positive value at $x = 0.004$ m, and negative value at $x = 0.054$ m). As the charge injected rapidly moves to the corners of the heated plate, these opposite electric fields appear since the voltage is applied and grows gradually since the heated plate absorbs charge. The charges progressively absorbed rearrange and redistribute themselves towards the corners of the heated plate, increasing the electric field modulus at the lower part of the channel. This is explained as a result of the electrical stability principles of conductor materials: the charges rearrange quickly in conductors in order to reach the equilibrium state (an in-depth explanation of this fact is given at chapter 6.2.1). Thus, the electric field grows with the injection of charge and the development of the voltage. This electric field at the edge of the heated plate ($x = 0.054$ m) will be the main reason of the fluid recirculation issues observed.

Figure 56 — Scenario 3.d: Charge density. $t = 0.1$ sFigure 57 — Scenario 3.d: Charge density. $t = 26$ s

This evolution of electric field means that the modification of the velocity profile is gradual. The negative electric field of the edge of the plate located at $x = 0.054$ m creates an undesired effect: it reintroduces fluid with high temperature towards the test section with a constant velocity along the cross-sectional area. This produces a localised augmentation of the temperature at the end of the

heated plate. Since z -coordinate grows in positive and negative values, the u -velocity decreases and the recirculation is more undesirable because the hot fluid increases the temperature at the corners of the heated plate. As it is explained for the case 3.a, the maximum temperature value is reached just in the corners of the plate due to the laminar velocity profile, so this recirculation does not help to cool down this localised zone. In addition, the recirculation increases with the voltage applied. As examples, two contours are plotted for scenarios 3.b and 3.d.

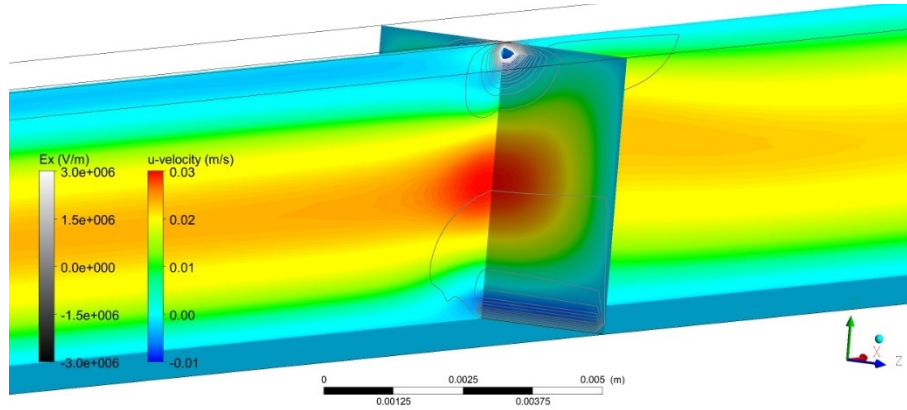


Figure 58 — Scenario 3.b: Recirculation problems. YZ plane at the outlet of the test section $x = 0.054$ m. $t = 26$ s

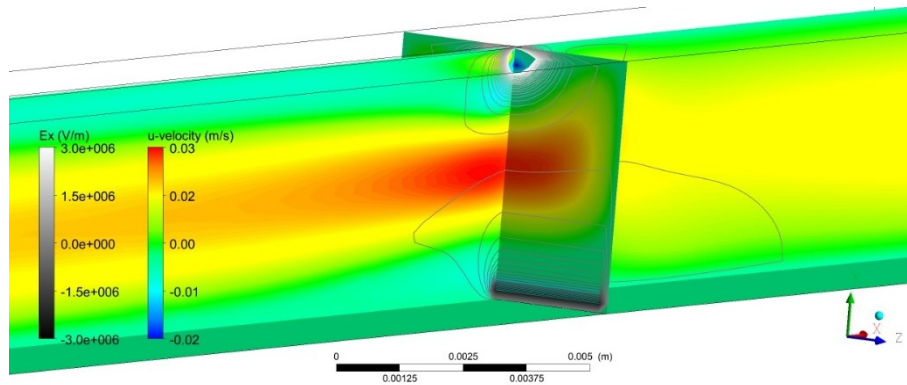


Figure 59 — Scenario 3.d: Recirculation problems. YZ plane at the outlet of the test section $x = 0.054$ m. $t = 26$ s

The u -velocity is “narrowed” at the exit of the test section due to the electrical Coulomb force present in this zone. The negative value of the charge and the positive electric field at the upper part of the channel, near the electrode, create a negative velocity. At the bottom, the u -velocity is also negative owing to the negative value of the electric field in x -direction and the positive value of the charge near the plate. Thus, a negative u -velocity is as well created and is constant along the z -axis. When the fluid moves away from the middle of the channel to the walls, in $\pm z$ -coordinate, the u -velocity due to the laminar profile reduces. So, this constant negative u -velocity that introduces the hot fluid is more detrimental with increasing and decreasing z -coordinates.

Regarding the v -velocity, the electrical force added to the momentum equation modifies the velocity profile. Near the wire, a negative v -velocity push the fluid towards the bottom of the minichannel. In addition, Figure 60 shows the buoyancy forces effects at the last part of the heated plate, where the

hot fluid rises due to its density variation (considered in the y -momentum equation with the Boussinesq approximation). The hot fluid tends to go up, extracting heat from the plate and flows towards the outlet section and a part of it is recirculated when it reaches the end of the test section.

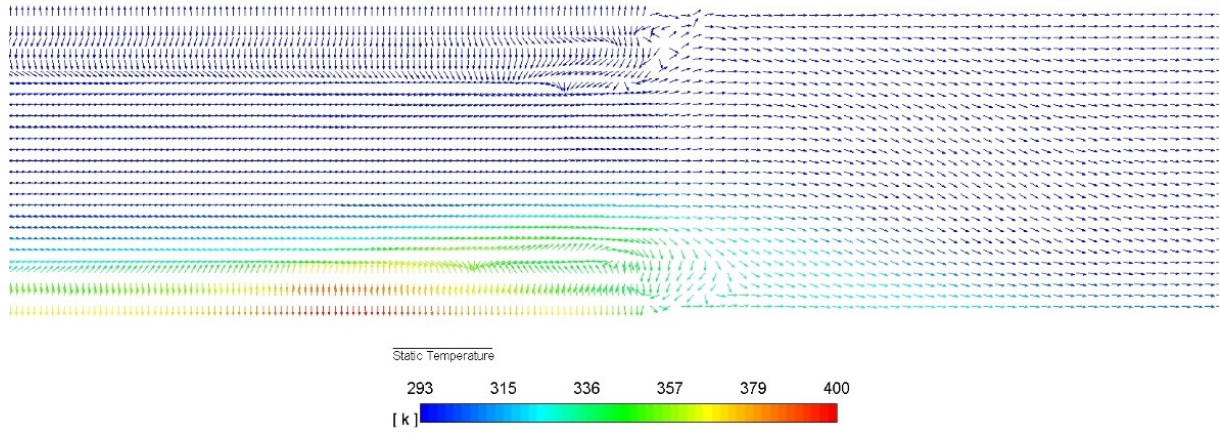


Figure 60 — Scenario 3.d: velocity modification at the middle plane of the minichannel. Velocity vectors coloured by temperature Plane XY, $x = 0.070$ m. $t = 26$ s

The u -velocity profile modified by the Coulomb force for scenario 3.b, as an example, is as follow:

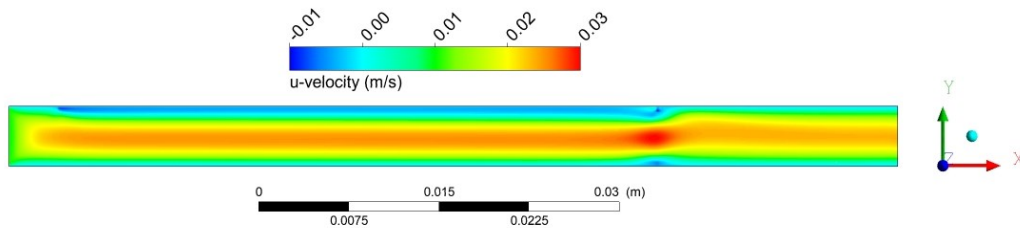


Figure 61 — Scenario 3.b: u -velocity modification at the middle plane of the minichannel. Plane XY, $t = 26$ s

Along the channel, the mean velocity of the fluid increases so the heat transfer rate is better than in case 3.a without the appliance of the voltage. The temperature profile of the heated plate for case 3.a (Figure 48) shows that the temperature of the heated plate at the corners increases gradually until its maximum value at the exit of the test section. With the appliance of the voltage, it can be observed a decrease in the temperature of the corners and a smoother temperature profile, avoiding the gap between the centre and the corners that we have noticed in case 3.a. A contour of scenario 3.b is plotted as an example of this fact.

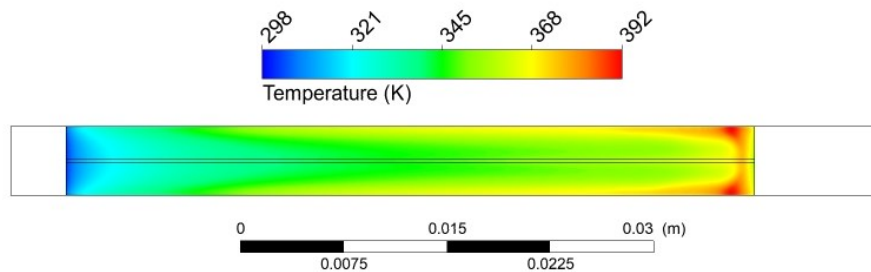


Figure 62 — Scenario 3.b: Temperature contour of the heated plate. Top view, plane XZ, $t = 26$ s.

RESULTS AND DISCUSSION

After the phenomenon of recirculation exposed and the analysis of the velocity profile, an in-depth analysis of the temperature of the heated plate is carried out. In order to determine if this recirculation affects to the temperature of all the plate, a segmented analysis of the heated plate is done. The heated plate goes from $x = 0.004$ m to $x = 0.054$ m and is divided in 4 different parts:

- HP-1: from $x = 0.004$ m to $x = 0.0165$ m
- HP-2: from $x = 0.0165$ m to $x = 0.029$ m
- HP-3: from $x = 0.029$ m to $x = 0.0415$ m
- HP-4: from $x = 0.0415$ m to $x = 0.054$ m

For each part of the heated plate, the maximum and the average temperature value is computed.

Case	Ave. Temp. (K)				Max. Temp. (K)			
	HP-1	HP-2	HP-3	HP-4	HP-1	HP-2	HP-3	HP-4
3.a	323.48	343.34	353.46	359.24	347.88	364.36	377.09	392.11
3.b	322.84	341.26	348.80	358.98	354.09	358.98	363.9	391.76
3.c	323.90	340.57	345.85	365.25	341.66	361.55	363.37	396.5
3.d	325.67	344.33	359.81	376.84	340.38	358.06	381.21	406.67

Table 15 — Scenarios 3.a to 3.c: Temperatures of the heated plate

As it can be observed in Table 15, the augmentation of the temperature is localised at the end of the plate. The maximum temperatures obtained at HP-1, HP-2 and HP-3 (from $x = 0.004$ m to $x = 0.0415$ m) are lower for scenarios 3.b and 3.c (unless for HP-1 in scenario 3.b). In scenario 3.d, a reduction of the maximum temperature measured is as well noticed but in HP-3 is higher. The maximum temperature for the last part of the plate (HP-4) is higher for cases 3.c and 3.d and very similar for case 3.b. This fact is a result of the recirculation at the end of the test section. Hot fluid re-enters to this zone and locally increases the temperature value of the plate. Besides, due to the lower Reynolds number of these cases, the Coulomb force contribution in the momentum equation is significant and the recirculation increase with the voltage applied.

Another cause of the temperature increase of the plate is the Joule heating. As it is explained in chapter 6.2.1, part of the charge injected from the wire is absorbed by the heated plate due to the conductor material that it is made of. Charges can move easily through the surface of the heated plate, so electric current flows through its surface. This current creates a source of Joule heating that contributes to increase the temperature at the final part of the plate. As the heated plate is not considered as an adiabatic wall (it introduces the heat flux into the domain), Joule heating is generated. The charge is accumulated in the surface of the plate and tends to be higher in the last part, where the temperatures are higher and where the Joule heating is higher too. Nevertheless, this contribution in the increase of the temperature is lower than the contribution of the recirculation phenomenon. Figure 63 shows the Joule heating at the plate.

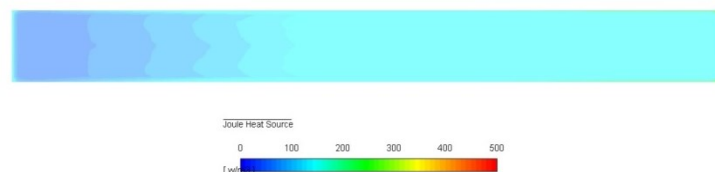


Figure 63 — Scenario 3.b: Joule Heat source at the heated plate. Plane XY, $t = 26$ s

The joule heating is introduced due to the potential applied. Regarding the equation (9), the heat generated is equal to the electric current that passes through an electrical conductor and the voltage applied. The biggest source of joule heating is the wire, but it is considered as an adiabatic wall in the model. So, the fluid does not increase its temperature in the upper part of the channel due to the voltage applied at the wire.

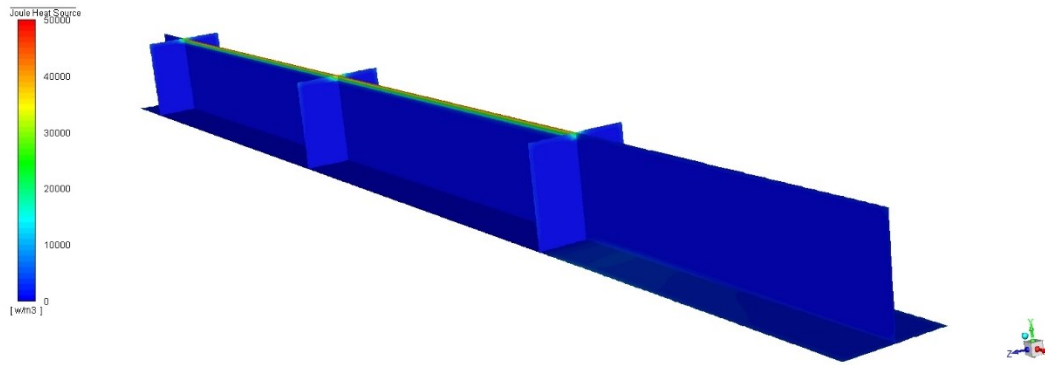


Figure 64 — Scenario 3.b: Joule heat source at the middle plane. XY planes at $x = 0.004$ m, $x = 0.029$ m, $x = 0.054$ m, $t = 26$ s

6.3.4 3D Parametric study: scenarios 3.e to 3.h

Scenario 3.e

These next four scenarios consider another velocity inlet: 0.05 m/s ($Re = 7.13$). The device reaches the steady state at 11 s. The same comments done for the velocity profile for scenario 3.a can be applied in this case.

The dimensionless numbers presented in chapter 2.1 are computed for this scenario.

Prandtl Number	Grashof Number	Rayleigh Number
396.43	22.22	8.88×10^3

Table 16 — Scenario 3.e: Dimensionless numbers

However, some aspects should be mentioned about the v -velocity. As the mean velocity of the fluid is higher in these scenarios, the v -velocity in all the YZ planes considered along the minichannel is lower than the observed in scenario 3.a. This fact can be explained by analysing the dimensionless numbers computed. Grashof number in scenario 3.a is 35.28 and in this case its value is 22.22 (7). As it is explained in chapter 2.1, the Grashof number computes the ratio between the buoyancy forces and viscous forces acting on a fluid. Even if both Grashof numbers are not too high, the value in scenario 3.a is higher than in this case, so the buoyancy forces acting in the fluid in this scenario are lower. Regarding the Rayleigh number, a lower value is as well obtained, so the convection heat transfer is lower than in case 3.a, but we the convection occurs since this value is higher than the critical value mentioned in chapter 2.1 (1700). Nevertheless, the same behaviour is observed when analysing v -velocity contours. The v -velocity is increasing near the walls of the minichannel and a decreasing value is observed in the centre of the duct. Therefore, the same detailed explanation of the buoyancy forces at the different YZ planes done for scenario 3.a can be applied to the scenario 3.e. Figure 65 shows that the maximum v -velocity is lower than in scenario 3.a, as it is expected. In

RESULTS AND DISCUSSION

addition, as the temperature gap between the plate and the fluid is lower, the contribution of the density variation to the y-momentum equation is lower.

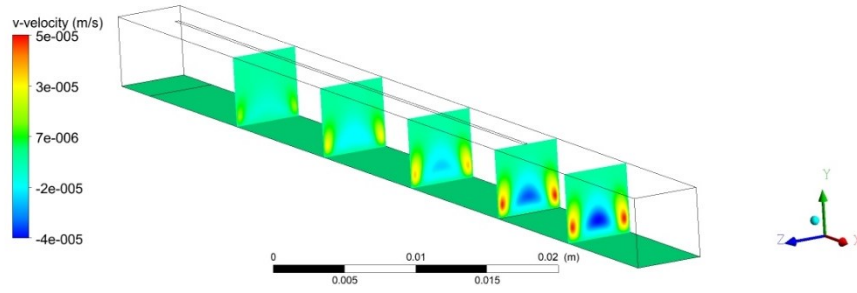


Figure 65 — Scenario 3.e: v -velocity profile along the minichannel. XY planes at $x = 0.0165$ m, $x = 0.029$ m, $x = 0.0415$ m, $x = 0.054$ m and $x = 0.064$ m, $t = 11$ s.

Next figures show the temperature of the heated plate after the steady state is reached. As it is explained for scenario 3.a, the maximum temperature is reached at the corners but its value is lower.

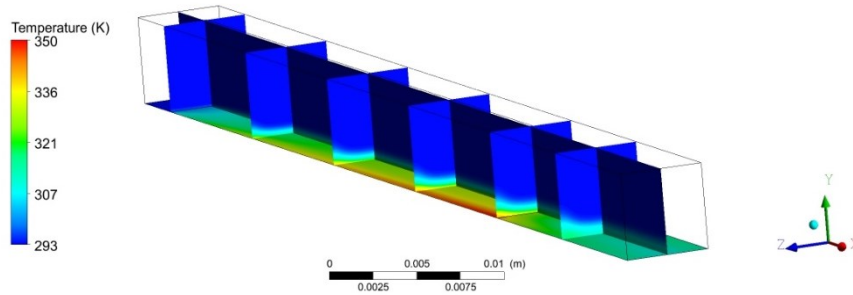


Figure 66 — Scenario 3.e: Temperature contour along the minichannel. YZ planes at $x = 0.004$ m, $x = 0.0165$ m, $x = 0.029$ m, $x = 0.0415$ m, $x = 0.054$ m and $x = 0.064$ m, $t = 13$ s.

The same pressure profile is observed than in scenario 3.a but with a bit higher values in order to drive the fluid at this velocity.

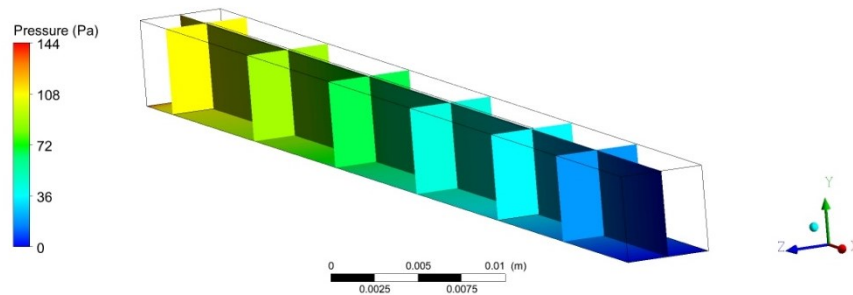


Figure 67 — Scenario 3.e: Pressure contour along the minichannel. YZ planes at $x = 0.004$ m, $x = 0.0165$ m, $x = 0.029$ m, $x = 0.0415$ m, $x = 0.054$ m and $x = 0.064$ m

Scenario 3.f to 3.h

These three scenarios are simulated for 19.5 s, when the temperatures measured stabilize. Next graphs (Figure 68) show the maximum temperature value and average temperature of the plate.

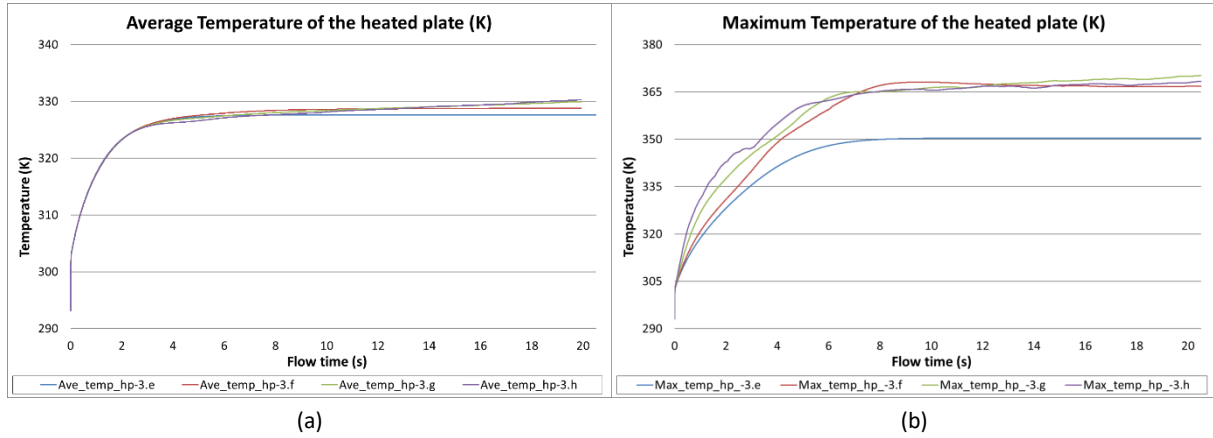


Figure 68 — Scenarios 3.f to 3.h: Average temperature of the heated plate and maximum temperature of the heated plate

The maximum temperature and the average temperature of the heated plate are higher than in scenario 3.e. This fact is due to three different factors: the modification of the velocity profile done with the appliance of the voltage at the wire is not enough, the same recirculation problems at the outlet of the test section mentioned above and the Joule heating generated at the plate.

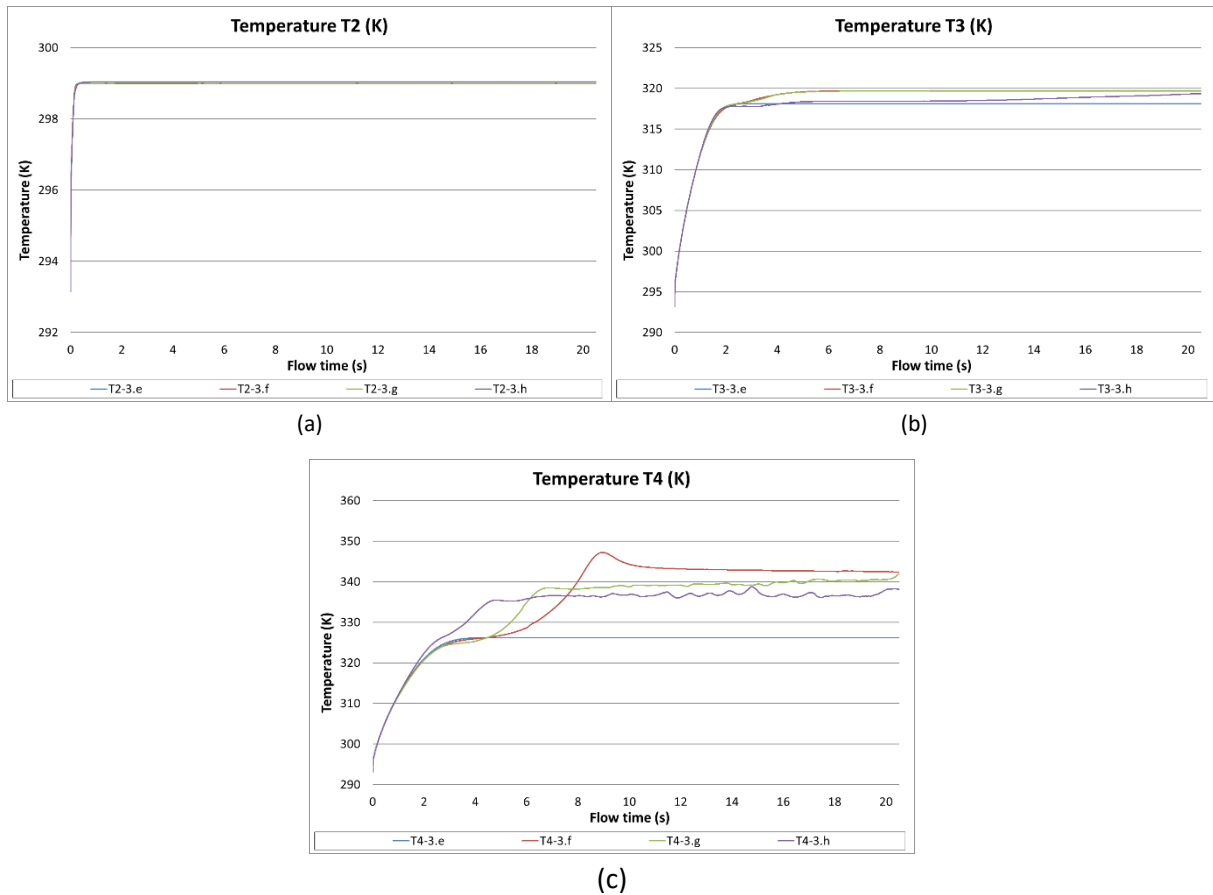


Figure 69 — Scenarios 3.f to 3.h: (a) Temperature measured at point T2 (4, -2.5, 0) mm. (b) Temperature measured at point T3 (29, -2.5, 0) mm. (c) Temperature measured at point T4 (54, -2.5, 0) mm.

RESULTS AND DISCUSSION

First, the problem of the modification of the velocity profile is treated. The same comments for the generation of the electric field made for scenarios 3.b to 3.d can be applied. The mechanism of charge injection, the evolution of the voltage and the electric field development are analogue. Nevertheless, the velocity inlet used (0.05 m/s) is dominant over the modifications that introduce the electrohydrodynamic system. The electrical Coulomb force contribution to the momentum equation is not enough. In addition, the same recirculation problems are observed at $x = 0.054$ m (end of the test section), so hot fluid re-enters to the test section increasing the temperature. Next contour shows, as an example of these cases, the velocity modification and the recirculation of the fluid for scenario 3.h.

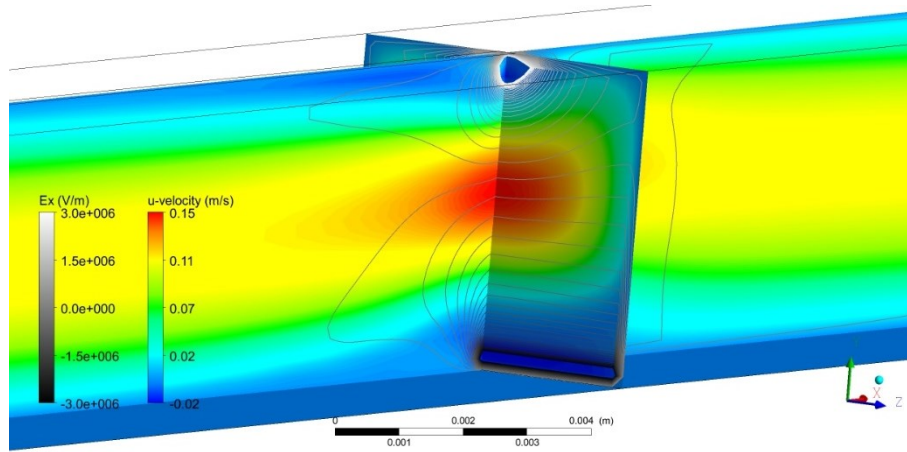


Figure 70 — Scenario 3.h: u -velocity modification at the outlet of the test section. $x = 0.054$ m, $t = 19.5$ s

By the analysis of the temperatures measured, the values obtained for T2 and T3 are similar. This is a result of the no modification of the flow velocity in this region of the test section. However, remarkable differences can be observed for T4. The biggest temperature increase at this point of the heated plate is achieved in scenario 3.f.

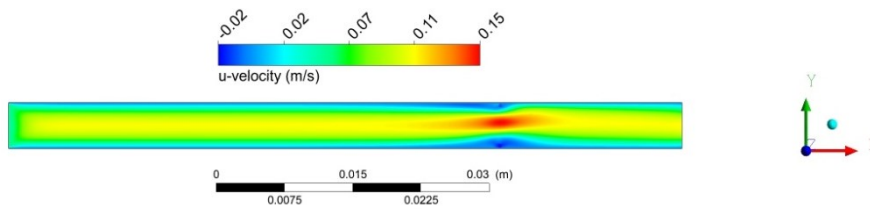


Figure 71 — Scenario 3.h: u -velocity modification at the middle plane of the minichannel. Plane XY, $t = 19.5$ s

The recirculation is highly undesirable. It is a result of the positive electric field at the right edge of the heated plate ($x = 0.054$ m). The same explanation done for scenarios 3.b to 3.d is applied to these cases: the negative electric field interacts with the positive charge injected in this zone of the fluid and produces a negative Coulomb force added to the momentum equation, creating these negatives values. As the velocity profile is laminar, this effect is more noticeable near the walls of the minichannel.

Regarding the v -velocity, Figure 72 shows the contribution of the electrical force in the momentum equation near the wire, modifying the fluid velocity with the purpose of pushing the fluid towards the bottom of the minichannel. In addition, this figure shows the buoyancy effects at the last part of the heated plate: the hot fluid rises up due to its density variations (considered in the y -momentum equation).

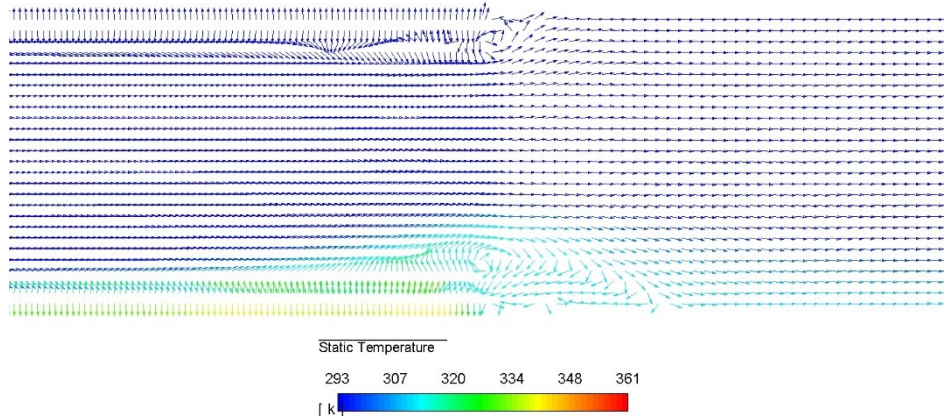


Figure 72 — Scenario 3.h: v -velocity modification at the middle plane of the minichannel. Vectors coloured by temperature Plane XY, $t = 19.5s$

In order to determine if the increase of the temperature is localised, the same procedure is performed as in scenarios 3.b to 3.d to calculate the average temperature and the maximum temperature of the four parts defined of the heated plate. Just to remind, the first part of the heated plate (HP-1) goes from $x = 0.004$ to $x = 0.0165$, the second part (HP-2) from $x = 0.0165$ to $x = 0.029$, third part (HP-3) from $x = 0.029$ m to $x = 0.0415$ m and the fourth part from $x = 0.0415$ m to $x = 0.054$ m. Next table shows the values obtained:

Case	Ave. Temp. (K)				Max. Temp. (K)			
	HP-1	HP-2	HP-3	HP-4	HP-1	HP-2	HP-3	HP-4
3.e	319.20	307.62	325.65	330.51	332.79	324.14	338.88	343.7
3.f	319.20	307.61	325.68	335.58	339.65	321.59	336.41	359.99
3.g	319.23	307.62	326.02	339.48	330.16	320.33	342.43	362.94
3.h	319.22	307.68	326.52	339.99	333.09	316.68	341.61	361.20

Table 17 — Scenarios 3.f to 3.h: Temperatures of the heated plate

The values of the average temperature of the first part, second and third part of the heated plate are very similar. As it can be observed in the previous contours, the velocity profile is not highly modified in these parts of the test section, so the same temperatures are expected. However, due to the recirculation, the average temperature at the last part of the plate increases with the potential applied. As it is explained before, a higher value of the potential applied means a higher value of the electric field, so the modulus of Coulomb force added to the conservation of momentum equation is higher when the voltage increases, becoming more significant the recirculation.

Regarding the maximum temperatures measured, it can be observed a decrease in the second part of the heated plate. Even if the modification of the velocity profile do not reach to lower the average

RESULTS AND DISCUSSION

temperature of the plate, its contribution near the corners help to decrease the temperature at these points where the velocity due to the laminar flow is almost null. Next contour show, as an example of this fact, the v -velocity at the end of the second part in the cross sectional area (just in the middle of the heated plate, plane YZ, $x = 0.029$ m). It is higher when the voltage applied is greater.

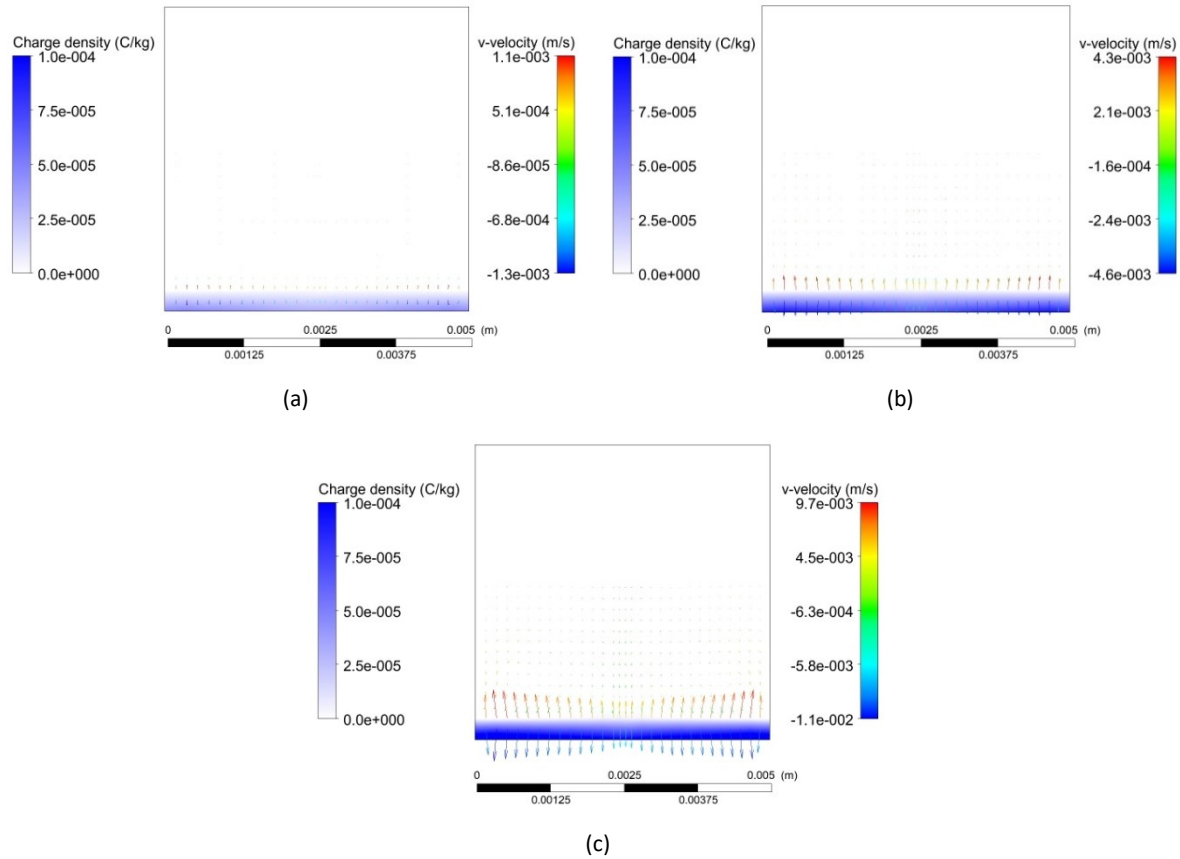


Figure 73 — Scenario 3.f to 3.h: v -velocity and charge density at the middle plane of the minichannel: (a) Scenario 3.f (5 kV). (b) Scenario 3.g (10 kV). (c) Scenario 3.h (15 kV). Plane YZ, $t = 19.5$ s

In the third part of the heated plate, the increase of the temperature starts to be noticeable and in the final part, from $x = 0.0415$ m to 0.054 m the augmentation is more pronounced.

The augmentation of the temperature is produced not only for the recirculation phenomenon at the end of the heated plate but also due to the Joule heating. When electric current flows through a conductor produces Joule heating. As it is mentioned before, the material used for the walls is a perfect electrical insulator, with a value of electrical conductivity very low. The mineral oil has as well a low value of electrical conductivity that is one of the properties preferred to use it for cooling applications. But, material conductors are characterised for having a high value of this property. The wire and the heated plate are made of copper. As it is mentioned before, the biggest source of Joule heating is the wire, due to its value of charge density and so its electrical current. However, the thermal boundary condition applied for this part of the device is as an adiabatic wall, so it does not introduce heat to the fluid.

On the other hand, the heated plate introduces a heat flux into the domain, so it is not treated as an adiabatic wall. As it is made of conductor material, it absorbs charge that remains on its surface and could move through it. This charge produces a Joule heat source at the different points of the heated plate since it moves through it. As it can be noticed in the next contour, the maximum values of the temperatures match with the maximum values of the charge absorbed by the heated plate. Thus, it can be concluded that part of the temperature increase of the heated plate is due to the Joule heating generated as a result of the flow of charge on its surface. Once again, a contour of scenario 3.h is provided to show this phenomenon.

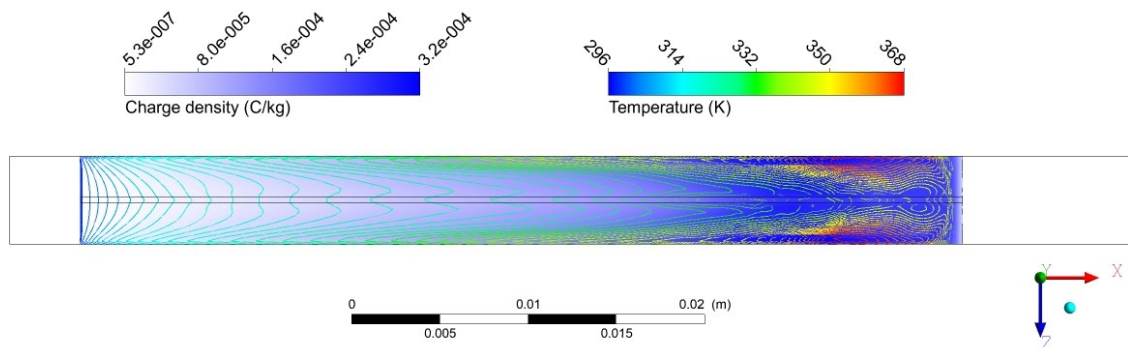


Figure 74 — Scenario 3.h: Temperature and charge density at the heated plate. Plane XY, $t = 19.5s$

Analysis of parameters

The parameters presented in chapter 5.2 are computed for all the scenarios in 3D. As we can see in Figure 75 shows that scenarios 3.b and 3.c have an enhancement in the heat transfer due to the positive ratio Nu/Nu_0 . The heat transfer coefficient computed for those scenarios are higher than the one of the scenario without the appliance of the voltage. The final average temperature of the heated plate measured is lower than the one obtained in scenario 3.a (0 kV). Nevertheless, in scenario 3.d, the heat transfer coefficient decreases due to the higher value of the average temperature of the plate obtained, and thus, the Nusselt number is lower as well.

On the other hand, ratios lower than unity for the scenarios 3.e to 3.h are observed. This is evidence since the heat transfer coefficient computed for these cases are lower than the one without the appliance of the voltage (3.e). These configurations not only do not reach to lower the average temperature of the plate, but also increase it due to the problems abovementioned.

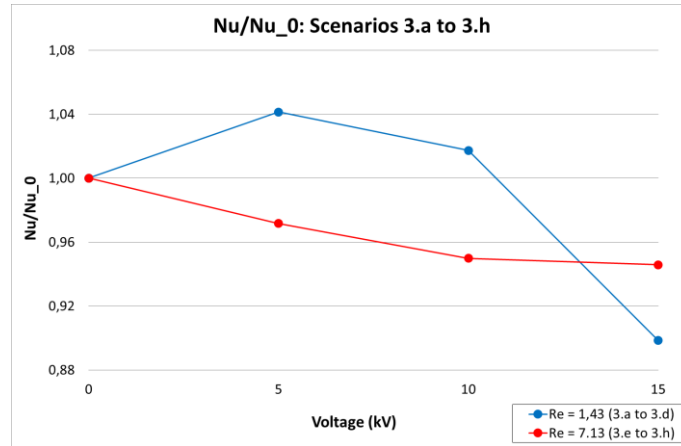


Figure 75 — Ratio of Nu/Nu_0 vs. applied voltage (kV) for scenarios 3.a to 3.h

Figure 76 shows the ratio of the pressure drop between different scenarios and the case without the appliance of the voltage. As it is expected, an increase in the pressure drop is observed. The pressure drop is related to the modification of the velocity profile. As this modification rises with the voltage applied, the pressure drop increases as well with it.

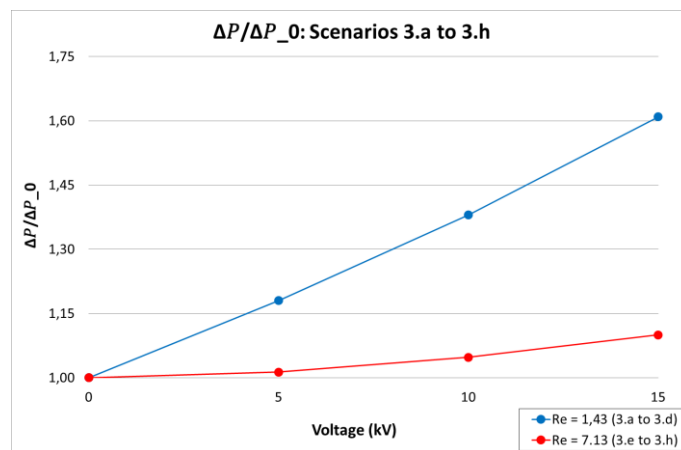


Figure 76 — Ratio of $\Delta P/\Delta P_0$ vs. applied voltage (kV) for scenarios 3.a to 3.h

The pressure drop is higher in scenarios with lower Reynolds number. The electric field in these cases achieve to have more influence in the flow and its contribution to the conservation of momentum equation is higher. In addition, as it is explained before, the modification in the velocity profile is greater when higher voltages are applied, so it is evident that the pressure drop will increase as well with the increase of the voltage. By increasing the Reynolds number, the influence that the electric field has in the conservation of momentum is lower. The effects of electric field are more noticeable at low Reynolds numbers.

As it is explained before, the Performance Evaluation Criterion is a parameter that relates the heat transfer enhancement to the pumping power needed. Even if the heat transfer coefficient shows a better heat transfer behaviour of the devices, it is important to consider the pumping power increase to assess the efficiency of the cooling technique. So, values greater than unity mean that the pressure drop generated with the use of the cooling method is lower than the rate of heat transfer enhancement.

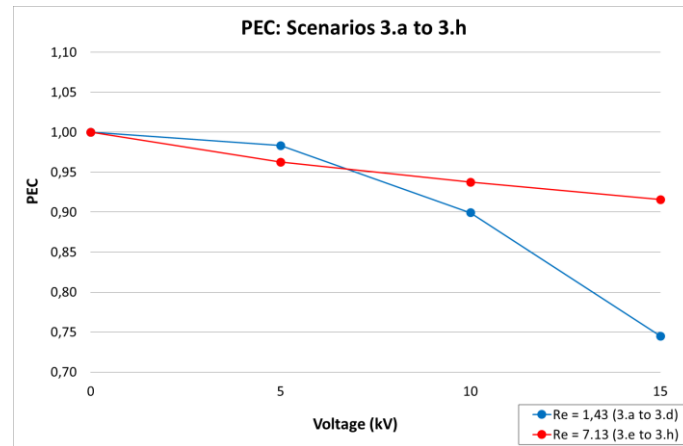


Figure 77 — PEC vs. applied voltage (kV) for scenarios 3.a to 3.h

Regarding the PEC values, none of the scenarios studied achieves that the heat transfer improvement is greater than the pressure drop generated. Even if the heat transfer rate is higher in scenarios 3.b and 3.c, the pressure drop generated is greater than the enhancement of heat transfer observed and they cannot be considered as efficient cooling techniques. However, regarding the segmented analysis of the plate for scenarios 3.b to 3.d, we can conclude that the maximum temperatures in the HP-1, HP-2 and HP-3 (from $x = 0.004$ m to $x = 0.0415$ m) of the heated plate are reduced and the average temperature of the plate is also reduced for these parts is scenario 3.b and 3.c.

By the analysis of scenarios of the higher Reynolds number, we can conclude that none of the configuration achieves to enhance the overall heat transfer coefficient, so the PEC values obtained are lower than unity. Nevertheless, regarding the segmented analysis performed, some reduction in the maximum temperatures reached is observed.

6.4 Weak points

Thanks to the analysis of the results obtained, some weak points have been noticed.

First, special mention can be done regarding the mesh of the model 3D. The 3D model mesh has a multizone configuration: the test zone has smaller elements than the inlet and the outlet section. This refinement of the mesh in the zone where the biggest modification of the fluids occurs is a good practice and a good strategy to simulate this CFD study. However, concerning the cross-sectional area (Figure 25), the transition between these smaller elements and the adjacent bigger ones is not progressively. The expansion ratio is too high and to have better results and improve the simulation setup, a refinement of this part should be done.

In addition, the maximum element size is also reduced at the wire but the same problem is observed at the top of the channel, the expansion ratio between the faces of the wire and the faces of the wall of the top of the channel is too high and not progressive (Figure 26).

In order to improve the mesh near the heated plate, the same comments can be done and a refinement of the adjacent layers should provide accurate results in the zone where the heat flux enters to the domain.

RESULTS AND DISCUSSION

The use of ANSYS Fluent Academic version is limited and do not allow creating meshes with more than 512k cells. For further work related to these simulations, a professional ANSYS Fluent version should be use to avoid the limitations of the mesh generation.

7. CONCLUSIONS

This Master Thesis is focused in the study of the heat transfer enhancement using electrohydrodynamically induced convection to cool down an electronic component.

First of all, some conclusions can be done by the qualitative analysis of use a minichannel with a reduced hydraulic diameter. On one hand, the heat transfer coefficient is a function of Nusselt number, fluid thermal conductivity and hydraulic diameter. Considering fully developed laminar flow through a pipe, the Nusselt number is considered as constant value (3.61 for square ducts [80]). Thus, for a given fluid, the heat transfer coefficient is inversely proportional to the hydraulic diameter of the minichannel. If the hydraulic diameter is reduced, the heat transfer coefficient increases. On the other hand, the pressure drop varies inversely with the Reynolds number following the friction factor expression (35). So, pressure drop increases when the hydraulic diameter decreases. Therefore, it is essential to find the balance between improving the heat transfer coefficient and not excessively increasing the pressure drop necessary to pump the fluid.

Regarding now the simulations performed in 2D, the first conclusion is that the use of a mineral oil as a refrigerant guarantees the cooling of a plate. By the appliance of a voltage at the wire of the top section, a secondary flow appears from the wire to the heated plate. This secondary flow pushes the neutral molecules of the fluid towards the plate, increasing the heat transfer coefficient. Electrostatics and hydrodynamics are coupled in the conservation of momentum equations and the Coulomb force, produced by the electric field and the charge injected, modifies the velocity profile of the flow. This modification of the velocity profile is noteworthy 2D and increases with the strength of the electric field: when the voltage applied is higher, the electric field computed is also higher and therefore, the modification of the velocity profile is more noticeable. A recirculation phenomenon is observed at the end of the test section due to the interaction of the electric field and the charge present at the fluid. This interaction is also responsible for the different modifications observed in the fluid along the test section.

Thanks to the analysis of the ratio between the Nusselt number of different scenarios, it can be concluded than the heat transfer rate increases with the voltage applied, and thus with the strength of the electric field. This fact is confirmed by analysing scenarios 1.a to 1.c. Nevertheless, a recirculation issue is observed due to the negative velocity that creates the secondary flow. This recirculation increases with the strength of the electric field (its contribution to the momentum equation is higher), so it is more significant in scenario 1.d than in the other ones. Contrary as it is expected (higher voltages mean higher secondary flow and higher heat transfer coefficient), this fact explains that the ratio Nu/Nu_0 for scenario 1.d (15 kV) is lower than the one for scenario 1.b (10 kV).

Regarding now the pressure drop, since the voltage applied rises, the secondary flow is greater. Thus, the modification of the velocity profile is higher as well, increasing the pressure drop. Figure 41 shows this augmentation of the pressure drop ratio. By the analysis of Performance Evaluation Criterion for these scenarios, it can be concluded that the efficiency of scenarios 1.b and 1.c (5 kV and 10 kV, respectively) is greater than unity and thus the heat transfer enhancement is higher than the pressure drop produced. Besides, as it is expected, the PEC is greater for the scenario 1.c with a

CONCLUSIONS

higher voltage applied (higher electric field computed). So, these two configurations are desirable to cooling down the heated plate. Nonetheless, PEC value for scenario 1.d is lower than unity and despite the greater heat transfer rate, the pressure drop needed is very large so it is not an efficient cooling method.

In the same way, some 3D cases are simulated. Before analysing the same device in three dimensions, a mesh study is done in order to determine the influence of the discretization of the domain in the accuracy of the results obtained. Three different hexahedral meshes are studied with 10x10, 20x20 and 29x29 (limit of cells reached) cells at the cross-sectional area. By analysing the results obtained (Figure 44 and Table 13), it is confirmed that the most accurate results are those of the finest mesh generated. The mesh (2.1) with the poorest quality (10x10 elements at the cross sectional area) is discarded due to the disparity of the results compared to the two other cases. The values of the frictional factor computed for scenarios 2.2 and 2.3 are similar and when comparing them with the classical correlation for laminar flows through square ducts, it can be concluded that the finest mesh (case 2.3) guarantees the minimum error.

Furthermore, eight different simulations in 3D are computed for two different Reynolds numbers to analyse the effect of the cooling technique studied. First, the buoyancy forces are more noteworthy near the walls due to the laminar velocity profile of the flow: the u -velocity component is dominant over the two other components and follows a parabolic profile. Since the fluid moves away from the middle plane of the channel towards the walls, the u -velocity is reduced and the buoyancy forces are more noticeable. Concerning now the scenarios with the lower Reynolds number (3.a to 3.d), the average temperature of the heated plate is reduced in scenarios 3.b and 3.c and only the maximum temperature is reduced with the 3.b configuration. Similar values of temperature are measured at the three different points located along the first part of the heated plate, and a temperature increase is observed at the final part of the heated plate.

This fact is explained because of the recirculation issue observed at the outlet of the test section. The charge is injected from the wire, creating a secondary flow that pushes the flow towards the heated plate. At the end of the wire ($x = 0.054$ m, $y = 0.0025$ m, $z = 0$ m) it can be observed a positive value of electric field in x direction at the top of the channel. As the charge is injected through the fluid, the heated plate becomes positively charged due to the material that it is made of, in our case copper. A negative electric field in x -direction is observed at the bottom part of the channel at the end of the test section, just in the edge of the heated plate ($x = 0.054$ m, $y = -0.0025$ m, $z = 0$ m). The interaction of this negative value of electric field and the positive charge present at the plate produces a negative u -velocity that reintroduces hot fluid towards the test section. This recirculation is more noticeable since the fluid moves away from the middle plane in $\pm z$ coordinates, due to the laminar velocity profile. This phenomenon explains that the maximum temperatures observed at the heated plate are at its corners near the end of the test section. The fluid recirculated in this zone is greater than the main velocity of the fluid, so this hot fluid increases the temperature of the plate. The segmented analysis of the heated (Table 15) confirms that this increase of temperature is localised at the end of the heated plate. This recirculation increases when the voltage applied is greater and thus when the strength of the electric field is higher.

The same comments about the velocity profile modification and the secondary flow can be applied for the scenarios 3.e to 3.h. Nevertheless, as the Reynolds number is higher, the momentum applied by the injection of charge has less relevance than in scenarios 3.b to 3.d.

Regarding now the analysis of the ratio Nu/Nu_0 , none of the scenario 3.e to 3.h reaches to improve the heat transfer rate. Only scenarios 3.b and 3.c have values greater than unity of this ratio and suppose an enhancement of heat transfer rate. The comparison of the pressure drop ratio shows that it increases when the voltage applied is higher. In fact, as the electric field is computed as the gradient of the voltage, a higher value of electric field means a higher modification of the velocity profile and therefore a higher pressure drop. Another fact visible is that the effects of the electric field are more noticeable at low Reynolds numbers. Both ratios have more significant changes in scenarios with the lower Reynolds number, and the slope of the ratios lines is more accentuated in the cases with low Reynold number (3.b to 3.d).

In addition, the analysis of the PEC values obtained shows that none of the 3D configurations is efficient. Even if scenarios 3.b and 3.c reach to lower the average temperature of the heated plate, the pressure drop is greater than the heat transfer improvement and they cannot be considered as more efficient cooling techniques. Nevertheless, regarding the segmented analysis performed, some reductions at the maximum temperature are achieved for the plate for both Reynolds number studied, unless for the final part, where the recirculation occurs and thus, its effects are greater.

Summarizing, the effect of electric field is noteworthy when the Reynolds number decrease due to the values obtained for Nusselt number and pressure drop ratios. In addition, the recirculation phenomenon observed confirms that the effects of the electric field increase when the voltage applied is higher, and thus with the strength of the electric field. In fact, ratios of Nu/Nu_0 greater than unity are observed for scenarios with the lower Reynolds number with 5kV and 10 kV applied. In the zones not influenced by the recirculation phenomenon, the heat transfer rate enhances with the voltage applied owing to the lower temperatures observed. Besides, the pressure drop increases with the voltage applied. Due to the recirculation issues, none of the 3D configurations reaches to have a value of efficiency greater than unity. On the other hand, the 2D configuration with 5 and 10 kV applied can be considered as more efficient cooling techniques than the zero voltage 2D configuration.

To conclude, the reliability and safety of the electronic components must be maximized and, thus, the problem of overheating has to be minimized since it is the main cause of the reduction of the lifespan of these components. So, the research and the development of these new cooling techniques should continue in order to meet all these cooling requirements.

CONCLUSIONS

8. BIBLIOGRAPHY

- [1] Liao, B. et al. (2016). *Photo-excited charge carriers suppress sub-terahertz phonon mode in silicon at room temperature*, Nature Communications.
- [2] Korotkov, A. N. et al. (1994). *Effects of overheating in a single-electron transistor*. Journal of Applied Physics.
- [3] Yang, W. B. et al. (2014). *Numerical study of overheat fault in copper wire caused by bad contact base on multi-physics coupling*. Paper presented at the 7th International Conference on Intelligent Computation Technology and Automation, Changsha, China.
- [4] Almubarak, A. A., (2007). *The Effects of Heat on Electronic Components*. Int. Journal of Engineering Research and Application ISSN : 2248-9622, Vol. 7, Issue 5, (Part -5), pp.52-57.
- [5] The Editors of Encyclopaedia Britannica, Erik Gregersen. (n.d.). *Heat*. Retrieved March 10, 2019 from <https://www.britannica.com/science/heat>
- [6] (1999). *Heat transfer*. Retrieved March 10, 2019 from <http://physics.bu.edu/~duffy/py105/Heattransfer.html>
- [7] Davies, T. W. (2011). *Fourier's Law*. Retrieved March 10, 2019 from <http://www.thermopedia.com/fr/content/781/>
- [8] Khan Academy. (n.d.) Retrieved March 10, 2019 from <https://www.khanacademy.org/science/physics/thermodynamics/specific-heat-and-heat-transfer/a/what-is-thermal-conductivity>
- [9] *Natural Convection. Forced Convection*. Retrieved March 10, 2019 from <https://www.cradle-cfd.com/glossary/>
- [10] *Newton's Law of Cooling*. Retrieved March 10, 2019 from <https://www.nuclear-power.net/nuclear-engineering/heat-transfer/convection-convective-heat-transfer/newtons-law-of-cooling/>
- [11] *Effect of heat on electronic devices*. Retrieved 15 May, 2019 from http://www.apiste-global.com/enc/technology_enc/detail/id=1262
- [12] (2003). *Dimensionless numbers*. Retrieved March 12, 2019 from https://ocw.mit.edu/courses/materials-science-and-engineering/3-185-transport-phenomena-in-materials-engineering-fall-2003/study-materials/handout_numbers.pdf
- [13] *What is Grashof Number*. Retrieved March 12, 2019 from <https://www.nuclear-power.net/nuclear-engineering/heat-transfer/introduction-to-heat-transfer/characteristic-numbers/what-is-grashof-number/>
- [14] Hagedoorn H. (2008). *Thermaltake PW850i Internal DIY Liquid cooling*. Retrieved March 10, 2019 from <https://www.guru3d.com/news-story/thermaltake-pw850i-internal-diy-liquid-cooling.html>
- [15] Markets and Markets (2016). *Thermal Management Market by Material Type (Adhesive, Non-Adhesive), Devices (Conduction, Convection, Advanced, Hybrid), Service (Installation & Calibration, Optimization & Post Sales), End-Use Application - Global Forecast to 2022*. Retrieved March 10, 2019 from <https://www.marketsandmarkets.com/Market-Reports/thermal-management-market-155049228.html>

-
- [16] Chen, D. (n.d.) *Need for Thermal Management in Electronic Systems*. Retrieved March 10, 2019 from <https://www.connector.com/need-thermal-management-electronic-systems/>
 - [17] Cengel, Y. Ghajar, A. (2015). Cooling of electronic equipment. *Heat and Mass Transfer: Fundamentals and Applications*. McGraw Hill. pp. 15.1 -15.70.
 - [18] C-Therm Technologies. *Tech Library*. Retrieved March 10, 2019 from https://ctherm.com/products/tci_thermal_conductivity/helpful_links_tools/thermal_resistance_thermal_conductance/
 - [19] APS News (2009). *December 1840: Joule's abstract on converting mechanical power into heat*. Retrieved March 25, 2019 from <https://www.aps.org/publications/apsnews/200912/physicshistory.cfm>
 - [20] TME. Retrieved March 25, 2019 from https://www.tme.eu/fr/katalog/memoires-eprom-circuits-integres_100279/
 - [21] Techopedia. Retrieved March 25, 2019 from <https://www.techopedia.com/definition/2267/printed-circuit-board-pcb>
 - [22] Thomas. Retrieved March 25, 2019 from <https://www.thomasnet.com/articles/automation-electronics/processing-printed-circuit-boards>
 - [23] Moghanlou, F. S. et al. (2014). *Experimental study on electrohydrodynamically induced heat transfer enhancement in a minichannel*. *Experimental Thermal and Fluid Science* 59, 24–31.
 - [24] Antonio Ramos, (2014). *Microfluidic Technologies for Miniaturized Analysis Systems 2* (1, 2), 60-85
 - [25] Partha Kumar Das and A. B. M. Toufique Hasan. (2017). *Mechanical Micropumps And Their Applications: A Review*. 7th BSME International Conference on Thermal Engineering. AIP Conference Proceedings 1851
 - [26] Williams, M. (2014). *What is heat conduction?* Retrieved 20 March, 2019 from <https://phys.org/news/2014-12-what-is-heat-conduction.html>
 - [27] Kiumars A. et al. (2016). *A review on Electrohydrodynamic (EHD) pumps*. 24th Annual International Conference on Mechanical Engineering-ISME2016, Yazd University, Yazd, Iran
 - [28] Seyed-Yagoobi, J. (2005). *Electrohydrodynamic pumping of dielectric liquids* *J. Electrostatics* 63(6–10), pp. 861-869.
 - [29] Melcher, J. R. and M. S. Firebaugh, (1967). *Traveling-Wave Bulk Electroconvection Induced across a Temperature Gradient*, *Physics of Fluids* 10(6): 1178-1185.
 - [30] Siddiqui, M., Seyed-Yagoobi, J. (2009). Experimental study of liquid film pumping based on conduction phenomenon. *IEEE Trans. Ind. App* 45, 3.
 - [31] Yazdani, M., Seyed-Yagoobi, J. (2009). Electrically induced dielectric liquid film flow based on electric conduction phenomenon. *IEEE Trans. Dielectr. Electr. Insul.*, 16.
 - [32] Jimil M. Shah, (2017). *Recent Advancements in Oil Immersion Cooling*. Electronic cooling blog. Retrieved March 10, 2019 from <https://www.electronics-cooling.com/2017/04/recent-advancements-oil-immersion-cooling/>
 - [33] Jimil M. Shah et al. (2016). *Effects Of Mineral Oil Immersion Cooling On IT Equipment Reliability And Reliability Enhancements To Data Center Operations*. Conference paper.
 - [34] Liquid cool solutions. Retrieved March 28, 2019 from <http://www.liquidcoolsolutions.com/>
 - [35] Green revolution cooling. Retrieved March 28, 2019 from <https://www.grcooling.com/>

-
- [36] Nishikawaraa, M. et al. (2016). *Numerical investigation of the characteristics of an ion drag pump*. Journal of Electrostatics Vol 84, p 23-31
 - [37] Jimil M. Shah et al. (2016). *Design Considerations Relating to Non-Thermal Aspects of Oil Immersion Cooling*. Proceedings of the ASME 2016 International Mechanical Engineering Congress and Exposition
 - [38] Harvey, H. *Types of mineral Oil*: Retrieved 28 March, 2019 from <https://itstillruns.com/types-mineral-oil-7456026.html>
 - [39] ANSYS Manual: Theory Guide. Retrieved April 8, 2019 from https://www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/flu_ug/flu_ug_sec_hxfer_buoy.html#flu_ug_uns_sec_boussinesq
 - [40] Tritton, D. J. (1977). *Physical fluid dynamics*. New York: Van Nostrand Reinhold Co.
 - [41] Antonio Castellanos (1998). The Boussinesq approximation in Electrohydrodynamics. *Electrohydrodynamic*. pp 81.
 - [42] *Incompressible and Compressible Flow*. Chegg Study. Retrieved March 10, 2019 from <https://www.chegg.com/homework-help/definitions/incompressible-and-compressible-flow-5>
 - [43] European Comission: *Waste Electrical & Electronic Equipment (WEEE)*: Retrieved 15 May, 2019 from http://ec.europa.eu/environment/waste/weee/index_en.htm
 - [44] Thermal Engineering Blog. *How Rayleigh's number links conduction and convection*. Retrieved May 10, 2019 from <https://www.pirobloc.com/en/blog-en/how-the-rayleigh-number-links-conduction-convection/>
 - [45] Bejan, A. (2013): *Convection Heat Transfer*: John Wiley & Sons.
 - [46] Baldé, C.P. et al. (2017). *The Global E-waste Monitor – 2017*, United Nations University (UNU), International Telecommunication Union (ITU) & International Solid Waste Association (ISWA), Bonn/Geneva/Vienna. Retrieved 15 May, 2019 from <https://www.itu.int/en/ITU-D/Climate-Change/Documents/GEM%202017/Global-E-waste%20Monitor%202017%20.pdf>
 - [47] J.S. Paschkewitz, D.M. Pratt, (2000). The influence of fluid properties on electrohydrodynamic heat transfer enhancement in liquids under viscous and electrically dominated flow conditions, *Exp. Thermal Fluid Sci.* 21 (4) 187–197.
 - [48] *Choosing the Right Fluid for Electronics Cooling: Seven Dimensions of a Decision*. Retrieved March 10, 2019 from <https://dsiventures.com/library/>
 - [49] ASTM Standard Specification D3487 “Standard Specification for Electrical Insulating Oil of Mineral Origin”, American Society for Testing and Materials
 - [50] D. Sundin, Ph. D., Ambit (2013). *Thermodynamic Analysis of Dielectric Heat Transfer Fluids*, Technical Consulting.
 - [51] Aluyor, E. Ori-Jesu, M, (2009). *Biodegradation of mineral oils – A review*, African Journal of Biotechnology.
 - [52] Rajput, S., & Thakur, N. K. (2016). *Geological Controls for Gas Hydrate Formations and Unconventionals*.
 - [53] (2018). *IEA releases World Energy Outlook 2018*. Retrieved 10 May, 2019 from <https://dieselnet.com/news/2018/11iea.php>

- [54] D. Prucnal, (2013). *Doing More With Less: Cooling Computers with Oil Pays Off*, The Next Wave, vol. 20, no. 2, pp. 20 - 29.
- [55] Luke Werner, P. E. (2014). *Submersion Cooling Evaluation*, PG&E's Emerging Technologies Program
- [56] Engineers Edge (n.d.). *Naphthenic Oils Review*. Retrieved 20 March, 2019 from https://www.engineersedge.com/lubrication/naphthenic_oils.htm
- [57] (2007). *Debiotech's Insulin Nanopump™*. Retrieved 3 March, 2019 from https://www.medgadget.com/2007/04/debiotechs_insulin_nanopump.html
- [58] Melcher, J. R. (1981). *Continuum Electromechanics*, 1st edn. MIT Press, Cambridge.
- [59] J.M. López-Herrera et al. (2011). A charge-conservative approach for simulating electrohydrodynamic two-phase flows using volume-of-fluid. *Journal of Computational Physics* 230.
- [60] D.A. Saville, (1997). Electrohydrodynamics: the Taylor–Melcher leaky dielectric model, *Annu. Rev. Fluid Mech.* 29, 27–64.
- [61] ANSYS website. Retrieved 5 May, 2019 from <https://www.ansys.com/about-ansys>
- [62] Kuzmin, D. *Introduction to Computational Fluid Dynamics*. Institute of Applied Mathematics, University of Dortmund. Retrieved 5 May, 2019 from <http://www.mathematik.uni-dortmund.de/~kuzmin/cfdintro/lecture1.pdf>
- [63] ANSYS *Fluent User's Guide: UDS Theory*. Retrieved 6 May, 2019 from https://www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/flu_ug/flu_ug_sec_uds_theory.html
- [64] ANSYS *Fluent User's Guide: Mesh Quality*. Retrieved 6 May, 2019 from https://www.sharcnet.ca/Software/Ansys/17.0/en-us/help/flu_ug/flu_ug_mesh_quality.html
- [65] ANSYS *Fluent User's Guide: Overview of Flow Solvers*. Retrieved 6 May, 2019 from https://www.sharcnet.ca/Software/Ansys/17.2/en-us/help/flu_th/flu_th_SolveOverview.html
- [66] ANSYS *Fluent Theory Guide: Pressure-Based Solver*. Retrieved 6 May, 2019 from <http://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node361.htm>
- [67] ANSYS *Fluent User's Guide: Judging Convergence*. Retrieved 6 May, 2019 from <https://www.sharcnet.ca/Software/Fluent6/html/ug/node1067.htm>
- [68] Kuron, M. (2015). *3 Criteria for Assessing CFD Convergence*. Retrieved 6 May, 2019 from <https://www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/9296/3-Criteria-for-Assessing-CFD-Convergence.aspx>
- [69] ANSYS *Fluent User's Guide. Boundary Conditions*. Retrieved 6 May, 2019 from <https://www.sharcnet.ca/Software/Fluent6/html/ug/node203.htm>
- [70]
- [71] Vázquez González, P. *Electrohidrodinámica con carga eléctrica en volumen: conceptos y aplicaciones*.
- [72] Crowley, J.M. (1999). *Fundamentals of Applied Electrostatics*, 1st edn. Laplacian Press/Morgan Hill, California

- [73] ANSYS *Fluent User's Guide: What is a User-Defined Function?* Retrieved 6 May, 2019 from https://www.sharcnet.ca/Software/Ansys/15.0.7/en-us/help/flu_udf/flu_udf_WhatIsAUDF.html
- [74] Hughes, S. (2005). *Lecture 5: Fields and Potentials around conductors. The electrostatic uniqueness theorem.* Massachusetts Institute of Technology, Department of Physics.. Retrieved 20 May, 2019 from <http://web.mit.edu/sahughes/www/8.022/lec05.pdf>
- [75] Evans, M. (2013). *Electromagnetism 3 Section 3: Gauss's Law.* University of Edinburg. Retrieved 20 May, 2010 from <https://www2.ph.ed.ac.uk/~mevans/em/lec3.pdf>
- [76] *Capacitance and Dielectrics.* University Central of Florida, Department of Physics. Retrieved 20 May, 2019 from https://physics.ucf.edu/~roldan/classes/Chap24_PHY2049.pdf
- [77] Kandlikar, S. et al (2006). Single-phase liquid flow in minichannels and microchannels. *Heat Transfer and Fluid Flow in Minichannels and Microchannels.* pp. 90-96.
- [78] 42U Data Center Solutions. Retrieved 21, May 2019 from <https://www.42u.com/measurement/pue-dcie.htm>
- [79] *Oil Cooling System for Data Centers.* Gesab. Retrieved 21, May 2019 from <https://gesab.com/oil-cooling-system-for-data-centers/>
- [80] Zohuri, B. (2017). *Laminar Incompressible Forced Convection.* 10.1007/978-3-319-53829-7_5.
- [81] Caminha, G. (2019). *The CFL Condition and How to Choose Your Timestep Size.* Retrieved 10 March, 2019 from <https://www.simscale.com/blog/2017/08/cfl-condition/>
- [82] Cimbala, Y. et al. (2006). *Fluid mechanics: fundamentals and applications (1st ed.).* Boston: McGraw-Hill Higher Education. pp. 321, 322, 323, 324, 325, 326, 327, 328, 329.
- [83] ANSYS *Fluent User's Guide: Electric Potential.* Retrieved 6 May, 2019 from https://www.sharcnet.ca/Software/Ansys/17.0/en-us/help/flu_th/flu_th_models_elec_potential.html
- [84] ANSYS *Fluent User's Guide: Heat Transfer Theory.* Retrieved 6 May, 2019 from https://www.sharcnet.ca/Software/Ansys/17.0/en-us/help/flu_th/flu_th_sec_hxfer_theory.html#flu_th_eq_energy_E

UNIVERSITÉ CATHOLIQUE DE LOUVAIN
École polytechnique de Louvain

Rue Archimède, 1 bte L6.11.01, 1348 Louvain-la-Neuve, Belgique | www.uclouvain.be/epl