Simulation of a micro jet cooling array

Pierre Montanier
Louisiana State University and Agricultural and Mechanical College, pmonta2@lsu.edu
SIMULATION OF A MICRO JET COOLING ARRAY

A Thesis

Submitted to the Graduate Faculty of the
Louisiana State University and
Agricultural and Mechanical College
in partial fulfilment of the
requirements for the degree of
Master of Science in Mechanical Engineering

in

The Department of Mechanical Engineering

by

Pierre Montanier
ENSICA, Toulouse, France, 2005
May 2005
Pour un Monde Meilleur!
Acknowledgments

My research would not have been completed without my advisor, Dr. Dimitris E. Nikitopoulos, who offered funding, motivation and personal support throughout the project. It was a great pleasure to work with him. Besides, I am honored that Dr. Kevin Kelly and Dr Ram Devireddy accepted my invitation to be part of my committee.

I would like to give a special thanks to Amit Maha who helped me a lot during the completion of my thesis. At the same time, I want to thank the students in the lab for providing a pleasant atmosphere.

I will not forget to thank my roommate Thomas for sharing with me all kind of situations that we encountered when we arrived in the United States. Then, I would like to thank Steven for many flights over Louisiana, and also Mauricio and Olivier for making a lot of parties during the week end. Another special thanks to Jean and Nada.

Finally, I am eternally grateful to my parents and all my family for their outstanding support and love.
Table of Contents

Acknowledgments........................................................................................................................................... iii

List of Tables.................................................................................................................................................. vi

List of Figures................................................................................................................................................ vii

Abstract......................................................................................................................................................... xi

Chapter 1 Introduction................................................................................................................................. 1
  1.1 Preliminary Tests on the Micro Jet Cooling Array................................................................. 4
  1.2 Plan of Study...................................................................................................................................... 6

Chapter 2 Principle of Numerical Simulations.......................................................................................... 8

Chapter 3 Surface Cooling with One Impingement Jet....................................................................... 10
  3.1 Problem Set Up............................................................................................................................... 10
     3.1.1 Meshing.................................................................................................................................... 12
     3.1.2 Boundary Conditions................................................................. ........................................... 12
     3.1.3 Material Properties................................................................................................................ 13
  3.2 Results for Re=452......................................................................................................................... 14
     3.2.1 Flow Features....................................................................................................................... 14
     3.2.2 Heat Transfer Features......................................................................................................... 17
  3.3 Results for Re=1804......................................................................................................................... 20
     3.3.1 Flow Features....................................................................................................................... 20
     3.3.2 Heat Transfer Features......................................................................................................... 23
  3.4 Conclusion........................................................................................................................................ 25

Chapter 4 Surface Cooling with a Micro Jet Cooling Array................................................................. 27
  4.1 Problem Set Up............................................................................................................................... 27
     4.1.1 Meshing.................................................................................................................................... 28
     4.1.2 Boundary Conditions.............................................................................................................. 29
  4.2 Results for Re=1033......................................................................................................................... 30
     4.2.1 Flow Field Only...................................................................................................................... 30
     4.2.2 Comparison of the Flow for two Different Grids................................................................. 37
     4.2.3 Flow Field and Heat Flux....................................................................................................... 41
  4.3 Results for Re=1160......................................................................................................................... 49
     4.3.1 Flow Field Features................................................................................................................ 49
     4.3.2 Heat Transfer Features......................................................................................................... 54
  4.4 Results for a Micro Jet Cooling Array with a Smaller Impingement Region.............................. 57
     4.4.1 Evolution of the Problem Set Up............................................................................................ 57
     4.4.2 Flow Field Features................................................................................................................ 59
     4.4.3 Heat Transfer Features......................................................................................................... 62
  4.5 Results for the Micro Jet Cooling Array without the Return Holes............................................. 65
     4.5.1 Change in the Problem Set Up.............................................................................................. 66
     4.5.2 Flow Field Features................................................................................................................ 66
     4.5.3 Heat Transfer Features......................................................................................................... 69
4.6 Comparison between Volume Heat Source and Surface Heat Flux Results for the Micro Jet Cooling Array

4.7 Comparison between Experimental and Numerical Results

4.8 Conclusion

Chapter 5 Discussion and Future Work

5.1 Discussion on the Best Size for the Impingement Region

5.2 Discussion on the Interest of Return Holes

5.3 Future Work

Chapter 6 Conclusion

References

Appendix A: UDF Function for One Impingement Jet

Appendix B: UDF for the Micro Jet Cooling Array

Appendix C: Thermal and Mechanical Properties of Liquid Water

Vita
List of Tables

Table 3.1: Comparison of Heat Transfer Coefficient Value.................................26
Table 4.1: Numerical Values of Micro Jet Cooling Array Dimensions.....................28
Table 4.2: Pressure Drop through MJCA in respect to the Inlet Reynolds Number.........53
Table 4.3: Pressure Drop through the Device in respect to the Inlet Reynolds Number...61
Table 4.4: Comparison of Heat Transfer Coefficient and Nusselt Number Average.......64
Table 4.5: Set Up Values for the two problems.....................................................71
Table 4.6: Geometrical Parameters for the Experimental Cases.............................74
Table C.1: Variation of Density, Kinematic Viscosity and Thermal Conductivity of Liquid Water with Temperature.................................................................84
Table C.2: Variation of Specific Heat of Liquid Water with Temperature....................84
List of Figures

Figure 1.1: Schematic depiction of cross flow from neighboring jets overwhelming impingement.................................................................2
Figure 1.2: Schematic showing flow of the Micro Jet Cooling Array..........................3
Figure 1.3: Cross section of the bottom plate of the Micro Jet Cooling Array..............4
Figure 1.4: Schematic of experimental apparatus to quantify MJCA performance..........5
Figure 1.5: Micro Jet Cooling Array data..................................................6
Figure 3.1: X-Z or Y-Z cut of the one impingement jet geometry.............................10
Figure 3.2: Three dimensional drawing of the one impingement jet problem............11
Figure 3.3: XY faces split in two parts......................................................12
Figure 3.4: Axial velocity on the X=0 plane.................................................15
Figure 3.5: Radial velocity on the X=0 plane...............................................16
Figure 3.6: Azimuthal vorticity on the X=0 plane..........................................17
Figure 3.7: Dimensionless Temperature Distribution on the X=0 Plane.................18
Figure 3.8: Dimensionless Temperature Distribution on the Bottom Wall.............18
Figure 3.9: Heat transfer coefficient on the bottom wall...................................19
Figure 3.10: Nusselt Number on the Bottom Wall........................................20
Figure 3.11: Axial Velocity (m·s⁻¹) on the X=0 Plane....................................21
Figure 3.12: Radial Velocity (m·s⁻¹) on the X=0 Plane..................................22
Figure 3.13: Azimuthal Vorticity (s⁻¹) on the X=0 Plane..................................22
Figure 3.14: Dimensionless Temperature Distribution on the Bottom Wall...........23
Figure 3.15: Heat Transfer Coefficient (W·m⁻²·K⁻¹) on the Bottom Wall...............24
Figure 3.16: Nusselt Number Distribution on the Bottom Wall............................24
Figure 3.17: Description of the One Jet Bottom Wall......................................25
Figure 4.1: Transversal Cut of the Micro Jet Cooling Array...............................27
Figure 4.2: Longitudinal cut of the Micro Jet Cooling Array………………………………...28

Figure 4.3: Transversal cut of the MJCA wedge I have solved………………………………29

Figure 4.4: Axial Velocity (\(m \cdot s^{-1}\)) on the \(Z = 40 \mu m\) Plane…………………………………31

Figure 4.5: Axial Velocity (\(m \cdot s^{-1}\)) on the \(Z = 1710 \mu m\) Plane…………………………………32

Figure 4.6: Azimuthal Velocity on the \(Z = 2164.8 \mu m\) Plane…………………………………33

Figure 4.7: Radial Velocity on the \(Z = 2164.8 \mu m\) Plane…………………………………33

Figure 4.8: Axial Velocity on the \(Z = 2164.8 \mu m\) Plane…………………………………34

Figure 4.9: Radial Velocity (\(m \cdot s^{-1}\)) on the \(Z = 858 \mu m\) Plane in the Outlet Region……35

Figure 4.10: Axial Velocity (\(m \cdot s^{-1}\)) on the \(Z = 1450 \mu m\) Plane……………………………36

Figure 4.11: Radial Velocity (\(m \cdot s^{-1}\)) on the \(Z = 1450 \mu m\) Plane……………………………36

Figure 4.12: Radial Velocity (\(m \cdot s^{-1}\)) on the \(Z = 374 \mu m\) Plane……………………………37

Figure 4.13: Comparison between the Denser and Regular Grid Axial Velocity Results at the Location \(Z = 1710 \mu m\) and \(Y = 1024 \mu m\) ………………………………………………………………38

Figure 4.14: Comparison between the Denser and Regular Grid Radial Velocity Results at the Location \(Z = 1710 \mu m\) and \(Y = 1024 \mu m\) ………………………………………………………………38

Figure 4.15: Comparison between the Denser and Regular Grid Tangential Velocity Results at the Location \(Z = 1710 \mu m\) and \(Y = 1024 \mu m\) ………………………………………………………………39

Figure 4.16: Comparison between the Denser and Regular Grid Axial Velocity Results at the Location \(Z = 1826 \mu m\) and \(Y = 1508 \mu m\) ………………………………………………………………39

Figure 4.17: Comparison between the Denser and Regular Grid Radial Velocity Results at the Location \(Z = 1826 \mu m\) and \(Y = 1508 \mu m\) ………………………………………………………………40

Figure 4.18: Comparison between the Denser and Regular Grid Tangential Velocity Results at the Location \(Z = 1826 \mu m\) and \(Y = 1508 \mu m\) ………………………………………………………………40

Figure 4.19: Comparison between Axial Velocity Results with and without Heat Flux on the Bottom Wall at the Location \(Z = 1710 \mu m\) and \(Y = 1024 \mu m\) ………………………………42

Figure 4.20: Comparison between Radial Velocity Results with and without Heat Flux on the Bottom Wall at the Location \(Z = 1710 \mu m\) and \(Y = 1024 \mu m\) ………………………………42
Figure 4.21: Comparison between tangential Velocity Results with and without Heat Flux on the Bottom Wall at the Location $Z = 1710 \mu m$ and $Y = 1024 \mu m$ .................43

Figure 4.22: Comparison between Axial Velocity Results with and without Heat Flux on the Bottom Wall at the Location $Z = 1826 \mu m$ and $Y = 1508 \mu m$ .........................43

Figure 4.23: Comparison between Radial Velocity Results with and without Heat Flux on the Bottom Wall at the Location $Z = 1826 \mu m$ and $Y = 1508 \mu m$ .........................44

Figure 4.24: Comparison between tangential Velocity Results with and without Heat Flux on the Bottom Wall at the Location $Z = 1826 \mu m$ and $Y = 1508 \mu m$ .........................44

Figure 4.25: Dimensionless Temperature Distribution on the Bottom Wall .....................45

Figure 4.26: Heat Transfer Coefficient ($W \cdot m^{-2} \cdot K^{-1}$) Distribution on the Bottom Wall.....46

Figure 4.27: Heat Transfer Coefficient ($W \cdot m^{-2} \cdot K^{-1}$) Distribution on the Bottom Wall for the whole Micro Jet Cooling Array.........................................................46

Figure 4.28: Nusselt Number Distribution on the Bottom Wall of the Micro Jet Cooling Array........................................................................................................47

Figure 4.29: Dimensionless Temperature Distribution on the $Z = 1710 \mu m$ Plane.............48

Figure 4.30: Dimensionless Temperature Distribution on the $Z = 1584 \mu m$ Plane.............48

Figure 4.31: Axial Velocity ($m \cdot s^{-1}$) on the $Z = 40 \mu m$ Plane........................................49

Figure 4.32: Axial Velocity ($m \cdot s^{-1}$) on the $Z = 1584 \mu m$ Plane..................................50

Figure 4.33: Radial Velocity ($m \cdot s^{-1}$) on the $Z = 374 \mu m$ Plane in the Outlet Region......51

Figure 4.34: Azimuthal Velocity ($m \cdot s^{-1}$) on the $Z = 2164.8 \mu m$ Plane......................52

Figure 4.35: Radial Velocity ($m \cdot s^{-1}$) on the $Z = 2164.8 \mu m$ Plane......................52

Figure 4.36: Pressure Loss Coefficient in respect to the Inlet Reynolds Number.................54

Figure 4.37: Dimensionless Temperature Distribution on the Bottom Wall.......................55

Figure 4.38: Dimensionless Temperature Distribution on the Whole MJCA Bottom Wall.....55

Figure 4.39: Nusselt Number Distribution on the Whole MJCA Bottom Wall.............57

Figure 4.40: Longitudinal Cut of the Micro Jet Cooling Array with a Smaller Impingement Region..........................................................58
Figure 4.41: Axial Velocity ($m \cdot s^{-1}$) on the $Z = 1800 \mu m$ Plane..........................59

Figure 4.42: Azimuthal Velocity ($m \cdot s^{-1}$) on the $Z = 1800 \mu m$ Plane..........................60

Figure 4.43: Radial Velocity ($m \cdot s^{-1}$) on the $Z = 1800 \mu m$ Plane..........................60

Figure 4.44: Pressure Loss Coefficient in respect to the Inlet Reynolds Number......................62

Figure 4.45: Dimensionless Temperature Distribution on the Whole MJCA Bottom Wall...........63

Figure 4.46: Nusselt Number Distribution on the Whole MJCA Bottom Wall.........................64

Figure 4.47: Transversal Cut of the New Wedge.........................................................65

Figure 4.48: Axial Velocity ($m \cdot s^{-1}$) on the $Z = 1650 \mu m$ Plane..............................67

Figure 4.49: Axial Velocity ($m \cdot s^{-1}$) on the $Z = 1584 \mu m$ Plane..............................67

Figure 4.50: Radial Velocity ($m \cdot s^{-1}$) on the $Z = 132 \mu m$ Plane...............................68

Figure 4.51: Radial Velocity ($m \cdot s^{-1}$) on the $Z = 374 \mu m$ Plane...............................68

Figure 4.52: Azimuthal Velocity ($m \cdot s^{-1}$) on the $Z = 132 \mu m$ Plane...........................69

Figure 4.53: Nusselt Number Distribution on the Whole MJCA Bottom Wall.........................70

Figure 4.54: Dimensionless Temperature Distribution on the Whole MJCA Bottom Wall........70

Figure 4.55: Comparison of Temperature (K) Profiles on the Bottom Wall.........................72

Figure 4.56: Comparison of the Three Components of Velocity at $Z = 2164 \mu m$ and $Y = 1496.2 \mu m$.................................................................................................72

Figure 4.57: Comparison between Experimental and Numerical Results..............................73

Figure C.1: Variation of Density ($kg \cdot m^{-3}$) with Temperature (K)..............................85

Figure C.2: Thermal Conductivity ($W \cdot m^{-1} \cdot K^{-1}$) Variation with Temperature (K)......86

Figure C.3: Dynamic Viscosity ($kg \cdot m^{-1} \cdot s^{-1}$) Variation with Temperature (K)........86

Figure C.4: Specific Heat ($J \cdot kg^{-1} \cdot K^{-1}$) Variation with Temperature..........................87
Abstract

The new generation of electronic devices are more powerful, and they produce more heat. Hence, there is a need for micro cooling systems for removing the heat from these micro chips. This research focused on one micro cooling system-the micro jet cooling array.

Some test simulations were made on a single jet cooling system. In addition, the inlet Reynolds number was varied in order to obtain the variation of the heat transfer coefficient. These simulations gave a basic idea of how the impingement cooling method works on the micro jet cooling array.

Numerical simulations were made on the actual micro jet cooling array. Simulations consisted of variation of parameters (geometry, Reynolds number, heat flux). When the micro jet cooling array from MEZZO systems was simulated with no wall separating the outlet region from the impingement area, performance was enhanced. In fact, this system presents the lowest pressure drop through the device and has the same heat transfer rate on the cooling surface.

The relationship between the heat transfer rate and the inlet Reynolds number was established for the micro jet cooling array system (for instance an inlet Reynolds number of 1033, the heat transfer coefficient average on the cooling surface is $9.9 \frac{W}{cm^2 \cdot K}$).
Chapter 1 Introduction

The new generation of semiconductor amplifiers and high-power systems create an important amount of heat that cannot be removed by classical cooling systems. The heat created is in the order of $1kW \cdot cm^{-2}$.

The classical systems are used when the heat flux is low. Indeed, in this case, it is possible to spread the heat into a high conductive material in order to increase the surface area and then to decrease the local heat flux. But, in our case, when the heat flux is too important, we cannot spread the heat because the thermal resistance of high conductivity material like copper will become too high. So we have to remove the heat directly without an intermediate surface. We put a low thermal resistance very close to the heat source. For the micro jet cooling array, the coolant used is liquid water at ambient temperature. This kind of coolant gives an important rate of heat removal (approximatively $1kW \cdot cm^{-2}$), but also a low temperature difference between the coolant and the surface.

In industry, three different methods-internal, impingement and film cooling-are used to cool. For the micro jet cooling array, we use the impingement method. We have an array of impingement jets which cool the surface requiring thermal protection. But we cannot have the array itself because we lose some efficiency in the cooling. Indeed, if one reservoir feeds the entire array, the velocity is much higher for the jets at the edge of the array compared to those on the center. This is due to the pressure drop between the center and the edges of the array. We also have some cross flow from interior micro jets which dilutes the impingement flow on the edge of the array. Both problems are illustrated in figure 1.1 below.
Unequal flow through jets is a function of the distance from the edge of the array

Cross-flow interferes with impingement at the edge of array

To avoid these two problems and to increase the efficiency of the system, some return holes are added around each jet of the micro jet cooling array. So the coolant can escape the annulus by these holes and thus prevent the cross flow on the edge of the array. Moreover this technology used in the micro jet cooling array gives a more uniform pressure differential in the impingement jets and thus a uniform velocity. This is due to the fact that the gap between the micro jet cooling array and the surface being cooled is much lower than the gap between the two plates of the array where the flow escapes. Figure 1.2 below shows the basic principle of the micro jet cooling array.

Figure 1.1: Schematic depiction of cross flow from neighboring jets overwhelming impingement (Project Proposal, Mezzo Systems)
The actual micro jet cooling array is made by a laser and LIGA micromachining process. The size of the cross section is 1.5cm * 1.5cm and the diameter of an impingement jet is 500 μm. Moreover, the bottom plate of the array is perforated by some return holes whose diameter is 350 μm. Jets and holes are arranged in a hexagonal structure; for instance,
each jet is surrounded by six other jets to form a hexagon, and between two jets there is one returned hole. Figure 1.3 below shows this hexagonal structure of the micro jet cooling array.

![Cross section of the micro jet cooling array. The red circles are the returned holes and the blue ones are the impingement jets.](image)

Figure 1.3: Cross Section of the Bottom Plate of the Micro Jet Cooling Array

1.1 Preliminary Tests on the Micro Jet Cooling Array

Mezzo Systems made some tests on the micro jet cooling array in order to know the performance of the prototype and also to prove the benefit of the returned holes. In fact, they tested two different prototypes: the first one is the actual micro jet cooling array and the second one is the micro jet cooling array without any returned holes. The geometry of these two prototypes is basically the same except for the returned holes. Figure 1.4 shows a cross section of the experiment set-up of these tests.

The results of these tests are the heat transfer coefficient (convection coefficient $h$) for the two prototypes and for different values of flow rate. The convection coefficient is found with the knowledge of thermocouple’s temperatures ($T_1$ and $T_2$). The formulas we need to find $h$ are below.
Figure 1.4: Schematic of experimental apparatus to quantify MJCA performance (Project Proposal, Mezzo Systems)

\[ q^* = k_{aluminum} \cdot \frac{(T_1 - T_2)}{\Delta x_1} \]

\[ T_{inter} = T_2 - k_{aluminum} \cdot \frac{q^* \cdot \Delta x_2}{q^*} \]

\[ h = \frac{T_{inter} - T_{cooler}}{q^*} \]

where \( T_{cooler} \) = the temperature of the coolant

\( k_{aluminum} \) = the thermal conductivity of aluminum

These preliminary tests were completed for different flow rates (from 0.5GPM to 1GPM) and for a heat flux of 70 \( W \cdot cm^{-2} \). As a result, we get Figure 1.5 below that shows the heat transfer coefficient function of the flow rate for both prototypes.
The heat transfer coefficient is better for the prototype with the return ports and the difference of convection coefficient increases with the flow rate. This experience gives a good idea of the micro jet cooling array concept.

But, with these results, we just know the local heat transfer coefficient in the center of the array. In addition to these tests, we are going to study the flow field of the micro jet cooling array. This study will be split into two parts. The first part will consist of the experimentation and visualization of the flow field with the µPIV (micro Particle Image Velocimetry), and the second part will focus on the numerical simulation using commercial software FLUENT.

1.2 Plan of Study

In this thesis, I will focus on the numerical part of the project. Before making a simulation on a module of the micro jet cooling array, I have studied a case with only one impingement jet (chapter 3). This first simulation will give us a possible comparison with the study of a single periodic module of the micro jet cooling array (chapter 4). For both cases, I studied the flow field itself, and then I added the heat flux in order to know the difference
between the cold and hot field. This comparison will be useful for the experimentation part to validate the visualization we will find without any heat flux. I have also made some changes in the geometry of the micro jet cooling array; for instance, I have studied different values of the distance between the bottom plate of the MJCA and the surface being cooled. The numerical simulations have been made for different values of the inlet Reynolds number.

Chapter 5 will present the discussion of results for the different kind of geometry and possibly make a conclusion on which geometry is better based on the numerical simulation. The future work will consist of comparison with experimental results.
Before showing any problems I have solved for this project, I will give an introduction of FLUENT, the commercial software I used. The equations solved by Fluent are based on the Navier-Stokes model of fluid dynamics. This model consists of four conservation equations: conservation of mass and conservation of the three momentum components. There is an additional conservation equation which is needed for flows involving heat transfer or compressibility. This last equation is based on the principle of total energy conservation.

The first equation is the continuity equation which follows the principle of mass conservation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho \cdot u_i) = 0$$

where $\rho$ is the density of the fluid and $u_i$ is the velocity in the i direction.

The following equation is the conservation of momentum in the i direction in a non-accelerating reference frame (a Galilean reference frame):

$$\frac{\partial}{\partial t} (\rho \cdot u_i) + \frac{\partial}{\partial x_i} (\rho \cdot u_i \cdot u_j) = \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho \cdot g_i + F_i$$

where $p$ is the static pressure, $\rho \cdot g_i$ is the gravitational body force in the i direction, $F_i$ represents other external body forces in the i direction and $\tau_{ij}$ is the stress tensor.

The stress tensor for a Newtonian fluid is represented by the following formula:

$$\tau_{ij} = \mu \left[ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] - \frac{2}{3} \mu \cdot \frac{\partial u_i}{\partial x_j} \cdot \delta_{ij}$$

where $\mu$ is the dynamic viscosity of the fluid.

The first part of the stress tensor is stress due to deformation and rotation, and the second part is the volume dilation effect on the stress.
When species diffusion and volumetric heat source are not present within the fluid, the conservation of energy is represented by the following equation:

\[
\frac{\partial}{\partial t}(\rho \cdot E) + \frac{\partial}{\partial x_i}(u_i \cdot (\rho \cdot E + p)) = \frac{\partial}{\partial x_j}\left(k \cdot \frac{\partial T}{\partial x_i} + u_j \cdot (\tau_{ij})\right)
\]

where \(k\) is the thermal conductivity.

We get an expression for the total energy \(E\) as a function of temperature, static pressure and velocity.

\[
E = \int_{T_{ref}}^{T} C_p \cdot dT + \frac{u_i^2}{2}
\]

where \(T_{ref} = 298.15K\) (reference temperature) and \(C_p\) is the specific heat of the considered fluid.

Before running FLUENT and starting iterations, we first make and discretize the geometry with Gambit. Gambit is an associated software with FLUENT in order to obtain a grid of the geometry.
Chapter 3 Surface Cooling with One Impingement Jet

3.1 Problem Set Up

The first simulation I made is of a cooling module with just one impingement jet. This first part will focus on the problem set up.

The geometry of the problem is cylindrical. There is one inner cylinder which corresponds to the inlet channel. This inlet channel is surrounded by two big cylinders. The first one next to the inner one is the wall of the channel and the external one corresponds to the outlet. And, in the bottom of these cylinders, there is an impingement region where the flow turns from the inlet to the outlet. The bottom wall is the surface being cooled of the Microsystems. The major part of the heat transfer is located in this region, so we decide to more precisely study this part of the geometry. Figure 3.1 below shows a longitudinal cut (in the Z direction) of the geometry.

![Figure 3.1: X-Z or Y-Z cut of the one impingement jet geometry](image-url)
In three dimensions, the geometry looks like three coaxes cylinders: the inner one corresponds to the inlet channel, the middle one to the wall and the outer one to the exit. The length of the two inner cylinders is $5d$.

Fig 3.2: three dimensional drawing of the one impingement jet problem

In this problem, the thickness “t” of the wall is $0.4mm$ and the diameter $d$ of the inner cylinder is $4mm$. Then the diameter $D$ of the outer cylinder is such that the outlet area equals to the inlet area. The following equations show the calculations I have made to get $D$.

\[
\frac{\pi \cdot D^2}{4} - \frac{\pi \cdot (d + t)^2}{4} = \frac{\pi \cdot d^2}{4}
\]

\[
D^2 - 2 \cdot d^2 - t^2 - 2 \cdot d \cdot t = 0
\]

Finally, we found $D = 5.95mm$. 
3.1.1 Meshing

Before starting the simulation, the geometry has to be meshed. The first thing I do is mesh faces in XY plane. But, we have to split the three cylinders in two parts so I can use a wedge primitive mesh in Gambit. Thus, faces in XY plane are either a quarter disk or a three quarter disk. The figure 3.3 below shows a generic XY face just before meshing.

![Figure 3.3: XY faces split in two parts](image)

Then, I mesh the two edges which are orthogonal in the figure 3.3 above and also the circular edges. And, I used the gambit wedge primitive scheme to mesh these faces. To finish the mesh of the entire volume, I extruded this XY face mesh through the all volume with the cooper scheme.

3.1.2 Boundary Conditions

First, I specified a fully-developed velocity profile at the inlet in order to get a quicker convergence. The expression of the velocity profile is just below.

\[ u(r) = 2 \cdot U_\infty \cdot \left[ 1 - \left( \frac{r}{r_\infty} \right)^2 \right] \]
where \( r_o \) (actually \( r_0 = \frac{d}{2} \)) is the radius of the inlet channel, \( U_\infty \) is the velocity of the flat velocity profile and \( r \in [0, r_o] \).

I ran two different cases with two different values of \( U_\infty \) which correspond to two different Reynolds numbers (451 and 1804).

\[
\text{Re} = 451 \Rightarrow U_\infty = 0.125 \text{ m} \cdot \text{s}^{-1} \\
\text{Re} = 1804 \Rightarrow U_\infty = 0.5 \text{ m} \cdot \text{s}^{-1}
\]

Then, for the energy equation, all the walls are adiabatic except for the bottom wall where I put a constant heat flux. But, the water cannot boil anywhere at any moment; so, there exists a maximum limit for the bottom wall heat flux. This limit depends on the Reynolds number. First, I made a heat transfer calculation for a stagnation point flow which is close to an impingement flow, and I found a first estimation of the maximum heat flux. Then I ran a case with this first value of heat flux, and I increased the heat flux step by step until the maximum temperature of the water reached 370K. The two limits are written below.

\[
\text{Re} = 451 \Rightarrow q^* = 340kW \cdot m^{-2} \\
\text{Re} = 1804 \Rightarrow q'' = 2800kW \cdot m^{-2}
\]

### 3.1.3 Material Properties

The material is liquid water with a temperature dependent property. The tables of density, specific heat, thermal conductivity and kinematic viscosity with respect to temperature are in the Appendix C. For the first three properties, I interpolated the tables to find a polynomial of temperature. The following equations are the three interpolated polynomials.

\[
\rho = -0.0033 \cdot T^2 + 1.6997 \cdot T + 783.33 \\
C_p = -4.5972 \cdot 10^{-8} \cdot T^5 + 7.746 \cdot 10^{-5} \cdot T^4 - 5.2122 \cdot 10^{-2} \cdot T^3 + 17.516 \cdot T^2 - 2.9407 \cdot 10^3 \cdot T + 201.53 \cdot 10^3 \\
k = -9 \cdot 10^{-6} \cdot T^2 + 0.0072 \cdot T - 0.721
\]
In these three polynomials, the temperature $T$ is in Kelvin and the units of results are in the international system: $\rho$ in $kg \cdot m^{-3}$, $C_p$ in $J \cdot kg^{-1} \cdot K^{-1}$ and $k$ in $W \cdot m^{-1} \cdot K^{-1}$.

For the viscosity, I deduced the formula below.

$$\mu = \mu_0 \cdot \exp[-1.94 - 4.8 \cdot \left(\frac{273.15}{T}\right) + 6.74 \cdot \left(\frac{273.15}{T}\right)^3]$$

where $\mu_0 = 1.792 \cdot 10^{-3} kg \cdot m^{-1} \cdot s^{-1}$ and $T$ is in Kelvin.

Before running the simulation, I patched the inlet and outlet fluid zones with a fully-developed profile. This was done to get a faster convergence. The two formulas below correspond to the inlet and the outlet patches, respectively. The proof can be seen in Appendix D.

$$u_{inlet} = 2 \cdot U_\infty \cdot [1 - \left(\frac{r}{r_0}\right)^2]$$

$$u_{outlet} = \frac{-2 \cdot U_\infty \cdot \left[ r_0^2 - r_i^2 + \frac{r_o^2 - r_i^2}{\ln\left(\frac{r_0}{r_i}\right)} \ln\left(\frac{r}{r_0}\right) \right]}{(r_i^2 + r_o^2) \ln\left(\frac{r_0}{r_i}\right) \ln\left(\frac{r}{r_0}\right)}$$

where $r_i$ in the first formula is the radius of the inner channel; $r_o$ and $r_i$, in the second formula, are the outer and inner radiuses of the annulus, respectively.

The convergence criterion is $10^{-5}$ for continuity and momentum equations and $10^{-6}$ for energy equation.

### 3.2 Results for Re=452

The first interesting thing I am interested in is the flow pattern itself, and then I looked at the heat transfer features, which are the goal of such a system.

#### 3.2.1 Flow Features

First, in this system, we have a simple flow in a pipe for the inlet and the outlet. That is why I patched the solution in these two parts of the geometry. Indeed, we know the solution by a simple calculation of flow inside a cylindrical pipe. The flow, at the end of the inlet pipe,
goes to the impingement region like one in a fountain. All the results are presented in a dimensionless form. The length scale is the actual diameter of the inlet pipe $d$, the velocity scale is the average velocity at the inlet $U_\infty$, and the dimensionless temperature is defined by the following formula.

$$T^* = \frac{T - T_{\text{max}}}{T_{\text{inlet}} - T_{\text{max}}}$$

where $T_{\text{mac}}$ and $T_{\text{inlet}}$ are respectively 372K and 290K.

Figure 3.4 below shows the velocity in the axial direction close to the end of the inlet channel and the impingement region.

In the bottom of the figure, there is pipe flow which is actually a parabolic profile. One is upward (in the inner cylinder) and the other one is backward (in the annulus). And then, close to the impingement region (top of the figure), the flow turns to the external annulus like a fountain flow.
In the annulus and in the inlet pipe, the major component of the velocity is the axial one. But, in the impingement region, the axial component decreases and the radial velocity increases, especially close to the surface being cooled. The radial velocity starts gradually increasing at the end of the inlet channel and decreasing regularly at the beginning of the annulus. The next figure corresponds to the radial component of the velocity in the $X = 0$ plane (figure 3.5).

![Figure 3.5: Radial Velocity ($m \cdot s^{-1}$) on the X=0 Plane](image)

Moreover, the azimuthal velocity value is not significant. Indeed, its order of magnitude approximatively equals to the convergence criterion for momentum and continuity equations. Actually, this component of the velocity is lower than $10^{-5} \ m \cdot s^{-1}$. It is just some numerical artifact.

Besides, the flow creates some vortices within the boundary layers of the inlet pipe and the external annulus. The principal direction of these vortices is azimuthal; the other two components are very small and are not really significant. The vortices are more important on
the boundary layers of the annulus than those of the inlet channel; indeed, the width of the annulus is smaller than the diameter of the pipe. The highest value of vorticity exists at the end of the wall that separates the inlet cylinder from the outlet annulus because at this point, the flow has to turn around. In this area, the vortices are bigger on the annulus side because the vortices due to the shear stress are added to the one created by the “turning” effect. Figure 3.6 plots the azimuthal vorticity on the $X = 0$ plane.

Figure 3.6: Azimuthal Vorticity ($s^{-1}$) on the X=0 Plane

### 3.2.2 Heat Transfer Features

One of the first heat transfer features of this flow is the repartition of temperature within the flow. The maximum temperature change is concentrated in a small area close to the surface being cooled. In the other flow region, the temperature equals to the inlet temperature ($290 \, K$), except for some regions where the temperature is a little bit higher (the maximum
temperature is approximately $298 \, K$). The temperature distribution for one impingement jet in the $X = 0$ plane is shown in the figure 3.7 below.

![Image](image1.jpg)

**Figure 3.7: Dimensionless Temperature Distribution on the X=0 Plane**

Figure 3.8 shows a close-up of the temperature distribution in the region next to the bottom wall. This region presents a high temperature gradient.

![Image](image2.jpg)

**Figure 3.8: Dimensionless Temperature Distribution on the Bottom Wall**
The highest temperature is on the circumference of the surface being cooled. Indeed, at this location, there is some recirculation which causes small velocity. It is exactly the same case at the stagnation point. The lowest heat transfer rate \( h = 0.4 \, W \cdot cm^{-2} \cdot K^{-1} \) is located at these two previous locations. On the contrary, the best rate is around the stagnation point where the bottom wall boundary layer starts. This value approximatively equals to \( 1.19 \, W \cdot cm^{-2} \cdot K^{-1} \). The complete distribution of the heat transfer coefficient through the bottom wall is displayed on the figure 3.9.

![Heat Transfer Coefficient on the Bottom Wall](image)

**Figure 3.9: Heat Transfer coefficient \((W \cdot m^{-2} \cdot K^{-1})\) on the Bottom Wall**

With this previous distribution and the distribution of the thermal conductivity, I obtained the distribution of the Nusselt number which is basically an adimentional heat transfer coefficient. The formula which defines this number is just below.

\[
N_u = \frac{h \cdot d}{k}
\]

where:
- \( h \) is the heat transfer coefficient
- \( d \) is the diameter of the inlet channel
- \( k \) is the thermal conductivity
The values of the Nusselt are between 24 and 70. These two extreme values are for the circumference and the circle just around the impingement point, respectively. The distribution on the surface being cooled of this number is displayed on the figure below.

![Figure 3.10: Nusselt Number on the Bottom Wall](image)

3.3 Results for Re=1804

These new results are for the same problem: surface cooling with one impingement jet, but the Reynolds number has been changed. Actually, I increased this number value in order to find the relationship between the heat transfer coefficient and the Reynolds number.

3.3.1 Flow Features

Like in the previous case with the other Reynolds number, the flow is a simple pipe flow for the one inside the inlet channel (the inner cylinder) and a flow inside an annulus for the outer cylinder. In these two basic flows, the velocity profile is parabolic. The only things which have changed are the variable values. The shape of flow distribution variables is the same in both cases. The major component of velocity is still the axial velocity for the inlet
pipe and the outlet annulus, and the radial velocity for the impingement region. The azimuthal velocity is still negligible; indeed its value is around $10^{-5} \text{ m} \cdot \text{s}^{-1}$ which is actually the value of the convergence criterion for the momentum and continuity equation.

Moreover, there are some vortices within the flow field. The vortices are located in the boundary layers of the inner cylinder and the annulus. The numerical values of the vortices are more important in the shear layer of the annulus than those in the shear layer of the inlet pipe. This is due to the fact that the width of the annulus is much smaller than the diameter of the inlet channel, and so, the stress is more important within the boundary layer of the annulus. Besides, the major component of the vorticity is the azimuthal vorticity. Indeed, the two other components are very small; for instance, their values are around the value of the convergence criterion of the momentum and continuity equation. The following figures (3.11, 3.12 and 3.13) show the distribution of axial velocity, radial velocity and azimuthal vorticity on the $X = 0$ plane, respectively.

![Figure 3.11: Axial Velocity ($m \cdot s^{-1}$) on the X=0 Plane](image-url)
Figure 3.12: Radial Velocity ($m \cdot s^{-1}$) on the X=0 Plane

Figure 3.13: Azimuthal Vorticity ($s^{-1}$) on the X=0 Plane
3.3.2 Heat Transfer Features

The first heat transfer feature is the temperature distribution on the bottom wall.

![Dimensionless Temperature Distribution on the Bottom Wall](image)

Figure 3.14: Dimensionless Temperature Distribution on the Bottom Wall

There is almost no difference between this temperature distribution and the one for the previous case because I adapted the heat flux value in order to get $370 \, K$ for the maximum temperature on the bottom wall. The location of the maximum temperature is a circle whose radius approximatively equals to $2 \, mm$, and the lowest temperature is located on a circle close to the stagnation point.

The distribution of the heat transfer coefficient of this case looks like the same as that for the previous case. Figure 3.15 shows this distribution on the bottom wall.

The best heat transfer rate is located where the temperature reaches its minimum value close to the stagnation point. This is where the boundary layer on the bottom wall starts growing. At this point, the heat transfer coefficient $h$ equals to $74000 \, W \cdot m^{-2} \cdot K^{-1}$ or $7.4 \, W \cdot cm^{-2} \cdot K^{-1}$.
To characterize the heat transfer rate, there is also the Nusselt number which is basically the heat transfer coefficient without any dimension. I gave the definition of this number in the previous part at the bottom of page 19. In this case, the Nusselt number is between 200 and 430 and its distribution is shown in the figure below.
3.4 Conclusion

In conclusion, I would like first to compare the heat transfer results I have found in these two cases with the heat transfer formulas of a stagnation point flow. Indeed, this one jet flow looks like a flow around a stagnation point especially in the impingement area. The formula below is the expression of the heat transfer coefficient in function of the fluid properties, the inlet velocity and a distance.

\[
h = 0.332 \cdot \left[2 \cdot P_r \right]^{2/3} \cdot k \cdot \frac{U_{\infty}}{\nu \cdot x}
\]

where \( P_r \) is the Prandtl number which is defined by \( P_r = \frac{\mu \cdot C_p}{k} \), \( \mu \) is the dynamic viscosity, \( C_p \) is the specific heat, \( k \) is the thermal conductivity, \( \nu \) is the kinematic viscosity, \( U_{\infty} \) is the inlet velocity and \( x \) is the distance from the centre of the bottom wall which actually is the location of the stagnation point. Figure 3.17 below shows the distance \( x \) on the bottom wall.

![Diagram of One Jet Bottom Wall](image)

**Figure 3.17: Description of the One Jet Bottom Wall**

This formula above can be integrated in order to find the Nusselt number average and the heat transfer coefficient average as well.

\[
\bar{N}_x = \frac{4 \cdot \sqrt{2}}{3} \cdot 0.332 \cdot (2 \cdot P_r) \sqrt{\frac{U_{\infty} \cdot d}{\nu}} \cdot \frac{d}{V} \cdot \frac{d}{D}
\]

\[
\bar{h} = \frac{4 \cdot \sqrt{2}}{3} \cdot 0.332 \cdot (2 \cdot P_r) \sqrt{\frac{U_{\infty}}{D}} \cdot k \cdot \frac{d}{D}
\]
The following Table 3.1 shows the comparison between the heat transfer coefficient and Nusselt number from the numerical simulation and the ones from the analytical formula.

<table>
<thead>
<tr>
<th></th>
<th>$R_e = 451$</th>
<th>$R_e = 1804$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\overline{N}_u$ from simulation</td>
<td>60.2</td>
<td>405</td>
</tr>
<tr>
<td>$\overline{N}_u$ from analytical formula</td>
<td>26.3</td>
<td>52.6</td>
</tr>
<tr>
<td>$\frac{\overline{N}_u}{(\overline{N}<em>u)</em>{theory}}$</td>
<td>2.3</td>
<td>7.6</td>
</tr>
<tr>
<td>$\overline{h}$ from simulation in $W \cdot m^{-2} \cdot K^{-1}$</td>
<td>8181</td>
<td>55071</td>
</tr>
<tr>
<td>$\overline{h}$ from the analytical formula in $W \cdot m^{-2} \cdot K^{-1}$</td>
<td>8626</td>
<td>17253</td>
</tr>
</tbody>
</table>

The heat transfer rate of one impingement jet cooling system is different from the rate of a simple stagnation point flow. Indeed, there is a stagnation point in this flow but the whole flow pattern of this problem is not completely similar to the stagnation point flow.

Finally, I am interested in the evolution of the Nusselt number and the heat transfer coefficient with the Reynolds number. I looked for formulas with the following shape.

$$\overline{N}_u = \beta \cdot (R_e)^\alpha$$

$$\overline{h} = \lambda \cdot (R_e)^\delta$$

where $\alpha$, $\beta$, $\lambda$, $\delta$ are constants to be determined.

These constants are determined by writing a system of two equations and two unknowns with the two values of Nu, h and Reynolds number. The final expressions for both relationships with $P_r = 7.02$ are just below.

$$\overline{N}_u = 0.0071 \cdot (P_r)^{0.5} \cdot (R_e)^{1.375}$$

$$\overline{h} = 3.95 \cdot (R_e)^{1.31}$$
Chapter 4 Surface Cooling with a Micro Jet Cooling Array

4.1 Problem Set Up

This part deals with the Micro Jet Cooling Array surface cooling which is a new kind of cooling system for Microsystems. This system consists of an array of inlet jets which end on an impingement region just above the surface being cooled. Each jet gets six other jets in its neighbourhood. These six jets are arranged like a hexagon around the jet in the middle. Moreover, between two jets, there is one returned hole, so each jet is surrounded by six returned holes. Figure 4.1 below shows the basic structure of the micro jet cooling array.

![Figure 4.1: Transversal Cut of the Micro Jet Cooling Array](image)

where the blue, green and red circles are the impingement jets, the walls around the inlet tubes and the return holes, respectively.
Figure 4.2 shows the flow path in the micro jet cooling array in the longitudinal cut. But, in order to simplify the figure, I drew one jet and its returned holes.

Figure 4.3 below represents the geometry of the problem I solved in fluent.

| Table 4.1: Numerical Values of Micro Jet Cooling Array Dimensions |
|------------------|-------|------|--------|-------|------|-----|
| D               | $D_0$ | $S_t$ | d      | H     | $t_s$ | $L_e$ |
| 500 µm          | 700 µm| 1250 µm| 350 µm | 500 µm | 100 µm| 1600 µm |

4.1.1 Meshing

For the actual simulation, I decided to simulate just one twelfth of the hexagon. Indeed, the maximum number of nodes for Fluent is lower than a million and a half, and I cannot greatly reduce the number of nodes. Moreover, the outlet cannot be too close to the end of the hexagon because in the case where the outlet is very close to the end of the hexagon, the outlet will be in the middle of a recirculation area, and it will be a problem for the convergence. That is why I decided to put the outlet far away from the end of the actual hexagon. The distance between the end of the hexagon and the actual outlet is about 15 mm. Figure 4.3 below represents the geometry of the problem I solved in fluent.
For the two grids I made, I first meshed a XY face like the one in figure 4.3, and then I extruded this mesh through the whole volume. In these XY faces, I first meshed the disks of the inlet channel and the returned hole in the same way I meshed the circles in the previous case with the wedge primitive scheme in Gambit. Then, I finished meshing the whole XY faces with a triangular grid. Finally, I extruded the whole mesh through the complete volume of the wedge with the Gambit cooper scheme.

In order to study the grid independence of my results, I made two different grids. The first one is a classical mesh with regular space between nodes and the second one has more nodes in the boundary layer region. Globally, the classical mesh is less dense than the second mesh.

4.1.2 Boundary Conditions

First, all the material properties are exactly the same as before (the case with only one impingement jet). The User Defined function program is a little bit different because, in the wedge I simulated, there are two inlet faces. So, I had to create two different expressions for
the velocity at the beginning of the inlet channel. These two formulas below correspond respectively to the expression of the velocity profile for the bottom and top inlet.

\[
\begin{align*}
    u_{\text{bottom}}(r) &= 2 \cdot U_\infty \left(1 - \left(\frac{r}{r_0}\right)^2\right) \quad \text{with } r_0 = 250 \, \mu m \\
    u_{\text{top}}(x, y) &= 2 \cdot U_\infty \left(1 - \left(\frac{x^2 + (y - 0.001250)^2}{r_0^2}\right)\right)
\end{align*}
\]

where \(U_\infty = 1.8 \, m \cdot s^{-1} \) for \( Re = 1033 \) and \( U_\infty = 2.03 \, m \cdot s^{-1} \) for \( Re = 1160 \)

The first expression is exactly the same as the one I used in the one jet simulation because the inlet in the bottom of the wedge is in the centre of the hexagon. On the contrary, the second inlet is located just above the first one. That is why there is a difference in these two previous expressions.

Like in the one impingement jet simulation, I patched the fully developed solution in the two inlet channels in order to get a quicker convergence. The fully-developed profiles I used to patch correspond to the formulas above. I have used \( u_{\text{bottom}} \) for the bottom channel and \( u_{\text{top}} \) for the top channel.

Since I solve for a wedge, I have to put symmetry conditions on the two straight edges of the wedge. For the energy equation, I made all the walls adiabatic except the bottom wall, where I apply a constant heat flux. The value of this heat flux is found so that the water temperature reaches 370 K. Indeed, water boils at 373 K.

For all results, the length scale is \( D \) the inlet jet diameter, the velocity scale is the inlet velocity average, and the dimensionless temperature is defined by \( T^* = \frac{T - 372}{300 - 372} \).

### 4.2 Results for \( Re=1033 \)

The first thing I am interested in is the flow field in the micro jet cooling array only, and then I will compare some profiles of velocity and pressure obtained from the two different grids to see if these results are independent from the grid. Finally, I focused on the flow field
when a heat flux is applied on the bottom wall and also on the heat transfer features of the micro jet cooling array.

4.2.1 Flow Field Only

The flow inside the two inlet channels is fully developed flow inside a cylinder pipe. The velocity profile is parabolic, and its definition is the one I have used in my user define function program. Because we know before doing any simulation the flow result in this part of the micro jet cooling array, we decided to patch the volume of these two channels with the fully developed solution of a flow inside a pipe. The goal is to obtain a quicker convergence. Figure 4.4 shows the parabolic profile of the axial velocity inside the two inlet pipes.

![Figure 4.4: Axial Velocity (m·s⁻¹) on the Z = 40 μm Plane](image)

The location of blue spots in the figure above corresponds to the position of the returned ports. The intensity is low because this slide is just at the beginning of the micro jet cooling array, and the wall with the returned holes is at the end. The distance between this slide and the returned hole wall is 1560 μm. Besides, the outlet is located on the top of figure 4.4, and so the major part of the flow which exits the returned holes has to turn on the left to
reach the outlet. That is another reason why the velocity is not very important compared to the inlet one in this region.

Before discussing more on the features of the outlet flow, I will focus first on the impingement region, which is the actual heat transfer area. The length of the inlet channel is 1.6 mm and the flow starts changing just after the end of the inlet channels. Actually, the magnitude of the axial velocity is reduced, and there is creation of azimuthal and radial velocity. Indeed, the flow has to turn in the impingement region so as to reach the holes which are distributed around the inlet channels. Figure 4.5 describes the axial velocity distribution on a plane just after the end of the inlet pipes.

![Figure 4.5: Axial Velocity (m·s⁻¹) on the Z = 1710 μm Plane](image)

Just 110 μm after the middle wall, the average of the axial velocity magnitude is already reduced. When we come closer and closer to the bottom wall, the axial component of the velocity decreases. To balance, the azimuthal and radial components increase. This change is well illustrated especially on planes next to the surface being cooled. The three following figures show the flow on the Z = 2164.8 μm plane.
Figure 4.6: Azimuthal Velocity on the \( Z = 2164.8 \, \mu m \) Plane

Figure 4.7: Radial Velocity on the \( Z = 2164.8 \, \mu m \) Plane
Figure 4.8: Axial Velocity on the $Z = 2164.8 \mu m$ Plane

The last figure shows the two stagnation points which are located at the projection on the bottom wall of the inlet channel end. From the other two figures, we can deduce that the flow exits the inlet pipes like a fountain flow.

When the flow turns from the inlet channels to the returned holes like the one in a fountain, some vortices are created in all directions (azimuthal, axial and radial). A fluid particle rotates on itself when it moves from the inlet channel to a returned hole. There are also some vortices within the boundary layers of the bottom wall and the inlet channel walls.

The outlet flow has to circulate in the space where there are the inlet channels. For example, when you have a flow around a cylinder, there is always a wake behind it. So, in our case, a wake behind each cylinder is created because of the outlet flow. These wakes can cause a convergence problem if you put the outlet surface too close from the end of the hexagon. That is why I decided to put this surface far enough from the end of the hexagon. Figure 4.9 below describes the radial velocity distribution on a plane located in the middle between the beginning of the micro jet cooling array and the middle wall.
Figure 4.9: Radial Velocity \( (m \cdot s^{-1}) \) on the \( Z = 858 \mu m \) Plane in the Outlet Region

The two inlet channels are actually not displayed on this figure, but they are on the right below this figure. The flow exits only on the left part of the actual outlet surface. Indeed, the right part is on the wake of the inlet cylinder; it is a recirculation area. It is possible to see that the radial velocity is sometimes negative in the right part of the outlet. And, for example, if you put the outlet surface closer to the end of the hexagon, the major part of this surface can have a negative radial velocity and the continuity equation might be unsatisfied.

When flow just exits from the returned holes, its velocity is principally axial, but further away from the holes the velocity becomes in major part radial. Indeed, the flow has to turn slightly to reach the outlet surface. It takes more than half of the width of the outlet region to have the radial velocity more important than the axial one. Figures 4.10 and 4.11 show the axial and radial velocity, respectively, on the plane just above the returned holes \( (Z = 1450 \mu m \) plane) where the velocity is principally axial.
Figure 4.10: Axial Velocity ($m \cdot s^{-1}$) on the $Z = 1450 \, \mu m$ Plane

Figure 4.11: Radial Velocity ($m \cdot s^{-1}$) on the $Z = 1450 \, \mu m$ Plane
The radial velocity on the $Z = 374 \mu m$ plane is described in figure 4.12 below. In this plane, the radial component is the principal part of the velocity because the flow is far enough away from the bottom of the micro jet cooling array.

![Figure 4.12: Radial Velocity ($m \cdot s^{-1}$) on the $Z = 374 \mu m$ Plane](image)

The figure which shows the axial velocity on this plane looks like the same as figure 4.4. We can actually see that the flow goes radially to reach the outlet instead of axially.

### 4.2.2 Comparison of the Flow for two Different Grids

Before analyzing the heat transfer features of the micro jet cooling array based on my numerical simulation, I made a comparison between the flow field obtained with the regular mesh and the one from the denser mesh. If these two flow fields match, we can say that my results are grid independent. To compare precisely, I have extracted data (axial, radial and azimuthal velocity) along some lines on some extracted planes. The points shown on these different graphs do not correspond to the actual nodes of the grid. The following figures display the profiles of the three velocity components on two lines. The first three ones are
located at $Z = 1710 \mu m$ and $Y = 1024 \mu m$, and the second three ones are located at $Z = 1826 \mu m$ and $Y = 1508 \mu m$.

Figure 4.13: Comparison between the Denser and Regular Grid Axial Velocity Results at the Location $Z = 1710 \mu m$ and $Y = 1024 \mu m$

Figure 4.14: Comparison between the Denser and Regular Grid Radial Velocity Results at the Location $Z = 1710 \mu m$ and $Y = 1024 \mu m$
Figure 4.15: Comparison between the Denser and Regular Grid tangential Velocity Results at the Location $Z = 1710 \, \mu m$ and $Y = 1024 \, \mu m$

Figure 4.16: Comparison between the Denser and Regular Grid Axial Velocity Results at the Location $Z = 1826 \, \mu m$ and $Y = 1508 \, \mu m$
Figure 4.17: Comparison between the Denser and Regular Grid Radial Velocity Results at the Location $Z = 1826 \, \mu m$ and $Y = 1508 \, \mu m$

Figure 4.18: Comparison between the Denser and Regular Grid Tangential Velocity Results at the Location $Z = 1826 \, \mu m$ and $Y = 1508 \, \mu m$
With these two figures, we can deduce that the results from both grids are compatible. Moreover, the other profiles I extracted in other flow region give the same conclusion. So we can conclude that the results I got for the micro jet cooling array are grid independent. In the simulation I have made afterwards, I have used the mesh with regular space between nodes because the simulation takes less time for converging compared to the one with the denser mesh.

4.2.3 Flow Field and Heat Flux

First I am interested in the comparison between the “cold” flow field and the “hot” one. And then, I focused on the heat transfer rate we have obtained.

4.2.3.1 Comparison between Flow Field with and without Heat Flux Applied on the Bottom Wall

In order to make a comparison between these two flow fields, I created data profiles on different Z-planes for different lines at Y=constant. In these different profiles, I have extracted the three components of velocity (axial, azimuthal and radial). I used the same profile location as the one for the grid comparison. Like in the previous comparison, the points do not correspond to the actual nodes of the mesh.

The following figures show the velocity profiles for both cases at two different locations. The first three figures represent the three velocity components at \( Z = 1710 \, \mu m \) and \( Y = 1024 \, \mu m \) and the second three figures show the velocity components in the impingement region at \( Z = 1826 \, \mu m \) and \( Y = 1508 \, \mu m \). We have extracted some other profiles at different locations, for example in the outlet and inlet regions. All these profiles are coherent, and so they induce the same conclusion.

We can see that the values of the three velocity components are very close in both cases. So as a conclusion the flow field obtained in a case where there is a heat flux applied on the surface being cooled is the same as the one from a different case without any applied heat flux.
Figure 4.19: Comparison between Axial Velocity Results with and without Heat Flux on the Bottom Wall at the Location $Z = 1710 \mu m$ and $Y = 1024 \mu m$

Figure 4.20: Comparison between Radial Velocity Results with and without Heat Flux on the Bottom Wall at the Location $Z = 1710 \mu m$ and $Y = 1024 \mu m$
Figure 4.21: Comparison between Tangential Velocity Results with and without Heat Flux on the Bottom Wall at the Location $Z = 1710 \, \mu m$ and $Y = 1024 \, \mu m$

Figure 4.22: Comparison between Axial Velocity Results with and without Heat Flux on the Bottom Wall at the Location $Z = 1826 \, \mu m$ and $Y = 1508 \, \mu m$
Figure 4.23: Comparison between Radial Velocity Results with and without Heat Flux on the Bottom Wall at the Location $Z = 1826 \mu m$ and $Y = 1508 \mu m$

Figure 4.24: Comparison between Tangential Velocity Results with and without Heat Flux on the Bottom Wall at the Location $Z = 1826 \mu m$ and $Y = 1508 \mu m$
4.2.3.2 Heat Transfer Features

The first interesting thing in the heat transfer features of the micro jet cooling array is the temperature distribution on the surface being cooled. Figure 4.25 below displays this distribution on the bottom wall.

![Dimensionless Temperature Distribution on the Bottom Wall](chart)

Figure 4.25: Dimensionless Temperature Distribution on the Bottom Wall

We can see that the lowest temperature on this surface is located next to the two stagnation points. And we can verify that these two locations have the best heat transfer rate on the bottom wall. This verification is displayed on figure 4.26 that actually shows the bottom wall distribution of the heat transfer coefficient.

In addition, the figure 4.27 and figure 4.28 show respectively the heat transfer coefficient distribution and the Nusselt number distribution on the surface being cooled of the whole micro jet cooling array.
Figure 4.26: Heat Transfer Coefficient (\( W \cdot m^{-2} \cdot K^{-1} \)) Distribution on the Bottom Wall

Figure 4.27: Heat Transfer Coefficient (\( W \cdot m^{-2} \cdot K^{-1} \)) Distribution on the Bottom Wall for the whole Micro Jet Cooling Array
Moreover, the lowest heat transfer rate is located on the two top corner of the wedge because, in these regions, there is some recirculation. For this Reynolds number \( R_e = 1033 \), the heat transfer coefficient is between \( 8.4 \, W \cdot cm^{-2} \cdot K^{-1} \) and \( 10.35 \, W \cdot cm^{-2} \cdot K^{-1} \). For comparison, the best heat transfer coefficient for a cooling system with only one impingement jet is about \( 3.5 \, W \cdot cm^{-2} \cdot K^{-1} \). This value has been obtained with the formula which establishes the relationship between the heat transfer coefficient and the Reynolds number for a one jet cooling system. This formula is at the end of the chapter 3 (page 28). So, with the micro jet cooling array, we have a heat transfer rate 3.5 times greater than the one obtained with a one impingement jet system. Besides, the Nusselt number is between 60 and 76 and the Nusselt number average is 73.07.

Another interesting thing is the distribution of temperature within the whole micro jet cooling array. When we look at the temperature anywhere in the micro jet cooling array, we see that the temperature is lower than 305 K except for the area which is pretty close to the

![Image](image_url)

Figure 4.28: Nusselt Number Distribution on the Bottom Wall of the Micro Jet Cooling Array

[47]
bottom wall. Figures 4.29 and 4.30 show the temperature distribution on the $Z = 1710 \, \mu m$ and $Z = 1584 \, \mu m$ planes, respectively.

Figure 4.29: Dimensionless Temperature Distribution on the $Z = 1710 \, \mu m$ Plane

Figure 4.30: Dimensionless Temperature Distribution on the $Z = 1584 \, \mu m$ Plane

In these two figures above, we can see that the temperature above and below the middle wall is almost the same (within one degree Kelvin). So there is no interest to solve for
the energy equation inside the middle wall and the walls around inlet cylinders. It is better to leave them adiabatic.

4.3 Results for Re=1160

One of the first interesting results is the flow field feature and the difference with the Re=1033 flow field (especially the relationship between the pressure drop through the whole micro jet cooling array and the Reynolds number). Then, for the heat transfer features I compare between this case and the other case with Re=1033, especially the relationship between the heat transfer coefficient and the Reynolds number.

4.3.1 Flow Field Features

Like in the case with Re=1033, the flow inside the two inlet channels is just a regular fully developed profile in a cylindrical pipe. The major component of the velocity is the axial component, and figure 4.31 below shows its distribution right after the inlet.

![Figure 4.31: Axial Velocity (m/s) on the Z=40 μm Plane](image)

Figure 4.31: Axial Velocity (m·s⁻¹) on the Z = 40 μm Plane
We can effectively see on the figure above the parabolic profile on the two inlet channels. But, around the jets, at the exact locations of the returned holes, there are some dark blue spots of axial velocity. Indeed, when the flow exits from the returned holes the velocity is principally axial and then becomes radial. This plane is far from the middle wall, so the major component of the velocity is the radial component. On the contrary, for a plane just above the middle wall, the velocity is principally axial. The following picture displays the axial velocity distribution on a plane just above the middle wall (in fact 16 $\mu$m above). And we can actually see the fact that the axial velocity is the dominant component of velocity. On the other hand, the radial component is pretty small at this location.

**Figure 4.32: Axial Velocity ($m \cdot s^{-1}$) on the $Z = 1584 \mu m$ Plane**

In the outlet part of the micro jet cooling array, the fluid actually exits principally in the left part of the outlet face because the right part of this face is located at the end of the inlet cylinder wake. Indeed, this wake causes some recirculation in this area. In figure 4.33, we can see the distribution of the radial velocity in the outlet region.
The radial velocity distribution in the outlet region looks the same in both cases (Re=1033 and Re=1160). Now, we focus on the flow characteristics in the impingement region. In this part of the micro jet cooling array, like in the previous case, we have a fountain flow with some vortices especially close to the wall within its boundary layer. The vortices are important in the bottom wall boundary layer as well as in the beginning of the impingement region. In other words, the flow turns on itself and also in the same way as a fountain flow. The following two figures (4.34 and 4.35) display respectively the azimuthal and radial velocity on the $Z = 2164.8 \mu m$ plane to show the fountain flow.

In this figure 4.34, the azimuthal velocity is more important on the left of the middle inlet jet. Indeed, to reach the two returned ports which are on the right of the wedge, the velocity has two components: radial and azimuthal. On the contrary, for the port on the right
hand corner and the one just below the middle inlet jet, the velocity between the inlet jet and
the port is just radial as you can see on the figure 4.35 below.

Figure 4.34: Azimuthal Velocity \( (m \cdot s^{-1}) \) on the \( Z = 2164.8 \mu m \) Plane

Figure 4.35: Radial Velocity \( (m \cdot s^{-1}) \) on the \( Z = 2164.8 \mu m \) Plane
Before focusing on the heat transfer features of the micro jet cooling array at this Reynolds number, I am interested in the variation of the pressure drop through the whole micro jet cooling array with the Reynolds number. The pressure drop that you apply between the inlet and the outlet of the actual device governs the entire flow field. So it is an important parameter for the micro jet cooling array. Table 4.1 below shows the pressure loss coefficient in respect to the inlet Reynolds number.

Table 4.2: Pressure Drop through MJCA in respect to the Inlet Reynolds Number

<table>
<thead>
<tr>
<th>Reynolds Number</th>
<th>1033</th>
<th>1084</th>
<th>1136</th>
<th>1142</th>
<th>1148</th>
<th>1160</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \Delta P ) (Pa) between 1\textsuperscript{st} inlet and outlet</td>
<td>950</td>
<td>1038</td>
<td>1136</td>
<td>1146</td>
<td>1157</td>
<td>1179</td>
</tr>
<tr>
<td>( \Delta P ) (Pa) between 2\textsuperscript{nd} inlet and outlet</td>
<td>856</td>
<td>933</td>
<td>1018</td>
<td>1027</td>
<td>1037</td>
<td>1056</td>
</tr>
<tr>
<td>Pressure Drop (Pa)</td>
<td>903</td>
<td>984</td>
<td>1077</td>
<td>1087</td>
<td>1097</td>
<td>1117</td>
</tr>
<tr>
<td>( \frac{\Delta P}{\frac{1}{2} \cdot \rho \cdot U_{\infty}^2} )</td>
<td>0.5571</td>
<td>0.5513</td>
<td>0.5494</td>
<td>0.5487</td>
<td>0.5480</td>
<td>0.5465</td>
</tr>
</tbody>
</table>

I interpolated the values of this table in order to find an analytical relationship between the pressure loss coefficient and the inverse of the inlet Reynolds number. The equation below corresponds to this relationship.

\[
\frac{\Delta P}{\frac{1}{2} \cdot \rho \cdot U_{\infty}^2} = 90.07 \cdot \frac{1}{Re} + 0.4694
\]

where \( \Delta P \) is the pressure drop in Pa and Re is the Reynolds number. The correlation factor of this linear interpolation is about 0.9609. This linear relationship between the pressure drop through the device and the inlet Reynolds number agrees with the theoretical relationship for a laminar flow inside a pipe. (Introduction to Fluid Mechanics, 4\textsuperscript{th} edition,
Robert W. Fox and Alan T. McDonald, p348). The figure below shows the evolution of the pressure drop with the Reynolds number.

4.3.2 Heat Transfer Features

First, the distribution of temperature for this Reynolds number looks the same as the distribution in the first case. The maximum temperature is actually reached on the two top corners of the wedge which are recirculation areas. On the other hand, the minimum temperature and also the best heat transfer rate is located close to the two stagnation points. We can actually see that on Figure 4.37 below which represents the temperature distribution on the bottom wall of the micro jet cooling array.

Both figures below display the temperature distribution on the bottom wall of the micro jet cooling array. The first one is the distribution on the wedge and the last one on the whole micro jet cooling array.
Figure 4.37: Dimensionless Temperature Distribution on the Bottom Wall

Figure 4.38: Dimensionless Temperature Distribution on the Whole MJCA Bottom Wall
Now, we focus on the heat transfer rate of the micro jet cooling array for this particular Reynolds number. But before displaying the distribution of the heat transfer coefficient on the bottom wall, I make the heat transfer coefficient average comparison between the two cases. In the previous case where the Reynolds number equals to 1033 the heat transfer coefficient value is between $84000 \, W \cdot m^{-2} \cdot K^{-1}$ and $103500 \, W \cdot m^{-2} \cdot K^{-1}$, and thus, the average heat transfer coefficient equals to $99374 \, W \cdot m^{-2} \cdot K^{-1}$. For the case with Re=1160, the values are between $88000 \, W \cdot m^{-2} \cdot K^{-1}$ and $104400 \, W \cdot m^{-2} \cdot K^{-1}$. The average, in this case, equals to $101109 \, W \cdot m^{-2} \cdot K^{-1}$.

I found a relationship between the heat transfer coefficient and the inlet Reynolds number. The shape of this relationship is exactly the same as the one I found for the one impingement jet cooling system. The generic equation of this relationship is just below.

$$\bar{h} = \alpha \cdot (Re)^{\beta}$$

where $\bar{h}$ is the average heat transfer coefficient on the bottom wall, $Re$ is the Reynolds number, and $\alpha$ and $\beta$ are two positive constants to be determined.

The two constants $\alpha$ and $\beta$ are the solutions of a two equations-two unknowns system. The values for $\alpha$ and $\beta$ and the final relationship are just below.

$$\alpha = 35245 \, W \cdot m^{-2} \cdot K^{-1}$$
$$\beta = 0.149$$
$$\bar{h} = 35245 \cdot (Re)^{0.149}$$

Besides, the distribution of the Nusselt number looks the same as the heat transfer coefficient distribution. The values of this number vary from 64 to 77 and the average value equals to 74.35. The relationship between the Nusselt number and the inlet Reynolds and Prandtl numbers is established.

$$N_u = 13.496 \cdot (P_r)^{\frac{1}{2}} \cdot (Re)^{0.1497} \text{ where } P_r = 7.02 \text{ at the inlet}$$
Figure 4.39 displays the distribution of the Nusselt number on the whole micro jet cooling array bottom wall.

4.4 Results for a Micro Jet Cooling Array with a Smaller Impingement Region

In the previous part, I studied the heat transfer and flow features of the micro jet cooling array for different values of the Reynolds number. The two interesting parameters are the pressure drop through the whole device and the heat transfer coefficient on the bottom wall. Now, we have changed a geometric feature of the micro jet cooling array: we reduce the size of the impingement region. The distance between the surface being cooled and the middle wall is decreased to \( \frac{d}{4} \) instead of \( d \) where \( d \) is the diameter of the inlet channel. This is to study the evolution of the important parameters when a characteristic dimension of the micro jet cooling array is changed.

4.4.1 Evolution of the Problem Set Up

From the previous set up of the problem, I kept everything except the impingement region. First, I removed the mesh in this area, and then, I changed the geometry in reducing the distance between the bottom wall and the middle wall. Secondly, I again grid this area in
extruding the mesh already in the XY faces with the cooper scheme in Gambit. Indeed, if the XY face meshes change, we have to mesh everything again. I have reduced the number of nodes in the longitudinal direction by a factor 4. That is done to keep the aspect ratio of the elements identical because this parameter of the mesh is a very critical parameter for a numerical simulation. By definition, the aspect ratio is the ratio of the cell length over the cell width. The aspect ratio of numerical cells plays an important role in the convergence.

Finally, in Fluent, I set up the problem in the same way as before. I used the same boundary conditions. The same UDF program is used for the definition of the inlet velocity, the material properties remain the same and all the wall are adiabatic except the bottom wall where a constant heat flux is applied. But because of convergence problem, I started the simulation for $Re = 269$, and then I increased step by step the Reynolds number in order to reach 1033 to make a comparison between the two geometries. When the final Reynolds number is reached, we applied a constant heat flux on the bottom wall. Figure 4.40 below displays the new geometry of the system in a longitudinal cut (XZ face). Everything remains the same except the width $H$ of the impingement region which is equalled to $125 \mu m$.

![Figure 4.40: Longitudinal Cut of the Micro Jet Cooling Array with a Smaller Impingement Region](image-url)
4.4.2 Flow Field Features

In a general way, the flow within this new micro jet cooling array has the same characteristics as before. For instance, in the impingement region, we have a fountain flow as well. But, since the region where the flow can turn is smaller, the values of azimuthal and radial velocity are a little bit higher. Moreover, the change from axial to radial and azimuthal velocity is quicker for the case where the impingement region is smaller. In less than one hundred microns, the velocity is principally radial and azimuthal as we can see in the following figures which show the three velocity components in the \( Z = 1800 \ \mu m \) plane.

![Figure 4.41: Axial Velocity (\( m \cdot s^{-1} \)) on the \( Z = 1800 \ \mu m \) Plane](image)
Figure 4.42: Azimuthal Velocity ($m \cdot s^{-1}$) on the $Z = 1800 \, \mu m$ Plane

Figure 4.43: Radial Velocity ($m \cdot s^{-1}$) on the $Z = 1800 \, \mu m$ Plane
From the \( Z = 1750 \mu m \) plane, the axial velocity at the returned holes location starts increasing, and on the other hand, the azimuthal and radial velocity decrease. After the middle wall, in the outlet region, the axial component starts again decreasing, and at the same time the radial velocity increases.

Another interesting thing in the flow field features of the micro jet cooling array is the evolution of the pressure drop through the device with the Reynolds number. The Table 4.2 below shows the pressure loss coefficient in respect to the Reynolds number.

Table 4.3: Pressure Drop through the Device in respect to the Inlet Reynolds Number

<table>
<thead>
<tr>
<th>Reynolds number</th>
<th>269</th>
<th>314</th>
<th>358</th>
<th>448.5</th>
<th>538.2</th>
<th>627.9</th>
<th>717.6</th>
<th>807.3</th>
<th>1033</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \Delta P ) (Pa) between 1\textsuperscript{st} inlet and outlet</td>
<td>240</td>
<td>290</td>
<td>331</td>
<td>446</td>
<td>588</td>
<td>759</td>
<td>958</td>
<td>1189</td>
<td>1431</td>
</tr>
<tr>
<td>( \Delta P ) (Pa) between 2\textsuperscript{nd} inlet and outlet</td>
<td>214</td>
<td>257</td>
<td>290</td>
<td>390</td>
<td>514</td>
<td>659</td>
<td>822</td>
<td>1015</td>
<td>1199</td>
</tr>
<tr>
<td>Pressure drop Average (Pa)</td>
<td>227</td>
<td>273.5</td>
<td>310.5</td>
<td>418</td>
<td>551</td>
<td>709</td>
<td>890</td>
<td>1102</td>
<td>1315</td>
</tr>
<tr>
<td>( \frac{\Delta P}{\frac{1}{2} \cdot \rho \cdot U_x^2} )</td>
<td>2.06</td>
<td>1.83</td>
<td>1.59</td>
<td>1.37</td>
<td>1.25</td>
<td>1.18</td>
<td>1.14</td>
<td>1.11</td>
<td>0.81</td>
</tr>
</tbody>
</table>

Like I did for the first case, I interpolated the values of this table to find the relationship between the Reynolds number and the pressure loss coefficient.

\[
\frac{\Delta P}{\frac{1}{2} \cdot \rho \cdot U_x^2} = 408.71 \cdot \frac{1}{Re} + 0.5094
\]

where \( \Delta P \) is the pressure drop in Pa, \( Re \) is the Reynolds number and the correlation factor is 0.9754.
The following figure 4.44 displays the evolution of the pressure loss coefficient with the inlet Reynolds number.

![Pressure Loss Coefficient vs Reynolds number](image)

Figure 4.44: Pressure Loss Coefficient in respect to the Inlet Reynolds Number

### 4.4.3 Heat Transfer Features

The first thing that is interesting is the distribution of temperature in the surface being cooled where a heat flux is applied. In this particular case, the maximum heat flux that can be removed by the device without boiling at any location is equalled to $650 \, W \cdot cm^{-2}$.

Figure 4.45 below represents the temperature distribution on the bottom wall of the whole micro jet cooling array. In this picture, we can see the locations of the highest temperature and heat transfer rate.

As we can see on the figure below, this distribution looks the same as the one obtained with the first micro jet cooling array. The hottest points are located close to the external wall, and the lowest ones are next to the stagnation points just below each impingement jet.
Figure 4.45: Dimensionless Temperature Distribution on the Whole MJCA Bottom Wall

But, there is an important difference with the previous micro jet cooling array. In the temperature distribution obtained for the first device, the temperature values are between 356K and 371K, and in this case, the temperature values are between 346K and 371K. That is why the highest heat transfer coefficient value is better for the micro jet cooling array with a smaller impingement region and also why the heat transfer coefficient average is more important for the second case. On the other hand, the heat flux that is applied on the bottom wall is lower for the case with the smaller impingement region. For instance, the heat flux equals to $700 \, W \cdot cm^{-2}$ for the first case and $650 \, W \cdot cm^{-2}$ for the smaller impingement region case. Indeed, the highest temperature is reached faster in the recirculation area because the distance is smaller and the intensity of the recirculation is more important. The distribution of the Nusselt number is shown in figure 4.46 below.
In this case, the heat transfer coefficient is between $7.4 \, W\cdot cm^{-2}\cdot K^{-1}$ and $11.5 \, W\cdot cm^{-2}\cdot K^{-1}$. Comparatively, in the first case, the values of the heat transfer rate are in the following interval $[8.4, 10.35] \, W\cdot cm^{-2}\cdot K^{-1}$. So, the maximum heat transfer rate is higher for the smaller impingement region case, and the minimum rate is lower for the same case. But, the area where the heat transfer rate is maximum is more important than the surface where the rate is minimum. That is why the heat transfer coefficient average is basically more important for the smaller impingement case. In the table below I summarize all the heat transfer results for the two previous cases. There are the heat flux applied on the bottom wall, the heat transfer coefficient and the Nusselt number average for both cases.

<table>
<thead>
<tr>
<th>Bottom Wall Heat Flux ($W\cdot cm^{-2}$)</th>
<th>Heat Transfer Coefficient Average ($W\cdot cm^{-2}\cdot K^{-1}$)</th>
<th>Nusselt Number Average</th>
</tr>
</thead>
<tbody>
<tr>
<td>Regular Impingement Region Case</td>
<td>700</td>
<td>9.937</td>
</tr>
</tbody>
</table>

(Table Continued)
### 4.5 Results for the Micro Jet Cooling Array without the Return Holes

In all the cases I have previously simulated there is a wall which actually separates the inlet and outlet region in one side and the impingement region in the other. On this wall, there are holes for the inlet channels but also for the returning flow. Indeed, the flow returns from the impingement region to the outlet through these returned holes.

In this part, I tested a micro jet cooling array without any hole for the returning flow. The flow exits the impingement region by the entire previous wall surface except the area used for the inlet channels. The figure below shows the actual middle wall.

![Transversal Cut of the New Wedge](image)

The size of the impingement region is the same as the one for the first case because the heat flux that can be removed from the micro chip is a little bit higher.
4.5.1 Change in the Problem Set Up

In this new problem I solved I just changed one thing from the first case. I created a zone for the middle wall itself. In doing that, I can specify this new zone as a fluid zone instead of a solid area.

Everything else in the set up of the problem remains the same. I used everything I already used for the first case.

4.5.2 Flow Field Features

The principal difference between this case without the returned holes and the basic micro jet cooling array is on the flow features. As you could see in the next part, the heat transfer features of both cases are very similar.

One of the first differences is about the pressure drop through the whole device. In the basic micro jet cooling array, the pressure drop is approximatively 900 Pa, and, in this case, it is about 300 Pa. So, the pump we need to drive the flow through the device has to be more important in the case with the returned holes.

The second difference deals with the distribution of axial component of velocity. The flow exits principally the impingement region on the left part of the wedge, that is to say the furthest away from the two inlet channels. There is quite no axial velocity between the two channels. This feature of the flow field without holes is shown on the following two figures which display the axial velocity distribution on the $Z = 1650 \ \mu m$ and $Z = 1584 \ \mu m$ planes, respectively.

Another difference is the radial velocity distribution in the outlet region. Indeed, the flow exits on the right part of the outlet face instead of exiting on the left. It is simply because there is no flow between the two inlet channels, so the wake of the cylinders is much smaller. Basically, the flow exits on the left part of the wedge, and when it comes in the outlet region,
it turns to the right part of the outlet area. Moreover, the flow starts turning above the $Z = 616 \, \mu m$ plane. Before that plane, the velocity is principally axial.

![Figure 4.48: Axial Velocity ($m \cdot s^{-1}$) on the $Z = 1650 \, \mu m$ Plane](image1)

![Figure 4.49: Axial Velocity ($m \cdot s^{-1}$) on the $Z = 1584 \, \mu m$ Plane](image2)
On these two figures below, the flow exits on the left part of the wedge in the impingement region, and then it turns to the right part when it arrives in the outlet part of the device. The next two figures show respectively the radial velocity on the $Z = 132 \, \mu m$ and $Z = 374 \, \mu m$ planes.

Figure 4.50: Radial Velocity ($m \cdot s^{-1}$) on the $Z = 132 \, \mu m$ Plane

Figure 4.51: Radial Velocity ($m \cdot s^{-1}$) on the $Z = 374 \, \mu m$ Plane
Figure 4.52 shows the azimuthal velocity on the $Z = 132 \, \mu m$ plane. It is another way to see the flow when it turns.

4.5.3 Heat Transfer Features

The heat transfer features of the micro jet cooling array are not quite affected by the elimination of the returned holes from the middle wall. Indeed, the heat flux that we can apply on the bottom wall is equalled to $690 \, W \cdot cm^{-2}$ instead of $700 \, W \cdot cm^{-2}$ for the basic device with the returned holes. Moreover, the heat transfer coefficient average on the bottom wall equals to $9.89 \, W \cdot cm^{-2} \cdot K^{-1}$ which is pretty close to the average of the heat transfer coefficient for the basic micro jet cooling array ($\bar{h}_{basic\,mjca} = 9.94 \, W \cdot cm^{-2} \cdot K^{-1}$). The difference between these two values is about 0.5%.

Besides, the Nusselt number average is 72.72, and its distribution on the bottom wall of the micro jet cooling array is the same as the heat transfer coefficient distribution. This distribution is displayed on figure 4.53 below.
The following figure represents the evolution of temperature on the bottom wall of the device.
4.6 Comparison between Volume Heat Source and Surface Heat Flux Results for the Micro Jet Cooling Array

In all the simulations I ran before, I applied a constant surface heat flux on the bottom wall. But, in reality, the micro chip beneath the bottom wall which has to be cooled by the micro jet cooling array has a certain thickness. So the micro chip releases an amount of heat per unit volume which is removed by the micro jet cooling array. Instead of applying a surface heat flux on the bottom wall, it might be better to apply an amount of heat per unit volume.

I keep constant in both cases the heat power (in Watt) released by the micro chip so I can make a comparison between the surface heat flux and volume heat source results. Table 4.5 summarizes the values I have used to set up the heat transfer in this new problem.

<table>
<thead>
<tr>
<th></th>
<th>Surface Heat Flux</th>
<th>Volume Heat Source</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bottom Wall Surface (m²)</td>
<td>9.3406 \times 10^{-7}</td>
<td>9.3406 \times 10^{-7}</td>
</tr>
<tr>
<td>Thickness of the Bottom Wall (mm)</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>Volume (m³)</td>
<td>Not defined</td>
<td>9.3406 \times 10^{-10}</td>
</tr>
<tr>
<td>Power (Watt)</td>
<td>6.5384</td>
<td>6.5384</td>
</tr>
<tr>
<td>Heat</td>
<td>7 \cdot 10^6 W \cdot m^2</td>
<td>7 \cdot 10^9 W \cdot m^3</td>
</tr>
</tbody>
</table>
| Properties of the Wall Material (Aluminium) | Not defined | $C_p = 871 \, J \cdot kg^{-1} \cdot K^{-1}$  
                                         |                    | $k = 202.4 \, W \cdot m^{-1} \cdot K^{-1}$ |

To compare these two cases I have extracted some velocity and temperature profiles on different planes. Figure 4.55 makes a comparison of the temperature distribution on the bottom wall.
Figure 4.55: Comparison of Temperature (K) Profiles on the Bottom Wall

We can easily see that the temperature distribution on the bottom wall is exactly the same in both cases. For the other temperature profiles within the whole device I extracted, there is almost no difference between the two cases. The next figure displays the comparison of the velocity components on a certain profile.

Figure 4.56: Comparison of the Three Components of Velocity at $Z = 2164 \, \mu m$ and $Y = 1496.2 \, \mu m$
For the velocity components, we can deduce the same conclusion as for temperature. In other words, we can say that both ways to set up the heat transfer on the bottom wall have very similar results.

To verify this conclusion, another simulation has to be run in a future work. A solid volume will be added beneath the bottom wall and a volume heat source will be applied inside this volume. Then a comparison will be made between this case with the new heat flux set up and the two cases just above.

4.7 Comparison between Experimental and Numerical Results

Some experiments have been already made for different flow rates. The following figure displays these experimental results with the numerical results.

![Comparison between Experimental and Numerical Results](image)

Figure 4.57: Comparison between Experimental and Numerical Results

We can actually see that the order of magnitude of the heat transfer coefficient from the numerical results is the same as the one obtained from the experimental results. We can make a complete comparison for the case $\frac{H_{\text{imp}}}{D_{\text{in}}} = 1$ and $\frac{d_{\text{hole}}}{D_{\text{in}}} = 0.7$. And we can conclude
that the numerical results are close to the experimental results when the geometrical parameters are exactly the same.

The following table summarizes the geometrical parameters of the different experimental cases.

<table>
<thead>
<tr>
<th>Table 4.6: Geometrical Parameters for the Experimental Cases</th>
</tr>
</thead>
<tbody>
<tr>
<td>$H_{imp}$ (µm)</td>
</tr>
<tr>
<td>$D_{in}$ (µm)</td>
</tr>
<tr>
<td>$d_{hole}$ (µm)</td>
</tr>
<tr>
<td>$\frac{H_{imp}}{D_{in}}$</td>
</tr>
<tr>
<td>$\frac{d_{hole}}{D_{in}}$</td>
</tr>
</tbody>
</table>

4.8 Conclusion

In this part, I have simulated different kind of geometry for the micro jet cooling array. These different devices have some advantages and disadvantages and in the next part I will discuss about that in order to know which case is the best compromise in terms of heat transfer rate. Moreover I have made the comparison between the flow field obtained with heat flux on the bottom wall and the one without any heat flux. This comparison is interesting for the experimental part which will be done in a future work.
Chapter 5 Discussion and Future Work

This part deals with the discussion between the different geometry I have tested for the micro jet cooling array. First, we focus on the best size for the impingement region which is the location of the heat transfer in the micro jet cooling array. Secondly, we put our interest on another parameter (fluid or solid for the middle wall) which plays an important role on the flow field and the heat transfer features. And finally, we conclude on the future work which will be done on this project.

5.1 Discussion on the Best Size for the Impingement Region

In the discussion about the size of the impingement region, we focus on three points of interest. The first one is related to the flow field only, indeed it is the pressure drop through the whole device. The second one is the heat transfer coefficient on the bottom wall of the micro jet cooling array. And the last one is the value of the surface heat flux that we apply on the surface being cooled.

First, when the distance between the bottom and middle wall is $d$ ($d$ corresponds to the diameter of the inlet jets), the pressure drop through the device is equal to 903 Pa. In comparison, when the distance is smaller (actually $\frac{d}{4}$), the pressure drop increases to be equal to 1315 Pa. This corresponds to a relative increase of 35%. Practically, the mechanical pump which has to drive the flow through the device has to be more powerful for the same flow rate in the case where the distance is smaller. So we need more electrical power to use the device with the smaller impingement region. The cost of the micro jet cooling array when we use it increases because of that important increase in the pressure drop.

Secondly, the heat transfer rate is a little bit better when the impingement region is smaller. Indeed, the heat transfer coefficient average on the bottom wall is equal to $9.9 \, W \cdot cm^{-2} \cdot K^{-1}$ with the regular size for the impingement region; in the other case, the
heat transfer coefficient average is $10.63 \, W \cdot cm^{-2} \cdot K^{-1}$. We basically have a little gain in the heat transfer rate, but we lose some efficiency with the increase of the pressure drop.

Moreover, the total surface heat flux applied on the bottom wall is lower for the small impingement region case. Indeed, in this last case, the total heat flux is equal to $650 \, W \cdot cm^{-2}$ instead of $700 \, W \cdot cm^{-2}$ for the regular device. That might be due to the fact that the recirculation area in the top corner of the wedge is more intense, and thus the maximum temperature is reached faster.

Therefore, we can say that the first device is a better system for cooling than the small impingement region micro jet cooling array. That is the reason why the simulation for the test of the second parameter has been based on the first device for the size of the impingement region.

5.2 Discussion on the Interest of Return Holes

First, we can say that there is almost no decrease or increase in the heat transfer rate on the bottom wall between the micro jet cooling array with and without the returned holes. Indeed, for the device without the returned holes, the heat transfer coefficient average is equal to $9.89 \, W \cdot cm^{-2} \cdot K^{-1}$; and for the basic device, the heat transfer coefficient average is $9.94 \, W \cdot cm^{-2} \cdot K^{-1}$.

Secondly, if the flow exits the impingement region by the entire middle wall surface, the pressure drop decreases. Indeed, the pressure drop in this last case is $300 \, Pa$ which is much lower than the one in the regular device ($\Delta P_{basic \, device} = 900 \, Pa$). As I said previously, it can be a great advantage to have a low pressure drop because the mechanical pump does not need to be very powerful for driving the flow.

But, for the case without the returned holes, the total heat flux applied on the bottom wall is a little bit lower than the one applied on the bottom surface of the regular micro jet
cooling array. For instance, the surface heat flux on the case without the returned holes is equalled to \( 690 \, W \cdot cm^{-2} \) instead of \( 700 \, W \cdot cm^{-2} \) for the basic micro jet cooling array.

Therefore, we have a loss of 0.5% for the heat transfer rate and 1.5% for the surface heat flux applied on the bottom wall if the flow exits on the entire surface of the middle wall. But, the pressure drop through the whole device is very much lower in this new case. This new geometry might be a good compromise for the next micro jet cooling array.

5.3 Future Work

In the future work, some experiments have to be made on the micro jet cooling array. These experiments consist principally of making some flow field visualizations with the micro particle image velocimetry. These visualizations show the “cold” flow field that is to say without any flux applied on the bottom wall because the bottom wall is replaced by a transparent surface (the light can pass through this surface for the visualizations) where we cannot apply any heat flux. But we can say that the “hot” and “cold” flow field are very similar. Indeed, I have made a comparison in the chapter 4 (pages 38 and 39) between these two flow fields and the conclusion is that there is no significant difference. So every visualization on the “cold” flow field would be the same as the ones on the “hot” flow field if we could make visualizations on the “hot” flow.

These experiments will check the results I found on my simulations to confirm or maybe cancel the results. Then, some conclusions will be drawn to optimize the performance of the micro jet cooling array.

Finally, on the new device, some numerical simulations and experiments will be run in order to confirm the optimization of the heat transfer performance.
Chapter 6 Conclusion

The numerical simulation on the surface cooling with one impingement jet has been made as a preliminary test. That gives us a basic idea of impingement cooling. For instance, we know the shape of the flow within the cooling device and the heat transfer rate that we can actually expect. In addition, we have some values in order to make a comparison between one jet cooling system and the micro jet cooling array.

First, the simulation on the basic module of the micro jet cooling array shows the complete flow field in the hexagon with and without heat flux applied on the bottom wall. These two flow fields are very similar, so the experiments on the “cold” flow field will be valid for the flow field obtained with a heat flux applied on the bottom wall.

Secondly, we know a relationship between the inlet Reynolds number and the average of the heat transfer coefficient on the surface being cooled, and also a relationship between the inlet Reynolds number and the pressure drop through the device. These two relationships help to find the best geometry with the best heat transfer rate and the lowest pressure drop. I have tested two different geometries for the micro jet cooling array.

The first one is a micro jet cooling array with a smaller impingement region. The width of this region is divided by four compared to the one in the basic device. But, in this new case, the pressure drop increases and there is no significant increase in the heat transfer rate. So the basic micro jet cooling array is a better compromise.

The second geometry is based on the basic device except that the middle wall is transformed into a fluid zone. In other words, the flow exits the impingement region by the entire middle wall surface. In this case, there is almost no decrease in the heat transfer rate compared to the one obtained with the regular geometry. But the pressure drop through the
whole device decreases. So this last geometry is, based on the numerical simulation, the best compromise.

Finally, these conclusions will be verified by the experiments which will be made on the micro jet cooling array in a future work.
References


7. Steven A Soper, Sean M Ford, Shize Qi, Robin L. McCarley, Kevin Kelly, Michael C. Murphy, “Polymetric Microelectromechanical Systems”, Analytical Chemistry Report, October 1, 2000


15. “Fluorescent dyed Microspheres” TechNote 103, Rev.003, Active 15/MAR/2001 Bangs laboratories


18. Mortimer Abramowitz, “Microscope Basics and Beyond”, Volume 1, for Olympus Corporation, 1985


Appendix A: UDF Function for One Impingement Jet

#include "udf.h" /* must be at the beginning of every UDF you write */

DEFINE_PROPERTY (cell_water_dyn_vis, cell, thread)
{
    /*Definition of the viscosity formula in function of temperature*/
    real mu_lam;
    real temp = C_T (cell, thread);
    mu_lam=0.001792*exp(-1.94-.8*(273.15/temp)+6.74*(273.15/temp)*(273.15/temp));
    return mu_lam;
}

DEFINE_PROFILE(inlet_velocity, thread, index)
{
    /* loops over all faces in the thread passed in the DEFINE macro argument */
    real x[ND_ND]; /* this will hold the position vector */
    real y;
    real z;
    face_t f;
    begin_f_loop(f, thread)
    {
        F_CENTROID(x,f,thread);
        y = x[0];
        z = x[1];
        F_PROFILE(f, thread, index)=*2*0.125*(1-y*y+z*z)/(0.002*0.002));
    }
    end_f_loop(f, thread)
}
Appendix B: UDF for the Micro Jet Cooling Array

#include "udf.h"  /* must be at the beginning of every UDF you write */

DEFINE_PROPERTY(cell_water_dyn_vis, cell, thread)
{
    real mu_lam;
    real temp = C_T(cell, thread);
    mu_lam = 0.001792*exp(-1.94-4.8*(273.15/temp)+6.74*(273.15/temp)*(273.15/temp);
    return mu_lam;
}

DEFINE_PROFILE(inlet_velocity_1, thread, index)
{
    real x[ND_ND];  /* this will hold the position vector */
    real y;
    real z;
    face_t f;
    /* loops over all faces in the thread passed in the DEFINE macro argument */
    begin_f_loop(f, thread)
    {
        F_CENTROID(x,f,thread);
        y = x[0];
        z = x[1];
        F_PROFILE(f, thread, index) =2*1.8*(1-(y*y+z*z)/(0.00025*0.00025))/0.00025*0.00025);
    }
    end_f_loop(f, thread)
}

DEFINE_PROFILE(inlet_velocity_2, thread, index)
{
    real x[ND_ND];  /* this will hold the position vector */
    real y;
    real z;
    face_t f;
begin_f_loop(f, thread)
{
    F_CENTROID(x,f,thread);
y = x[0];
z = x[1];
    F_PROFILE(f, thread, index) = *2*1.8*(1-(y*y+(z-0.001250)*(z-0.001250))/(0.00025*0.00025));
}
end_f_loop(f, thread)
### Appendix C Thermal and Mechanical Properties of Liquid Water

Table C1: Variation of Density, Kinematic Viscosity and Thermal Conductivity of Liquid Water with Temperature

<table>
<thead>
<tr>
<th>TEMPERATURE (K)</th>
<th>$\rho, , kg \cdot m^{-3}$</th>
<th>$k, , W \cdot m^{-1} \cdot K^{-1}$</th>
<th>$\nu, , m^2 \cdot s^{-1}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>273.15</td>
<td>1002.28</td>
<td>0.552</td>
<td>$1.788 \cdot 10^{-6}$</td>
</tr>
<tr>
<td>293.15</td>
<td>1000.52</td>
<td>0.597</td>
<td>$1.006 \cdot 10^{-6}$</td>
</tr>
<tr>
<td>313.15</td>
<td>994.59</td>
<td>0.628</td>
<td>$0.658 \cdot 10^{-6}$</td>
</tr>
<tr>
<td>333.15</td>
<td>985.46</td>
<td>0.651</td>
<td>$0.478 \cdot 10^{-6}$</td>
</tr>
<tr>
<td>353.15</td>
<td>974.08</td>
<td>0.668</td>
<td>$0.364 \cdot 10^{-6}$</td>
</tr>
<tr>
<td>373.15</td>
<td>960.63</td>
<td>0.680</td>
<td>$0.294 \cdot 10^{-6}$</td>
</tr>
<tr>
<td>393.15</td>
<td>945.25</td>
<td>0.685</td>
<td>$0.247 \cdot 10^{-6}$</td>
</tr>
</tbody>
</table>

Table C.2: Variation of Specific Heat of Liquid Water with Temperature

<table>
<thead>
<tr>
<th>TEMPERATURE (K)</th>
<th>$C_p, , kJ \cdot kg^{-1} \cdot K^{-1}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>273.15</td>
<td>4.2174</td>
</tr>
<tr>
<td>278.15</td>
<td>4.2019</td>
</tr>
<tr>
<td>283.15</td>
<td>4.1919</td>
</tr>
<tr>
<td>288.15</td>
<td>4.1855</td>
</tr>
<tr>
<td>293.15</td>
<td>4.1816</td>
</tr>
<tr>
<td>298.15</td>
<td>4.1793</td>
</tr>
<tr>
<td>303.15</td>
<td>4.1782</td>
</tr>
<tr>
<td>308.15</td>
<td>4.1779</td>
</tr>
<tr>
<td>313.15</td>
<td>4.1783</td>
</tr>
</tbody>
</table>

(Table Continued)
<table>
<thead>
<tr>
<th>Temperature (K)</th>
<th>Density ($kg \cdot m^{-3}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>318.15</td>
<td>4.1792</td>
</tr>
<tr>
<td>323.15</td>
<td>4.1804</td>
</tr>
<tr>
<td>328.15</td>
<td>4.1821</td>
</tr>
<tr>
<td>333.15</td>
<td>4.1841</td>
</tr>
<tr>
<td>338.15</td>
<td>4.1865</td>
</tr>
<tr>
<td>343.15</td>
<td>4.1893</td>
</tr>
<tr>
<td>348.15</td>
<td>4.1925</td>
</tr>
<tr>
<td>353.15</td>
<td>4.1961</td>
</tr>
<tr>
<td>358.15</td>
<td>4.2002</td>
</tr>
<tr>
<td>363.15</td>
<td>4.2048</td>
</tr>
</tbody>
</table>

Figure C.1: Variation of Density ($kg \cdot m^{-3}$) with Temperature (K)
Figure C.2: Thermal Conductivity ($W \cdot m^{-1} \cdot K^{-1}$) Variation with Temperature (K)

Figure C.3: Dynamic Viscosity ($kg \cdot m^{-1} \cdot s^{-1}$) Variation with Temperature (K)
Figure C.4: Specific Heat \( (J \cdot kg^{-1} \cdot K^{-1}) \) Variation with Temperature
Vita

Pierre was born on July, 23rd 1980 in Aurillac, France. He obtained his “Baccalaureat S” (High School degree in mathematics and physics) with honors in July 1998.

After three years of post-secondary program leading to the nation-wide competitive examinations to “Grandes Ecoles”, he entered L’Ecole Nationale Supérieure d’Ingénieurs en Constructions Aéronautiques, an aerospace engineering school located in Toulouse, France. He convinced his school to let him study abroad for his last year as an exchange student in order to get a Master of Science in Mechanical Engineering.

Pierre enrolled a Master of Science Program in Mechanical Engineering in August 2003 at the Louisiana State University in the Mechanical Engineering Department with Dr. Dimitris E. Nikitopoulos as major professor. His research focused on numerical simulations of a micro jet cooling array. He will be graduating in May 2005.