Table of contents

CHAPTER 1

1. INTRODUCTION

Computational Fluid Dynamics (CFD) is the branch of fluid mechanics science that predicting the fluid flow, transfer of heat, transfer of mass, chemical reactions and related activities by solving the numerical equations which obey these processes using a numerical simulation process. CFD analyses is relevant data used in: :

(i) Conceptual and systematic studies of new designs.

(ii) Detailed data of product development.

(iii) Troubleshooting in the system..

(iv) Redesign computational analysis complements data and experimentation.

(v) Minimize the total effort required in laboratory work . .

Analysis and Design

1. Numerical Simulation-based design instead of build and test
   (i) Cost effective and very rapid than EFD.
   (ii) CFD provides high-fidelity database study for analyzing flow field.
2. Numerical Simulation of physical fluid phenomena that are more difficult for experiments.
   (i) Full simulations (e.g., ships and airplanes).
   (ii) Environmental effects on the system (wind, weather, etc.).
   (iii) Hazards by the system (e.g., explosions, radiation, pollution).
   (iv) Physics of the system (e.g., planetary boundary layer, stellar evolution).

Where is CFD used?

1. Aerospace system
2. Automotive system
3. Biomedical system
4. Chemical Processing system
5. HVAC system
6. Hydraulics system
7. Marine system
8. Oil and Gas application
9. Power Generation system
10. Electronics system at micro level system

Steps to CFD

Step 1. Separate the fluid volume (surface) up into manageable chunks (gridding).

Step 2. Simplify and solved the equations to be solved.

Step 3. Fix boundary conditions.

Step 4. Proceed the other grid values.

Step 5. Steps through the grid establishing that at the grid points and nearest neighbours, these equations are satisfied .

Advantages of CFD

1. **Less Cost**
   - Using tangible experiments and tests to achieve vital engineering results for design can be extravagant.
   - Deterministic numerical simulations are relatively less costly, and expenses are likely to reduce as computers become robust and dynamic.
2. **Speed**
   - CFD numerical simulations can be executed in less time.
   - Quick means engineering inputs can be imported early in the design process system .
3. **Ability to Simulate Real Conditions**
   - Some flow and heat transfer processes can not be (easily) tested - e.g. hypersonic flow.

Types of CFD codes

1. Commercial CFD code used: FLUENT, Star-CD, CFDRC, CFX/AEA, etc.
2. Research CFD code used: CFDSHIP-IOWA.
3. Public domain software used (PHI3D, HYDRO, and WinpipeD, etc.).
4. Other CFD software includes the Grid generation software used (e.g. Gridgen, Gambit) and flow visualization software (e.g. Tecplot, Field View).

CFD process

1. The main Purposes of CFD codes will be differ for different applications: investigate the bubble-fluid interactions for bubbly flows, data of wave induced separated flows for free-surface, etc.
2. Depend on the individual purpose and flow characteristic of the problem, Various CFD codes can be chosen for various applications (aerospace, marines, combustion, multi-phase flows, etc.)
3. Once aims and CFD codes are chosen, CFD process is the points to set up the IBVP problem and run the code:

Steps

1. Geometry of the specified model
2. Physics
3. Mesh of the geometry.
4. Solve the model.
5. Reports get after simulation.
6. Post processing of the model.

1.1. ***BACKGROUND INFORMATION***

Reducing the size of VLSI circuits in power electronics creates challenges with regard to dissipating the high value heat flux, to motivate the young scientists to develop new cooling technique at the micro level. The air-cooling methods have their limitations but liquid cooling through a micro level channel has given exceptional and improved results for the reduction of high heat fluxes in electronics.

Researches and technical studies in the use of micro-channel along with jet impingement heat sink cooling technology in high value of heat flux management in areas such as high-performance electronics chips, military avionics system, hybrid-vehicle field , power electronics system , radars, defence laser techniques and microwave oriented energy weapon systems is improving and gaining rapid popularity.

As technology advances, demands for smaller electronic systems and low cost of products increases. Similarly, the miniaturization of electronic systems is on the rise to maintain the linear relationship with the demand and In fulfilling these demands. One thing that cannot be avoided is the increase in the amount of heat generated by these electronic systems.

The study of micro-channels for use in cooling applications is accepted to have begin when Tuckerman and Pease produced a publication that stated the experimental ideas benefits of using micro-range diametric channels for cooling of very-large-scale integrated circuits. They observed that decrease in the hydraulic diameter of a channel increases the heat transfer coefficient and also showed an ample increase in heat transfer capabilities.

Through the narrow small parallel channels, the liquid flows from inlet end to the other output end. It leaves high temperatures at the inlet-side and lower temperatures at the outlet-side. The non-uniformity of temperature distribution on the heated surface induces temperature gradients, that reduces the life span of the electronics systems.

Other than jet impingement heat sink and micro-channel , there is a new field of research trying to harness the two cooling technologies to achieve greater high heat flux removal and hence reduce the high temperature and pressure gradient along channel flow. Jet impingement heat sink cooling offers new innovative technology for high value of heat flux systems of whole facial and also in power electronics system, hot spots are found more homogeneity in distribution of temperature across the chip surface.

In this current study, single-phase flow of fluid and transfer of heat in a micro-channel jet impingement is modeled with the use of CFD and the fluid flow characteristics investigated. Numerical model with the assumption of fully developed flow which is used to analyze the three dimensional micro-channel jet impingement fluid flows. Various parameters such as Reynolds number, channel geometry, substrate material, and working fluid effects on the performance of the configuration were investigated and results obtained used to compare existing literatures.

Further steps were also made to improve the model by studying the optimal values of the important parameters (values just mentioned) used during the study. Fully developed flow assumption is applied to reduce flow analysis complexity. Currently, the implementation of jet impingements heat sink on substrates was considered for getting the higher temperature uniformity in the substrate material and also improved hot spot management of electronics systems.

1.2. ***OBJECTIVES***

The recent work has main objectives as mentioned below:

1. Model of fluid flow in a jet impingement heat transfer in a micro-channel.
2. Investigate transfer of heat and flow characteristics parameters in the micro-channel
3. To study the effect of some key parameters such as Reynolds number, channel geometry, working fluid, and substrate material on the cooling performance of the configuration.
4. To improve the cooling model using the optimum parameters obtained from Proceeding studies (simulations).

1.3. REPORT LAYOUT

The report has 5 chapters. The content of each of the chapters are highlighted below:

1. **Chapter 1 : Introduction**
   This chapter gives a brief background of the project in which objectives of the research are briefly as well as numerical method used also mentioned.

2. **Chapter 2 : Literature Review**
   This chapter gives a review on previous works on micro-channel heat sink, jet impingement heat sink cooling and micro-channel/ jet impingement cooling technology. Advantages and disadvantages of both technologies are also mentioned.

3. **Chapter 3 : Methodology**
   This chapter gives a brief outline on how the work was carried out and the methods are used and also sheds light on the boundary condition settings.

4. **Chapter 4 : Results and Analysis**
   The outcomes obtained from the numerical solutions are displayed and analyzed graphically. In some cases, the brief explanation of results are given.

5. **Chapter 5 : Conclusion and suggestions**
   This chapter briefly summarizes the result obtained from the research work and suggestions for future work.

CHAPTER 2

2. LITERATURE REVIEW

In this section of the report review of the previous researches papers that have been Focused in the areas of micro-channel and jet impingement heat sink. Over the years both experimental and numerical simulation studies which have been carried out and to know how to manage amount of heat is produced by micro level electronics systems. Also, heat flow parameters affecting the execution of heat transmission using the various technologies (jet impingement and micro-channel heat sink) are also covered. Review of some of this works would be carried out in two sub part:

(i) Micro-channel

(ii) Jet impingement

2.1. *MICRO-CHANNEL*

Last decade micro-channel system has been mostly used in various field for better cooling of devices such as electronic systems and VLSI system. Effectiveness of the micro channel system is due to its advantages less coolant demands and small dimensions [1]; Hence ,it has fascinated the study of fluid discharge channel and transfer of heat in parallel-channels. This is of vast use in engineering as well as various practical applications such as medical instruments [2, 3].

Different experiments conducted had reported to reduce the geometrical parameters such as height, width and length of system (channel) to micro-level to improve heat transfer rate [4]-[10]. Due to the difficulties in machining of the microstructure of the system and measuring of 3-D properties along the micro-channel wall and simulations were adopted since it could predict the some flow parameters such as drop in pressure and temperature difference along the direction of flow of fluid. Though, combination of velocity and temperature values made modified the thermo physical properties of the channel [11, 12].

USES OF MICRO-CHANNEL

Tuckerman and Pease were the first studied the flow of fluid in the micro channel for heat removal, within the scope of a Ph.D. study. Their publication titled High Performance Heat Sinking for VLSI is credited as the first study on micro channel heat transfer. Their good work in the micro level has motivated researchers to focus on the field of micro channel and flow in the channel has been recognized as a high performance removal of heat tool ever since.

Before proceeding with flow in the micro channel and transfer of heat by the flowing of fluid in channel, it is sufficient to introduce a definition for the term micro channel. The main scope of the micro channel among the debate in this topic between researchers in the field. Mehendale et al. [13] used the following classification based on manufacturing techniques required to obtain various ranges of channel dimensions, D, being the smallest channel dimension:

$$\begin{array}{ccc} 1\mu m < D < 100\mu m & : & \text{Micro channels} \\ 100\mu m < D < 1mm & : & \text{Mini channels} \\ 1mm < D < 6mm & : & \text{Compact Passages} \\ D > 6mm & : & \text{Conventional Passages} \end{array}$$

Easiest classification was introduced by Obot [14] which is totally based on the hydraulic diameter rather than the smallest channel dimension. Obot classified channels of hydraulic diameter which is range under 1 mm ($D_h < 1mm$) as micro channels, which was also adopted by some other researchers such as Bahrami and Jovanovich (2006), Bahrami et al.[15].

This definition is included to be more importance for the purposes of this thesis. Higher volumetric transfer of heat densities need improved manufacturing techniques and lead to more complicated manifold designs. Many of the researchers have the same manufacturing techniques for the fabrication of electronic circuits which are being used for the making of compact heat exchangers.

Micro channel mainly contains an improved cooling technology for the removing of very large amount of heat through a micro channel whose cross section area at micro level. This is the best alternative for the replacement of conventional fin tube heat exchangers which is mostly used in industries. The heat removed is mainly from a high thermal conductivity solids such as silicon or copper with the channels joined into its surface where removal of heat takes place by the either precision machining or fabrication technology at micro level.

Coolant which is in the form of fluid is to pass through the channels by the external force system for the removal of heat or carry away heat from a hot surface. In Micro channel heat removal process the flow is typically laminar and heat transfer coefficients(h) are proportional to velocity of the fluid.

It offers other benefits which is including more latent capacity for micro channel evaporators. Micro channel heat exchangers improve transfer heat in two ways. First, the smaller of the refrigerant flow into the channel that increase refrigerant-side transfer of heat. Second, the flat tube which is oriented in flat position that reduces the airside flow resistance, which leads to be either increased airflow or reduced fan power.

A micro reactor is a device in which the chemical reactions carried out in a confinement with lateral dimensions below 1.0 mm. When temperature of the reactor is needed to maintain, micro reactor also act as a micro channel heat exchanger. Micro reactors used in these devices are typically regular flow reactors rather than batch reactors, with the regular flow model providing good and efficient performance in material synthesis than is possible.

The device offers so many benefits over conventional reactors, including best improvements in energy efficiency, reaction speed and yield, safety in system and finer degree of process control. It enables miniaturization of the processor of fuel because they minimize both heat transfer and mass transfer resistance.

2.2. ***JET IMPINGEMENT***

Like micro-channel, jet impingement give more effective means for convective heat transfer coefficient due to its more transfer of heat and mass transfer rate. The use of micro channel is mostly used in industrial applications such as tempering of material, annealing, plastic cooling and paper fabric dry process [16, 17, 18]. Heat transfer parameters of multiple jets could differ substantially from single jets which is totally depending on the geometrical conditions [19, 20]. Also noted that spacing between the jet is very greater influence on the amount of heat transfer.

Multiple jets geometry design has been a mirage because of more number of militating factors in which is complexity in comparing one result to another [21, 22, 23]. So that, the complexity of comparing the result of one to another has led to development of correlations as in Sung and Mudawar [21]. Reynolds number (Re) and nozzle to impingement surface distance, H/D and angle of impingement are some of the parameters that determine the effectiveness of jet impingement configurations [24, 25].

Sung and Mudawar have carried out various experimental as well as numerical simulation on micro-channel and jet-impingement system to minimize the issue of pressure drop and temperature gradient in micro-channel on the electronic device , and stagnation of fluid flow and rapid development of thermal boundary layer away from impingement zone in jet impingement. Sung and Mudawar [26] noted that results obtained from the 3- dimensional heat transfer characteristics analysis of the hybrid scheme using water , DIUF water , PF-5052, HF-7100 as working fluid were agreement with results to improve the individual technologies.

Another research by Sung and Mudawar [21] postulated that working of the modified configuration in which fluid is flow for cooling is controlled more by jet impingement micro channel as compared to the influence of micro-channel flow. The test surface is divided into a part or grid system. It is controlled by jet impingement and another by micro-channel flow while implementing the more exact critical heat flux correlation for each part. By doing this they were able to predict the critical heat flux data for the modified cooling structure with an absolute error of about 8.42%.

Sung and Mudawar [22], experimentally and numerically simulated results investigated a fresh modified cooling system that proposed for high heat flux mainframe in the micro channel. They observed that when increasing jet velocity allows the jets to penetrate the micro-channel flow towards the heated surface and decrease the wall temperature. In spite of the poor thermo-physical properties of HFE-7100, the proposed cooling system that facilitated the dissipation of 304.9 W/cm2 without phase change in the system. In addition to the numerical predictions, they divided the heat transfer surface into the various zones that were each dominated by a different heat transfer process, at the same time of process assigning different heat transfer coefficient value to each zone.

Using this study Sung and Medawar developed a new correlation that fits the results obtained from study with an error of 6.037%. Sung and Mudawar [16] further studied the hybrid cooling system by investigating a single-phase cooling performance of the hybrid cooling system in which a series of jets that provide cooling into each channel of a micro-channel heat sink. This caused by symmetrical flow in each micro-channel, and the coolant expelled through in both ends of the micro-channel. Three (3) jet patterns examined in this case were: decreasing-jet-size, increasing-jet-size and equal-jet-size.

Also, the performances of each pattern were studied experimentally and numerical simulation using HFE-7100 as working fluid. Use of refrigeration in indirect way for cooling was applied to minimize the coolants temperature in order to provide low wall temperatures of micro channel during high heat flux dissipation. Their studies and numerical simulation showed excellent accuracy in predicting micro channel wall temperatures. The numerical results showed that the hybrid cooling system involves complex interactions of jet impingement and micro-channel fluid flow. It was also noticed that increasing the coolants flow rate strengthens more the contribution of jet impingement by using fluid to the overall cooling performance, and decrement in wall temperature.

Decreasing the jet-size pattern yields the most effective convective heat transfer coefficients (h) and the least wall temperatures, whereas equal-jet-size pattern gives the effective uniformity in wall temperature. Also, increasing the jet size pattern yields complex patterns of flow and more wall temperature gradients; which resulted from blockage of spent fluid flow due to the impingement of jet from larger jets near the channel outlets. Barrau et al[27] experimentally studied and noticed that a new proposed hybrid system for high flux thermal management and power devices. Also, they tested the geometry to characterize its cooling performances and also to check its capacity to provide this characteristic parameter.

Temperature distribution in the channel was measured using a matrix of thermocouples. Since the experimental study provides a global decrease of the temperature of the heat sink in the direction of the flow of fluid, they deduced the hybrid cooling system has the capacity to improve the temperature uniformity of the channel to be cooled.

From the various literature review, no investigation has been takes place either experimentally or numerical simulation to improve the parametric design of the hybrid cooling system . In present study, a hybrid micro-channel jet-impingement cooling system is to be modeled using Ansys 15.0 and the characteristics of the fluid flow also studied.

The influence of Reynolds number (Re) on thermal resistance and overall performance of the micro channel module was also investigated. Geometrical parameters such as height of the channel and width were respectively. Copper, silicon, aluminum, steel, nickel and titanium were the substrate materials used. Four kinds of working fluids (water,DIUF, PF-5052 and HFE-7100) were examine. Constant effective heat flux of 25 W/cm2 was supplied at the bottom of the substrate of the micro channel and inlet hydraulic diameter base velocity of the working fluids calculated and based on the working fluid Reynolds number (in the range of 100-120000).

CHAPTER 3

3. METHODOLOGY/ MODEL DESCRIPTION

*3.1. **MODEL DESCRIPTION***

A multiple micro-jet impingements heat sink system, as shown in Fig .1 was designed on the other side of working surface of a microchip which have cooling with multiple nozzles which work like jet. The nozzles were fabricated on the substrate part of the micro channel, and a fluid channel was created to allow liquid to pass through the nozzles. The working fluid jets strike at the heated surface of the channel and the flow of fluid converted into the radial direction while removing high heat from the heated surface, and reaches to the outlet port of the material, as shown in Fig. 3.

For conservative studies, the substrate on which these nozzles were fabricated was not examine , whereas the nozzles with a fluid channel and substrate base were included in the computational domain on which fluid flow. The jets and outlet channel part of substrate were fabricated separately and joined together with epoxy to form the designed structure.

The dimensions of the jet impingement micro channel were 10 mm x10 mm x 0.6 mm. Thickness of the solid domain substrate base was 100 mm and the depth of the fluid domain channel was 200 mm. The thickness of the nozzle plate on which nozzle fixed was 300 mm on which multiple nozzles were designed for the better performance. One half of the heat sink channel was taken as the numerical computational domains due to the symmetry of the heat sink about the central plane.

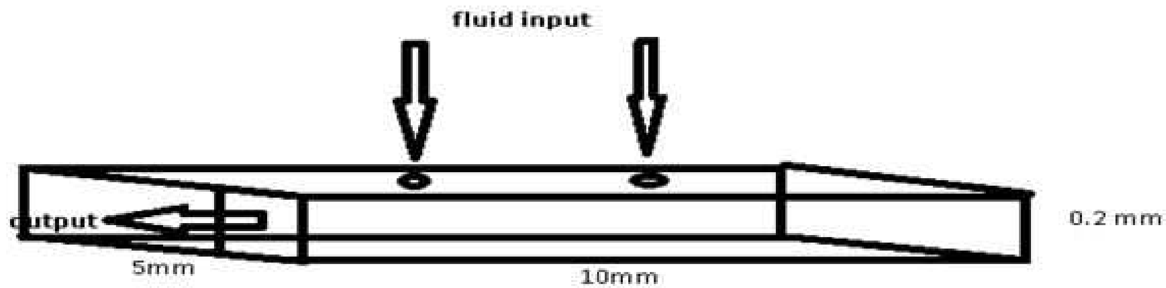

Figure 1: Fluid domain

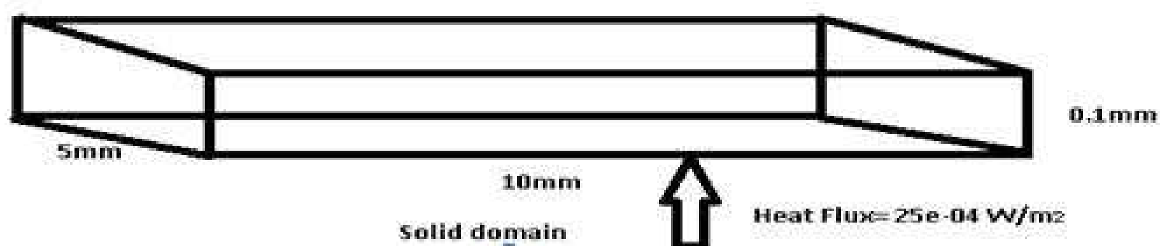

Figure 2: Solid domain

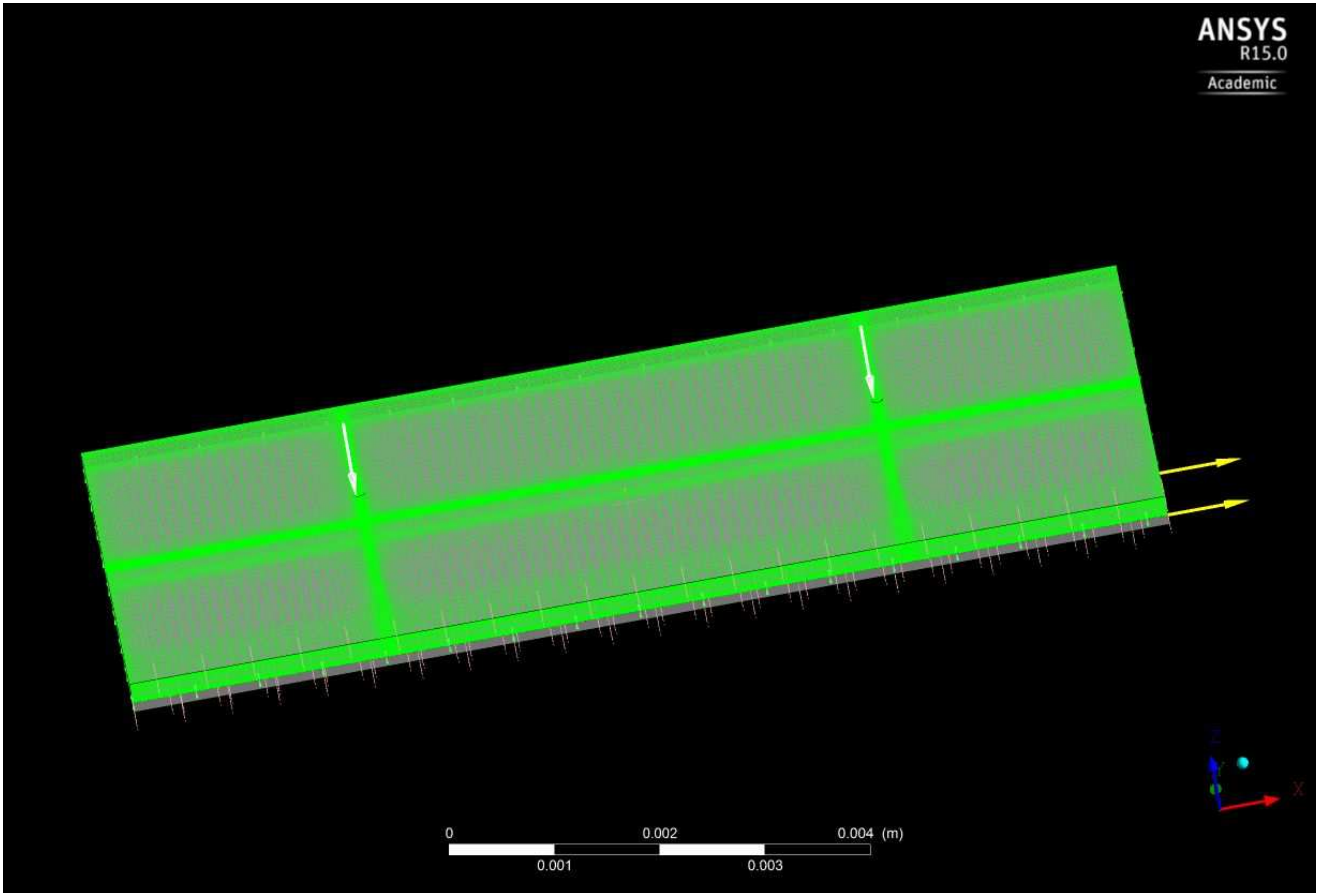

Figure 3: Solid and fluid domain interface

3.2. NUMERICAL SCHEME AND BOUNDARY CONDITIONS

3 D numerical analysis was carried out for flow of fluid in fluid domain and conjugate transfer of heat in the model. DIUF water, water, HF-7100 and Performance fluid (PF-5052) were used as a coolant and flow of fluid through the nozzles and domains substrate channel. The governing equations are used for the conservation of mass, momentum and energy for fluid flow and conjugate heat transfer can be written in vector form as follows:

$$\frac{\partial(\rho_f v_i)}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial(\rho_f v_i v_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i}\left(\mu_f \frac{\partial v_i}{\partial x_j}\right) + \frac{\partial}{\partial x_i}\left(\mu_f \frac{\partial v_j}{\partial x_i}\right) - \frac{2}{3}\frac{\partial}{\partial x_i}\left(\mu_f \frac{\partial v_k}{\partial x_k}\right) \tag{2}$$

$$\frac{\partial(\rho_f v_j c_p T_f)}{\partial x_j} = v_j \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}\left(k_f \frac{\partial T_f}{\partial x_j}\right) + \tau_{ij}\frac{\partial u_j}{\partial x_i} \quad \text{(For the fluid)} \tag{3}$$

$$\frac{\partial}{\partial x_j}\left(k_s \frac{\partial T_s}{\partial x_j}\right) = 0 \quad \text{(For the substrate conduction)} \tag{4}$$

The numerical simulations were carried out using finite volume CFD code Ansys 15.0 that used for the coupled algebraic multiple grid method. The physical properties of the liquids were allowed to varied with temperature distribution to take micro-scale effects into account.

Hexahedral mesh was generated for a particular specified computational solid and fluid domain on which fluid is flowing . For an study of the heat sink and its optimization the specified parameters, all walls of the fluid domain except for the fluidsolid interface at the bottom of the channel are kept adiabatic.

A uniform heat flux should provide as a heat source at the base of solid domain substrate heat sink. There is no-slip condition (V = 0) was applied at the internal walls of the nozzles and channel. The thermal conditions are:

$$
\begin{aligned}
-k_s \frac{\partial T_s}{\partial x} &= 0 \text{ at } x = -l_x/2 \text{ and } x = l_x/2 \text{ except at the outlet} \\
-k_s \frac{\partial T_s}{\partial y} &= 0 \text{ at } y = l_y/2 \\
-k_s \frac{\partial T_s}{\partial z} &= q \text{ at } z = 0 \text{ and} \\
-k_s \frac{\partial T_s}{\partial z} &= 0 \text{ at } z = H_c + t_s \text{ except at the nozzle exits} \\
-k_s \frac{\partial T_s}{\partial \overline{n}} &= 0 \text{ at the nozzle walls}
\end{aligned}
$$

Heat transfer coefficient were evaluated as:

$$h = q/(T_f - T_s) \tag{5}$$

3.3. *ASSUMPTIONS*

Due to the complexity of the 3-D conjugate heat sink, the following assumptions are made for the analysis:

1. Flow is fully developed along the micro-channel length and Steady state flow situation to be assumed.

2. In calculation of the velocity, in micro channel water is to be distributed uniformly throughout the fluid domain.

3. The velocities are zero in the heat sink substrate and assumed to be an adiabatic surface (V=0 no-slip boundary condition).

4. The temperature of the solid and liquid is assumed to be same at the interface between solid and liquid domain .

CHAPTER 4

4. RESULTS AND DISCUSSIONS

4.1. SUBSTRATE MATERIAL

Different types of substrate material used for the hybrid cooling in which heat sink plays an very important role in the amount of heat remove from the substrate material which is used in the present work. In this work Six different materials (table 1) were used and results obtained after the simulation are presented in terms of Temperature difference and Heat Transfer coefficient against the hybrid channel length.

It is observed that Copper has the highest value of heat transfer coefficient as compared to the other materials as shown in the table no. 1. We are used six different materials which is shown in the table and temperature distribution diagrams also provided as given.

We have seen that in the present work the temperature distribution is uniform after certain interval of time in every six different materials. So we can say that with the help of jet impingement we get the uniform temperature distribution as compared to the without jet used micro channel.

We also seen that the value of temperature drop and average temperature in the copper is very minimum as compared to the other material s used. Because of this the value of heat transfer coefficient is high in copper. As we studied in the literature when the heat transfer coefficient is more, more is the Nusselt number. So, finally we can say that copper is material which have the highest coefficient of heat transfer and Nusselt number.

Table 1: Material properties and results from numerical simulations

Material Properties				Results after Simulations		
Material	Density(kg/m^3)	Cp(J/Kg-K)	K(W/m-K)	Avg. temp.(K)	Temp. range(K)	Heat transfer coefficient (h) W/m^2K
Copper	8933	385	401	319.97	313-321	31250
Aluminum	2702	903	237	320	311 -322	22727.27
Steel	7854	434	60.5	321.43	303-324	11904.76
Silicon	2329	700	149	320.621	308-323	16666.66
Titanium	4500	520	21	322.42	299-326	9259.25
Nickel	8908	445	90	321.05	305-323	13888.88

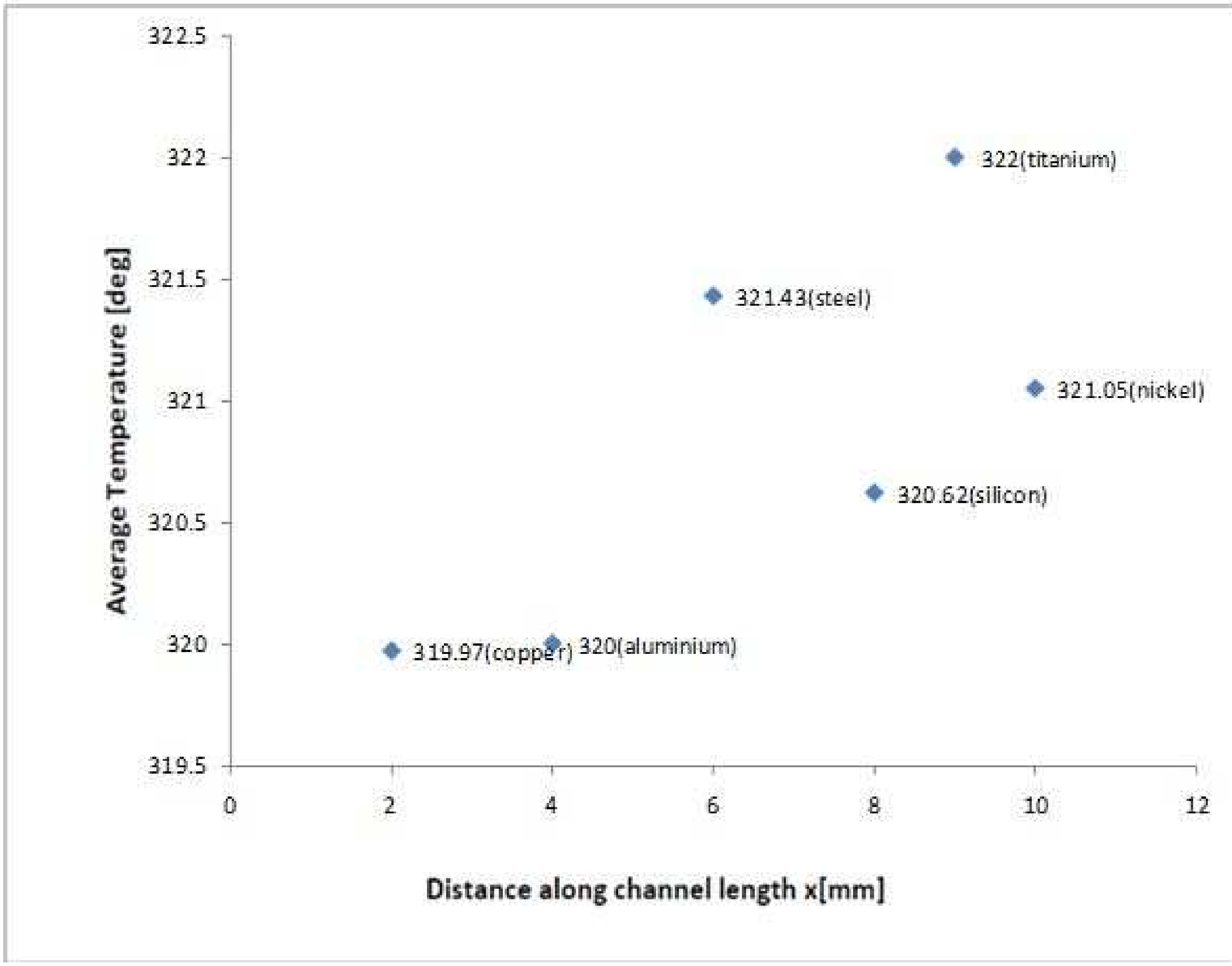

Figure 4: Graph between Average Temperature and Distance along the channel for different material.

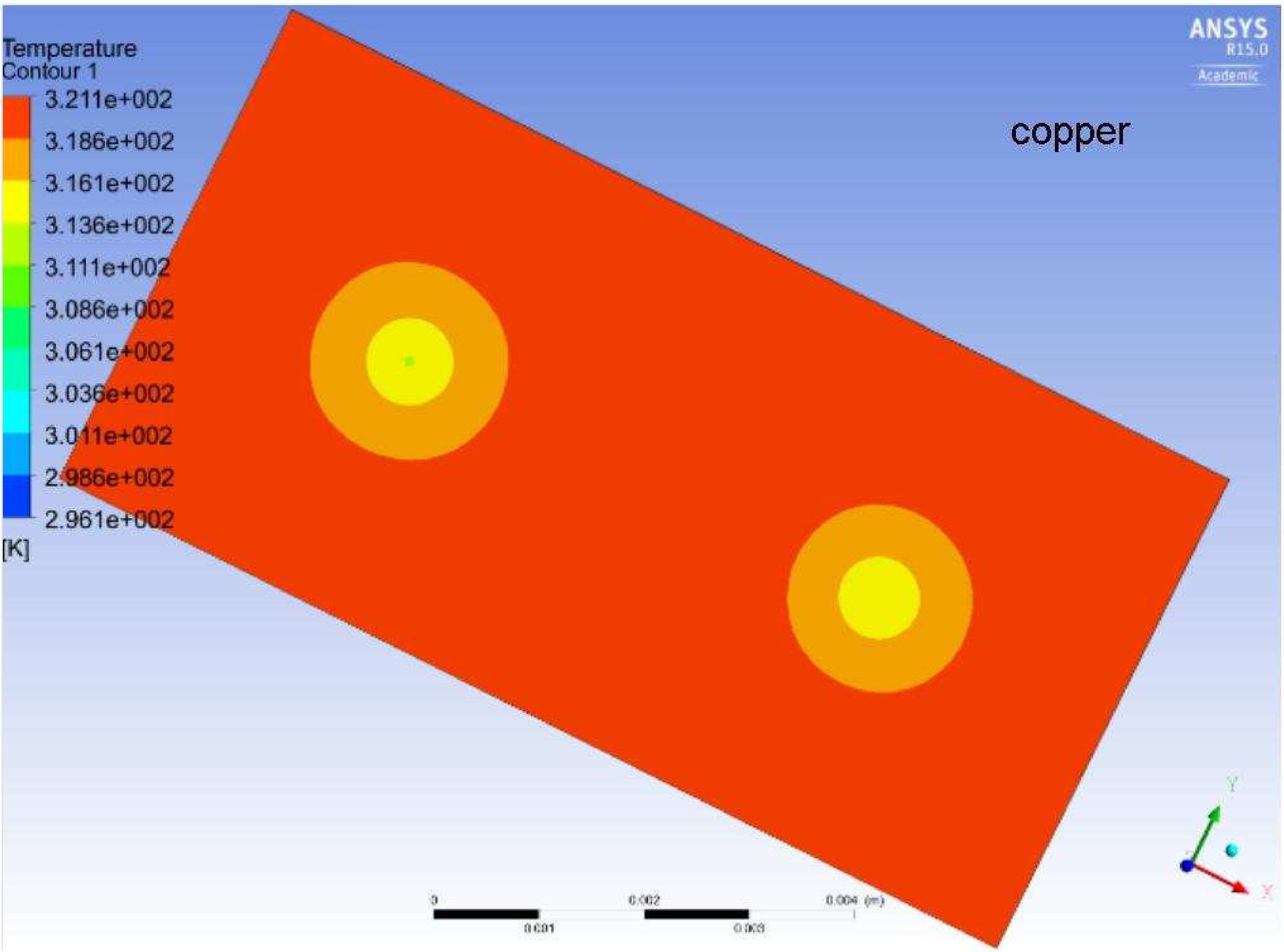

Figure 5: Temperature distribution for Copper

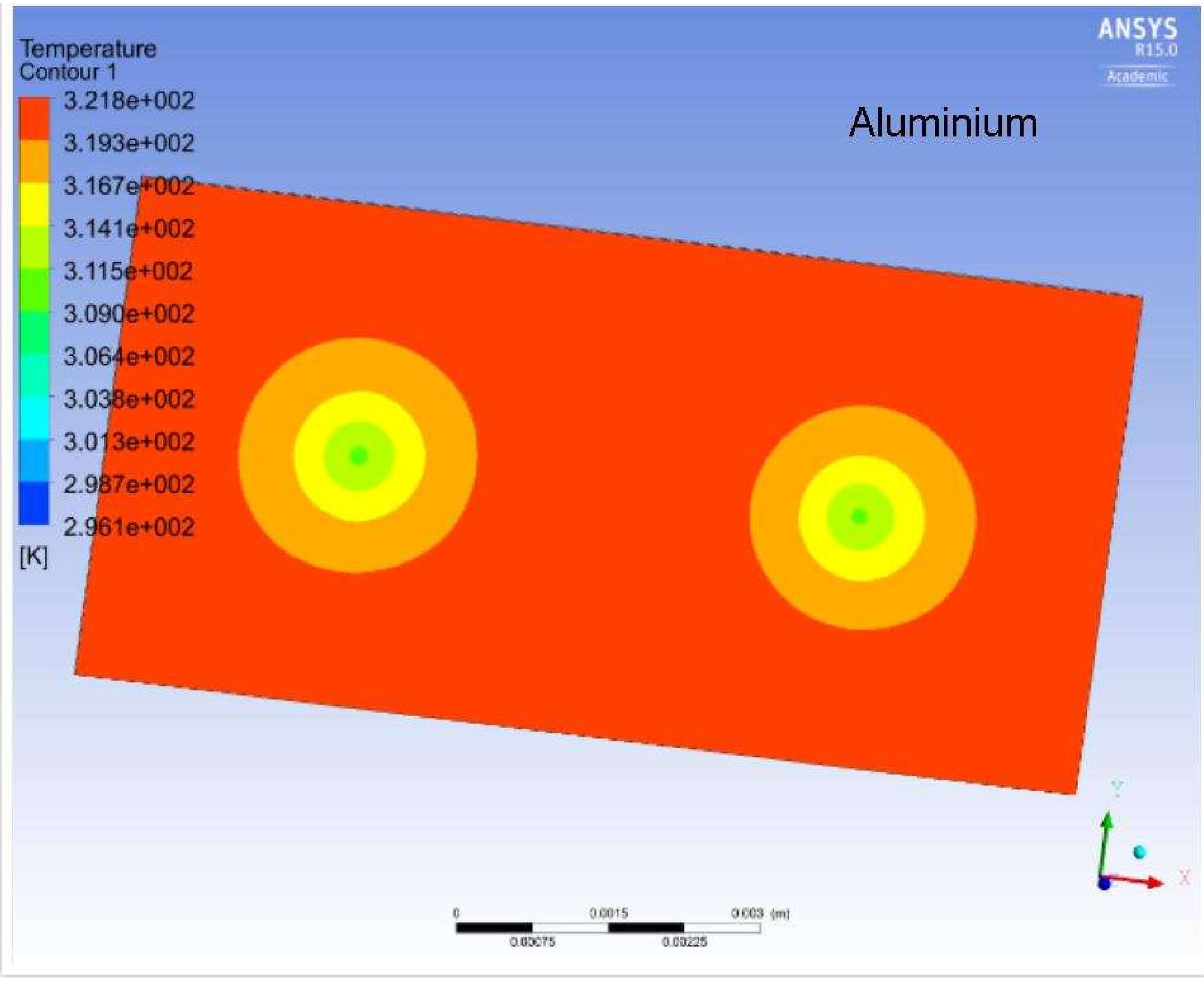

Figure 6: Temperature distribution for Aluminium

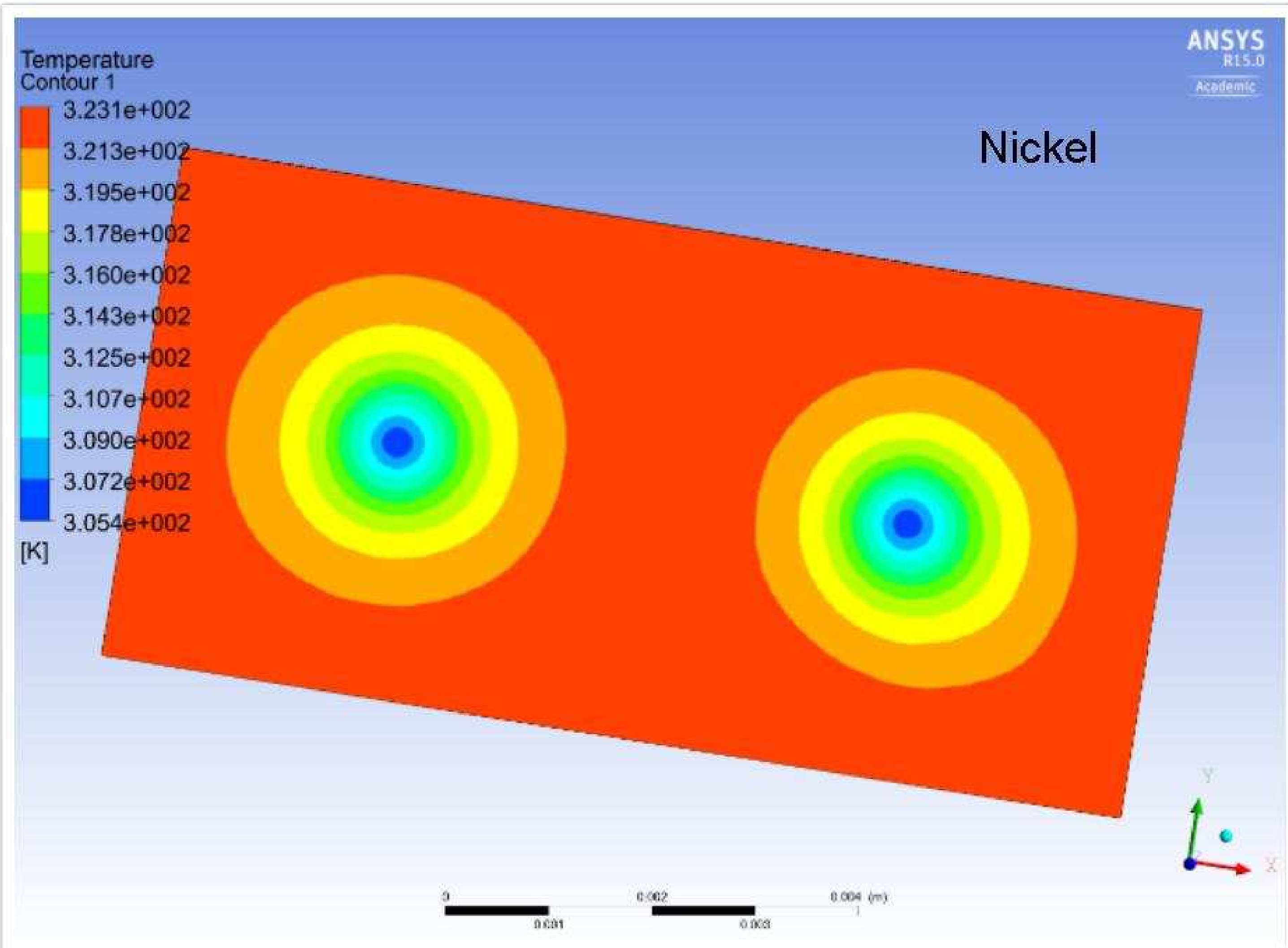

Figure 7: Temperature distribution for Nickel

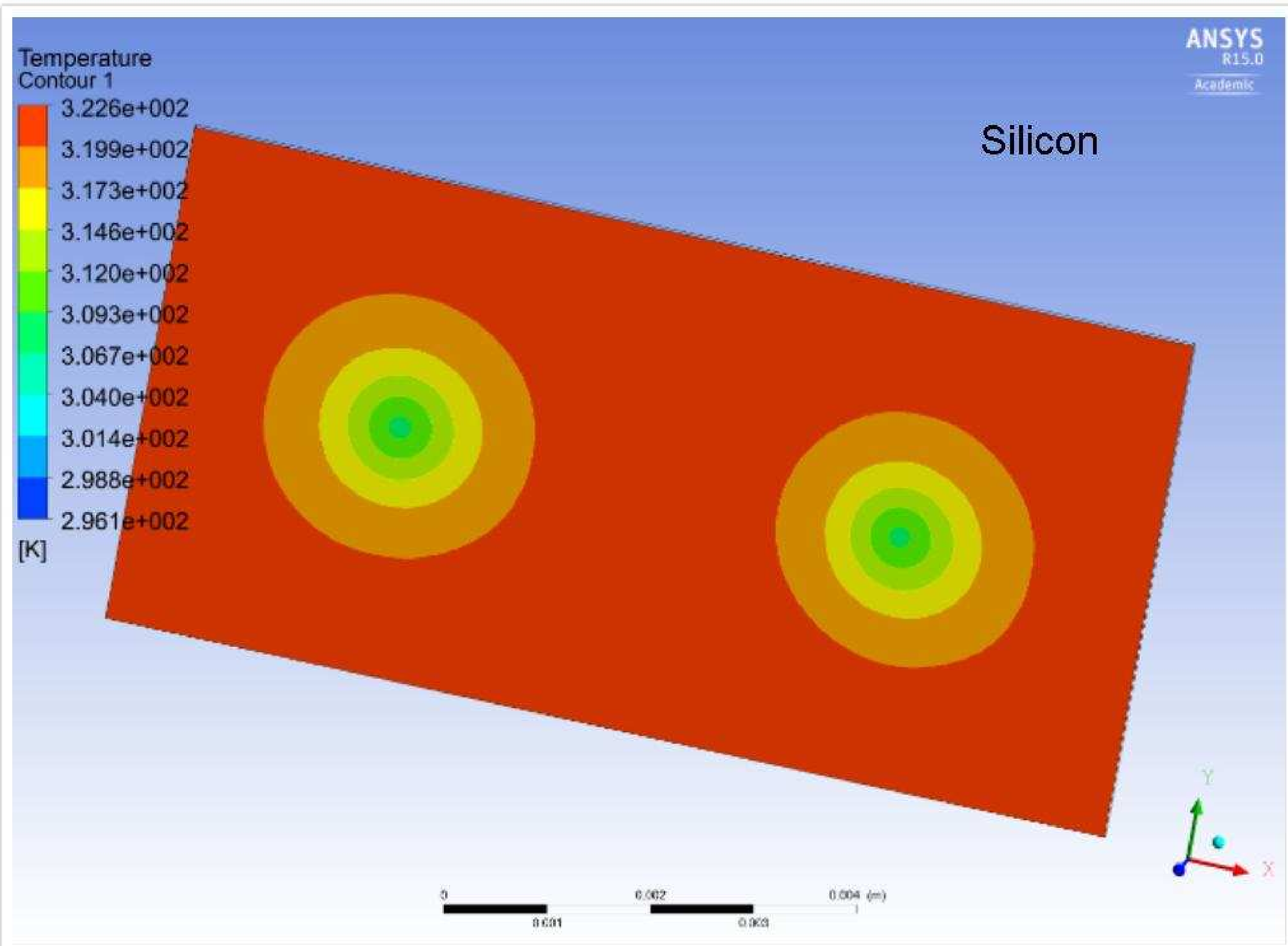

Figure 8: Temperature distribution for Silicon

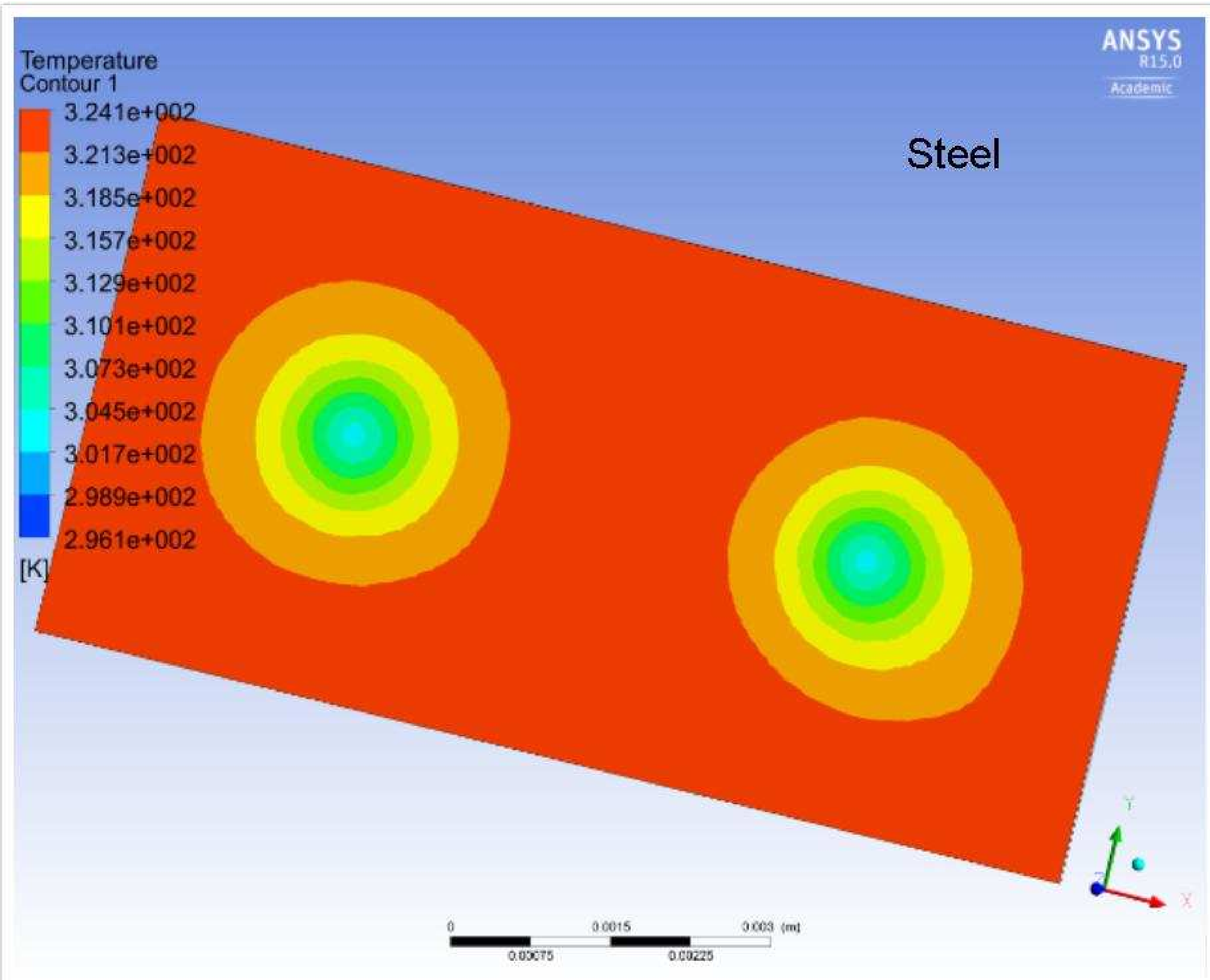

Figure 9: Temperature distribution for Steel

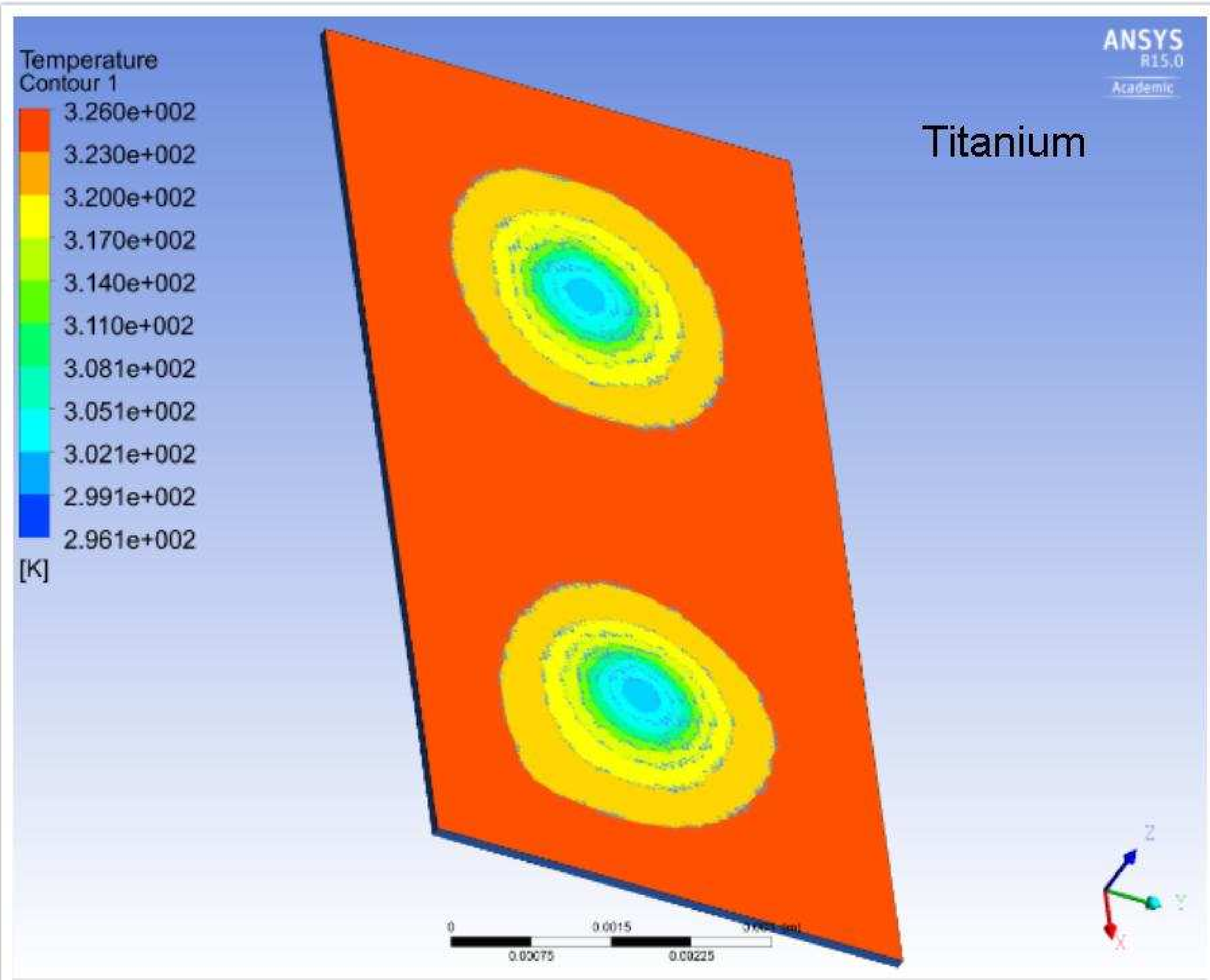

Figure 10: Temperature distribution for Titanium

As the substrate material the working fluid used to find out the behavior of the system Performance as well as cooling characteristics. We are taking four working fluid and were investigated in this study include: water, PF-5052, HF -7100 and DIUF. Table no. 2 gives details properties of the fluid used in this study and results obtained after the numerical simulation of computations. All numerical conditions were kept constant for all working fluids.

From the figure 2 it is apparent that DIUF yields the lowest temperature of the hybrid scheme with an average surface temperature of 9.00°Cfollowed by HFE-7100 with average surface temperature of 65°C and PF-5052 being the least with an average surface temperature of 69.23°C. This is due to high thermal conductivity of DIUF.

Obviously, increase in thermal conductivity of working fluid reduces the overall surface temperature and increases the heat transfer rate. This term is valid as thermal conductivity implies ability of the working fluid to transfer of heat energy.

Table 2: Fluid properties and summarized numerical results of working fluid used.

Liquid properties					Results obtained from numerical Simulations		
Fluid	$\rho(Kg/m^3)$	Cp(J/Kg K)	K(W/m-k)	$\mu(kg/m-s)\times10^{-5}$	Avg.Temp.($^\circ C$)	Avg. q(W/cm^2)	Average Number
Water	998	4182	0.6	100.3	9.35	11	20
DIUF water	990	4190	.582	100.2	9.00	11.2	21
PF-5052	1776	1014	.065	97.77	69.23	10.88	31.25
HF-7100	1538.3	1133	.074	76.99	65	1086	29

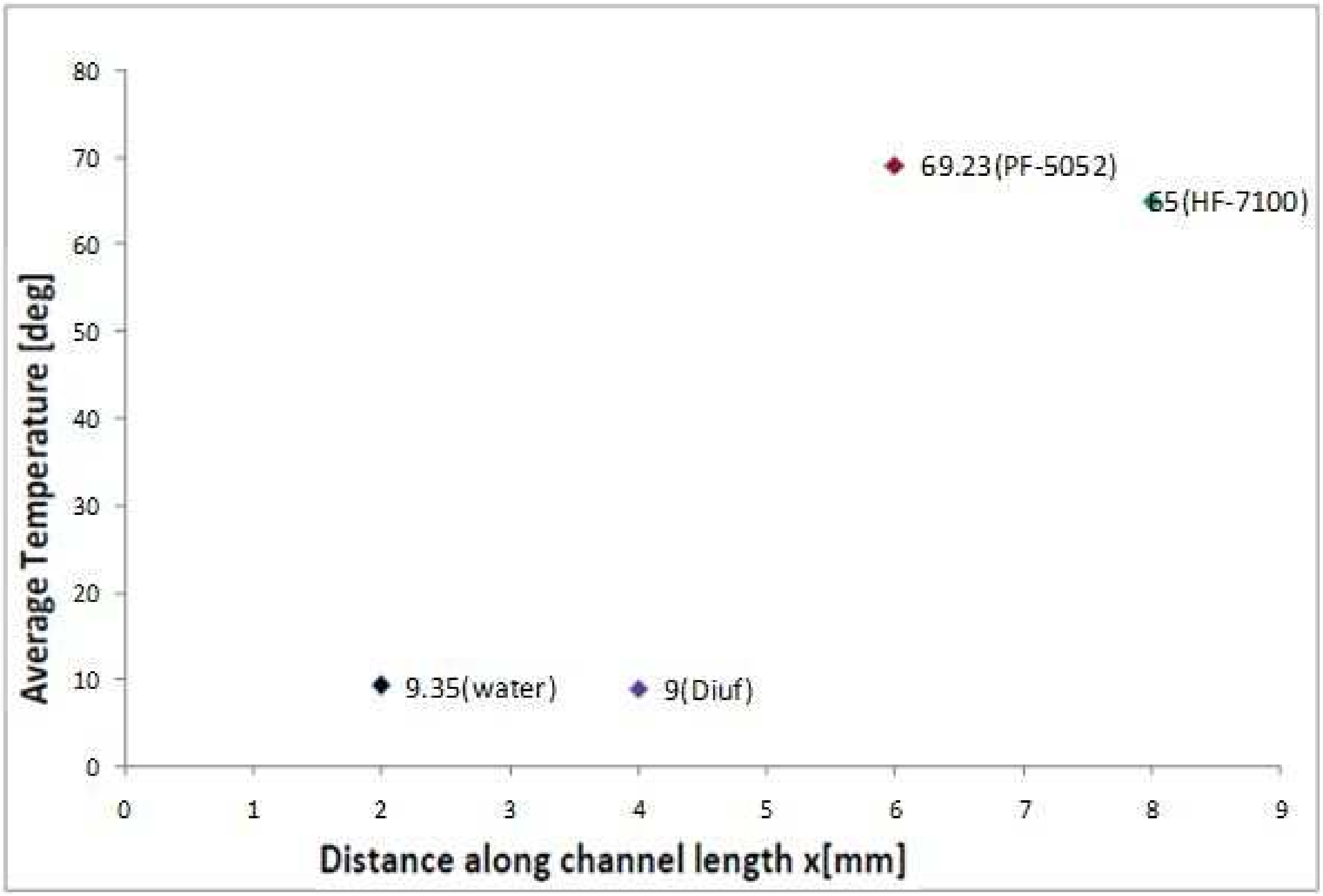

Figure 11: Graph between Average Temperature and Distance along the channel for different fluids.

CHAPTER 5

5. CONCLUSIONS AND SUGGESTIONS

5.1. *CONCLUSIONS*

After thorough inspection of the numerical simulation applied in each case and Assumptions which used in this present study the conclusions are drawn from the Numerical Simulations carried out.

1. Single-phase fluid flow characteristics of the micro-channel/jet impingement cooling system as been successfully modeled and studied numerically.

2. From the different substrate materials used in the present study Copper yielded the highest heat transfer coefficient with an average value of temperature which is about 2.9% and 3.9% better than that of Silicon and Aluminum respectively.

3. The least average surface temperature ($9°C$) of the hybrid cooling scheme was achieved using DIUF water as working fluid. This is about 77% less than that of HFE-7100 and PF-5052.

4. Hence, the improved hybrid system in this scope of study would be rectangular Channel wit Copper and DIUF water as substrate material and working fluid respectively.

5.2. *SUGGESTIONS FOR FUTURE STUDIES*

In focus of the results obtained from the numerical study, the future scope is suggested as given follows:

1. Impact of jet configuration.

2. Position of jet on the hybrid system.

3. Angle of impingement of flow and distance between impinging jets and surface to be impinged.

4. Phase change in the fluid flow.

REFERENCES

[1] C.M. Ho, Y.C- Tai, 1998. Micro-electro-mechanical systems (MEMS) and fluid flows. Annual Review of Fluid Mechanics, 79-612.

[2] T. Chovan, A. Guttman, 2002 Micro-fabricated devices in biotechnology and bio-chemicalprocessing. Review TREDN in Biotechnology ,116–122.

[3] B. Gromoll, 1998. Micro-cooling systems for high density packaging. Rev. Gen. Therm, 781–787.

[4] P. Gao, S.L. Person, M.F.-Marinet, 2002. Scale effects on hydrodynamics and heat transfer in two-dimensional mini and micro-channels. International Journal of thermal Science , 1017–1027.

[5] G. Gamrat, M.F-Marinet, D. Asendrych, 2005. Conduction and entrance effects on laminar liquid flow and heat transfer in rectangular micro-channels. International Journal of Heat and Mass Transfer, 2943–2954.

[6] W. Owhaib, B. Palm, 2004. Experimental investigations of single-phase convective heat transferin circular micro-channels. Experimental Thermal and Fluid Science, 105–110.

[7] S.G. Kandlikar, S. Joshi, S. Tian, 2001. Effect of channel roughness on heat transfer and fluidflowcharacteristics at low reynolds numbers in small diameter tubes. Proceedings of NHTC01 35th National Heat Transfer conference.

[8] J. Yao, M.K. Patel, Y. Yao, P.J Mason, 2006. Numerical Simulation of heat transfer in Rectangular Micro Channel. European Conference on Computational Fluid Dynamics ECCOMAS CFD.

[9] W. Qu, I. Mudawar, 2002. Experimental and numerical study of pressure drop andheat transfer in a single-phase micro-channel heat sink. International Journal of Heat and Mass Transfer ,25492565.

[10] X.F. Peng, 1996. Convective heat transfer and flow friction for water flow in micro-channel structures. Int. J. Heat Mass Transfer ,2599-2608.

[11] A.G. Fedorov, R. Viskanta, 2000 Three-dimensional conjugate heat transfer in themicrochannel heat sink for electronic packaging. International Journal of Heat and Mass Transfer 399–415.

[12] D.-Y. Lee, K. Vafai, 1999 . Comparative analysis of jet impingement and micro-channel cooling for high heat flux applications. International Journal of Heat and Mass Transfer Vol. 42, 1555–1568.

[13] Mehendale, S. S., Jacobi, A.M. and Shah, R. K. 2000. Fluid Flow and Heat Transfer at Micro- and Meso-scales with Application to Heat Exchanger Design. Applied Mechanics Reviews, Vol 53, 175–193.

[14] Obot, N.T. 2003. Toward a Better Understanding of Friction and Heat/Mass Transfer in Microchannels A Literature Review. Microscale Thermophysical Engineering. Vol 6, 155– 173.

[15] Bahrami, M and Jovanovich, M. M. 2006. Pressure Drop of Fully Developed Laminar Flow in Microchannels of Arbitrary Cross-Section. Journal of Fluids Engineering. Vol 128, 1036–1044.

[16] B. Weigand, 2009. Multiple Jet Impingement-A review. Interntional Symposium onHeat Transfer in Gas Turbine Systems . Antalya,Turkey.

[17] B. Han, R.J. Goldstein, Jet-impingement Heat Transfer in Gas Turbine Systems. Heat Transfer laboratory, Dept. of Mech. Eng. Uni. Of Minneapolis, MN55455, U.S.A.

[18] S.G. Kandlikar, A.V. Bapat, 2007.Evaluation of Jet Impingement, Spray and Micro-channel Chip Cooling Options for High Heat Flux Removal. Heat Transfer Engineering .Vol 28:11, 911–923.

[19] M.K. Sung, I. Mudawar,2008. Effects of jet pattern on single-phase cooling performance of hybrid micro-channel/micro-circular-jet-impingement thermal management scheme. International Journal of Heat and Mass Transfer .Vol 51 ,4614-4627.

[20] C. Glynn, T. ODonovan, D.B. Murray, Jet Impingement Cooling. Center for Telecommunications Value-Chain-Driven Research (CTVR), Department of Mechanical and Manufacturing Engineering, Trinity College Dublin.

[21] M.K. Sung, I. Mudawar,2006. Correlation of critical heat flux and in hybrid jet impingement/micro-channel cooling scheme. International Journal of Heat and Mass Transfer Vol49 , 2663–2672.

[22] M. K. Sung, I. Mudawar,2008. Single-phase hybrid micro-channel/micro-jet impingement cooling. International Journal of Heat and Mass Transfer Vol ,51 4342–4352.

[23] R.J. Goldstein , W.E. Ibele, S.V. Patankar, T.W. Simon, T.H. Kuehn, P.J. Strykowski, K.K. Tamma, J.V.R. Heberlein, J.H. Davidson, J. Bischof, F.A. Kulacki, U. Kortshagen, S.Garrick, V. Srinivasan, K. Ghosh, R. Mittal, Heat transfer-A review of 2005 literature, International Journal of Heat and Mass transfer .Vol 53 ,4397–4447.

[24] P. Naphon, S. Wongwises, 2010.Investigation on the jet liquid impingement heat transfer for the central processing unit of personal computers. International Communications in Heat and Mass Transfer .Vol 37, 822-826.

[25] T.S. ODonovan, D.B. Murray, EFFECT OF VORTICES ON JET IMPINGEMENT HEAT TRANSFER. Dept of Mechanical and Manufacturing Engineering, University of Dublin Trinity College Dublin, Ireland.

[26] M.K. Sung, I. Mudawar,2006. Experimental and numerical investigation of single-phase heat transfer using a hybrid jet-impingement/micro-channel cooling scheme. International Journal of Heat and Mass Transfer. Vol 49 , 682–694.

[27] J. Barrau, D. Chemisana, J. Rosell, L. Tadrist, M. Ibanez, 2010. An experimental study of a new hybrid jet impingement/micro-channel cooling scheme. Applied Thermal Engineering Vol 30, 2058-2066.

[28] Wang EN, Zhang L, Jiang L, Koo J-M, Maveety JG, Sanchez EA, Goodson KE, Kenny TW 2004. Micro machined jets for liquid impingement cooling of VLSI chips. J micro electromechanical 833-842.

[29] Herwig H, Mahulikar SP 2006 Variable property effects in single-phase incompressible flows through microchannels. Int J Therm Sci 45:977-981.