Available online www.ejaet.com

European Journal of Advances in Engineering and Technology, 2015, 2(1): 12-20

Research Article ISSN: 2394 - 658X

Numerical Analysis of Flow through MINI Channels

Anooplal B, Binoy Baby and C J Joseph

Department of Mechanical Engg, St Joseph's College of Engg &Tech., Palai, India - 686579
anooplalb@gmail.com

ABSTRACT

Overheating of components and devices in electronic systems led to the application of mini and microchannel technologies. The aim is to eliminate as fast as possible the maximum heat quantity from these systems in order to ensure an increased reliability and functional stability. Analysis of a laminar flow through a mini-channel has been done by a numerical model method by considering the surface irregularities. The process has been carried out as three phases and comparison is made on the variation of temperature on different cross-section. The objective of the work is to investigate fluid flow and temperature variation of regular cross-section with that of irregular cross-section. The result for the channel with surface irregularities is compared with those of a smooth channel with regular cross section shape and geometry. From the investigation, we obtain that the actual prediction of flow and heat transfer characteristics of channels of small cross-section requires the considerations of surface irregularities while performing the numerical computations.

Key words: Mini channel, CFD, flow analysis

INTRODUCTION

Electronic cooling is required to remove the waste heat produced by electronic components, to keep within permissible operating temperature limits. Components that are susceptible to temporary malfunction or permanent failure if overheated include integrated circuits such as CPUs, chipset, graphics cards, and hard disk drives. Components are often designed to generate as little heat as possible, and computers and operating systems may be designed to reduce power consumption and consequent heating according to workload, but more heat may still be produced than can be removed without attention to cooling. Use of heat sinks cooled by airflow reduces the temperature rise produced by a given amount of heat. Fans are very widely used to reduce temperature by actively exhausting hot air. There are also more exotic and extreme techniques, such as liquid cooling.

Integrated circuits (e.g., CPU and GPU) are the prime generators of heat in modern computers. Heat generation can be reduced by efficient design and selection of operating parameters such as voltage and frequency, but ultimately acceptable performance can often only be achieved by accepting significant heat generation. In operation, the temperature of an electronic component will rise until the heat transferred to the surroundings is equal to the heat produced by the component, i.e., thermal equilibrium is reached. For reliable operation, the temperature must never exceed a specified maximum permissible value for each component. For semiconductors, instantaneous junction temperature, rather than component case, heat sink, or ambient temperature is critical.

Cooling can be hindered by:

- Dust acting as a thermal insulator and impeding airflow, thereby reducing heat sink and fan performance.
- Poor airflow including turbulence due to friction against impeding components such as ribbon cables, or improper orientation of fans, can reduce the amount of air flowing through a case and even create localized whirlpools of hot air in the case.
- Poor heat transfer due to poor thermal contact between components to be cooled and cooling devices. This can be improved by the use of thermal compounds to even out surface imperfections, or even by lapping.

Until today the microprocessor industry has managed to keep pace with Moore's law of increasing transistor density on a single chip. However, as more and more transistors are being packed in a given chip area, the heat dissipation density of a typical chip has risen dramatically. High heat density leads to high chip temperature, which adversely affects the chip performance and raises concerns about thermal reliability of electronics. Single phase liquid cooling for microprocessors has been long recognized as an effective method to replace conventional air-cooling to handle the increasing heat densities of current and future microprocessors. The liquid best suited thermally for single-phase cooling is water due to its high specific heat and thermal conductivity, as well as high availability and environmental friendliness.

Heat sinks with micro-channels are now established as an effective approach to implement liquid cooling for electronics. A maximum power dissipation density of 790W/cm2 with a thermal resistance of $0.1^0C\ cm^2/W$ but a high pressure drop of 2 bar was measured. Following this, experimental and numerical studies have investigated the flow and heat transfer characteristics of micro-channel heat sinks. Fedorov and Viskanta reported a numerical study of a manifold micro-channel heat sink and analyzed the complex thermal transport inside a 3D micro-channel. Their results supported the idea of using a manifold micro-channel heat sink due to higher heat transfer coefficients near the channel inlets. Lee et al [5] studied heat transfer in rectangular micro-channels and concluded that conventional numerical analysis can be used to model thermal performance of micro-channels.

Apart from effective cooling of high heat dissipating electronic chips, an additional important issue of concern is the increasing energy consumption by large computing systems such as data centers. Direct electricity consumption by data centers had already reached 1% of the total world electricity consumption by 2005 due to increased demands for IT (Information Technology) related services such as internet and telephony. The introduction of the Green 500 list for supercomputers emphasized that performance can no longer be the sole motivation for development of microprocessors and that performance per unit energy consumption is the more appropriate metric for better computing. Today, the energy for cooling conventional air-cooled data centers comprises almost half of the total energy consumption of such systems. This portion of energy use can be significantly reduced by switching to liquid cooling. This is because the much lower thermal resistance inherent in liquid use enables cooling above the free cooling limit thus eliminating the need for coolant chillers.

Additionally, and perhaps more interestingly, if hot water in the temperature range of $50^0C – 70^0C$ is used to cool electronic chips, direct utilization of the collected thermal energy becomes feasible, either using synergies with district heating or specific industrial applications. In doing so, irrespective of the cooling system being used, thermal reliability of the electronic chip must be guaranteed. This requires maintaining the chip temperature below certain upper limits as the thermal reliability of the chip reduces exponentially with chip temperature. There are two classes of parameters that influence the performance of a cooling system for electronic chips. The first set of parameters, which are directly related to cooling system, include inlet flow rate and temperature of coolant fluid, the design of heat transfer surfaces in the heat sink, the nature of the coolant fluid (liquid or gas) and the nature of flow (single or two phase). The other category of parameters lies outside the envelope of cooling system and belongs to the design of the electronic system. These parameters include chip power dissipation, thermal reliability of chip, net wiring length on an electronic board etc.

LITERATURE REVIEW

Several investigation both experimental and theoretical have been reported on the flow and heat transfer behavior of micro and mini-channel and compact passages though performing theoretical studies, or in making comparisons with experimental studies, often it is assumed that channel have regular geometry with perfectly defined surfaces. In fact, the surface irregularities produced by the fabrication technique make the channel geometry different from assumed perfect geometries, so that the assumption of perfect geometry does not predicts the performance correctly in an analysis, unless the realistic domain is analyzed.

Experimental investigation conducted by Binoy Baby and C B Sobhan [1] to investigate the fluid flow and heat transfer characteristics of a compact passage with irregular geometry surface profiles used as the boundaries of the domain. In this study the coefficient of friction and Nusselt number for the channel with irregular cross section are observed to be lesser than those for channel with regular smooth surface geometry. The result obtained from this experiment is used to validate the result that will be obtained from CFD simulation. The calculations are performed using finite difference method on a compact passage assuming fully developed laminar flow, and benchmarked using experiments on the channel with the same surface structure using interferometric measurements. The computational model has been used for the analysis of mini-channels and micro-channel flow, where derivations in flow and heat transfer from conventional correlations could be of significance, due to surface irregularities.

Many investigators have reported that flow and heat transfer parameters are very sensitive to channel cross section profile and other geometric factors. Experimental studies by Jiang et al [2] on the laminar flow of liquid in silicon Micro channels with different cross sections has shown that, for non-circular ducts the value of C_f are smaller than that of conventional values. Experimental investigations were conducted by Wu and Cheng [3] to measure the friction factor of laminar flow of water in smooth Microchannels of trapezoidal cross section with a hydraulic diameter in the range 25.9-291µm. It was observed that the coefficient of friction of these Microchannels was greatly influenced by the cross sectional aspect ratio. Silva et al [4] investigated the influence of surface phenomena on micro-scale flow using micro-particle image velocimetry (MPIV). Experimental study of on a micro-channel of hydraulic diameter 637µm with rough walls having relative roughness 1.6% and a very irregular cross section shape showed that Poiseuille numbers differed by 11%with respect with those with smooth walls. This emphasized the need to account for wall roughness in micro flows. Most of the investigation reviewed by Garimella and Sobhan [5] pointed out that the friction factor and heat transfer measurement do not agree with the conventional theories. The flow was found to be most strongly affected by the hydraulic diameter and aspect ratio of the channel. It was summarized that research on single phase flow in Microchannel has to be focused on three aspects. (i) Effect of channel geometry, dimension of flow and heat transition (ii) Reynolds number, (iii) Correlation in terms of fluid properties and Microchannel geometry. The friction factor and Nusselt number can be expected.

The objective of the work is to investigate fluid flow and temperature variation of regular cross section with that of irregular cross section. The result for the channel with surface irregularities is compared with those of a smooth channel with regular cross section shape and geometry.

NUMERICAL ANALYSIS OF MINI-CHANNEL

Computational fluid dynamics, usually abbreviated as CFD uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight test.

Mini-channel in micro-technology is a channel with a hydraulic diameter below 3 mm. These are used in fluid control and heat transfer. They are tiny channels through which fluid is directed in some types of high-power electronic cooling systems. A channel serves to bring a fluid into intimate contact with the channel walls and to bring fresh fluid to the walls and remove fluid away from the walls as the transport process is accomplished.

A number of investigation have been undertaken in the recent past to understand the fundamentals of fluid flow in a mini channel, as well as to compare the heat transfer characteristics to those in conventional channels. This work has been driven in large part by the very high heat transfer rates that can be achieved with micro channel heat sinks for electronics cooling and other applications. Micro-electro-mechanical systems (MEMS) based devices find their applications in a wide variety of emerging technologies, ranging from the micro actuators, micro sensors, micro reactors to the micro-channel heat sinks and the thermo-mechanical data storage systems, to name a few. The mini channels considered ranged in width from 4mm with the channel length being nominally 12cm. The simulation was conducted with de-ionized water, with the Reynolds number ranging from approximately.

Meshing Theory

Meshing is the task of partitioning a spatial domain into simple geometric elements such as triangles (in 2D) or tetrahedrons (in 3D). Meshes typically have to conform to boundaries. Furthermore, for many applications there are quality criteria that need to be met, such as ensuring that angles are not too small. Meshing is a huge industry, with dozens of companies selling meshing software, hundreds of companies using software, and a diverse set of applications, including graphics, geographic information systems, computer vision, and air flow and structural simulations.

In the past decade there have been many important theoretical advances in algorithm design for meshing related problems, but only some of these have made it into meshing software. The goal of this PROBE is to build a stronger link between the theory and practice of meshing. The hope is both for the algorithms community to better understand the needs of applications and for applications community to more rapidly integrate algorithmic ideas into their software. A key step of the finite element method for numerical computation is mesh generation. One is given a domain (such as a polygon or polyhedron, more realistic versions of the problem allow curved domain boundaries) and must partition it into simple ‘elements’ meeting in well-defined ways. There should be few elements, but some portions of the domain may need small elements so that the computation is more accurate there. All elements should be 'well shaped' (which means different things in different situations, but generally involves bounds on the angles or aspect ratio of the elements). One distinguishes 'structured' and 'unstructured' meshes by the way the elements meet, a structured mesh is one in which the elements have the topology of a regular grid. Structured meshes are typically easier to compute with (saving a constant factor in runtime) but may require more elements or worse-shaped elements. Unstructured meshes are often computed using quad trees, or by Delaunay triangulation of point sets, however there are quite varied approaches for selecting the points to be triangulated.

PROBLEM FORMULATION

The CFD problem discussing in this paper is the comparison of flow parameters and temperature properties of a rectangular mini channel with its modifications in shapes. The application of these mini-channels usually include liquid cooling systems used in cooling of IC chips, servers etc. In reality, a sheet of these mini-channels, which containing usually water, are placed over the IC Chips or server where the cooling is required, and the fluid flow is produced by external methods which will absorb the heat and cooling is produced. In this paper we are estimating the properties of the flow and heat transfer in the above mentioned shapes by utilizing the application of Computational Fluid Dynamics (CFD) based softwares like Fluent, Ansys etc. At first, the Fig. of each phase is drawn with the help of designing software CATIA. Then this Fig. is exported to the Workbench for further evaluation of the problem in a flow simulating software Fluent. And after a number of iterations the result graphs are obtained.

The analysis of the problem is being conducted in three phases or stages as mentioned above. At first stage, we are doing the analysis in a perfectly rectangular duct and the results were obtained for the same. In the next phase the inner rectangular surface has been modified, and will be of irregular shape and contains zigzag projections which will increase the area of contact of the flow. At third stage, a real surface property is being evaluated.

DESIGN AND ANALYSIS

The Investigation of Flow in Rectangular Duct

The flow investigation in a rectangular duct is done at the first step. The Fig. 3 shows the various views of the assembled solid and liquid domains in the CATIA software interface.
The rectangular duct has the following dimensions.

Inner rectangular duct = 4 mm × 4 mm × 120 mm
Outer rectangular duct = 8 mm × 8 mm × 120 mm

The inner rectangular duct is called as fluid domain because it is the region in which the fluid flow actually taking place. The outer rectangular duct is called as solid domain because it is made of aluminum metal for our consideration. The solid portion at the bottom of the Fig. is where a constant heat flux is given, which in practical our IC's heat is given.

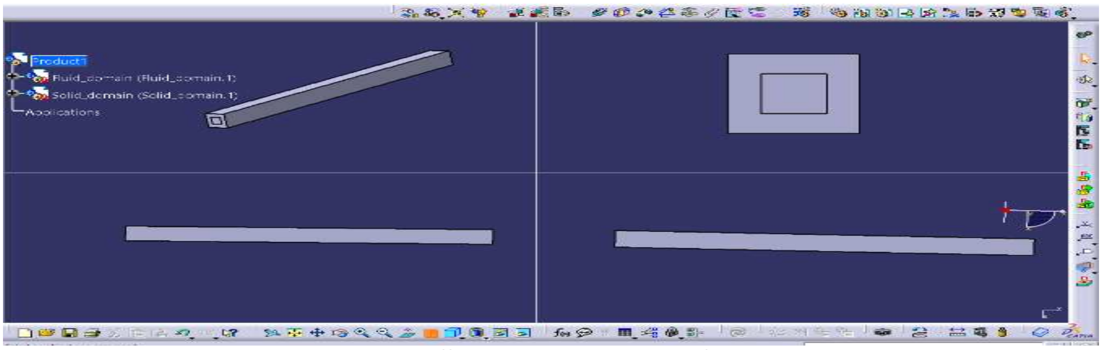

Fig. 1 Different views in CATIA interface

The Investigation of Flow in a Modified Surface with Irregularities

In this stage, as mentioned earlier the flow through a modified rectangular surface is being evaluated. The modification of rectangular surface mainly includes the presence of projections in the surface in zigzag manner. The Fig. 2 gives the idea of the modified surface. The dimensions of the Fig. 2 are the same as that of the above case. Boundary conditions and heating surfaces are the same. The fluid domain here represents the inner path at which the fluid is flowing and the solid domain is the outer metal body made of aluminum. The main difference of this case from the previous case is the presence of projected surfaces which will help the heat transfer rate by increasing the surface area.

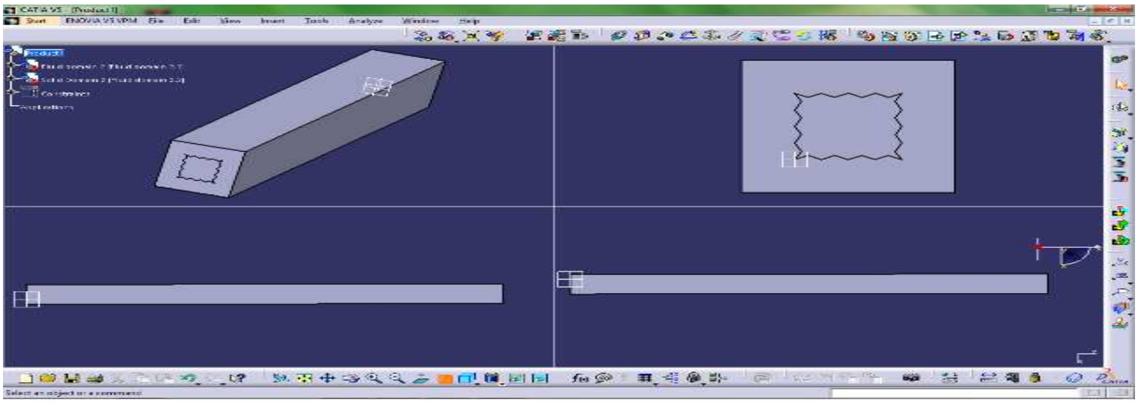

Fig. 2 Different views in CATIA interface

The Investigation of Flow in an Actual Surface

There are always a considerable difference between the actual possible rectangular channels that we can cut, in a metal and the ideal rectangular channel that we have considered in the first stage. By giving the space to consider the actual difficulties that we have to face in order to cut a rectangular duct in a metal, we can assume geometry with a random profile as shown below, as actual case of duct. In this figure the geometry's inner fluid domain has a dimension within the limits than that of the first case. The outer solid domain has the same dimension as that of the first case. All the boundary conditions and the heating values remain constant as that of the previous cases.

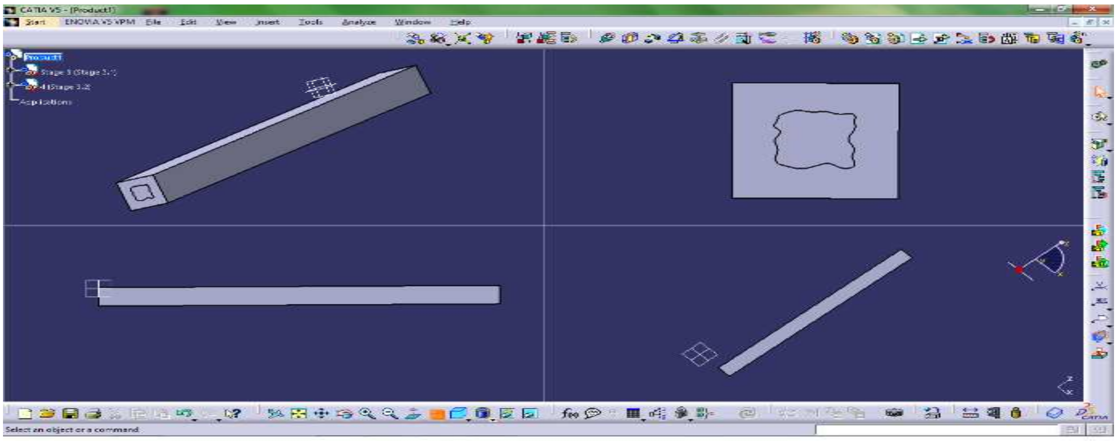

Fig. 3 Different views in CATIA interface

Boundary Conditions

Following are the boundary conditions defined for the simulation:

Inlet temperature	= 28.8^{0}C
Mass flow rate at inlet	= 1L/h.
Inlet pressure	= 101325 bar
Wall	= No slip wall
Fluid	= Incompressible fluid

When a real fluid flow past a solid body or a solid wall, the fluid particle adheres to the boundary and condition of no slip occurs. This means that the velocity of fluid close to the boundary will be same as that of the boundary.

Assumptions

The assumptions based on which simulation is formulated are the following:

- Velocity is considered to be zero at all boundaries except at channel inlet and outlet.
- There are no radiation effects.
- The flow is unidirectional.
- Buoyancy effect is neglected.
- Cavitation effects are ignored.
- The flow is incompressible and hydro dynamically and thermally fully developed.
- Steady state analysis.

The buoyancy effect is neglected as forced convection heat transfer characteristics are being investigated. Since the majority of the length of the mini-channel is in the fully developed condition. The process of heat transfer can only take place when different points in a body have different temperatures. Temperature field are defined as either steady or transient (unsteady). If in a heat flow problem the temperature at each point is constant in time, it is referred to as steady state, here temperature is a function of space coordinate. Fluid used for simulation is water, hence it is incompressible. Cavitation is the formation and then immediate implosion of cavities in a liquid *i.e.* small liquid-free zones ('bubbles') – that are the consequence of forces acting upon the liquid. It usually occurs when a liquid is subjected to rapid changes of pressure that cause the formation of cavities where the pressure is relatively low. Cavitation is a significant cause of wear in some engineering contexts. When entering high pressure areas, cavitation bubbles that implode on a metal surface cause cyclic stress through repeated implosion. This result in surface fatigue of the metal causing a type of wear also called 'cavitation'

RESULTS

The following results are obtained from the analysis after completing the calculations in the FLUENT. These results are obtained with the help of CFD-POST. The results for each phase of analysis are shown below.

Phase 1

The following results are obtained through CFD analysis of phase 1.The outlet temperature is found to be 6-7 degrees higher than inlet temperature. The images of the results are shown below. The figure shows the temperature distribution along the length. The flow is simulated along the Z axis and the above figure shows a sectional view along the length. The left end is the inlet and the right end is the outlet of the channel. The bottom portion comprises the heating surface. We observe a temperature rise of about 6 to 7 degrees for the fluid. This image shows the temperature variation on the cross-sections at two points along the length. This Fig. represents the flow behavior inside the fluid channel.

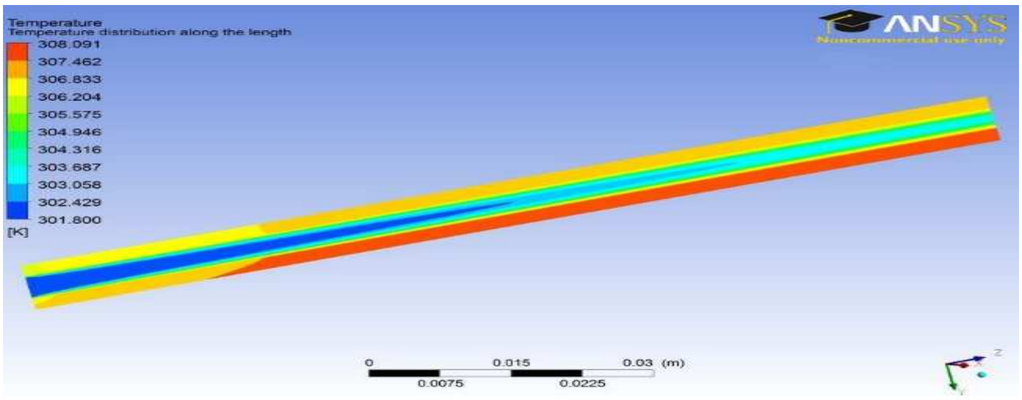

Fig. 4 Temperature distribution along the length

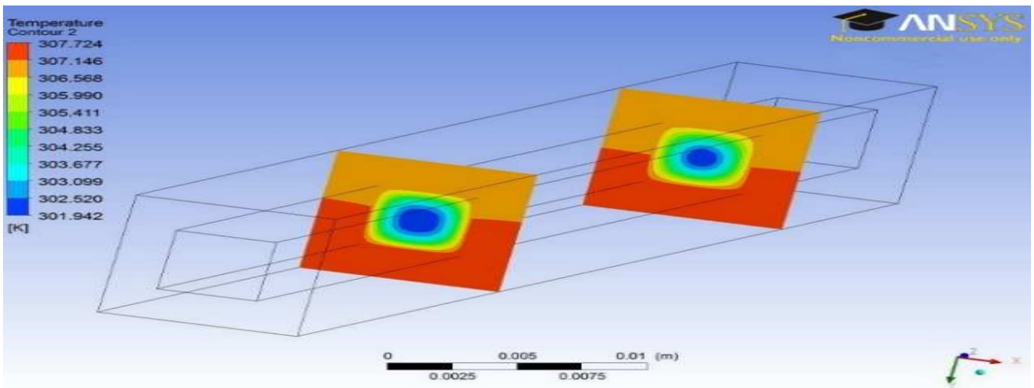

Fig. 5 Temperature variation along length

The variation of Nusselt number along the length is plotted in the above graph. From analyzing the results, it is found that the Nusselt number value was very high at the beginning and it reduces as the length proceeds. The Nusselt number varied from a value about 500 at the beginning and the value reaches a minimum of around 4 and after that the reduction diminishes and the number remains almost a constant value. These are the results obtained by the CFD analysis conducted in the first case problem.

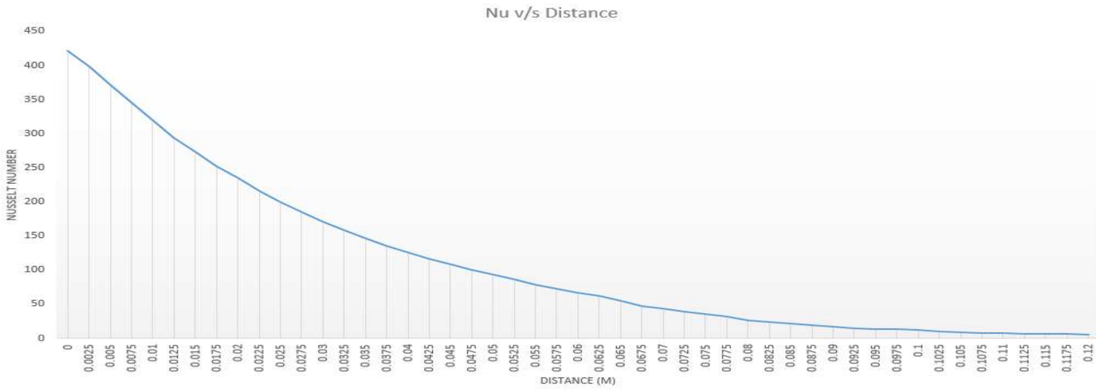

Fig. 6 Nusselt number v/s Distance

Phase 2

In the second phase we are evaluating the flow through the zigzag surface. In this evaluation the following results are obtained. There is a temperature difference of 4-5 degrees is observed between inlet and outlet. These results are plotted from the CFD-POST after completing the calculations in the FLUENT. The following Fig. shows the temperature distribution along the length. The flow is simulated along the Z axis and the above Fig. shows a sectional view along the length. The left end is the inlet and the right end is the outlet of the channel. The bottom portion comprises the heating surface. We observed a temperature rise of about 4 to 5 degrees for the fluid. This image shows the temperature variation on the cross-sections at two points along the length. This Fig. represents the flow behavior inside the fluid channel. The flow is in the direction away from the observer.

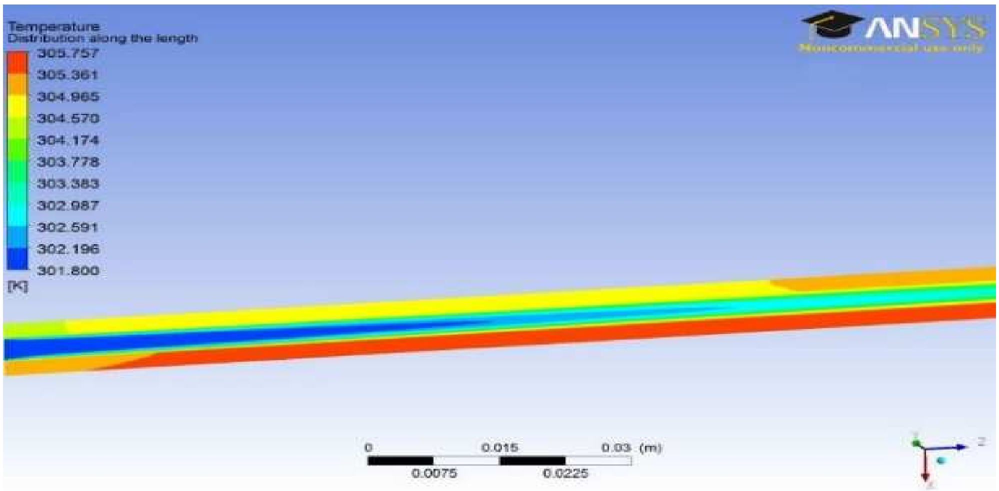

Fig. 7 Temperature variation along length

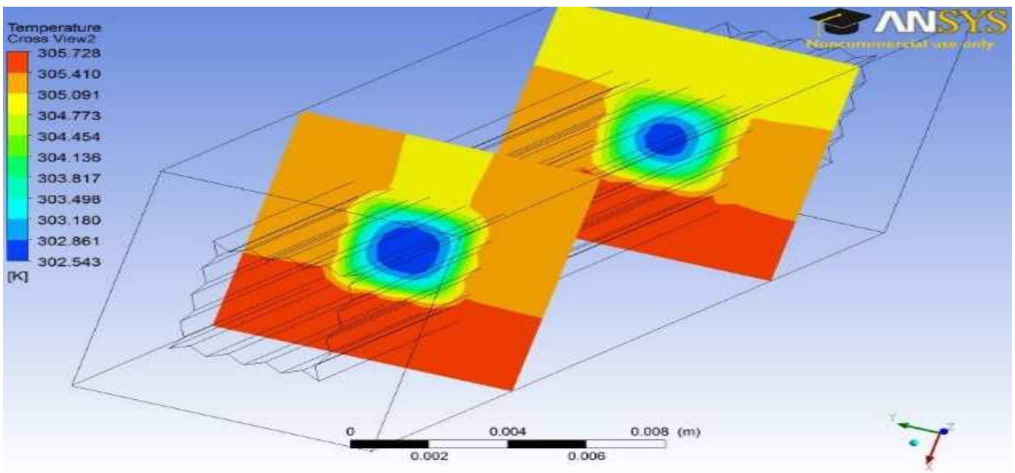

Fig. 8 Temperature variation in a cross section

The variation of Nusselt number along the length is plotted in the above graph. From analyzing the results, it is found that the Nusselt number value was very high at the beginning and it reduces as the length proceeds. The value reaches a minimum of around 6 and after that the reduction diminishes and the number remains almost a constant value. The value of Nusselt number is found greater compared to the first case, so we could expect a better heat transfer for the second case because of the Nusselt number variation. These are the results obtained by the CFD analysis conducted in the second case problem.

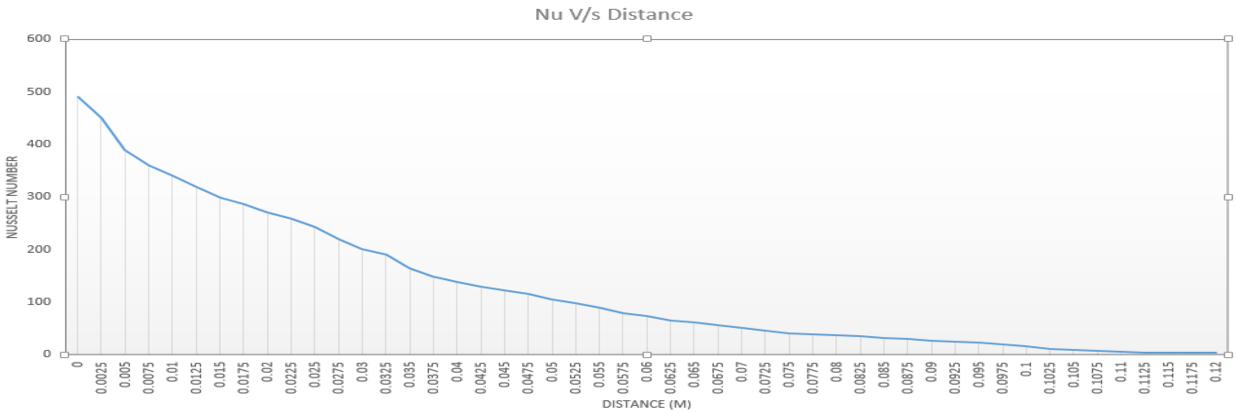

Fig. 9 Nusselt number v/s Distance

Phase 3

In phase 3, we are evaluating the flow through a real cross section channel. From the analysis a temperature difference of 1-2 degree is observed between inlet and outlet. The Fig. 10 shows the temperature distribution along the length. The flow is simulated along the Z axis and the above Fig. shows a sectional view along the length. The left end is the inlet and the right end is the outlet of the channel. The bottom portion comprises the heating surface. We observed a temperature rise of about 4-5 degrees for the fluid. This image shows the temperature variation on the cross-sections at two points along the length. This figure represents the flow behavior inside the fluid channel. The flow is in the direction away from the observer. The inlet of the channel is marked in blue color.

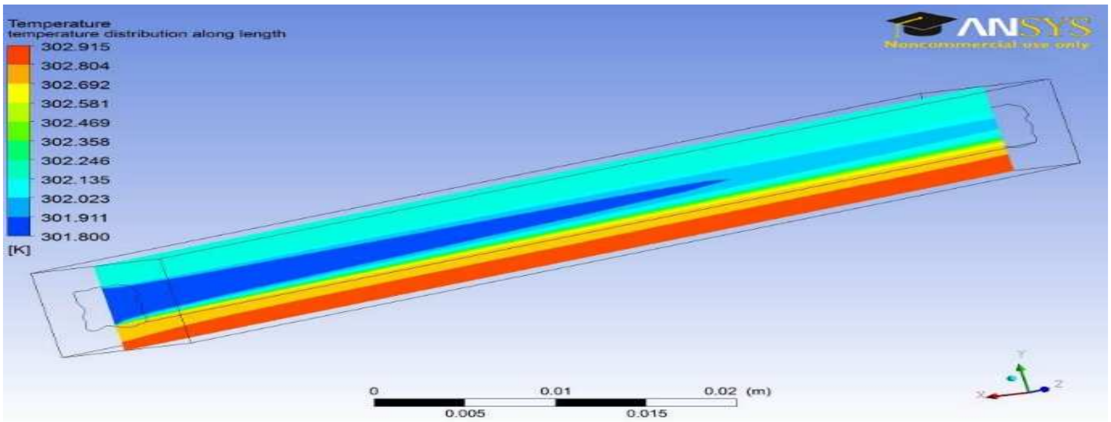

Fig. 10 Temperature variation along length

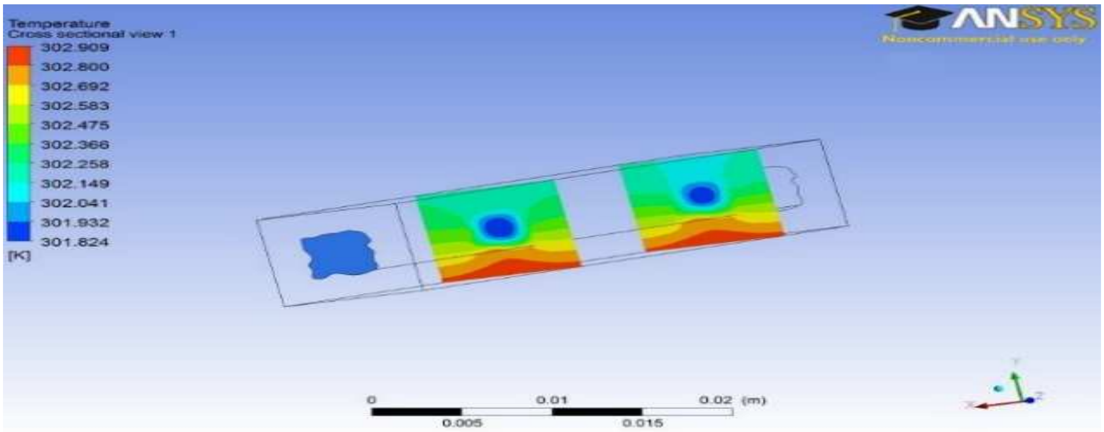

Fig. 11 Temperature variation in a cross-section

The variation of Nusselt number along the length is plotted in the above graph. From analyzing the results, it is found that the Nusselt number value was very high at the beginning and it reduces as the length proceeds. The value reaches a minimum of around 5.5 and after that the reduction diminishes and the number remains almost a constant value. The obtained values of the Nusselt number is found to be lesser than the values of the second phase, therefore we can expect a lesser heat transfer than the second case. These are the various results obtained from the analysis of the third phase.

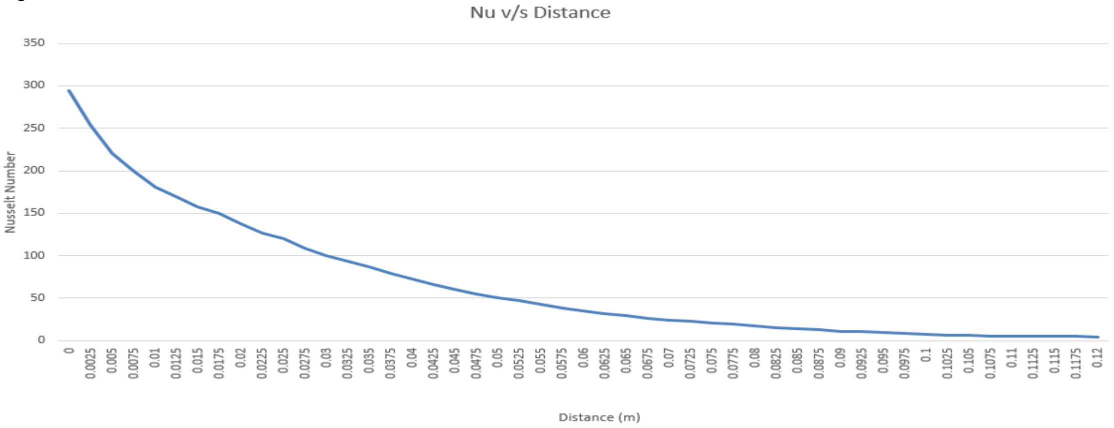

Fig. 12 Nusselt number v/s Distance

CONCLUSION

Analysis of a laminar flow through a mini-channel has been done by a numerical model method by considering the surface irregularities. The process has been carried out as three phases and comparison is made on the variation of temperature on different cross section. The flow characteristics and heat transfer of irregular cross-section channels are compared with smooth cross-section channels and the deviations are studied. From the investigation, we obtain that the actual prediction of flow and heat transfer characteristics of channels of small cross-section requires the considerations of surface irregularities while performing the numerical computations. The comparison of experimental results with computational results based on an assumption of regular surface geometries would often be mistaken as actual deviations which could be wrongly interpreted as due to fundamental differences in flow behavior in small cross-section channels.

REFERENCES

[1] Binoy Baby, C B Sobhan, Investigation on Forced Convention in Compact Passages with Surface Irregularities, *Heat Transfer Engineering*, 33, **2012**, 1105-1119.

[2] Jiang Zhou, Laminar Flow through Micro Channel used for Micro Scale Cooling System, *IEEE/CMPT Electronic Packaging Technology Conf*erence, Singapore, **1997**,119-122.

[3] Wu Cheng, Friction factors in Smooth Trapezoidal Silicon Microchannel with Different Aspect Ratios, *International Journal of Heat and Mass Transfer*, 46, **2003,** 2519-2525.

[4] Leal Silva and Semio, Micro-PIV and CFD Characterization of Flow in Microchannel, Velocity Profiles, Surface Roughness and Poiseuille Numbers, *International Journal of Heat and Fluid Flow*, 29, **2008,** 1211-1220.

[5] Sobhan Garimella, Transport in Microchannel-A Critical Review, *Annual Review of Heat Transfer*, Begell House, New York, **2003**, 1-50.