A

THESIS

ON

STUDY OF HEAT TRANSFER ENHANCEMENT IN MICROCHANNELS USING COMPUTATIONAL FLUID DYNAMICS MODELS

SUBMITTED

BY

SATYA SOUMENDRA BEHURA

ROLL NO:-109CH0467

SESSION:-2012-13

for the partial fulfillment of

Bachelor of Technology
in
Chemical Engineering

UNDER THE ABLE GUIDANCE
OF

Prof.(Dr.) BASUDEB MUNSHI

Department of Chemical Engineering, National Institute of Technology,Rourkela-769008

DEPARTMENT OF CHEMICAL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA

ORISSA, INDIA - 769008

CERTIFICATE

This is to certify that the project entitled "**Study of Heat Transfer Enhancement in Microchannels using Computational Fluid Dynamics (CFD) Models "** has been successfully completed by **Satya Soumendra Behura** (109CH0467) during the academic year 2012-2013 in partial fulfillment of the requirements for the award of the Degree of Bachelor of Technology in Chemical Engineering at National Institute of Technology, Rourkela. The entire work has been carried out under my supervision. He has fulfilled all the prescribed requirements. His project work during this period has been satisfactory. He bears a good honest character and is physically and mentally fit to get the above-mentioned degree.

DATE:-6th MAY, 2013

Prof. (Dr.) BASUDEB MUNSHI
Associate Professor
Department of Chemical Engineering
National Institute of Technology, Rourkela

ACKNOWLEDGEMENT

I would like to express my deepest gratitude and heartfelt thanks to Prof. Basudeb Munshi for his guidance and assistance in my project work. Without his sincere and kind effort, this project would not have successfully completed.

I am also thankful to all the staff and faculty of Chemical Engineering Department, National Institute of Technology, Rourkela for their constant encouragement.

I would like to extend my sincere thanks to all my friends and colleagues who assisted me completing the project directly or indirectly.

Last but not the least I am also thankful to Mr. Akhilesh Khapre for devoting his priceless hours in helping me learning ANSYS and providing relevant guidance for the successful completion of the project.

-Satya Soumendra Behura
Roll no:- 109CH0467
Session:-2012-2013

CONTENTS

ABSTRACT

Microchannel heat transfer operations involve reaction chamber whose sizes are typically in the range of micrometers (μm) with volumetric capacity in the range of micro liters (μL).While electronics now provide the ‘brains’ to the electrical world, microchannel devices act as ‘sensors and actuators’ to the outside world providing the same benefits as the microchips provide to the electronics. Their high surface to volume ratio, highly efficient heat and mass transfer characteristics and greatly improved fluid mixing allow precision control of reaction with enhanced conversions, selectivity and yields of desired products, easily repairable tendency have resulted in their increasing demand and they are the key components in automobile airbags, ink-jet printers, projection display systems, aircrafts etc. They can even perform reactions on a very small scale to determine the potential for dangerous situations. Reverse-flow action is used to utilize the thermal energy inside a reactor.

The present study emphasizes on the enhancement of heat transfer in microchannnels by using vortex promoters. The effects on thermal and hydrodynamic behaviour produced by vortex promoters of various shapes in a 2D, laminar flow in a microchannel are studied.The liquid is assumed to be a Newtonian fluid (water) and a Non-Newtonian fluid (Banana puree/CMC solution) .It is desired to obtain a suitable design criteria of microcooling systems, which should be both thermally efficient and not expensive in terms of the pumping power. Three reference cross sections, namely circular, rectangular, triangular are considered. The effect of the blockage ratio, the Reynolds number, and the relative position and orientation of the obstacle are also studied. The plots of Surface Nusselt Number vs Length of rectangular channel are studied to find the most heat efficient design. In addition to this, a microchannel with no obstacle is taken and the effect on heat transfer by reducing the height of the channel is also studied. Some design guidelines based on the above plots ,which could be used in an engineering environment are provided. The geometry of the problem and meshing of the problem have been made in ANSYS Workbench.The models have been solved by ANSYS Fluent 13.0 solver.

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

ρ	Density of fluid
p	Static pressure
∇	Gradient
E	Activation energy
μ	Viscosity
v	Velocity
t	Time
g	Acceleration due to gravity
M	Molecular weight
Re	Reynolds Number
T	Temperature
α	Aspect ratio
β	Blockage ratio
F	External Forces
d_h	Horizontal size
d_v	Vertical size
K	Thermal conductivity
k	Consistency index
Cp	Specific Heat
h	Heat Transfer Coefficient
Nu	Nusselt Number
n	Power law index

Subscripts

j	Species

CHAPTER 1

INTRODUCTION

CHAPTER-1

INTRODUCTION

Recently scientists have found out that not only electrical devices but also mechanical devices, may be miniatured and batch-fabricated providing the same benefits to the mechanical world as IC technology has given to the electronics world. While electronics now provide the “brains” for today’s advancing systems, micromechanical devices can provide the “sensors and actuators” -the eyes, ears, hands and feet-which connect to the outside world.

1.1 MICROCHANNEL AND ITS USES

Microchannel flow has been extensively studied during the last decade because of its interest from both the scientific and the technological point of view. Laminar flow in ducts with hydraulic diameters ranging from tenths to hundredths of microns provide some unique features that are still being discussed and remain somewhat controversial. One major thing we find is the high rate of heat transfer and enhancing the same has been studied in this context.

On the other hand, microchannel flow exhibits practical applications such as in the conceptual design of many systems/subsystems that are relevant in various industrial sectors. For example, providing more compact and lighter designs is very much attractive in aerospace engineering because, at the end of the day, we see economic revenues. Weight and space are primary factors considered in modern aircraft design, and microheat exchangers are the ideal choices to perform the thermal control of a new generation of high capability avionics characterized by the large heat dissipation.

In connection with specific aerospace engineering applications, if thermal efficiency is the main priority, as it happens in defence applications, active systems might be preferred, even though in this case the designer should not forget pressure drop, to avoid increasing too much the power of the pump and hence cost.In this context, optimization of micro-cooling systems behavior is an issue that may have a non-negligible economic impact.

Concerning heat transfer in microchannels, it should always be noted that steady, laminar flow in a standard channel is almost parallel (non-parallelism being only due to dependence of viscosity on temperature) which means that thermal conductivity (which is quite low in ordinary liquids) is the only mechanism for heat transport from the hot wall to the bulk. Nevertheless, heat transfer in these systems can be enhanced by vortex promoters (built-in obstacles in the micro-channels), since they produce transversal convection, which is a quite effective heat transport mechanism if the produced vortices exhibit appropriate size and intensity.

1.2 COMPUTATIONAL FLUID DYNAMICS & ITS USES

Computational Fluid Dynamics or CFD is the analysis of systems involving fluid flow phenomena, heat transfer and associated phenomena such as chemical reactions by means of computer based-simulation. The technique is very powerful and spans a wide range of industrial and non-industrial application areas. Some examples are:-

- Aerodynamics of aircrafts and automobiles.
- Hydrodynamics of ships.
- Powerplant: Combustion in IC engines and gas turbines.
- Electrical and electronic engineering.
- Chemical process engineering,etc.

Fluid (gas and liquid) flows are governed by partial differential equations (PDE) which represent conservation laws for the mass, momentum and energy. Computational Fluid Dynamics (CFD) is used to replace such PDE systems by a set of algebraic equations which can be solved using digital computers. These equations are solved iteratively until the solution reaches the desired accuracy (ANSYS Fluent 13I.0). CFD provides a qualitative prediction of fluid flows by means of

- Pre-processor:-Definition of geometry(The computational domain) , Grid generation ,Selection of chemical and physical phenomena that need to be modeled, Definition of fluid properties ,Specifying boundary conditions at cells.
- Solver:-Integration of the governing equations, Discretisation(converting the resulting integral equations into a system of algebraic equations, Solution of the algebraic equations using an iterative method.
- Post-processor:-Domain geometry and grid display, vector plots, particle tracking etc.

1.3 ANSYS FLUENT SOFTWARE

FLUENT is one of the widely used CFD software package. ANSYS FLUENT software contains the wide range of physical modeling capabilities which are needed to model flow, turbulence, reactions and heat transfer for industrial applications. Features of ANSYS FLUENT software:

- TURBULENCE: ANSYS FLUENT offers a number of turbulence models to study the effects of turbulence in a wide range of flow regimes.
- MULTIPHASE FLOW: It is possible to model different fluids in a single domain with FLUENT.
- ACOUSTICS: It allows users to perform sound calculations.

- REACTING FLOW: Modelling of surface chemistry, combustion as well as finite rate chemistry can be done in fluent.
- MESH FLEXIBILITY: ANSYS FLUENT software provides mesh flexibility. It has the ability to solve flow problems using unstructured meshes. Mesh types that are supported in FLUENT includes triangular, quadrilateral, tetrahedral, hexahedral, pyramid, prism (wedge) and polyhedral. The techniques which are used to create polyhedral meshes save time due to its automatic nature. A polyhedral mesh contains fewer cells than the corresponding tetrahedral mesh. Hence convergence is faster in case of polyhedral mesh.
- DYNAMIC AND MOVING MESH: The user sets up the initial mesh and instructs the motion, while FLUENT software automatically changes the mesh to follow the motion instructed.
- POST-PROCESSING AND DATA EXPORT: Users can post-process their data in FLUENT software, creating among other things contours, path lines, and vectors to display the data.

In this context, the geometry of the problem and meshing of it have been made in ANSYS Workbench. The models have been solved by ANSYS Fluent 13.0 solver.

1.4 OBJECTIVE OF THE PRESENT STUDY

1. A 2D, unsteady, laminar flow of liquid in a non-isothermal rectangular micro-channel is considered.
2. The liquid is first assumed to be a Newtonian Fluid (WATER) and then a Non-Newtonian fluid (BANANA PUREE).
3. The case of a single obstacle (vortex promoter) with various cross sectional shapes, namely circular, rectangular, triangular and different aspect ratios has been studied. In addition, the blockage ratio, the position and orientation of the obstacle are varied.
4. For the above mentioned cases, the heat transfer rate enhancement and the pressure drop increase are studied. To ensure that our predictions are as close as possible to practical applications, temperature dependent fluid viscosity and thermal conductivity are taken.
5. Finally from the Nusselt Number plots, the design with the most efficient heat transfer is selected which also reduces the pumping cost due to pressure loss.

CHAPTER 2

LITERATURE REVIEW

CHAPTER-2

LITERATURE REVIEW

Some experimental and theoretical work on microchannels and their heat transfer enhancement has been done in the last few decades. Both the industrial and academic people have taken interest in this area. The following is a review of the research that has been completed especially on heat transfer in microchannels.

2.1 STUDY OF HEAT TRANSFER IN MICROCHANNELS

Meis et al (2010) studied the heat transfer enhancement in microchannels using vortex promoters. Here a laminar 2D flow of water was considered with vortex promoters of circular and rectangular shapes. Different designs were studied and a useful design was chosen which both thermal efficient and not expensive in terms of pumping power based on the plots of thermal efficiency and pressure drop.

Rahnama and Moghaddam (2005) considered the case of a square vortex promoter who obtained a correlation between Nusselt and Reynolds numbers.

Turki et al (2003) studied two-dimensional laminar fluid flow and heat transfer in a channel with a built-in heated square cylinder and addressed mixed convection and generated engineering correlations between the relevant parameters.

Icoz and Jaluria (2006) considered a 3D rectangular channel with a transversal aspect ratio of 6 which took the wall region effects. The vortex promoters had circular, square, and hexagonal shapes, and were located ahead of two tandem heating sources. Different blockage ratios were considered. The authors reported that the hexagonal shape is best to remove heat from the first heat source and the circular one is optimal when heat removal from the second source is the main aim. The square vortex promoter gave a reasonable combination of heat transfer and pressure drop.

Abbasi et al (2001) studied forced convection in a lane channel with a built-in triangular prism. He saw an increase of 85% in the time averaged Nusselt number at Reynolds Number of 250.

Nitin and Chhabra (2005) considered a non-isothermal rectangular channel in which a rectangular vortex promoter is immersed in a non-Newtonian fluid (power-law fluid), and reported heat transfer variations of 10% and a strong sensitivity on the power law which is used to describe non-Newtonian behaviour.

Valencia (2000) studied the flow in the turbulent region (Re >2000) who reported a heat transfer enhancement of the order of 30%, with a fivefold increase in the friction factor. High Reynolds Number are really effective in enhance cooling. Small Reynolds Number (closer to 2000) have almost no effect on cooling and produce a large viscous dissipation that highly increases the pressure drop.

The use of tandem cylinders as mixing promoters is also the current subject of much research interest. Fluid dynamic issues have been studied by **Papaioannou et al and Tasaka et al (2006)** while heat transfer effects have been considered by **Niu et al. and by Valencia and co-workers (2006).**

Reyes et al (2010) studied heat transfer and pressure drop in microchannel based heat sinks having tip clearance. In this, optimisation of micro-heat sink configurations is studied when both thermal effects and pressure drop are taken into consideration. The working fluid taken was water and, according to typical power dissipation and system size requirements, the considered fluid regime was either laminar or transitional, and not fully developed from the hydrodynamics point of view. Five configurations were considered: a reference geometry (selected for comparison purposes) made up of square section micro-channels, and four alternative configurations that involved the presence of a variable tip clearance in the design. The performance of the different configurations was compared with regard to both cooling and pressure drop.

Wang and Peng (1995) investigated experimentally the single-phase forced convective heat transfer characteristics of water/methanol flowing through micro-channels with rectangular cross section of five different combinations, maximum and minimum channel size varying from (0.6 × 0.7 mm2) to (0.2 × 0.7 mm2). The results provide significant data and considerable insight into the behavior of the forced-flow convection in micro-channels.

Peng and Peterson (1996) also investigated experimentally the single-phase forced convective heat transfer micro channel structures with small rectangular channels having hydraulic diameters of 0.133–0.367 mm and distinct geometric configurations. The results indicate that geometric configuration had a significant effect on single-phase convective heat transfer and flow characteristics. The laminar heat transfer found to be dependent upon the aspect ratio i.e. the ratio of hydraulic diameter to the centre to centre distance of micro channels. The turbulent flow resistance was usually smaller than predicted by classical relationships.

Fedorov and Viskanta (2000) developed a three dimensional model to examine the conjugate heat transfer in a micro channel heat sink with the same channel geometry used in the experimental work done by Kanwano. This investigation indicated that the average channel wall temperature along the flow direction was nearly same except in the region close to the channel inlet, where very large temperature gradients were observed. This allowed them to conclude that the thermo–properties are temperature dependent.

Jiang et al. (2001) performed an experimental comparison of microchannel heat exchanger with microchannel and porous media. The effect of the dimensions on heat transfer was analyzed numerically. It was reported that the heat transfer performance of the microchannel heat exchanger using porous media is better than using of microchannels, but the pressure drop of the former is much larger.

Qu and Mudawar (2004) conducted a three-dimensional fluid flow and heat transfer analysis for a rectangular micro channel heat sink .This model considered the hydrodynamic and thermal developing flow along the channel and reported that the Reynolds number would influence the length of the developing flow region. It was also found that the highest temperature typically occurs at the heated base surface of the heat sink immediately adjacent to the channel outlet and that the temperature rise along the flow direction in the solid and fluid regions can both be approximated to be linear .

Mishan et al. (2007) worked on heat transfer and fluid flow characteristic of a rectangular microchannel experimentally, taking water as a working fluid. The experimental results of pressure drop and heat transfer show that including the entrance effects, the conventional theory is applicable for water flow through microchannels. They have developed new method for measurement of fluid temperature distribution and it gives the fluid temperature distribution inside the channel.

Jung et al (2009) have studied experimentally the heat transfer coefficients and friction factor of Al_2O_3 with diameter of 170 nm in a rectangular micro channel. Noticeable enhancement of the convective heat transfer coefficient of the nanofluids with the base fluid Literature Review of water and a mixture of water and ethylene glycol at the volume fraction of 1.8 volume percent was obtained without major friction loss. It has been found that the Nusselt number increases with increasing the Reynolds number in laminar flow regime, which is opposing to the result from the conventional analysis.

CHAPTER 3

COMPUTATIONAL FLUID DYNAMICS MODEL EQUATIONS

CHAPTER-3

COMPUTATIONAL FLUID DYNAMICS MODEL EQUATIONS

In this study only single phase model is considered. Hence this model will calculate one transport equation for the momentum and one for continuity for each phase, and then energy equations are solved to study the thermal behaviour of the system. The theory for this model is taken from the ANSYS Fluent 13.0.

3.1 SINGLE PHASE MODELING EQUATIONS

The single phase model equations include the equation of continuity, momentum equation and energy equation (ANSYS Fluent 12.0). The continuity and momentum equations are used to calculate velocity vector. The energy equation is used to calculate temperature distribution and wall heat transfer coefficient. The equation for conservation of mass, or continuity equation, can be written as follows:

3.1.1Mass Conservation Equation

The equation for conservation of mass, or continuity equation for compressible or incompressible flow can be written as follows:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\vec{v}) = 0 \qquad (3.1)$$

3.1.2Momentum Conservation Equation

Conservation of momentum in an inertial (non-accelerating) reference frame is described by

$$\frac{\partial(\rho\overline{v})}{\partial t} + \nabla.(\rho\vec{v}\vec{v}) = -\nabla P + \nabla.(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F} \qquad (3.2)$$

Where P is the static pressure, τ is the stress tensor (described below), and $\rho\vec{g}$ and $\vec{F}$ are the gravitational body force and external body forces (e.g., that arise from interaction with the dispersed phase), respectively. $\vec{F}$ also contains other model dependent source terms such as porous-media and user-defined sources.

The stress tensor is given by

$$\bar{\bar{\tau}} = \mu\left[(\nabla\vec{\upsilon} + \nabla\vec{\upsilon}^{T}) - \frac{2}{3}\nabla.\vec{\upsilon}I\right] \qquad (3.3)$$

3.1.3 Energy equation

ANSYS FLUENT solves the energy equation in the following form:

$$\frac{\partial}{\partial t}(\rho E)+\nabla.(\vec{\upsilon}(\rho E+p))=\nabla.\left(\kappa_{eff}\nabla T-\sum_j h_j\overrightarrow{J_j}+(\overline{\overline{\tau}}_{eff}.\vec{\upsilon})\right)+S_h \tag{3.4}$$

Where K*eff* is the effective conductivity (*K* + *Kt)* where K*t* is the turbulent thermal conductivity, defined according to the turbulence model being used), and *J j* is the diffusion flux of species *j*. The first three terms on the right-hand side of Equation represent energy transfer due to conduction, species diffusion, and viscous dissipation, respectively. S_h includes the heat of chemical reaction, and any other volumetric heat sources.

Also,

$$E=h-\frac{p}{\rho}+\frac{\upsilon^2}{2}$$

$$h=\sum_j Yjhj$$

$$hj=\int_{Tref}^{T} Cp,j\,dT$$

Tref is used as 298.15K

CHAPTER 4

SIMULATION OF 2D MICROCHANNEL

CHAPTER-4

SIMULATION OF 2D MICROCHANNEL

4.1 HEAT TRANSFER ENHANEMENT IN RECTANGULAR MICROCHANNELS BY USING VORTEX PROMOTERS

A study of the effects of heat transfer and pressure drop produced by vortex promoters of various shapes in a 2D, laminar flow in a micro-channel is carried out. The liquid is assumed to be a Newtonian fluid (water) and a Non-Newtonian Fluid (Banana Puree), with temperature dependent viscosity and thermal conductivity. It is intended to obtain useful design criteria of micro-cooling systems, taking into account that practical solutions should be both thermally efficient and not expensive in terms of the pumping power. Three reference cross sections, namely circular, rectangular and triangular, at various aspect ratios are considered. The effect of the blockage ratio, the Reynolds number, and the relative position and orientation of the obstacle are also studied.

4.2 PROBLEM STATEMENT-1

Consider a 2D, unsteady, laminar flow of water in a non-isothermal microchannel. A circular/rectangular/triangular obstacle is placed cross-flow wise inside the channel as shown in figure. Walls are adiabatic except in a portion of the bottom wall located downstream of the obstacle (a distance of 1.67 microns) where the temperature is maintained at Tmax= 353 K, while the temperature of the incoming flow is Tinlet =293 K. The blockage ratios were varied and the effect on Nusselt Number is studied. In addition to this , effect of obstacle's orientation and position on Nusselt Number is also studied.

4.3 GEOMETRY AND MESHING

The Computational domain of a circular micro channel is represented in two dimensional (2D) form by a rectangle and displayed below in Figure 4.1. The geometry consists of two walls, a centerline, an inlet, an outlet boundaries, a circular obstacle. The height (H), the length(L) of the pipe, Diameter of circlular obstacle (D) are specified in the Figure 4.2. It is constructed in ANSYS 13.0 Workbench and it is then fine meshed with quadrilateral meshing method. A Refinement of the order of 3 is also applied to get the final mesh as shown in Figure 4.3.

For the circular obstacle, L=3.5 micrometer, H=0.5 micrometer , D=0.05, 0.25, 0.125 micrometer

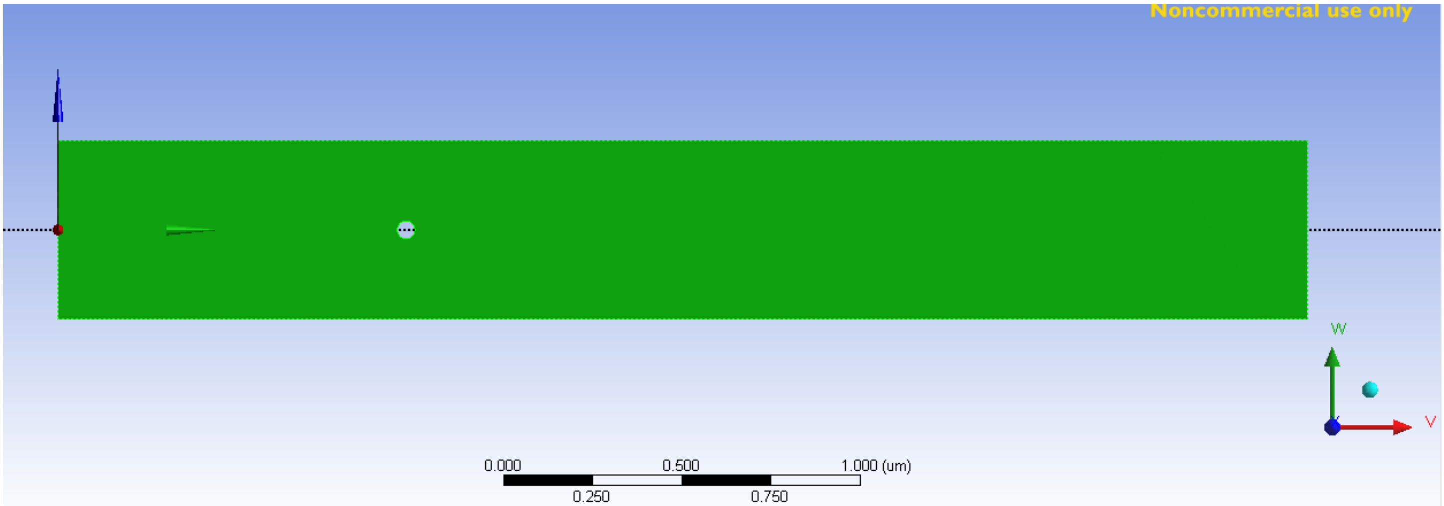


Figure 4.1:- Geometry of the problem showing a circular obstacle inside the microchannel

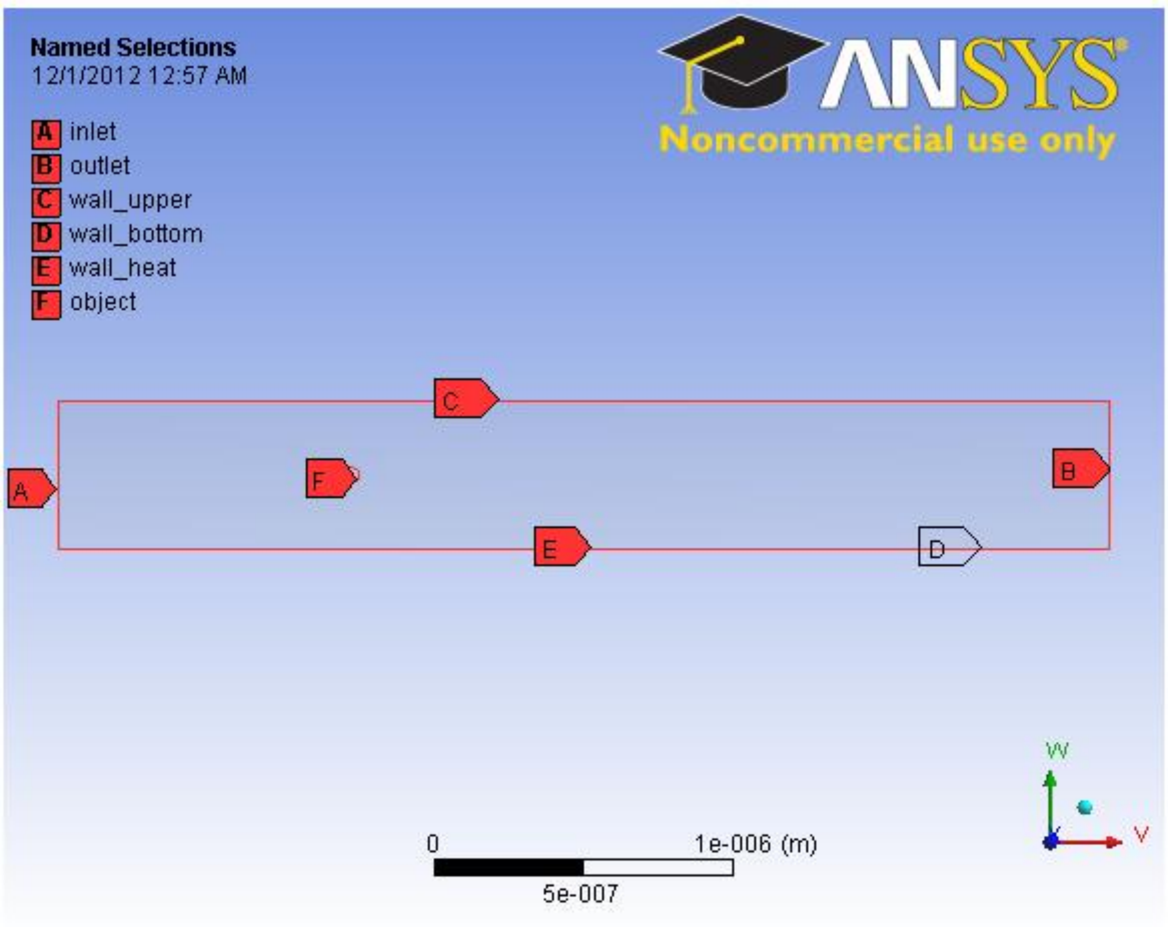
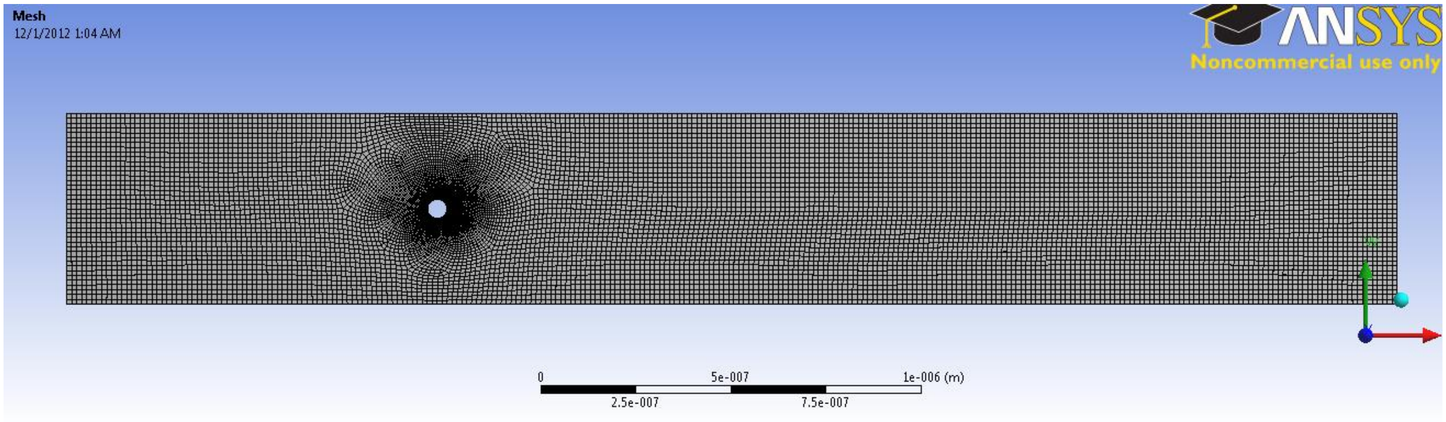


Figure 4.2: Geometry's Named Selections

The geometry is meshed into 14924 nodes and 14588 elements for D=0.05um using refinement tool and quadrilateral meshing method as shown in Figure 4.3 .



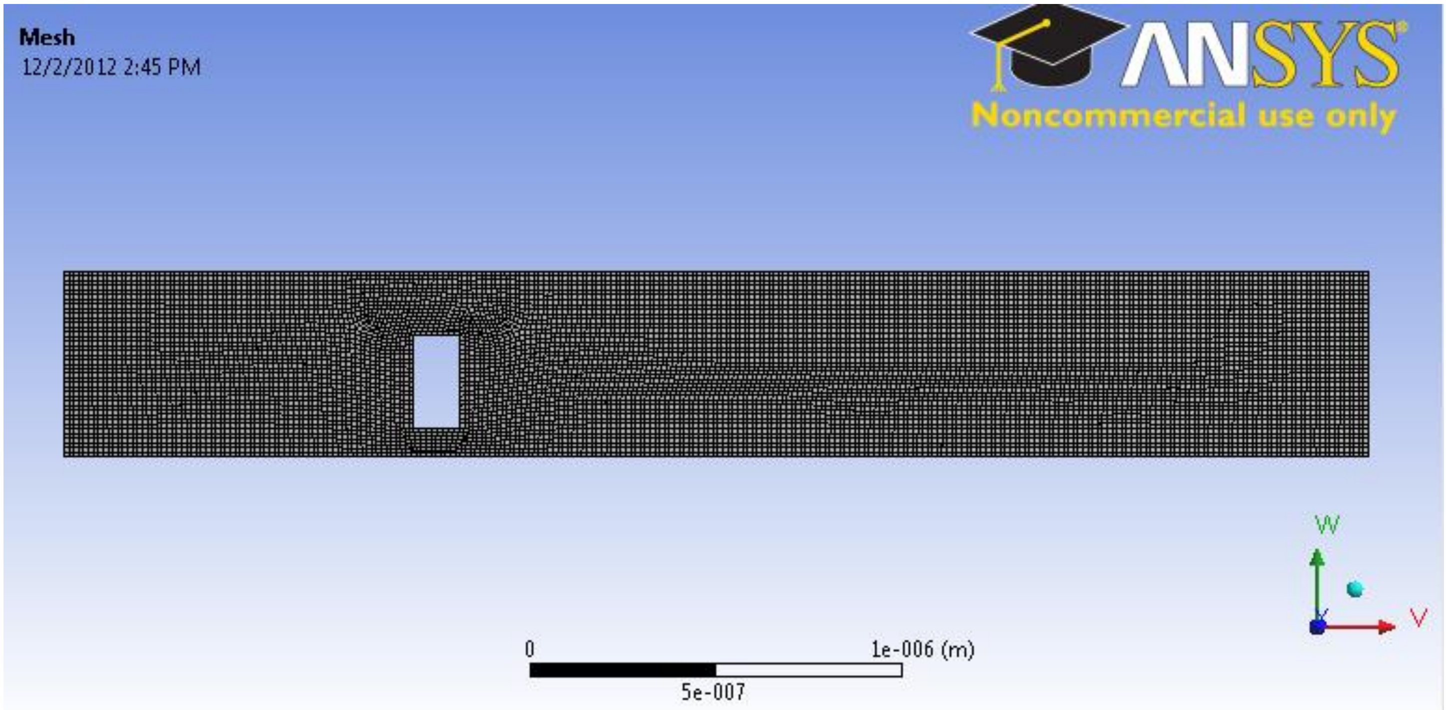


Figure 4.3:- Meshing of the microchannel with circular and rectangular obstacle.

4.4 GOVERNING EQUATIONS

The governing equations for continuity , momentum, energy are as follows:-

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{4.1}$$

$$\frac{\partial u}{\partial t} + u\,\frac{\partial u}{\partial x} + \,v\frac{\partial u}{\partial y} = \,-\frac{\partial p}{\partial x} + \frac{1}{Re}\left(\frac{\partial\,\left(\mu\frac{\partial u}{\partial x}\right)}{\partial x} + \frac{\partial\,\left(\mu\frac{\partial u}{\partial y}\right)}{\partial y} + \frac{\partial\mu}{\partial x}\frac{\partial u}{\partial x} + \frac{\partial\mu}{\partial y}\frac{\partial v}{\partial x}\right) \tag{4.2}$$

$$\frac{\partial v}{\partial t} + u\,\frac{\partial v}{\partial x} + \,v\frac{\partial v}{\partial y} = \,-\frac{\partial p}{\partial y} + \frac{1}{Re}\left(\frac{\partial\,\left(\mu\frac{\partial v}{\partial x}\right)}{\partial x} + \frac{\partial\,\left(\mu\frac{\partial v}{\partial y}\right)}{\partial y} + \frac{\partial\mu}{\partial y}\frac{\partial v}{\partial y} + \frac{\partial\mu}{\partial x}\frac{\partial u}{\partial y}\right) \tag{4.3}$$

$$\frac{\partial T}{\partial t} + u\,\frac{\partial T}{\partial x} + \,v\frac{\partial T}{\partial y} = \,-\,\frac{1}{RePr}\left(\frac{\partial\,\left(k\frac{\partial T}{\partial x}\right)}{\partial x} + \frac{\partial\,\left(k\frac{\partial T}{\partial y}\right)}{\partial y}\right) \tag{4.4}$$

4.5 BOUNDARY CONDITIONS

First a Reynolds Number of 600 is chosen based on the hydraulic diameter (twice the channel height) and the effects of following properties are studied for water coming in at a temperature of 293K and the non-adiabatic portion of the bottom wall being maintained at 353K.Inlet is velocity-inlet and outlet is pressure outlet. Fluid flowing is water with its viscosity and thermal conductivity increasing as a polynomial function of T as follows:-

$$\mu = 1 - \,\mu_1 T + \,\mu_2 T^2 \tag{4.5}$$

$$k = 1 + \,k_1 T + \,k_2 T^2 \tag{4.6}$$

$\boldsymbol{\mu_1}$**=1.1292** $\boldsymbol{\mu_2}$ **= 0.4904** $\boldsymbol{k_1}$**= 0.1572** $\boldsymbol{k_2}$ **=0.047**

The effect of the following properties of the configuration are studied:-

- Obstacle shape (circle, rectangle and triangle) and aspect ratio $\alpha = \frac{dh}{dv}$ where d_h and d_v are the horizontal and vertical sizes, when the obstacle is placed in symmetric position.
- Channel blockage ratio $\beta = \frac{dv}{h}$ where h is the channel height.
- Obstacle orientation and position.

4.6 SIMULATION WITH NON-NEWTONIAN FLUIDS

After carrying out the simulation with water , simulation with Non-Newtonian Fluids is carried out. The Non-newtonian fluids considered here are psuedoplastic fluids. Here Banana Puree and Carboxymethyl Cellulose (CMC solution (100ppm)) are considered which obey the power-law equation given below with the following properties in Table 4.1.

$$\mu = k\gamma^{n-1} \tag{4.7}$$

Where $\mu = viscosity, k = consistency\ index\ , \gamma = rate\ of\ shear$

Table 4.1:- Properties of Non-Newtonian Fluids [4]

Fluid	Power law index(n)	Consistency Index (k)	Density (ρ) kg/m^3	Specific Heat Cp (J/kg.K)	Thermal Conductivity (W/m.K)
Banana Puree	0.458	0.0065	1115	3642.5	0.692
CMC (100ppm)	0.951	0.0038	4000	4100	0.7

4.7 RESULTS AND DISCUSSIONS

SIMPLE scheme was used to find the solution with Second Order UPWIND spatial discretization method. Transient Formulation is First order IMPLICIT. Time step size =0.001 sec, No of time steps=100 , Maximum iterations per step=30.

4.7.1 SIMULATION USING A CIRCULAR OBSTACLE :-Following are the instantaneous (absolute value of) vorticity & velocity contours obtained for a circular obstacle at the indicated blockage ratios and Re = 600 when the obstacle is located at 1micro meter from the inlet side.

1. $\beta = 1/10$

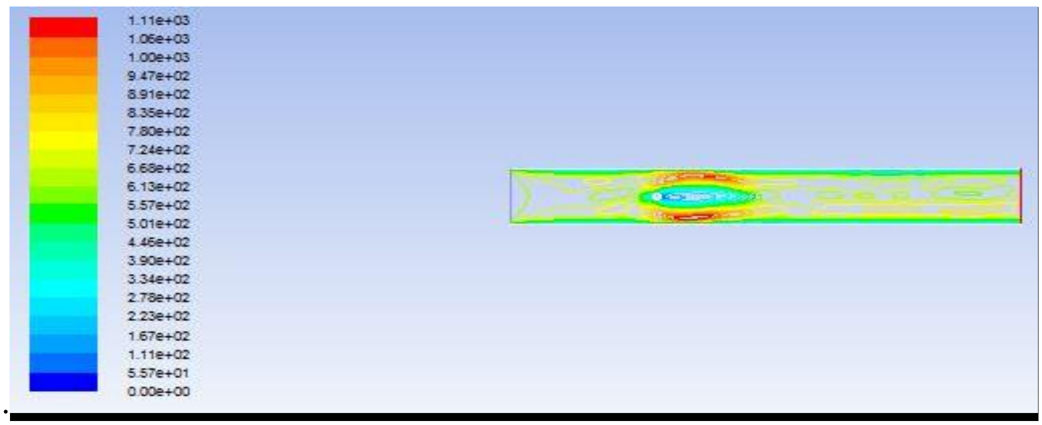


Figure 4.4:- Instantaneous (absolute value of) velocity contours for a circular obstacle at the mentioned blockage ratio and Re = 600

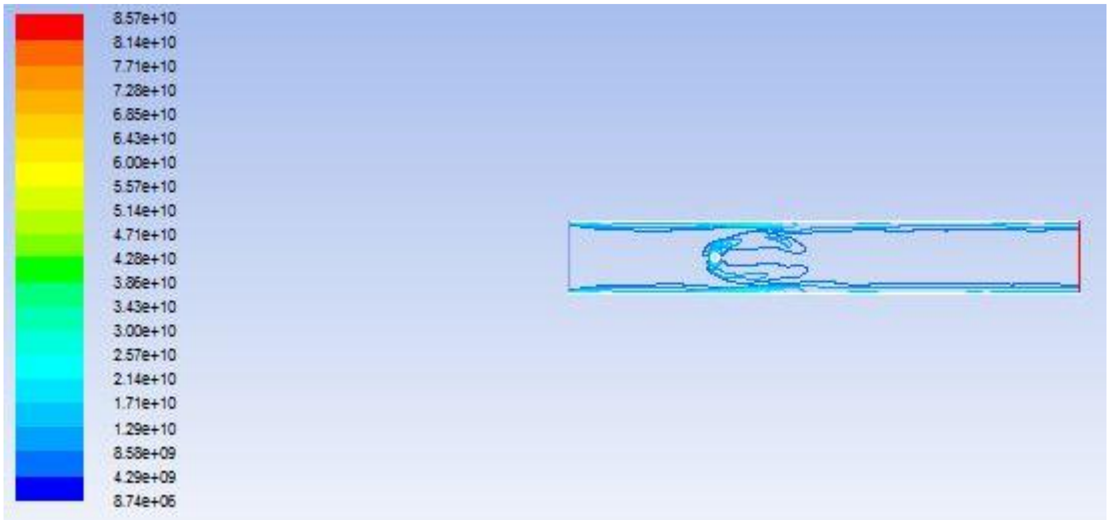


Figure 4.5 :- Instantaneous (absolute value of) vorticity contours for a circular obstacle at the mentioned blockage ratio and Re = 600

2. $$\beta = 1/4$$

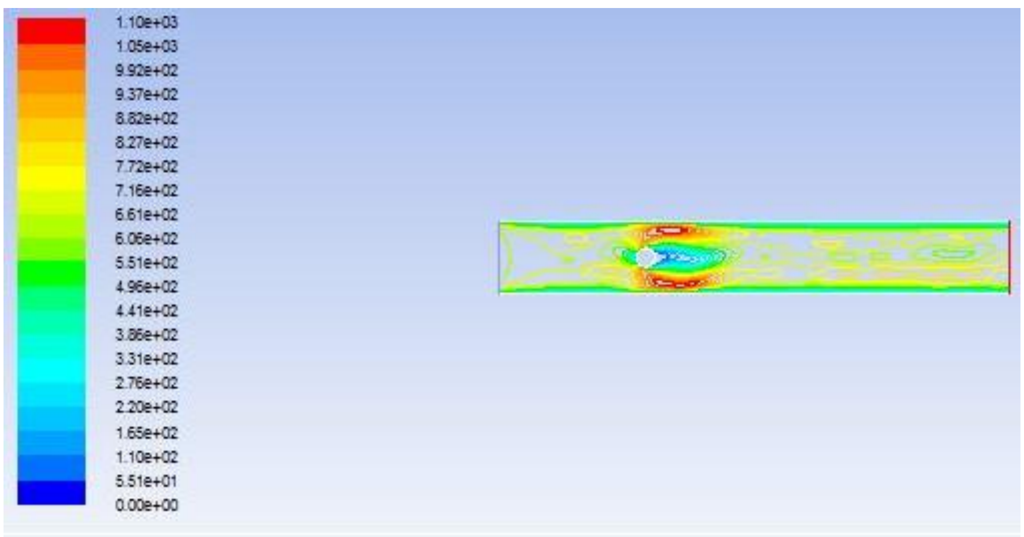


Figure 4.6:- Instantaneous (absolute value of) velocity contours for a circular obstacle at the mentioned blockage ratios and Re = 600

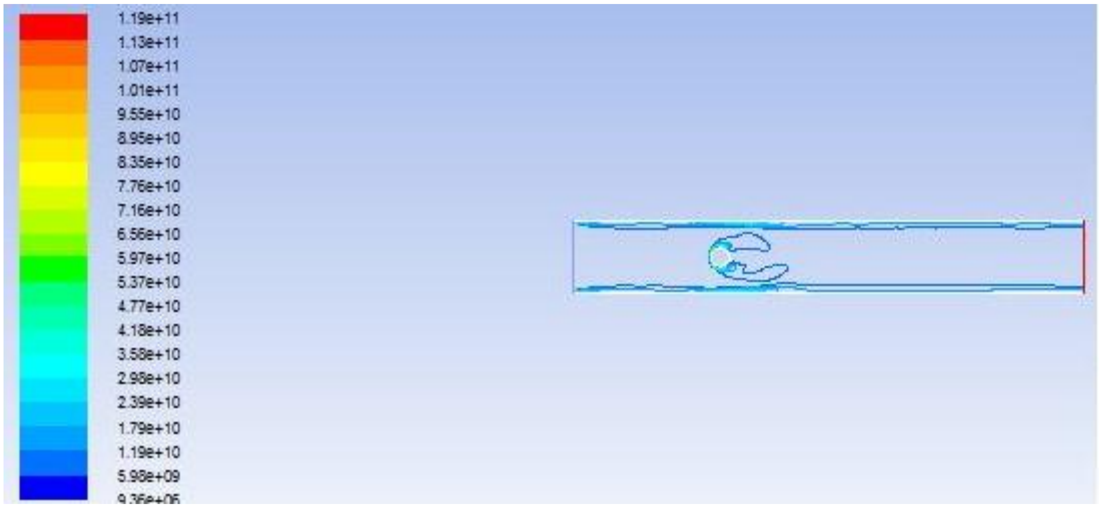


Figure 4.7:- Instantaneous (absolute value of) vorticity contours for a circular obstacle at the mentioned blockage ratios and Re = 600

3. $$\beta = 1/2$$

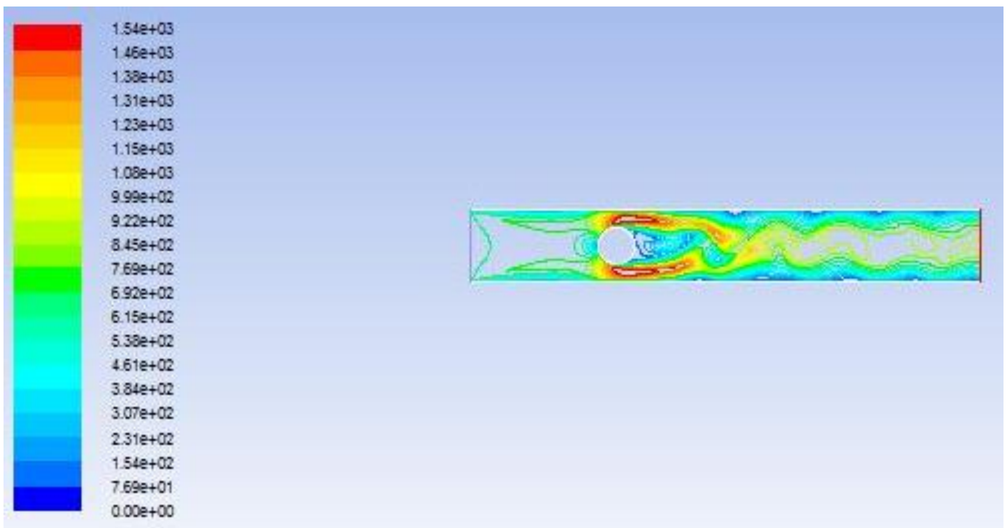


Figure 4.8:- Instantaneous (absolute value of) velocity contours for a circular obstacle at the mentioned blockage ratios and Re = 600

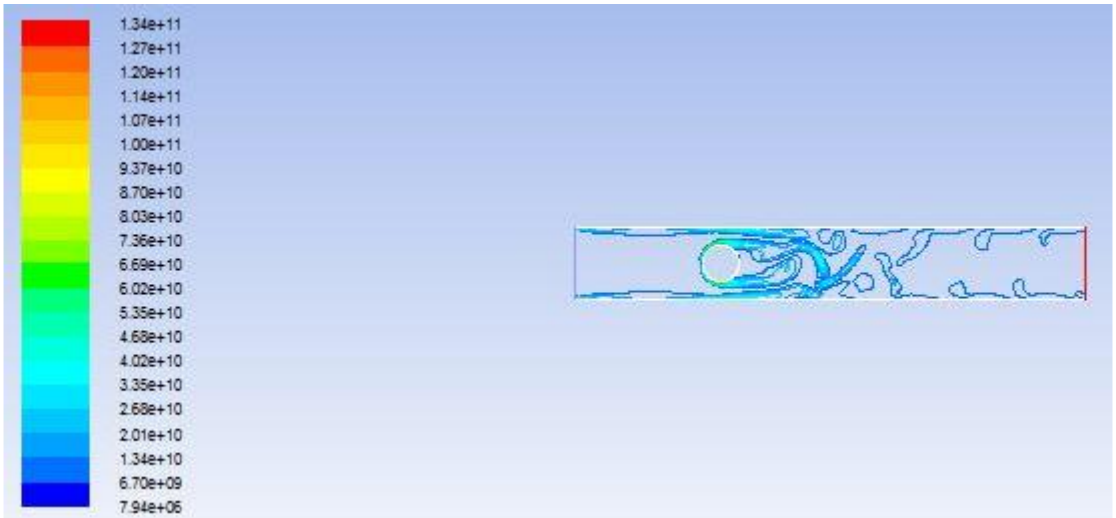
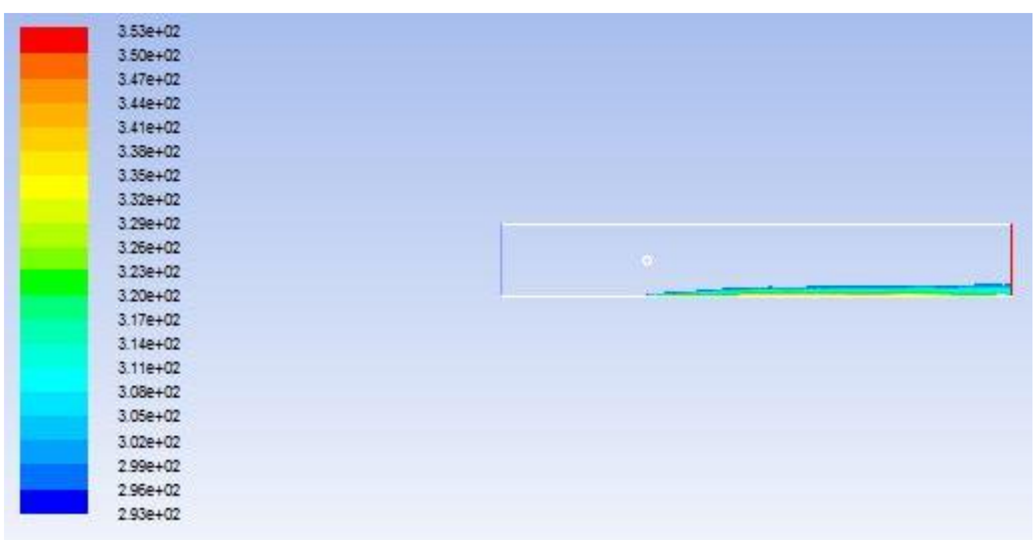


Figure 4.9:- Instantaneous (absolute value of) vorticity contours for a circular obstacle at the mentioned blockage ratios and Re = 600

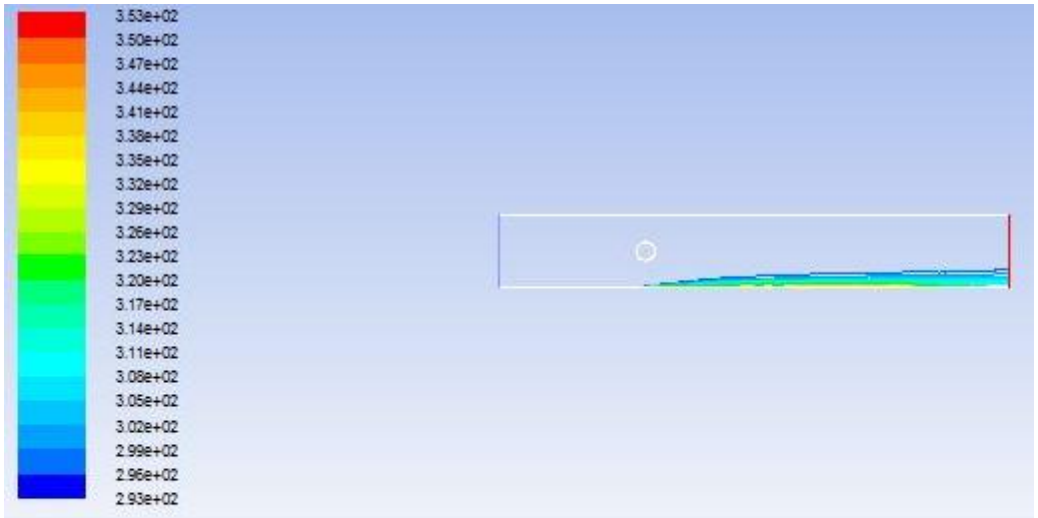
The vorticity and velocity snapshots in Figure 4.4-4.9 show that the intensity of the von Karman vortices increase as the blockage ratio increases, producing various transversal plumes at the interface between adjacent vortices, which are stronger near the (hotter) lower wall. Von Karman vortex street is a repeating pattern of swirling vortices caused by the unsteady separation of flow of a fluid around blunt bodies.

When these plumes are strong enough, they are quite effective in enhancing heat transport from the (thermal boundary layer attached to the) hot wall to the bulk, as discussed in the temperature contours plotted in Figure 4.10**.** These plots show the heat transfer is carried out downstream(due to presence of non-adiabatic wall) and thermal efficiency increases as blockage ratio increases.

$\beta = 1/10$



$\beta = 1/4$



$\beta = 1/2$

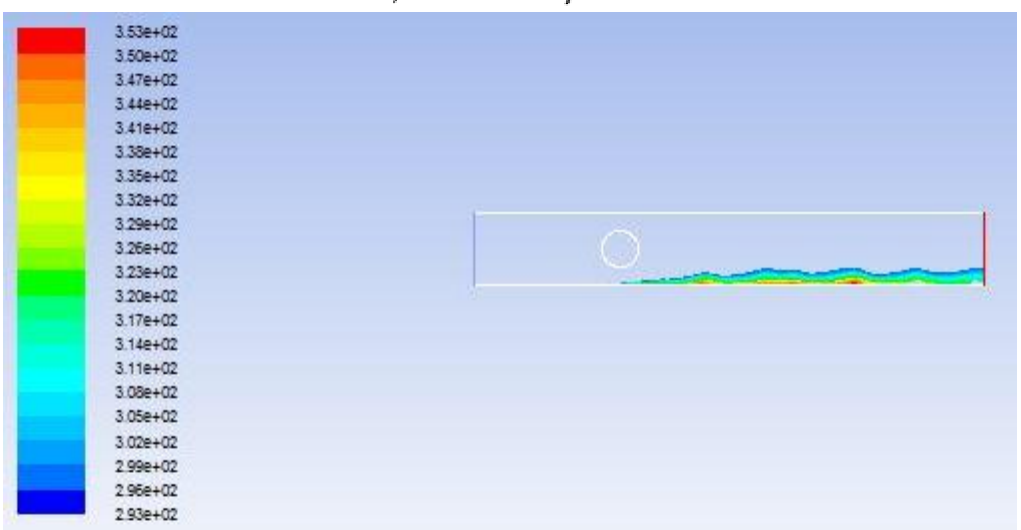


Figure 4.10:-Instantaneous temperature contours for a circular obstacle at the mentioned blockage ratios and Re = 600

This thermal efficiency increase is further appreciated in Figure 4.11,where the Surface Nusselt Number Vs Length of Microchannel is plotted. The plot shows that local Nusselt number diverges at x = 0, which is due to the weak singularity that appears at the wall as the boundary condition changes from zero heat flux to constant temperature. Since Nusselt Number is directly proportional to the heat transfer coefficient , a high value of Nusselt Number for most part of the graph represents a higher thermal efficient design which in this case is the circular obstacle of blockage ratio ½.

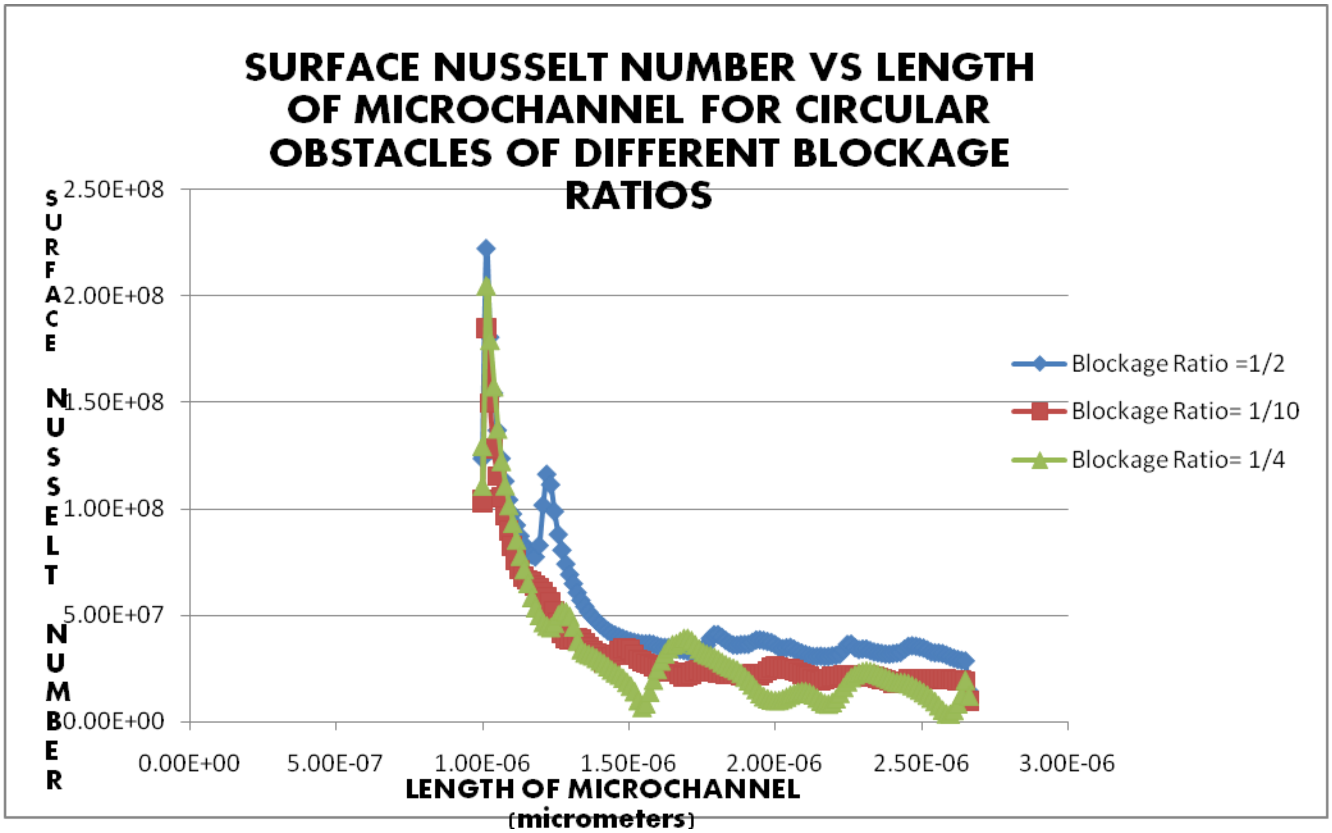


Figure 4.11 :- SURFACE NUSSELT NUMBER VS LENGTH OF MICROCHANNEL FOR CIRCULAR OBSTACLES OF DIFFERENT BLOCKAGE RATIOS OF 1/10 , ¼ & ½

4.7.2 SIMULATION USING A RECTANGULAR OBSTACLE

Here a rectangular obstacle in the path of flow is kept and its effect on heat transfer is studied. The position and the orientation of the obstacle is changed and the resulting impact on heat transfer efficiency is reported.

4.7.2.1 RECTANGLE(0.25 μm* 0.125 μm) PLACED AT THE CENTRELINE

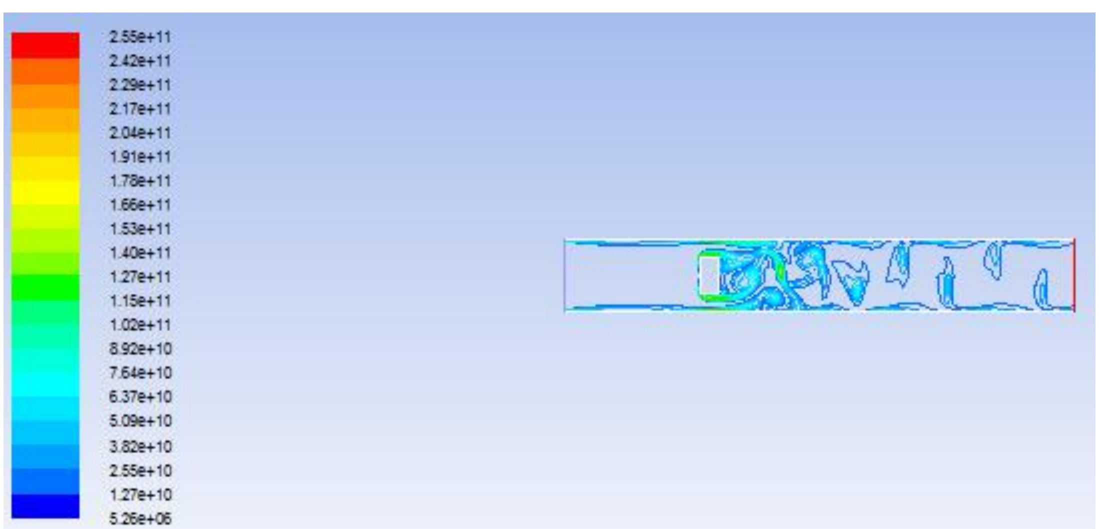


Figure 4.12:- Vorticity contour for a rectangle placed at the centre.

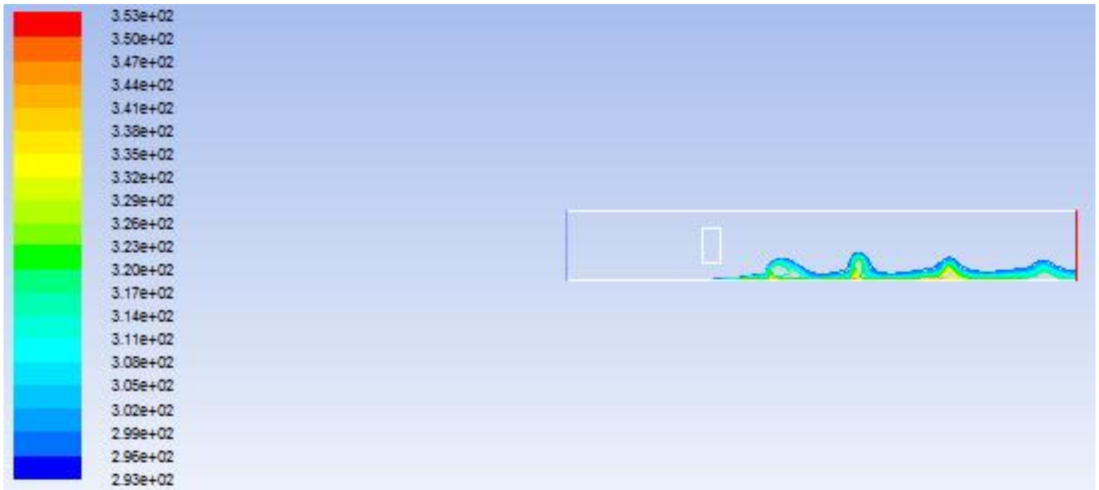


Figure 4.13:-Temperature contour for a rectangle placed at the centerline.

Comparision with the results of the circular obstacle , we find that a rectangular obstacle produces slightly better convective plumes which in turn will result in high heat transfer from the wall.

4.7.2.2 RECTANGLE PLACED AT A DISTANCE OF 0.1 μm BELOW THE CENTRELINE

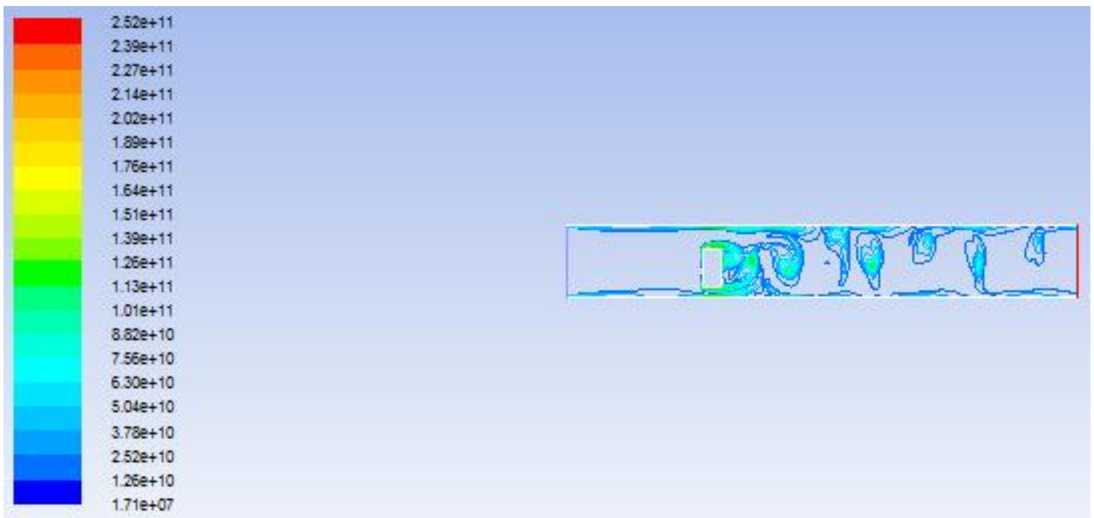


Figure 4.14:-Vorticity contour for rectangle placed at a distance of 0.1 micrometer below the centerline

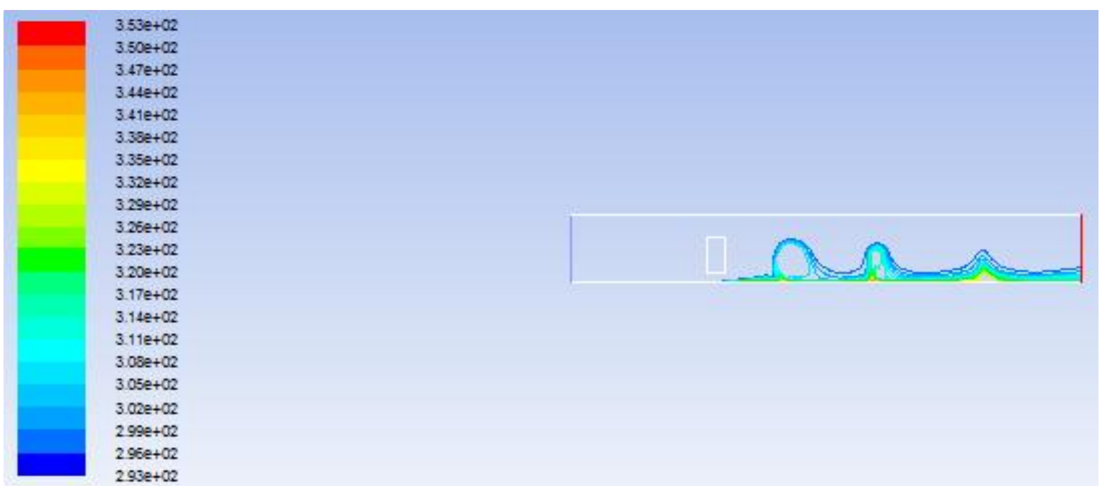


Figure 4.15:- Temperature Contour for Rectangle placed below the centerline

4.7.2.3 RECTANGLE PLACED AT A DISTANCE OF 0.1 μm ABOVE THE CENTRELINE

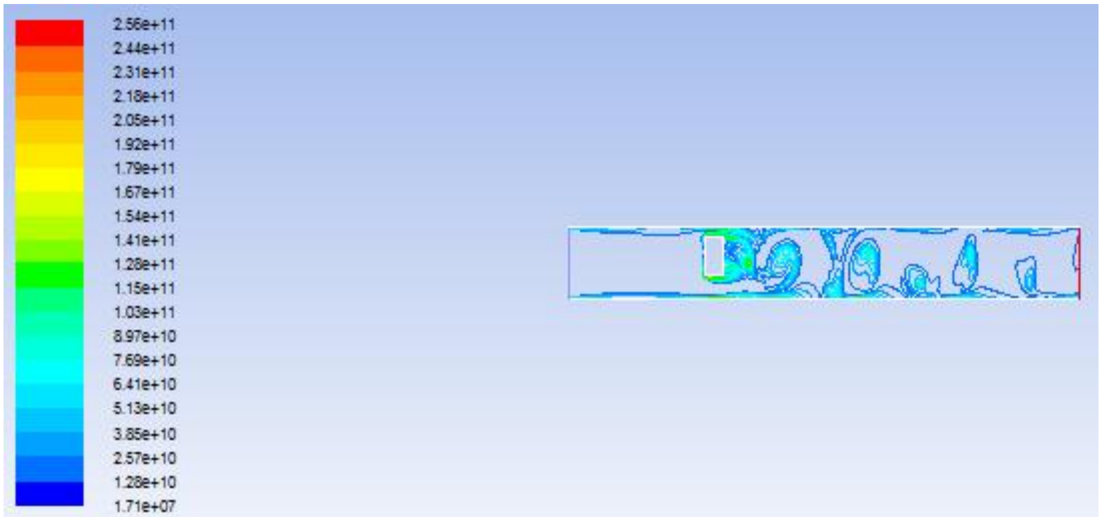


Figure 4.16:-Vorticity contour for rectangle placed at a distance of 0.1 micrometer above the centerline

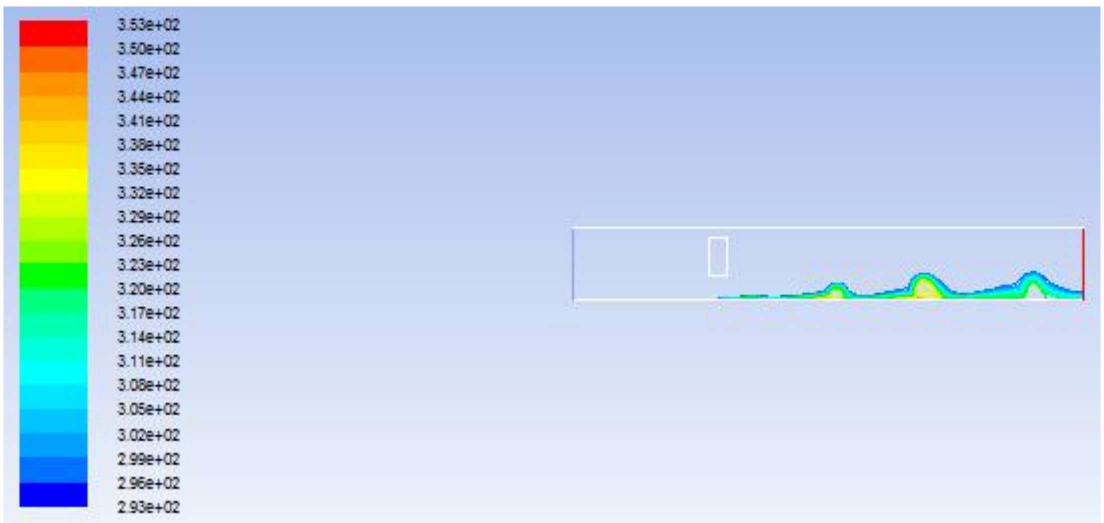


Figure 4.15:- Temperature Contour for Rectangle placed above the centerline

4.7.2.4 RECTANGLE ROTATED AT AN ANGLE OF 15^0 FROM THE VERTICAL

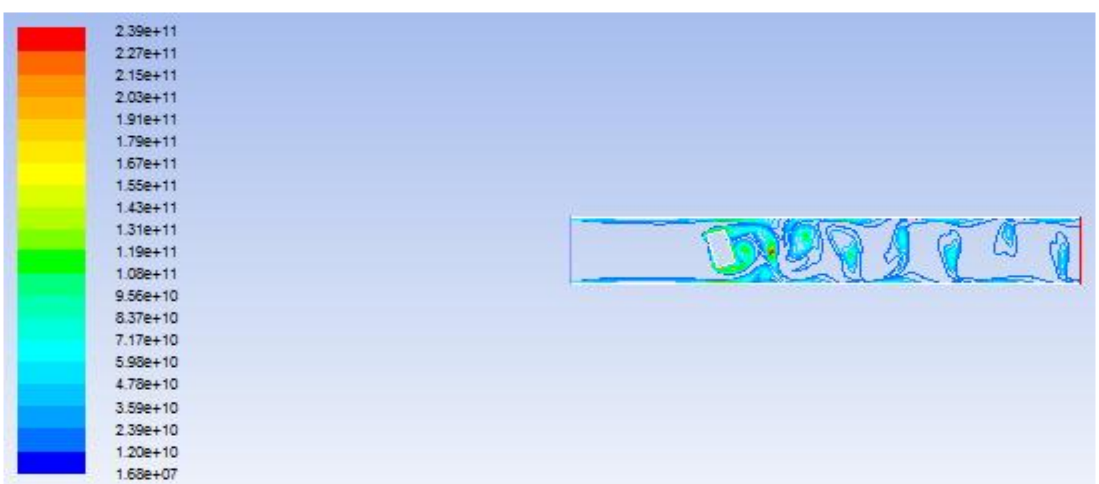


Figure 4.16:-Vorticity contour for the rectangle rotated at an angle of 15^0

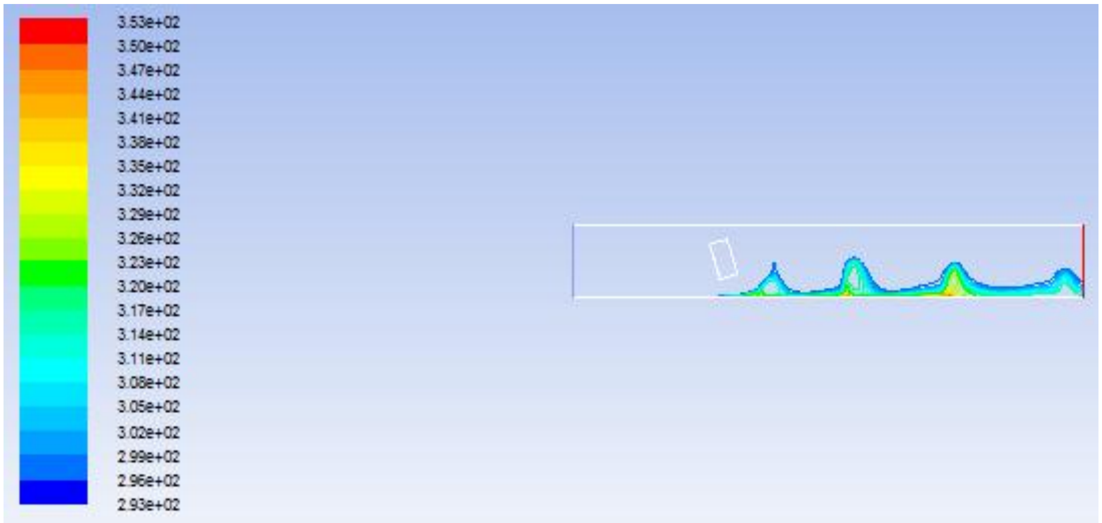


Figure 4.17:-Temperature contour for rectangle rotated at an angle of 15^0

Changing the position or orientation of the the rectangular obstacle has a significant impact on the heat transfer. The transversal plumes become more and more intense and hence the thermal efficiency also increases which is justified by the Nusselt Number plots below.

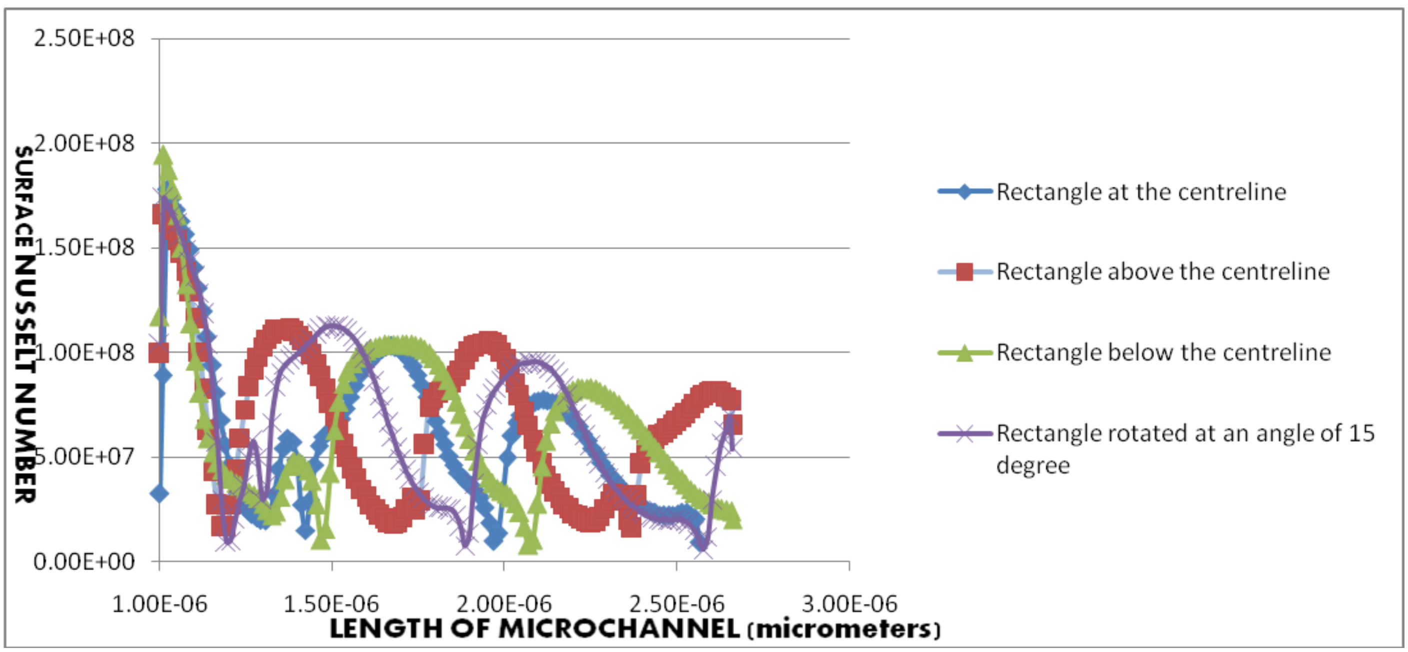


Figure 4.18:-Surface Nusselt Number Vs Length of Rectangle plot for various postions of rectangle.

It can be reported from the above graph that the Rectangle with rotation provides better thermal efficeiency mainly because a part of it is being streamlined to the flow producing intense vortices and hence the thermal plumes and the Nusselt Number are large enough.Then comes the rectangle positioned above , then the rectangle positioned below and finally the rectangle at the centerline.But an increase in thermal efficiency in these cases comes at the cost of huge prsseure drop. So the cost of pumping will be very high.

4.7.3 SIMULATION WITH TRIANGULAR OBSTACLE

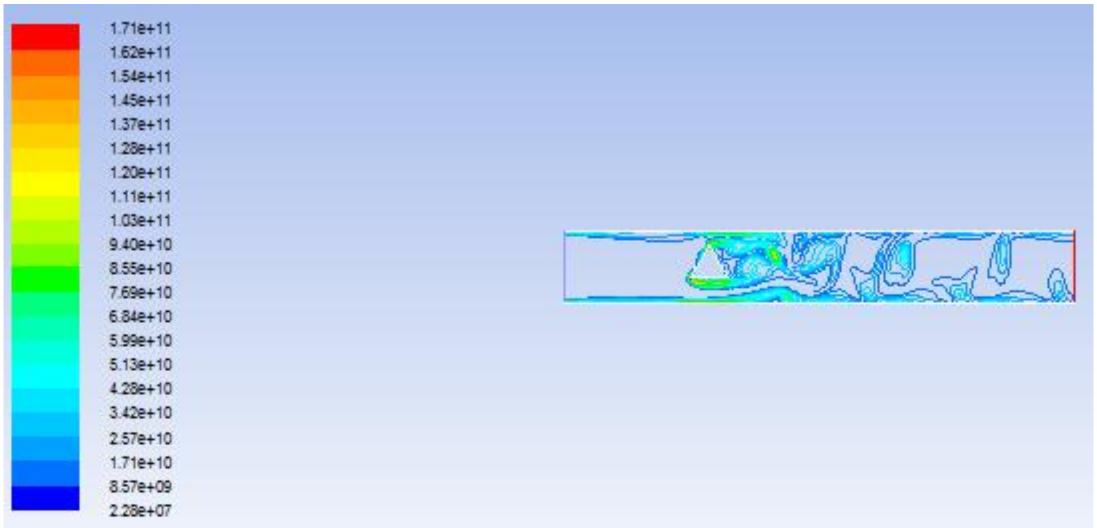


Figure 4.19:-Vorticity contour for triangular obstacle

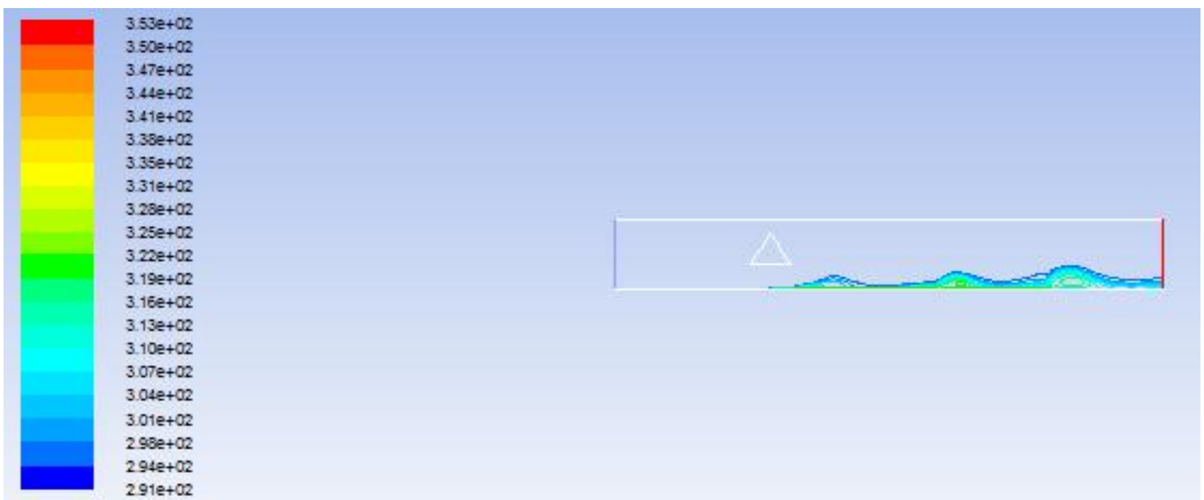


Figure 4.20:- Temperature Contour for a triangular obstacle

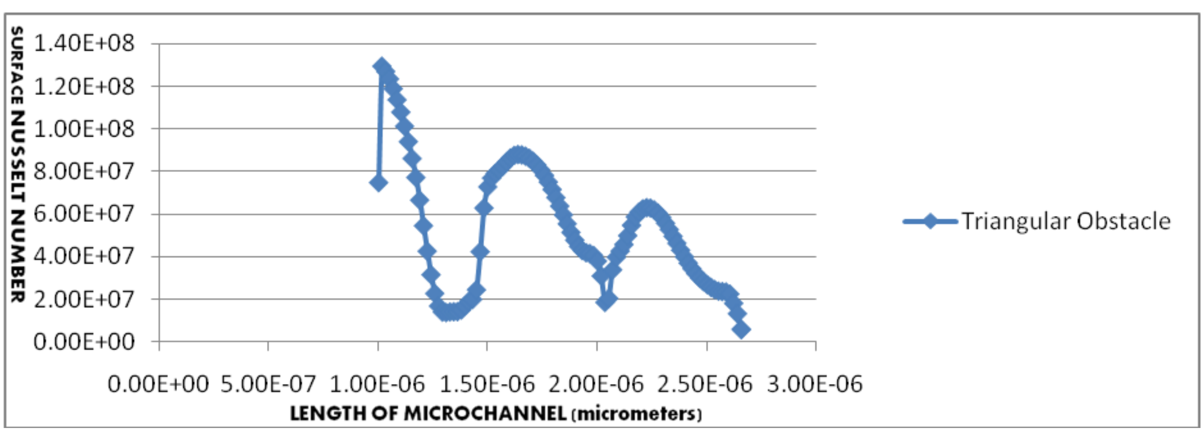


Figure 4.21:- Surface Nusselt Number Vs Length of microchannel plot for a triangular obstacle

The triangular obstacle provides the best result of all as can be seen from the above plot. It is mainly due to its streamline nature.The vortices produced are of huge intensity and hence facilitate extensive heat transfer but at the cost of mechanical penalty.

4.7.4 SIMULATION WITH NON-NEWTONIAN FLUIDS

To study the effect of non-newtonian fluid medium , a rectangular obstacle is taken at the centerline. Two Pseudoplastic (n<1) Non-Newtonian fluids namely, Banana Puree and CMC solutions(10ppm) are taken one by one and the results obtained are compared with that of water.

4.7.4.1 CMC (10ppm) Medium

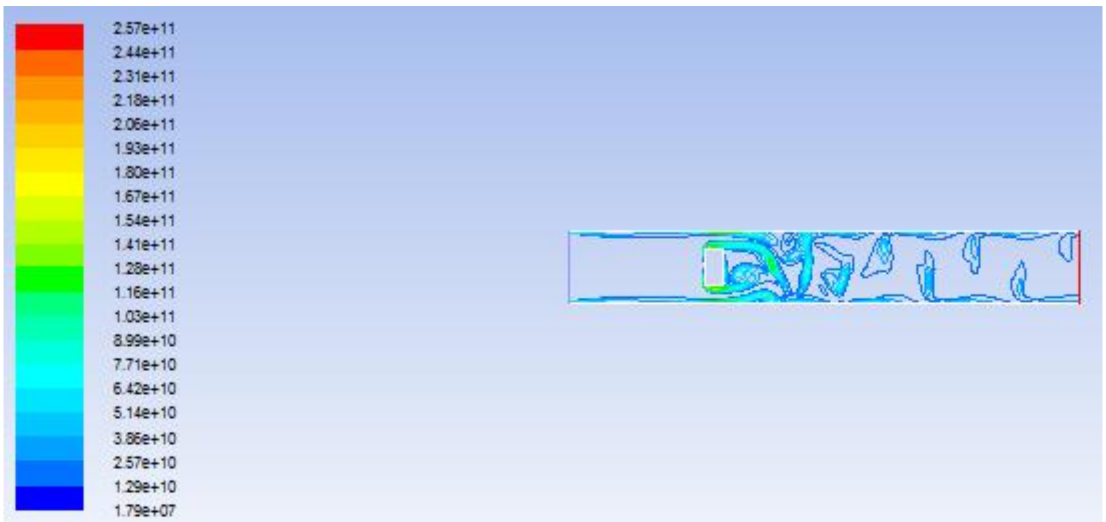


Figure 4.22:-Vorticity contour for CMC medium

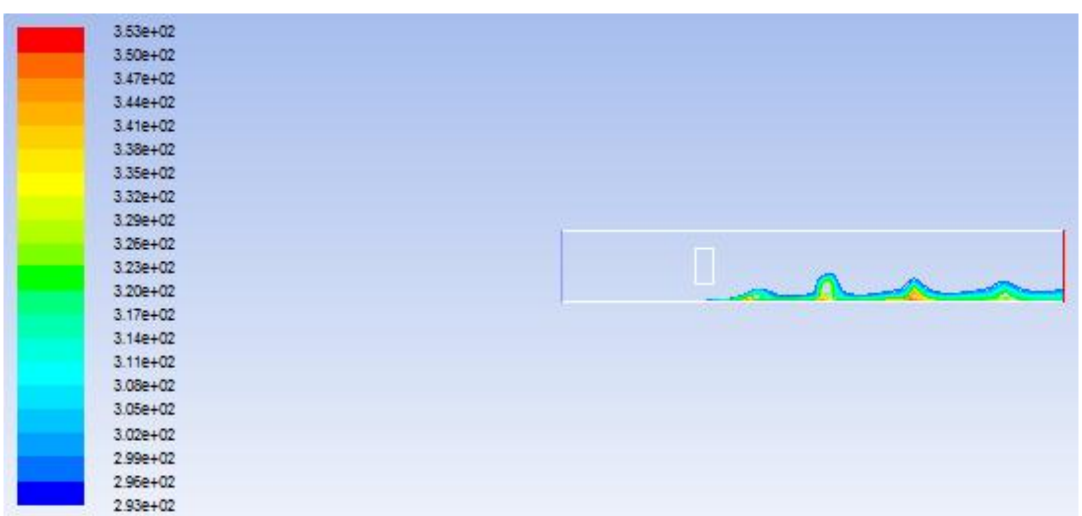


Figure 4.23:- Temperature Contour for CMC medium

4.7.4.2 BANANA PUREE MEDIUM

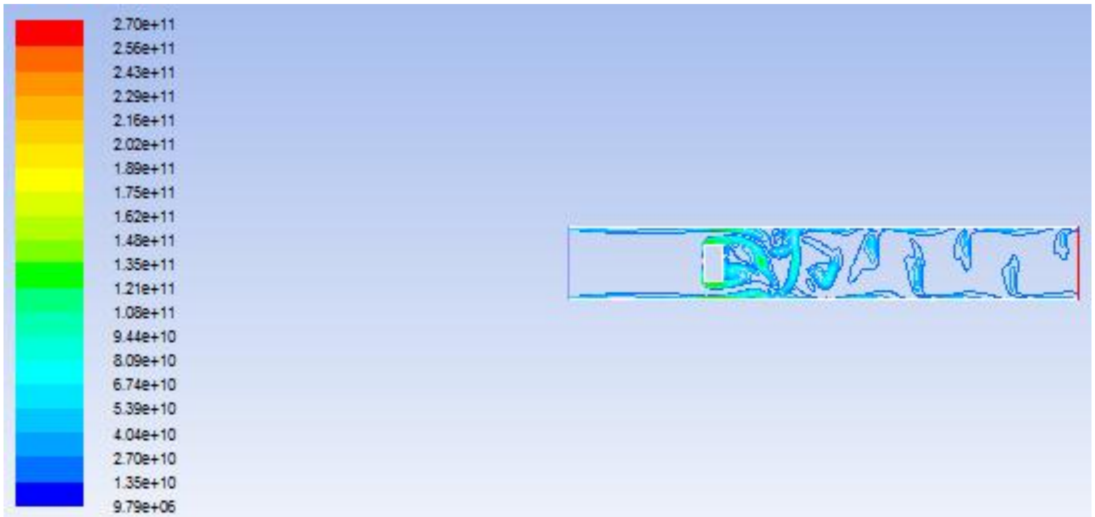


Figure 4.24:-Vorticity contour for BANANA Puree Medium

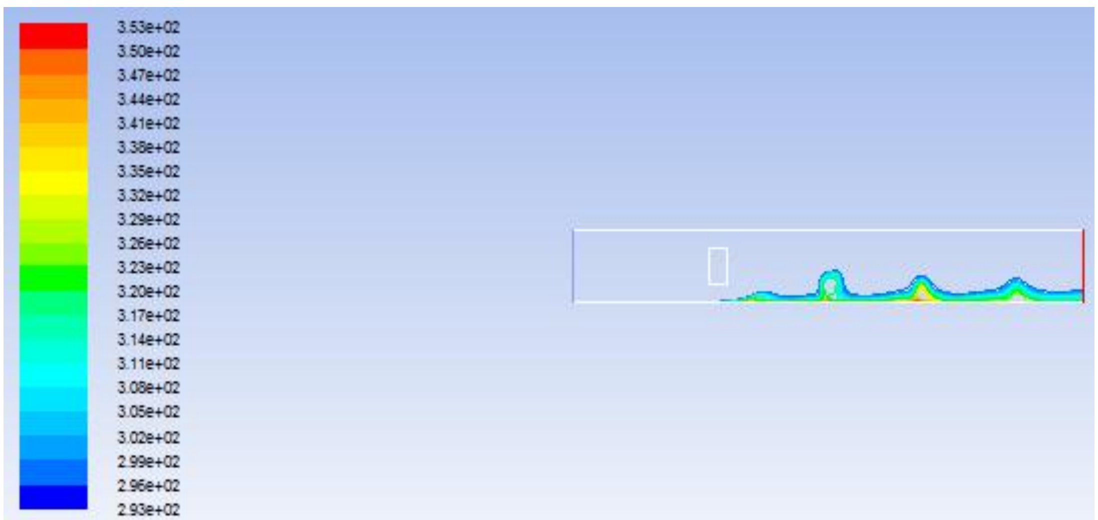


Figure 4.25:-Temperature Contour For BANANA Puree medium

The power-law fluid behaviour does not seem to alter the streamline in a significant manner, except for the fact that the shear-thinning behaviour not only delays the formation of a visible wake but also the resulting wake is also somewhat shorter than that in a Newtonian fluid. The shear thickening, on the other hand, has exactly the opposite influence on wake formation.

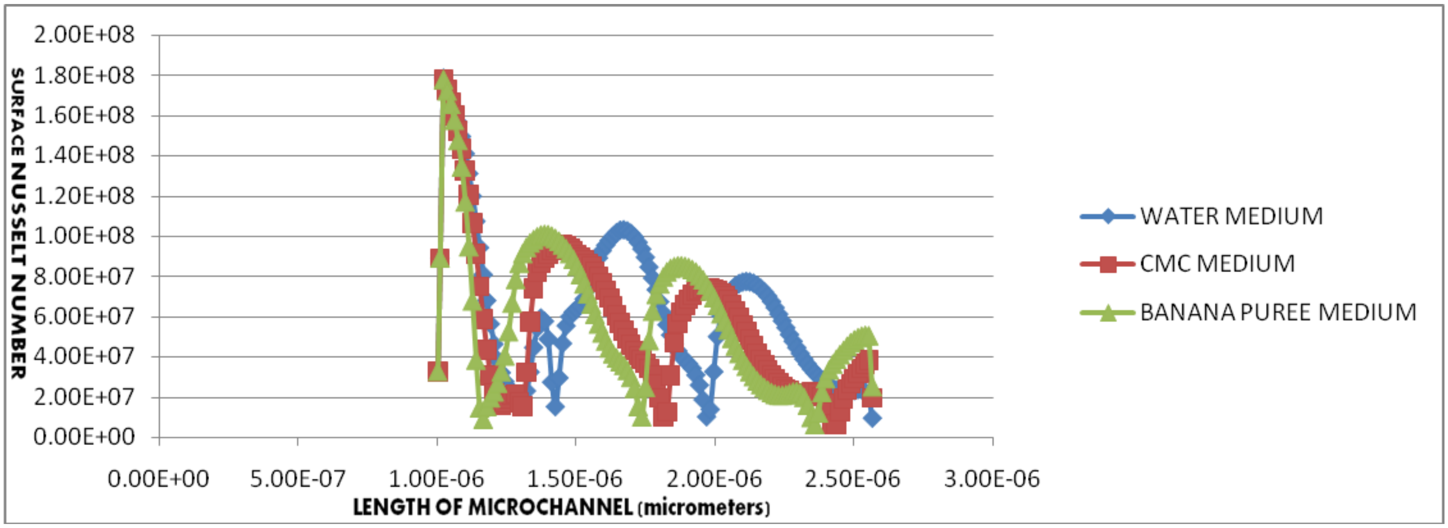


Figure 4.26:- Surface Nusselt Number vs Length of microchannel for different flow mediums.

The above plot shows that changing the flow medium from Newtonian to Non-Newtonian only slightly increases the heat transfer effect.

4.8 PROBLEM STATEMENT-2

Consider a 2D, unsteady, laminar flow of water in a non-isothermal microchannel. Walls are adiabatic except in a portion of the bottom wall located downstream of the obstacle (a distance of 1.67 microns) where the temperature is maintained at Tmax= 353 K, while the temperature of the incoming flow is Tinlet =293 K. The height of the rectangular channel is varied (0.5,0.4,0.3,0.2 microns) and the effect on Nusselt number is studied.

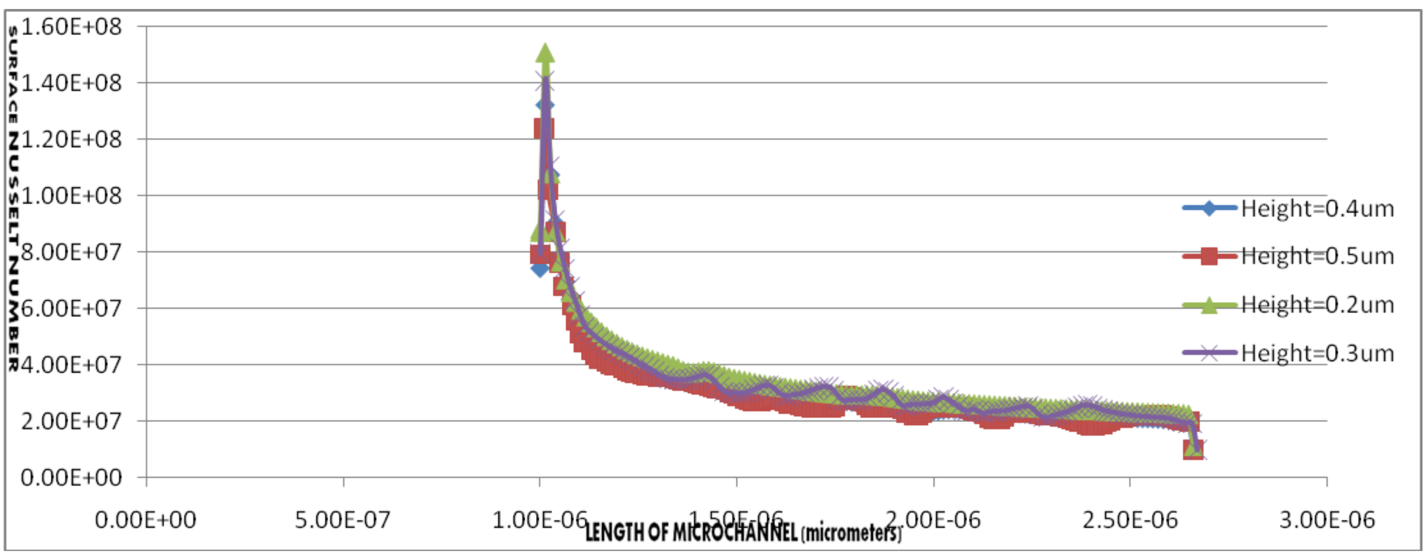


Figure 4.27:- Surface Nusselt Number Vs Length of Microchannel plots for different heights of microchannel.

The above plot shows that decreasing the height of the microchannel increases the heat transfer effect or thermal efficiency. From the Bernoullis theorem we know that when pressure decreases, velocity increases. So a decrease in the height of the channel increases the pressure drop which increases the velocity manifold and hence the vortices formed will be of higher intensity which facilitate higher rate of heat transfer.

CHAPTER 5

CONCLUSIONS AND FUTURE SCOPE OF THE WORK

CHAPTER-5

CONCLUSIONS & FUTURE SCOPE OF THE WORK

Heat Transfer Enhancement was studied using vortex promoters of circular , rectangular and triangular shapes with Newtonian and Non-Newtonian fluids and the transient state models were simulated using ANSYS 13.0 FLUENT SOFTWARE. The following conclusions were drawn from the above results obtained:-

1. Regardless of the obstacle's specific shape, heat transfer increases as the channel blockage ratio increases up to blockage ratio=1/2 . The reason for that is the expected one, namely as blockage ratio is increased, the initially steady configuration destabilizes and produces a von Karman vortex street that facilitates vertical heat convection from the lower wall.
2. Larger blockage ratios increase the pressure drop significantly. The relative mechanical penalty is larger than heat transfer benefit in all the above cases considered which increases the pumping cost. So the least loss should be considered.
3. Better results are obtained varying the obstacle shape. Considering elliptical, rectangular, and triangular obstacles shows that the latter provides the best results, then the rectangles, then the ellipses.
4. Thermal efficiency and mechanical penalty depend far more strongly on rotation angle than on position relative to the microchannel centerline.
5. For Non-newtonian Fluids, the heat transfer effect is slightly increased for pseudoplastic fluids.
6. For designing purposes mainly circles and rectangles are considered because triangles are trickier to design.
7. A complex question arises, namely how do the results above improve or deteriorate as all parameters are considered simultaneously. To answer this we require some more sophisticated mathematical tools like Galerkin projection which is currently under research.

FUTURE SCOPES:-

- Simulation taking Reynolds number in the turbulent region.
- Heat Transfer Enhancement taking streamlined objects in series.

REFERENCES

1. ANSYS Fluent 13.0 tutorial guide (2010)

2. Abbasi,H., Turki, S., Nasrallah, S.B. , Mixed convection in a plane channel with a built-in triangular prism, Numer. Heat Transfer A 39 (2001) 307–320.

3. Abbasi, H., Turki, S., Nasrallah, S.B. , Numerical investigation of forced convection in a lane channel with a built-in triangular prism, Int. J. Thermal Sci. 40 (2001) 649–658.

4. Pandey, A.K., and Munshi B., A Numerical Study of Newtonian and Non-Newtonian Fluid Flow and Heat Transfer over a Series of Submerged Circular/Non-Circular Objects using CFD methods, (2010),Department of Chemical Engineering , NIT Rourkela.

4. Ergu, O. B., Sara, O.N., Yapıcı, S. and Arzutug, S., Pressure drop and point mass transfer in a rectangular microchannel. International Communications in Heat and Mass Transfer (2009). 36. 618 – 623.

5. Gogineni A, Cfd Modelling Of Rectangular Microchannel With Increase In Heat Flux And Effect On Nusselt Number, Proceedings of the 6th Annual GRASP Symposium, Wichita State University, 2010

6. Hasan, M. I., Rageb, A. A., Yaghoubi, M. and Homayoni, H.Influence of channel geometry on the performance of a counter flow microchannel heat exchanger, International Journal of Thermal Sciences, (2009). 48, 1607-1618.

7. Icoz, T. , Jaluria, Y. ,Design optimisation of size and geometry of vortex promoter in a two-dimensional channel, J. Heat Transfer 128 (2006) 1081–1092.

8. Kang, S. and Tseng, S. Analysis of effectiveness and pressure drop in micro crossflow heat exchanger, Applied Thermal Engineering (2007) ,27, 877-885.

9. Kang, S. ,Characteristics of flow over two circular cylinders in a side-by-side arrangement at low Reynolds numbers, Phys. Fluids 5 (9) (2003) 2486–2498.

10. Meis M , Varas F, Velázquez A , Vega J. M., Heat transfer enhancement in micro-channels caused by vortex promoters, International Journal of Heat and Mass transfer (2009) 53 29-40.

11. Nitin, S. , Chhabra R.P., Non-isothermal flow of a power law fluid past a rectangular obstacle (of aspect ratio 1/2) in a channel: drag and heat transfer, Int. J. Eng. Sci. 43 (2005) 707–720.

12. Li, J., Peterson, G.P. and Cheng, P.. Three-dimensional analysis of heat transfer in a micro-heat sink with single phase flow. International Journal of Heat and Mass Transfer. (2004) 47, 4215–4231

13. Papaioannou, G., Yue, D.K.P. , Triantafyllou, M.S. , Karniadakis, G.E. , Three dimensionality effects in flow around two tandem cylinders, J. Fluid Mech. 558 (2006) 387–413.

14. Rahnama, M. , Moghaddam, H.H. , Numerical investigation of convective heat transfer in unsteady laminar flow over a square cylinder in a channel, Heat Transfer Eng. 26 (10) (2005) 21–29.

15. Reyes M , Arias J R, Velazquez A , Vega J R, Experimental study of heat transfer and pressure drop in micro-channel based heat sinks with tip clearance, Applied Thermal Engineering (2011) 31 887-893.

16. Tasaka, Y. , Kon, S. , Schouvelier, L. , Le Gal, P. ,Hysteretic mode exchange in the wake of two circular cylinders in tandem, Phys. Fluids 18 (2006). 084104-1–6.

17. Turki, S. , Abassi, H. , Nasrallah, S.B. , Two-dimensional laminar fluid flow and heat transfer in a channel with a built-in heated square cylinder, Int. J. Thermal Sci. 42 (2003) 1105–1113.

18. Valencia A., Turbulent flow and heat transfer in a channel with a square bar detached flow the wall, Numer. Heat Transfer A 37 (2000) 289–306.

19. . Versteeg, H. and Malalasekra, W.. An Introduction to Computational Fluids dynamics: The QUICK Method.(2007), 2nd Edition, Pearson.