CFD Analysis of Maldistribution in Mini-channel Heat Exchanger

Tankadhar Bhoi

Department of Mechanical Engineering
National Institute of Technology Rourkela
Sundargarh, Odisha, India – 769 008

CFD Analysis of Maldistribution in Mini-channel Heat Exchanger

A Thesis submitted in partial fulfillment of
the requirements for the award of the degree of

Master of Technology in
Cryogenics and Vacuum Technology

Submitted to

National Institute of Technology Rourkela

by

Tankadhar Bhoi

(Roll No. 214ME5338)

under the supervision of

Prof. R.K. Sahoo

Department of Mechanical Engineering
National Institute of Technology Rourkela
Sundargarh, Odisha, India – 769 008

Dedicated to

My Family
&
The Almighty God

Department of Mechanical Engineering
National Institute of Technology Rourkela
Rourkela-769 008 , Odisha , India. `www.nitrkl.ac.in`

Prof. R.K.Sahoo
Professor

May , 2016

Certificate

This is to certify that the work in the thesis entitled *CFD Analysis maldistribution in Mini-channel heat exchanger* by *Tankadhar Bhoi* , bearing Roll Number 214ME5338, is a record of an original research work carried out by him under my supervision and guidance in partial fulfillment of the requirements for the award of the degree of *Master of Technology* in *Cryogenics and Vacuum Technology, Department of Mechanical Engineering*. Neither this thesis nor any part of it has been submitted for any degree or academic award elsewhere.

R.K. Sahoo

Acknowledgement

Firstly, I would like to convey my heartily thanks and gratitude to my beloved guide, *Prof. R.K. Sahoo, (Dept. of Mechanical Engineering, NIT, Rourkela)* for his continuous guidance and support, with the help of which I have successfully been able to complete my research work.

I would like to acknowledge my special thanks to the Head of the Department, *Prof. S.S. Mahapatra* and all the faculty members of the Department of Mechnanical Engineering, for providing me the deep insight and discernments given through the various course they had taught and also for their valuable guidance during my work.

I am also thankful to all my well wishers and the staff of Mechanical Engineering Department whose timely help and co-operation allowed me to complete my research work in time and bring out this thesis. My special thanks to classmate *Mr.Abinash khandual* and *Piyush Rajput*.

Last, but not the least, I would like to profound my deepest gratitude to my family for their exceptional love and encouragement throughout this entire journey, without which I would have struggled to find the inspiration and motivation needed to complete this thesis.

Tankadhar Bhoi
Roll No. 214ME5338
Cryogenics and Vacuum Technology

Abstract

Heat exchanger plays an important role in engineering application. Many places find application of heat exchanger starting from the big one to Nano channel heat exchanger. Among the different types heat exchanger mini-channel heat exchanger finds wide application in engineering fields. Flow maldistribution is a phenomenon where the distribution of flow is not uniform in all the channel due to some defects in geometry, unhealthy header design, leakage, etc. , even some channel in the heat exchanger may have backflow and some does not flow at all. Due to the maldistribution, there is a decrease in the efficiency of the heat exchanger. Present research focuses on to know the maldistribution when more nos of channels were using tracer analysis in ANSYS FLUENT .Tracer analysis is carried out in two channels with ideal case and the result was validated with the theoretical analysis. With the help of the tracer analysis one can find residence time distribution ,Mach number and based on the Mach number the Maldistribution effect can be known .Some simple cases of Mini-channel were modeled with maldistribution; results show that with maldistribution heat transfer decreases.

***Keywords*:** ANSYS FLUENT ; Tracer Analysis; Maldistribution; Residence time; Mach no

Contents

Abbreviations

A	Heat transfer Area
j	counter of channels
k	overall heat transfer coefficient
L	nominal length of heat exchanger
N	number of tubes
NTU	number of transfer units
T	temperature
$\triangle T$	Tempearture difference
$\triangle T_m$	cross-sectional mean temperature difference
$\triangle T_{ad}$	adiabatic mean temperature difference
$\dot{W}$	thermal flow rate
w	Flow vwlocity
$\bar{w}$	mean flow velocity
x	space coordinate
φ	Peclet number
M	Mach number
t_{vj}	residence time
$\bar{t}$	average residence time
ψ	Dimenssionless function
σ^2	Variance
ϵ	dissipation rate of turbulent kinetic energy

κ	von Karman constant
σ_κ	turbulent prandtl numbers for κ
σ_ϵ	turbulent prandtl numbers for ϵ
ρ	density
μ_t	turbulent or eddy viscosity
τ_w	wall shear stresses
P	pressure
$P^{'}$	pressure correction
t	time
T	Temperature, Time averaging interval
$\bar{u}_t$	mean velocity parallel to the wall
u_τ	shear velocity
u, v, w	velocity
$u^{'}, v^{'}, w^{'}$	velocity corrections
u_j	velocity in jth direction
x_i	spatial co ordinate in the ith direction

Greek Symbols

θ	Dimenssionless temperature
ξ	Dimenssionless spatial variable

Subscript

1	fluid 1
2	fluid 2
$back$	backward flow in tubes
$forward$	forward flow in tubes
f	adiabatic mean
m	cross-sectional mean

List of Figures

List of Tables

Chapter 1

Introduction

1.1 Definition and usage of mini-channels heat exchanger

The heat exchanger is a mechanical element which transfers heat between two fluid with or without mixing .The heat exchanger can be cross flow or parallel flow or cross flow. According to the size of heat exchanger, they also can be divided into mini-channel, microchannel and conventional type. The mini-channel heat exchanger found wide application in the area electronic cooling, refrigeration systems where compactness is the main criteria. Due to external and internal fin arrangement of the mini-channel heat exchanger it provides an efficient cooling to the system. In the case of refrigerant application, it finds wide application because it flow very less refrigerant in the channels and it provides efficient heat transfer. Due to the advancement in the electronic industry, there is a decrease of the size of chips or components of the devices. On contrary due to the compactness the heating power of the heat exchanger increases within a small area. In order to keep the instrument in good conditions the temperature generated must be within the same limit . Hence the heat generated in the system must be taken out, and required cooling must be provided to the circuit. Hence, the conventional air cooling system will not be able to provide much amount of cooling to the system, The cooling system must be replaced with heat sinks which can use other better fluid for the removal of heat. This type of heat exchanger is called as a heat sink. The advantage of microchannel lies in its heat removal capacity within a small surface area. Heat sinks used in the electronic application particularly made up of high thermal conductivity material such as copper ,aluminum , etc. The heated surface generally the microprocessors are fixed with the copper plate or any highly thermal conductivity material above which many microchannels were fitted where the fluid is forced to pass through the channels and forced fluid will take the heat of the electronic processor. The mini-channel heat exchanger can be easily repairable as compare to conventional case. Due to smaller passage the fluid side heat transfer increases very much which is not found in conventional case.

1.2 Classification and effect of maldistribution in Mini Channel heat exchanger

Classification given by Khandker [1]an in-between channel size of 200 m to 3 mm can be considered as a mini channel. There is a different type of micro channel cross section available such as U,V,S,D several research has done on different type of channel to find out the flow and heat transfer. All the channels in mini-channel are connected to a general manifold where the fluid is forced to the channel ; similar arrangement also provided at the outlet of the channel. In the general case, it is assumed that whatever mass is coming from the header is uniformly distributed in all channels and all the channels has same mass flow rate. But in practically it is not the case rather channels may have different mass flow rate even some have to bask flow or no flow also.This phenomenon called as maldistribution of flow. Maldistribution predominantly found in conventional heat exchanger but in the case of mini-channel heat exchanger, it can be neglected. The reason for maldistribution is geometry defect, dispersion of fluid inside the channel, material used, the size of the channels, etc. As mini channels heat exchanger are very small in size so during the manufacturing of channel the need of brazing can occur due to the brazing there is change in the dimension of the channel hence mass coming inside the it changes or due to some leakage in the channel, all of it may have different mass flow rate. Maldistribution in the case of heat exchanger can never be zero practically, but it can be reduced to a certain acceptable limit with a proper analysis. Several research has been going on this to find out the factor responsible for the heat exchanger. For the first time Tuckerman and Pease finds a way to cool using the mini channel for the electronic circuit Which opens the doors of research on the area of mini-channels. The main obstacles of the mini channel design lie on factors like the maldistribution and higher pressure drop. Due to the pressure drop the requirement of cooling power increases and due to the non-uniform flow, there is a chance of formation of a hot spot inside the channel.Whenever a fluid is flowing inside a channel which is surrounded by the walls, there is boundary layer formation between from the wall towards the center of the channel .Due to this effect some of the mass is moving at a faster rate (central masses) and the mass nearer to the wall having a slower rate ,this is known as axial moment .Due to this the heat conduction also axially conducted. If all the masses is moving in same flow rate than the flow pattern is called as plug flow . Due to the axial movement maldistribution effect also observed in case of the heat exchanger. Different models have been developed to know the effect of maldistribution such as parabolic, hyperbolic. One of the parameters known as Mach number is used for understanding of this type maldistribution. The Mach number can be calculated by using residence time analysis in heat exchangers. .

1.3 Objectives of work

It is evident that the flow maldistribution is very much responsible for the effectiveness of heat exchanger.Literature review shows that their are so many research done on the area of mini channels and most of the research are experimental or anlytical.Very few research has done using Computational Fluid Dynamics(CFD) method. Following from the analytical analysis by Roetzel and Ranong [2] ,this study focuses on CFD simulation heat exchanger. Present research focuses on the following aspects

- Computational fluid dynamics modeling and simulation of two tube heat exchanger with back flow.
- Residence time analysis using computational tool for the case of maldistributed plug flow with 2 mini channel
- Understanding of thermal and flow behaviour in case of heat sink with constant heat flux at the base.

Chapter 2

Literature Review

Several research work has been done on Maldistribution in heat exchanger .Some of the research includes experimental works and some includes simulation work. Roetzel and Ranong [2]has done extensive research in the field of maldistribution. Under their research work, they tried to find out the axial temperature profile for shell and tube heat exchanger by taking the effect of maldistribution in shell and tube heat exchanger. For their analysis, they have taken the parabolic model and hyperbolic model considering the effect of maldistribution in tube side of the heat exchanger. According to their analysis, the hyperbolic model found good results compare to the parabolic model. By analyzing the heat exchanger in a transient mode with adiabatic condition they tried to find a parameter called as third sound Mach number. Different boundary conditions were taken such as backing, forward maldistribution, etc. to calculate the temperature profile in bundles of the tube.

Taylor [3] has done research work in axial dispersion model which could be applied to the heat exchanger. According to the research he tried to find how soluble method flows inside a system.

Danckwert P.V. [4]has done to find out how the flow is fluid flowing in a system ,In general, two types of flow were assumed first one is piston flow and the second one is axial mixing flow. But in practical cases, the flow is not ideal .In this paper the flow parameter was measured with the help of practical analysi,residence time distribution can be calculated. Open and packed tubes are taken for the analysis. This analysis can also be extended to more systems.

Sahoo R.K and W.Roetzel. [5] By taking the basic heat exchanger equation they Sahoo and Roetzel develop hyperbolic dispersion model. This paper focuses on the model for shell and tube heat exchanger by taking the hyperbolic and parabolic case. Different boundary conditions have been developing for the calculation of the cross-sectional temperature profile .Additional to the temperature profile Mach number also calculated and compare with the previous results.

Lalot et al. [6] have taken electrical heater with heat exchanger system for their experimental analysis where they found out what is the effect of gross flow maldistribution in the case of the heat exchanger. According to their observation, the header design is main factor contribution for the maldistribution if header design is perfect there will not be any backflow. The effect of maldistribution can be seen in the efficiency of the heat exchanger and it reduces drastically. Prabhakara et al. [7] have done an experimental investigation on the pressure drop in case of plate fin heat exchanger and effect of the pressure drop in the maldistribution. Set up consists on a Plate fin heat exchanger with pressure probe at different position of heat exchanger. Reynolds number also varies from 1000-17000 and the corresponding pressure drop is measured. from the experiment, they concluded that the overall pressure drop has a linear relationship with the maldistribution.

Anindya Roy, and Das, S.K. [8] has developed a technique on regenerators by using the hyperbolic model .In their study, they have taken the effect of madistribution and back-mixing for the analysis instead of taking ideal flow .For the analysis of dispersion, it assumed that disturbance is traveling in a finite propagation velocity. This model is can be conveniently used where maldistribution is significant as it is analyzing with the real case variables.

Roetzel et al. [9]have done pioneer work on axial conduction of heat when it flows in a system surrounded by the wall this is due to the boundary layer theory. This phenomenon generally terms as dispersion and due to the dispersion, it moves with a finite velocity. It has been observed that the flow seems to be similar to sound propagation hence a new term came here called as third sound wave flow. Different Boundary conditions were analyzed with the dispersive flow by taking two fluid as analysis domain. Sonic and subsonic flow have been observed at inlet and outlet of the boundary. The first theoretical analysis was done which compares with the numerical work and based on this an experimental set-up has been developed.

Zhe Zhang and YanZhong Li. [10] has done a numerical simulation using Ansys FLUENT where they have modeled a plate fin heat exchanger with a conventional header. According to their analysis, the maldistribution effect is very serious in a direction perpendicular to the fluid motion.And the result of simulation well obeyed with the experimental work. Again for the optimization a header design was modeled with two stage distributing structure and analyzed using FLUENT. The result shows that is the diameter of inlet and outlet header is same then there will be less Maldistribution.

Wilfried Roetzel, and Frank Balzereit. [11] has taken the axial dispersion model and they analyzed for the actual case rather for the 1D plug flow case .Here they tried to find out the packet number by doing residence time analysis of heat exchanger. Peclet number is assumed to be a measure of dispersion if peclet number is 0 the flow is completely mixing in the axial direction if it is infinity the flow is perfectly plug flow without any axial mixing. Investigation of water shows that peclet number varies between 15-160 and axial dispersion presents in many cases.

Muller [12] found the effect of maldistribution in the heat exchanger according to the observation turbulent flow in a heat exchanger has less effect on performance, but laminar flow shows a significant reduction in the performance. Several cases of maldistribution have taken for analysis.

Anil Kumar Dwivedi and Sarit Kumar Das. [13]] suggested a model for unsteady analysis in plate fin heat exchanger try to investigate the time dependent maldistribution effect. An experimental investigation was done with an input in the inlet and corresponding outlet temperature measurement with different flow rates. Then the result of theoretical analysis is compared with the experimental work which well obey to the experimental work.

Khandekar [1] has given the basic classification for the heat exchanger based on their size .According to the classification, a channel size greater than 3 mm is consider to be a conventional channel whereas if channel size is between 3mm to 200 micrometers is called as a mini channel and less than 200 micrometers is a microchannel. This provides a basic landmark to the classification of channels. Commenge et al. [14]] have done extensive research on the on the shape of the manifolds used in microchannel heat exchanger. Different shapes of manifold have analyzed at inlet to observe where the maldistribution effect is less and flow uniformity is more .

Chein and chen [15] have developed numerical models by analyzing different shapes of inlet and outlet header configuration .The shapes taken are U,Z,T,D,Z shapes and tried to find out the effect of the shapes on the flow pattern and temperature maldistribution effect on the microchannel systems.

Kumaraguruparan et al. [16]tried to find out the effect of the different parameter on maldistribution for their analysis they have done numerical analysis on a channel by varying the number of channels, by varying the viscosity of fluid used, by changing the shape of the header. They found that with presence of maldistribution effect there is an increase in maldistribution whereas viscosity holds an inverse relationship.

Amador et al. [17] have done research on the effect of manufacturing tolerance on the maldistribution of the heat exchanger. They also consider the effect of manufacturing defect on maldistribution. According to their finding geometry defects increases the maldistribution.

Chapter 3

Maldistribution in heat exchanger and Residence time Analysis

3.1 Introduction

In general cases, it is assumed that the flow in a heat exchanger is uniformly distributed in all channels of a heat exchanger but in practical cases, all channels or tubes does not have same mass flow rate this phenomena generally called as flow maldistribution. Due to the uneven distribution of mass in all tubes, the heat exchanger efficiency severely affected. So in order to get a good heat exchanger design one should know the maldistribution effect and how to get a uniform flow in the heat exchanger so that the heat transfer will be effectively done. Mass flow distribution is defined as different mass flow rate at both inlet and outlet header or at the inside of the heat exchanger, the main factors which cause maldistribution are heat exchanger design defects which include defect in geometry, manufacturing defects, leakage, etc. and operating conditions of heat exchanger such as fluid flow , fouling factors.

3.2 maldistribution due to geometry defects

Geometry based maldistribution generally refers to the flow maldistribution due to the defects in heat exchanger inlet header, manifold, nozzle design , etc. This type of flow maldistribution generally occurs during to the fabrication process of the header or due to some manufacturing tolerances.The manufacturing operations such as welding, brazing in pipes can also effect the uniform flow of mass. Any leakage within the system of heat exchanger also considers as geometry based flow maldistribution.

Geometry defects maldistribution can be categorized into three category

- Gross flow maldistribution
- Passage to passage flow maldistribution
- Manifold-induced flow maldistribution

3.2.1 Gross flow distribution

Gross flow maldistribution is generally occurred due to the non-uniformity in flow, the main reason for this is the improper design of the header , a blockage inside the pipes, etc. Due to this, there is a severe pressure drop inside the heat exchanger and effects the heat transfer rate.

3.2.2 Passage to Passage flow maldistribution

This type of maldistribution is generally occurred due to the improper size of the channels used in the heat exchanger , as of different sizes of the channels the mass flow rate varies , and a non-uniformity in flow is observed .The main reason for this type of channel defects is the improper manufacturing processes. This type of maldistribution severely affects the heat transfer.

3.2.3 Manifold-induced flow maldistribution

Manifolds are the important part in a heat exchanger it basically distributes the flow from a larger stream to a number of small streams and the streams are collected at the outlet as a single stream. Different types of manifolds are found application in industries such as dividing combining, U- type and Z-type .Due to flow pattern in manifolds such as reverse flow, manifold design, etc. .

3.3 Governing equations in shell-and-tube heat exchanger with maldistributional in tube side

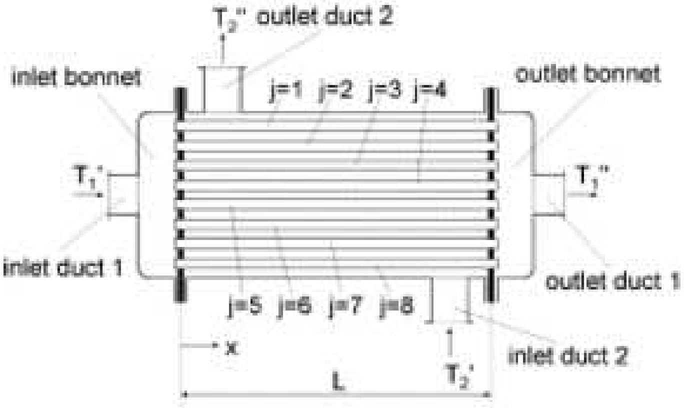

Figure 3.1: Schematic of shell and tube heatexchanger with tube channel

One of the basic assumption taken for the heat exchanger is that the flow in the tube side is assumed to be have constant velocity in the shell side it is assumed that it has plug flow .In practical case the velocity is not same in all of the tubes and certain maldistribution were observed. The equation given by

$$\frac{W_{1j}}{\bar{W}_1}\frac{d\theta_{1j}}{d\xi} + \frac{\frac{KA}{N}}{\frac{(\dot{W}_1)}{N}+}(\theta_{1j} - \theta_2) = 0 \tag{3.1}$$

with further simplification the equation looks like

$$\frac{W_{1j}}{\bar{W}_1}\ \frac{d\theta_{1j}}{d\xi} + NTU_1(\theta_{1j} - \theta_2) = 0 \tag{3.2}$$

The second term of equation 3.2 is an indicative of heat flux in a tube due to shell side fluid The above equation can further be expanded to N numbers of tubes and can be written as

$$\frac{1}{N} \sum_{j=1}^{N} \frac{d}{d\xi} \frac{W_{1j}}{\bar{W}_1} \theta_{1j} + NTU_1 \{ \frac{1}{N} \sum_{j=1}^{N} \theta_{1j} - \theta_2 \} = 0 \tag{3.3}$$

The shell side equation can be written as

$$\frac{d\theta_2}{d\xi} + NTU_2 \{ \frac{1}{N} \sum_{j=1}^{N} \theta_{1j} - \theta_2 \} = 0 \tag{3.4}$$

For the condensation and evaporation problem the $NTU2 = 0$,If the flow is parallel then $NTU1 < 0$ and for counterflow $NTU2 > 0$ In general cases the flow velocity in all the pipes is not same but some tube have back flow but the mean flow velocity in forward direction can be written as

$$\bar{w_1} = \frac{1}{N} \sum_{j=1}^{N} W_{1j} \tag{3.5}$$

The above equations 3.2 and 3.4 can be solved using different sets of boundary conditions Boundary condition without back flow both for concurrent flow and counter flow

1. With out back flow
$$\xi = 0, \qquad \theta_{1j} = 1 \tag{3.6}$$
2. Counter flow
$$\xi = 1 \qquad \theta_2 = 0 \tag{3.7}$$
3. For concurrent flow
$$\xi = 0 \qquad \theta_2 = 0 \tag{3.8}$$

for the case of back flow two tubes are taken for analysis where one is having forward flow and another is having back flow The tube taken are named as T1 and T2 The boundary conditions are given below

1. Outlet cross section

$$\xi = 0 \quad \sum_{j=1}^{N} \frac{W_{1j}}{\bar{W}_1}(\theta_1 - \theta_{1j}) = 0, W_{1j} > 0 \tag{3.9}$$

2. Inlet cross section

$$\xi = 1 \quad \sum_{j=1}^{N} \frac{W_{1j}}{\bar{W}_1}(\theta_1 - \theta_{1j}) = 0, W_{1j} > 0 \tag{3.10}$$

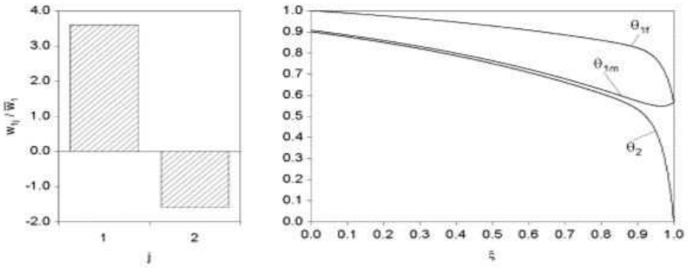

Figure 3.2: Temperature profile of two tubes with backflow

3.4 Maldistribution with back flow

Some cases may arise when due to maldistribution effect some pipes may have back flow which severely affect the heat exchanger performance.For the analysis of backflow simple two tubes Reference should added are taken for analysis one having a forward velocity and one having a backward flow or reverse flow .The problem is solved numerically by .Due to back flow in one channel the fluid in the back flow tube is adiabatically mixed with the inlet forward flow tube ,hence a sharp drop in temperature profile is observed at inlet of the heat exchanger.If a heat exchanger of counterflow type with perfect plug flow is taken keeping all other parameter constant exchanger then the area required is 11 percent of the actual case with maldistribution.

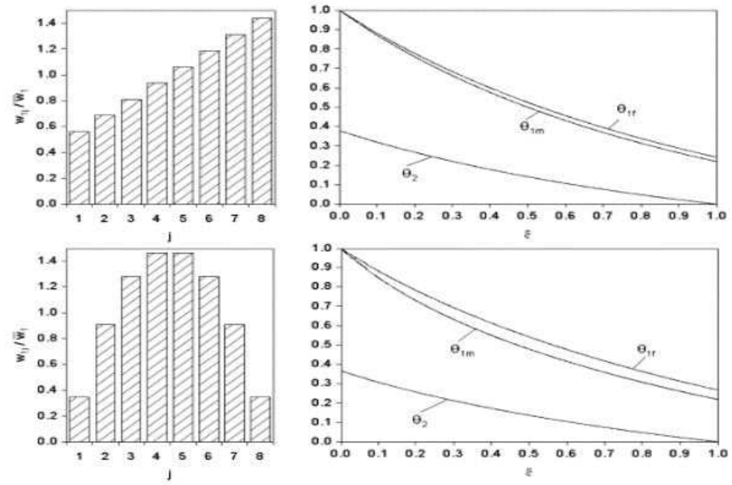

Figure 3.3: Temperature profile for maldistributin calculated numerically

3.5 Maldistribution without back flow

This is the general case of heat exchanger where the pipe have simple flow but the not the ideal plug flow.The steams coming from the header to the channel tube will not distributes the mass evenly in all channels and as it is not a plug flow due to the boundary layer formation some axial conduction takes place .A Simple analysis has been done [2] where they have analyzed 8 channels with different mass flow rates in different tubes , the above system is solved using numerical scheme and temperature profile was plotted against the length of heat exchanger

For the analysis of heat exchanger two types of temperature has been taken for analysis first ons is cross sectional mean and second one is adiabatic mean temperature. In case of plug flow the two temperature at particular cross section remains same because all the elements in flow have same velocity if maldistribbution occurs two cases may arise when in one case the temperature may same for all channels then the two temperatures remains same. In the third case when temperatures at different tubes are not same then the two temperatures at a particular cross section takes different value.The above fig shows the maldistribuiton affect and temperature variation.

3.6 Mach number

Mach number in heat exchanger analysis may be defined as the ratio of fluid flow velocity to velocity of propagation .A basic assumption in heat exchanger with out maldistribution is that the flowing velocity is much lower than the velocity of propagation , hence in general cases the Mach number is equal to zero.But when maldistribution effect is observed in a heat exchanger than the velocity of propagation is not high enough and it has a finite magnitude in some cases it is of the order of the flow velocity . Hence the Mach number is not zero but it takes a finite value and more than zero.Some cases in maldistribution may arise where in a bundle of tubes some may have back flow ,due to the back flow Mach number falls in between 0 and 1.This zone is called as subsonic zone.But in case of forward flow maldistribution bundle of pipes have different flow rates so the exit from a pipe is not in the same time.In this case the Mach number is coming to be greater than 1, this zone can be called as super sonic zone.In case where Mach number is 1 i.e Sonic then in the bundle of tubes the fluid is static. Some mathematical correlation has been given by [2] to calculate the Mach number

3.6.1 Hyperbolic dispersion model

In this model the effect of maldistribution is taken and the shell side modified equation is given by

$$\frac{d\theta_{1,hyp}}{d\xi} + \frac{1}{Pe}\frac{d\varphi}{d\xi} + NTU_1(\theta_{1,hyp} - \theta_{2,hyp}) = 0 \tag{3.11}$$

As the shell side is of constant heatflux so

$$\theta_{2,hyp} = 0$$

As the flow is maldistributed the conduction of heat also have finite value which is given by Chester equation as

$$\varphi + \frac{M^2}{Pe}\frac{d\varphi}{d\xi} = -\frac{d\theta_{1,hyp}}{d\xi} \tag{3.12}$$

The equations 3.11 and 3.12 can be solved by using following boundary conditions

$$\theta_{1,hyp}(0) = 1 \qquad \varphi(0 = 0)$$

The Mach number derived by(authors) with assumption that the flow is forward and no reverse flow is happening is given by

$$M^2 = \frac{\frac{1}{N}\sum_{j=1}^{N}\frac{\bar{W}_1}{W_{1j}}}{[\frac{1}{N}\sum_{j=1}^{N}\frac{\bar{W}_1}{W_{1j}}] - 1} \tag{3.13}$$

The Mach number can also be found out by using residence time measurement.In this method of measurement an arbitrary inlet signal of square nature it injected at the inlet with respect to time and the outlet response of the arbitrary signal is measured at outlet. For the analysis simple two channel have taken both having forward flow and the energy equation are used for the unsteady process. the governing differential equation used for solving is given by

$$-\frac{w_j}{\bar{w}}\frac{\partial\theta_j}{\partial\xi} = \frac{\partial\theta_j}{\partial\tau} \tag{3.14}$$

The temperature or concentration is taken for the analysis for a short period of time the temperature is maintain and the outlet signal comes out from the two individual channels with out any changes.The individual residence time can be calculated by

$$t_{vj} = \frac{L}{w_j}$$

The response at the outlet may be taken individually or by taking combinly at the header. Above model is analyzed for the case of hyperbolic dispersion with a peclect number being equal to 0. The below equations were solved to find out the outlet response .

$$-\frac{\partial\theta_{hyp}}{\partial\xi} - \frac{\partial\psi}{\partial\xi} = \frac{\partial\theta_{hyp}}{\partial\tau} \tag{3.15}$$

$$Pe\psi + \frac{\partial\theta_{hyp}}{\partial\xi} + M^2\frac{\partial\psi}{\partial\xi} = -M^2\frac{\partial\psi}{\partial\tau} \tag{3.16}$$

where

$$\psi = \frac{\varphi}{Pe}$$

The equation 3.15 and 3.16 can be solved by the following initial conditions and boundary conditions

$$\theta_{hyp}(\xi, 0) = 0\psi(\xi, 0) = 0\psi(0, \tau) = 0 \tag{3.17}$$

The equations 3.15 to 3.17 solved using the Laplace method and the mach number is equal to 5. Fig shows the inlet signal and outlet signal response

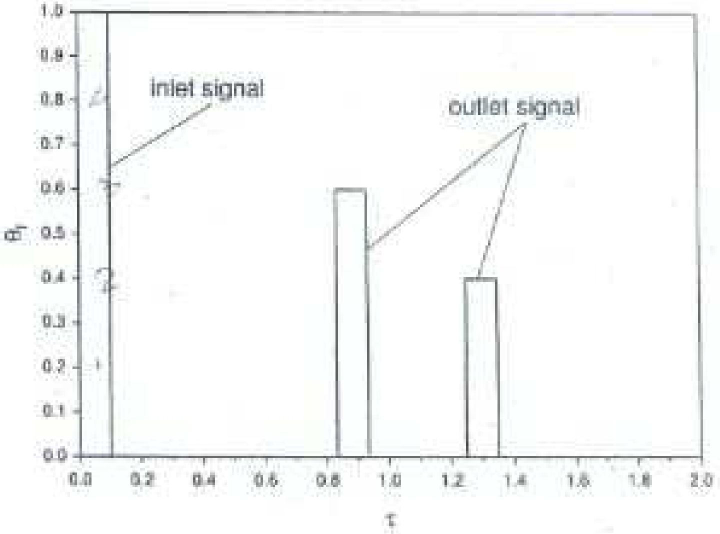

Figure 3.4: Inlet and Outlet response curve of a square signal in two channel which gives a Mach number =5

The residence time curve can be used to calculate Mach number for this two channel .Pulse inlet signal is seems like a Dirac delta function as for a small time the value is certain and after some time it is equal to zero. Mach number can be found by

$$M^2 = 1 + \frac{1}{\sigma_\delta^2} \tag{3.18}$$

By using the equation 3.18 Mach number can be calculated. Where σ_δ^2 can be obtained from the following formula.

$$\sigma_\delta^2 = \int_0^\infty \theta_{f,\delta}(\xi = 1)d\tau \tag{3.19}$$

Also σ_δ^2 is the difference between inlet variance of curve and outlet variance of the curve.

$$\sigma_\delta^2 = \sigma_{out}^2 - \sigma_{in}^2 \tag{3.20}$$

Where

$$\sigma_{in}^2 = \frac{\int_0^\infty (\tau - \bar{\tau}_{in})^2 \theta_f(\xi = 0)d\tau}{\int_0^\infty \theta_f(\xi = 0)d\tau} \tag{3.21}$$

$$t\bar{a}u_{in} = \frac{\int_0^\infty \tau\theta_f(\xi = 0)d\tau}{\int_0^\infty \theta_f(\xi = 0d\tau} \tag{3.22}$$

$$\sigma_{in}^2 = \frac{\int_0^\infty (\tau - \bar{\tau}_{out})^2 \theta_f(\xi = 1)d\tau}{\int_0^\infty \theta_f(\xi = 1)d\tau} \tag{3.23}$$

$$t\bar{a}u_{in} = \frac{\int_0^\infty \tau\theta_f(\xi = 1)d\tau}{\int_0^\infty \theta_f(\xi = 1)d\tau} \tag{3.24}$$

The equations can be used to calculate variance value and the Mach number can be found from the variance.For simple analysis two channel were taken and the Mach number can be found by the tracer experiment.Apart from this two channel the tracer experiment can be done on more than two channel where the Mach number can be calculated.

3.7 Residence time Analysis

3.7.1 Basics of Residence time Distribution

It is a technique where one can know much time a particle has spent inside a system under consideration .The residence time distribution [18] is generally is used for the analysis of the chemical reactors. The residence time distribution can be done on two reactors first one ideal plug flow reactors and another non-ideal reactors. The residence time curve does not provide any information about the mixing between the fluid elements. RTDs curve basically gives the information about the macromixing inside the fluid elements. Ideal reactors or plug flow reactor are those where the elements entering into the system all will spend the same time and comes out at the same time but in the case of non-ideal reactors all elements do not spend same time, but they will spend different times and came out in a different time. A function named as residence time distribution function (E(t)) is used to describe how much time a particle has spent inside a system. It describes the quantitative manner of particle spent inside a system. Residence time distribution can be done by tracer experiment .the tracer being injected are an inert particle, molecules or atoms. The tracer species used should be non-reactive in order to detect it, be colored one to easy detection and its properties is such that it should not absorb at the wall. Tracer is injected at a time t=0, and the concentration C is measured with respect to time.

Fig 3.5 shows a where the tracer is injected at inlet and the corresponding exit of the tracer is measured at the outlet.Inlet tracer injected will move through the stream and with the stream tracer will comes out after a time.

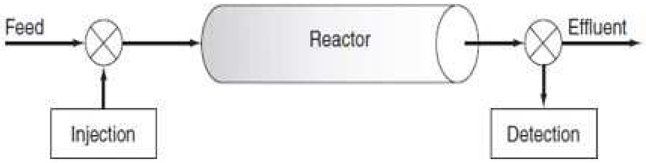

Figure 3.5: Schematic of tracer injection

Generally there are two methods to inject the tracer

1. Pulse input
2. Step Input

Pulse Input

In this method of tracer injection a tracer of known concentration is injected for a very short interval of time and the concentration of injected stream is track at the outlet.During flow the pulse signal spread according to the flow of primary stream.The injection concentration with respect to time seems like a pulse hence this method is called as pulse injection. Fig **??** shows a concentration versus the time curve and it is generally called as the concentration curve or in short C-Curve.

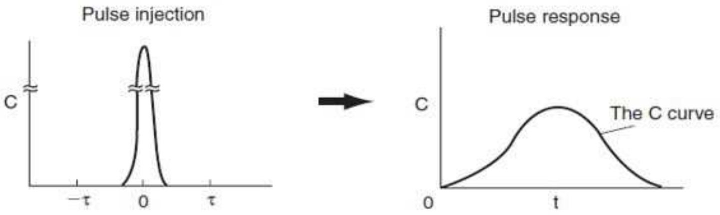

Figure 3.6: Schematic pulse injection at the inlet and corresponding response curve at the outlet

Let a amount of concentration N_0 is being injected for a short interval of time. Let for a small time Δt the amount of tracer leaving a particular area is

$$\Delta N = C(t)vdt$$

Where v= Volume flow rate of the effluent stream.By dividing with the value of initial concentration N_0 to above equation can be re written as

$$\frac{\Delta N}{N_0} = \frac{vC(t)}{N_0}\Delta t$$

Where

$$E(t) = \frac{vC(t)}{N_0}$$

E(t) is known as residence time distribution function.Further simplifying the above equations E(t) can be re written as

$$E(t) = \frac{C(t)}{\int_0^\infty C(t)dt}$$

The significance of E(t) is that it allow some one to measure the fraction of material outgoing through a particular system which spent a time between $t_1 to t_2$ which is equal to

$$\int_{t_1}^{t_2} E(t)dt$$

By using the C-Curve one can find some parameter related to the residence time .The Parameter used generally called as Moments .It includes variance ,skewness of the curve etc.. .

- Mean residence time
- Variance
- skewness

Mean residence time

It is the mean of time a substance spent in a system.It generally defined as the ratio between volume of system to hold and the flow rate of the substance.Mathematically mean residence time can be written as with relation to the E(t) curve as

$$\bar{t} = \int_{0}^{\infty} tE(t)dt \tag{3.25}$$

Where $\bar{t}$= Mean residence time

For the ideal case such as plug flow reactor all the atoms of tracer enters the system stays same time and comes out at a particular time all together.Plug flow is a flow pattern where all the tracer injected to a system must move have the same velocity and their will not be any mixing in the axial direction.What ever the inlet signal is present at inlet will comes out with same with out any change.

Variance

The variance of a curve is the indication of spread in the concentration with respect of time. Variance can be calculated by the following equations

$$\sigma^2 = \int_{0}^{\infty} (t - \bar{t})^2 E(t)dt \tag{3.26}$$

If the variance is more then the dispersion will be more i.e. the spreading of mass inside the system is more and vice-versa.By observing the variance macro mixing inside a system can be understood.

Skewness

According to the probability, theory skewness may be defined as asymmetry in the distribution curve by taking mean as a reference. Skewness may be of positive,negative and in some cases it is not defined at all.in case of positive skewness the curve has a tail at the right-hand side and in negative skewness the curve has tail towards left-hand side.It is also called as the 3rd moment. Skewness may be written as

$$Skewness = \int_0^\infty (t-\bar{t})^3 E(t)dt \tag{3.27}$$

Step input

In the step input method unlike the pulse method a known concentration of N_0 not injected for a very short period of time rather injected continuously maintained for a particular time ,at the outlet concentration is measured with respect time.The C-curve seems to be like a step so this method of injection is called as step input method.The figure below shows the how the concentration is kept at a particular time and it looks like a step at the inlet. At outlet due to some diffusion or axial mixing the step input curve is displaced.

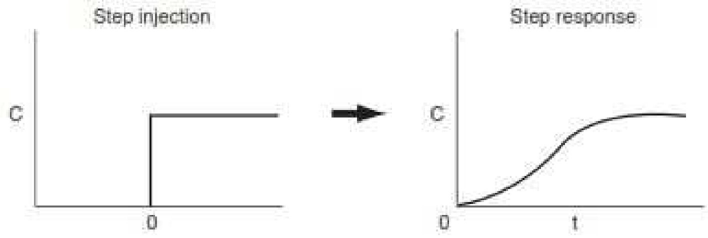

Figure 3.7: Schematic of Step injection and response at the outlet

Step method usually easy to conduct experimental analysis as it is easy to maintain a concentration for a long period time but in case of pulse input concentration is maintain for very short interval of time which is rather a difficult job to do.Secondly in step input method one need not bother about the amount of tracer.The disadvantage of step input is that it uses large amount of tracer which makes the system more costly and it involves lot of mathematical calculations compare to the pulse injection.So pulse injection is generally chosen for the analysis. Details about the step injection is not discussed here as it is beyond the scope of current research Work.

Chapter 4

Overview of Ansys Fluent

4.1 Introduction

Due to development in high-performance capable computers, it is possible to simulate the real time problems in simulation work. Computational Fluid Dynamics (CFD) is a numerical analysis method to solve the fluid flow. Several software has been developed to solve the fluid flow by using a numerical algorithm such as PHONICS [21], FLUENT [12], etc. the basis of all software to use finite volume method as a numerical scheme to solve the equation. CFD has the capability to handle the simulation environment like two-phase flow, evaporation, and melting, chemical reactions with different species, etc. But now a days FLUENT is the one of the best software to deal with the problems related to the fluid flow. Fluent is a part of Ansys , Inc. which is used for solving and designing problems it can simulate the flow in a virtual environment and it will give the results as per the solution obtain. Fluent solver has the capability to deal with the upcoming mesh.

Fluent uses the upcoming mesh file and solve using the finite volume method scheme .To solve any problem in Fluent the basic steps involve are

1. Geometry Design
2. Mesh generation
3. Fluent solver analysis

4.2 Fluent analysis

4.2.1 Geometry Design

Initially the Ansys workbench interface of the Ansys software was opened the window a new Ansys fluent was opened after that proper selection of units,2D or 3D were made etc. were done after doing all the prerequisites the design modular the tab of Fluent was opened according to the requirement the detail sketches and operation were done.

4.2.2 Mesh generation

Ones all the geometry operation did the geometry will be undergone for meshing operation where the total flow domain is divided into small volumes as required by the Ansys fluent. The meshing operation should be done according to the requirements.

4.2.3 Fluent solver analysis

TThe mesh file generated will be read by the FLUENT solver, In FLUENT solver proper discretization scheme, proper boundary conditions, required material, etc. were set for the given problem once all of the above over then the solution was initialized and the solution was calculated for the number of iteration. After the solution has completed, then the results can be calculated using the post processing. Ansys fluent solve according to the flow chat given below Ansys FLUENT basically uses two types of solver Ansys FLUENT basically uses two types of solver

- Pressure based solver
- Density based coupled solver

In pressure based solver by the help of continuity equations pressure velocity coupling algorithm were derived, the primary variable used by the solver is momentum and pressure. Again there are two algorithm to solve in pressure based solver

- Segregated solver
- Coupled Solver

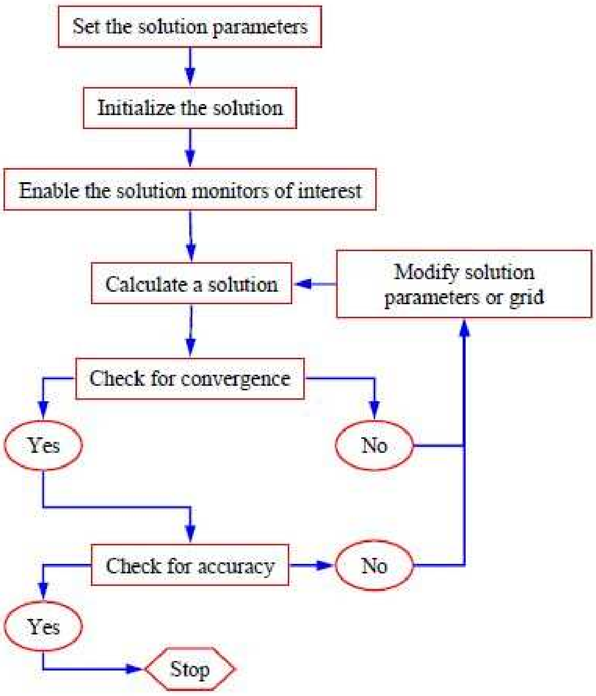

Figure 4.1: Ansys Fluennt process flow chat

In segregated solver pressure and momentum sequentially but in coupled solver pressure correlation and momentum taken simultaneously.segregated solver generally used for incompressible flow where as simpler problems consider but in case of coupled solver used for compressible flow with a high Mach number.Segregated solver runs the program at a faster than coupled solver. With the help of discretization scheme using finite volume method the computational domain is divided into small number of control volume.But the Ansys fluent linearizes the equations in two different way.

Coupled based solver

In coupled based solver equations like momentum,energy ,species,continuity etc . are solved using coupling to each other.If any case some more species are used for the calculation it has to solved using segregated method.As numerical scheme involve iteration so the all equations were solved for number of iteration until a converged reached,after each iteration the different parameter of the solution is obtained as follow

- All properties of fluid is updated after each iteration.If the solution has not started then the initial solution is used as the property.
- All differential equations used by the necessary is solved i coupled manner.
- Where some additional scalar is used then the the respective governing equation can be solved by the the earlier value.

All process is repeated until the solution get converged

Segregated solver

Instead solving as like the coupled based solver in coupled manner the segregated solver uses sequential operation to solve the differential equations.Like the coupled based solver the segregated solver also runs for numbers of iteration in order to get a converged solution.In each iteration the following steps has done by the solver

- Flow properties is being updated in iteration ,in case it is at initial condition the initial value is used for the next iteration.
- Updated value of pressure and mass flux is used for the calculation of velocity components used in continuity equations
- In case where the obtained velocity components does not match the continuity equation in that case the corrected value of the pressure and mass flux is obtained with the help of Poisson equation.

All process is repeated until the solution get converged

4.3 Linearizion Scheme used

Equations used in Ansys Fluent is of non linear so before solving that the equations must be linearize.There are two methods in Ansys Fluent i.e Explicit and implicit

- Explicit Scheme

Here the only known value is used to calculate the value known value.Here the equations are not solve one by one rather all equations are solved at a time.No such relation between the known and unknown value is developed,only one equation is solved for this method.

- Implicit Scheme

This explicit scheme can be used the neighborer node value can be used to calculate the unknown value and the equations are solved one after another by using the unknown value.Here the relation is between the known ,unknown value is taken into consideration.

In general the segregated methods uses the implicit scheme to solve the higher order differential equations where it linearizes the equation into simpler form.Each cell node in this scheme uses single equations to solve the unknown value.Simultaneous solution of the equations are used to solve the value. By the method of coupled solver one can use implicit or explicit scheme according to the requirement.Extra species can be solved using the segregated method as discussed earlier.

4.4 Fluent solver governing equations

fluent solves continuity ,Energy equation ,Navier-Strokes equation different species equations in numerical scheme and gives the result of different parameter such as the temperature ,pressure ,velocity etc. The differential equation used for the analysis are given below

Continuity

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \tag{4.1}$$

Momentum

$$\frac{\partial}{\partial t}(\rho u_i u_j) = -\frac{\partial \rho}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \tag{4.2}$$

Where

$$\tau_{ij} = \mu(\frac{\partial x_i}{\partial t} + \frac{\partial x_j}{\partial t}) - \frac{2}{3}\frac{\mu \partial u_k}{\partial x_k}\delta_{ij}$$

Energy

$$\frac{\partial \rho E}{\partial t} + \frac{\partial}{\partial x_i}(u_i(\rho E + \rho)) = \frac{\partial}{\partial x_i} k \frac{\partial T}{\partial x_i} \tag{4.3}$$

These above are the basic equations used for the analysis of fluid flow but in turbulent analysis turbulent K-ϵ model is used and the governing differential equation can be written as given by

K-ϵ Model

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \bar{u}_j k}{\partial x_j} = \frac{\partial}{\partial x_j}[\mu \frac{\partial k}{\partial x_j}] - \frac{\partial}{\partial x_j}(\frac{\rho}{2}\overline{\dot{u}_j \dot{u}_i \dot{u}_i} + \overline{\dot{p}\dot{u}_j}) - \overline{\dot{p}\dot{u}_j \dot{u}_i}\frac{\partial \bar{u}_i}{\partial \bar{x}_i} - \mu \overline{\frac{\partial \dot{u}_i}{\partial x}}\,\overline{\frac{\partial \dot{u}_i}{\partial x}} \tag{4.4}$$

$$\frac{\partial \rho \epsilon}{\partial t} + \frac{\partial \rho \epsilon u_j}{\partial t} = C_{\epsilon 1} P_k \frac{\epsilon}{k} - \rho C_{\epsilon 2}\frac{\epsilon^2}{k} + \frac{\partial}{\partial x_j}(\frac{\mu_t}{\sigma}{}_{\epsilon}\frac{\partial \epsilon}{\partial x_j}) \tag{4.5}$$

where

$$\mu_t = \rho C_\mu \sqrt{k} L$$

the equation of the model has a combination of several number of terms and it is a very highly complex equation.

Additional to the equation when ever a species use to solve the species transport model is used for analysis By considering the flow is incompressible the species transport equation can be given as

$$\frac{\partial C}{\partial t} + \frac{\partial}{\partial x_i}(u_i C) = \frac{\partial}{\partial x_i}(D\frac{\partial c}{\partial x_i}) + S \tag{4.6}$$

ce the coefficient of diffusion can be neglected $D = 0$ and as here no additional source is used so the source term S is also zero. Hence for the ideal case above differential equation can be re written as

$$\frac{\partial C}{\partial t} + \frac{\partial}{\partial x_i}(u_i C) = 0 \tag{4.7}$$

For the analysis of tracer it is assumed that the flow is ideal which does not consider the diffusion ,hence in real case earlier equation 4.6 is being solved by the Fluent This species transport equation is solved by the Fluent numerically and yields the value of local species and transport properties.The source term is basically used where generation or destruction of species occurs , where their is a need of physical problem. Species transport is generally used for chemical reactions.

4.5 Steps in Fluent to solve a problem

After proper analysis of physical problem the geometry followed by meshing is done then the file is imported to the Fluent solver where some steps are followed to solve the problem

1. Import of the meshing file as required by the solver i.e.wheather 2-D or 3-D
2. Select basic setting of the solver
3. Proper selection of the models are done as required by the problem specification which includes such as energy ,laminar ,turbulent ,species transport etc.
4. select the proper material and specify the property as per requirement.
5. Provide boundary conditions
6. Set the solution controls as per convenience
7. Initialize to get a initial gauss for the problem
8. Give number of iteration and Calculate the solution
9. Analysis of the result
10. Analise problem with a grid independent test.

Chapter 5

Numerical Simulation and Results

The numerical solution has been done on the heat exchanger using Ansys Fluent. Simulation work consists of both steady and unsteady problems .Based on the requirement several models are taken for analysis. For steady state analysis, two problems have taken for analysis.

5.1 Problem Specification

5.1.1 Case 1

For the first case, a steady state model has been taken for analysis The model used for analysis consists of two tube with where on the tube having positive flow and other tube having negative flow velocity. In the tube hot fluid is flowing, and the shell side is used for the cold fluid.

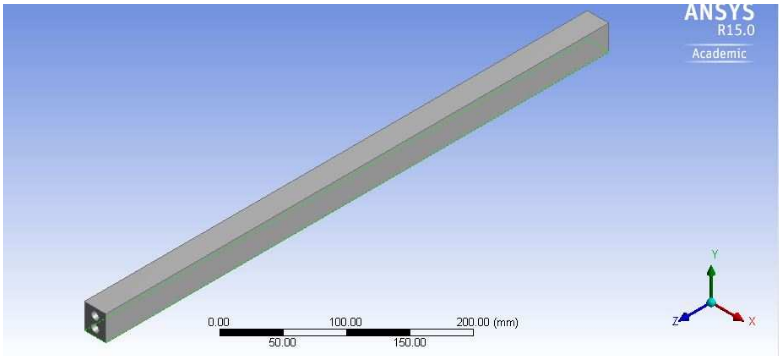

Figure 5.1

length of the heat exchanger=560 mm

Diameter of the tubes=8mm

Height = 26 Width =24

Boundary conditions

Mass flow rate channel 1 =5.4 kg/sec, channel 2= 2.5 kg/sec

In channel 1 forward flow is considered and in channel 2 backward flow is considered.The meshing for this simulation

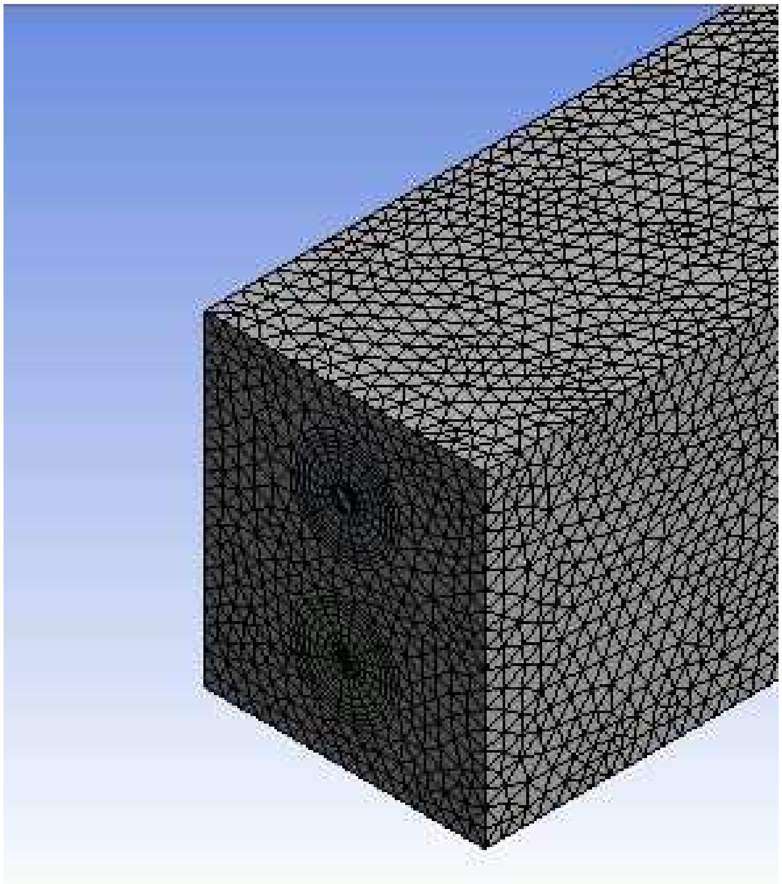

Figure 5.2: Meshing for the model

Fluent Solver Setting

After setting the boundary conditions and choosing the proper material in fluent solver setting the following under-relaxation factors are set.

- Pressure 0.3
- Energy 1.0
- Momentum 0.7
- Turbulent kinetic energy 0.8

- Turbulent dissipation rate 0.8
- For pressure choose, STANDARD
- For momentum choose, First order upwind
- For pressure-velocity coupling choose, SIMPLE
- For energy choose, First order upwind
- For turbulent kinetic energy choose, First order upwind
- For turbulent dissipation rate choose, First order upwind

The convergence criteria are

Continuity = 0.001 k = 0.001 = 0.001 x, y, z velocity = 0.001 Energy = 1e-6

5.1.2 Case 2

Another model has been developed with 2-D with inlet and outlet header , and a tracer experiment has performed using water as a primary fluid and tracer as the secondary fluid. The tracer is being injected for a certain time of 0.1 sec and after a certain time, the injection of the tracer stopped .The tracer signal is plotted at the inlet with respect to time it seems that a square signal is injected at the inlet, outlet signal monitored at outlet of two channels .All the analysis is performed is based on the assumption that the flow inside channel is plug flow. The results are compared with the analytical method analysis of the tracer analysis. Geometry for the simulation work designed such a way that the fluid should behave ideally. For the tracer analysis, Mini-channel of dimension less than 3mm is taken.

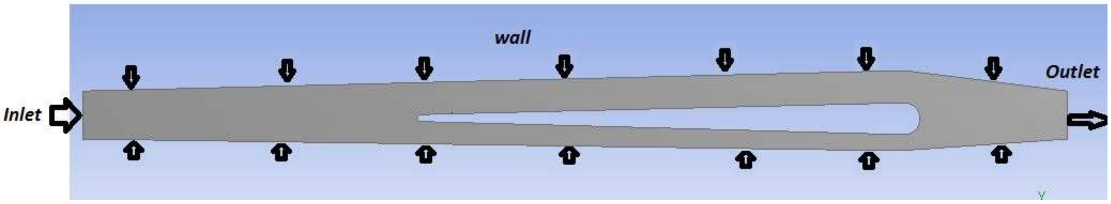

Figure 5.3

Length of the model=60 mm

Bigger diameter channel= 2 mm

Smaller diameter channel=0.9 mm

Inlet header diameter=3 mm

Outlet header diameter=3 mm

Mesh generation for the model done with a total number of nodes 197177 and number of elements is 193353 and the minimum orthogonal quality is o.55.

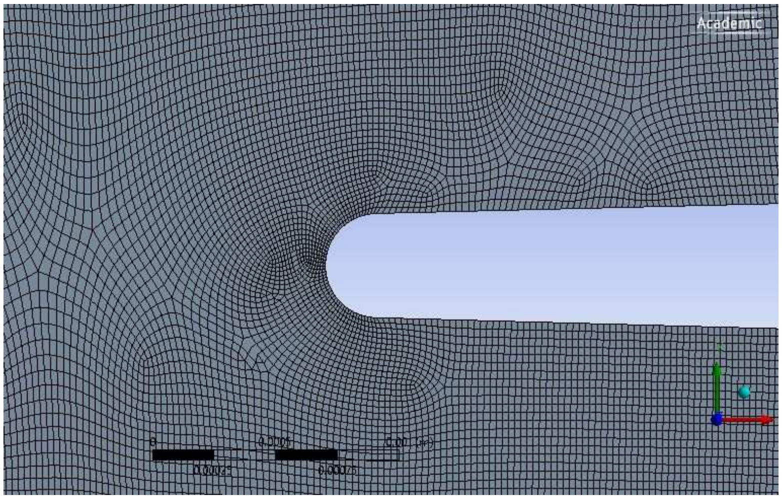

Figure 5.4: Meshing for the geometry

Material used for simulation

Simulation of the model carried out by taking water as a primary fluid and another material which is same property as water which is generally secondary fluid .Secondary fluid also called as tracer.

Table 5.1: Properties of Material

Property	Water liquid	tracer
Density	998.2 $\frac{Kg}{m^3}$	998.2 $\frac{Kg}{m^3}$
Specific Heat	4182 $\frac{j}{kgK}$	4182 $\frac{j}{kgK}$
Viscosity	0 $\frac{Kg}{m-sec}$	0 $\frac{Kg}{m-sec}$
Molecular Weight	18.015 $\frac{Kg}{Kgmol}$	18.015 $\frac{Kg}{Kgmol}$
Thermal Conductivity	0.6 $\frac{w}{mK}$	0.6 $\frac{w}{mK}$

The above two material in 5.1 are used to make a mixture which is used by the species transport model.Mixture property can be written as

Viscocity of mixture= 0 $\frac{Kg}{m-sec}$

Mass Diffusivity=0 $\frac{m^2}{sec}$

Above properties are taken considering the flow as ideal.

Fluent Solution Setup

1. General
   - Pressure-Based Solver
   - Transient
   - 2D Planer

1. Models
   - Viscous Laminar
   - Species Transport

1. Material

   - Mixture(Water + Tracer)

1. Boundary Conditions

   - velocity inlet=0.05 $\frac{m}{sec}$
     Tracer mass fraction for 0.1 sec=1 , after 0.1 sec=0
   - At wall slip boundary condition and shear stress =0 Pascal
   - pressure Outlet

1. Solution methods

   - Pressure-Velocity Coupling , Scheme=SIMPLE
   - Gradient=Least squares cell based
   - Pressure = Second Order
   - Momentum = Second Order Upwind
   - Tracer = Second order Upwind
   - Transient formulation=1st order implicit

1. General

   - Pressure=0.3
   - Density=1
   - Body Forces=1
   - Momentum=0.7
   - Tracer=1

1. Solution Initialization

   - Gauge Pressure= 0 pascal

- X velocity= 0 $\frac{m}{sec}$
- Y Velocity =0 $\frac{m}{sec}$
- Tracer = 0

5.1.3 Case 3

To know the temperature distribution in case of mini channel with maldistribution a Computational model was made.The model consist of 4 channels and each channel is considered to have different flow of masses.A constant heat flux at the base of channel is considered. The geometry can be shown as The fluent set up is similar to the first case except the mass flow rate boundary conditions is replaced by velocity inlet ,velocity in individual pipe taken different to consider maldistribution. Another boundar condition choosen i.e constant heat flux at the base of the channel.

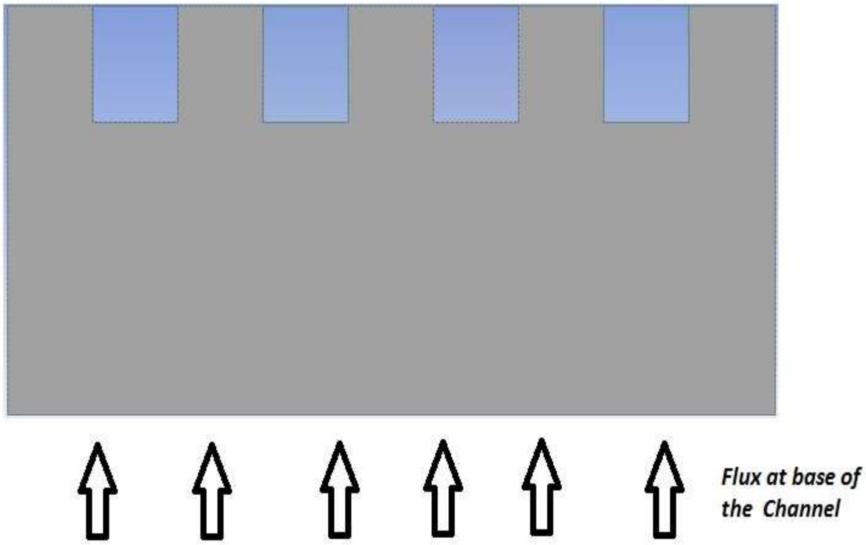

Figure 5.5: Front view of the model

length of channel= 50 mm

width of the channel=0.33 mm

height of the channel=0.4 mm

5.2 Results and Discussion

5.2.1 Case 1

From present study ,the mass weighted average of the temperature along the length of heat exchanger is plotted.

Mass weighted Average temperature

Mass weighted temperature may be defined as the ratio of the product of the selected temperature and the absolute value of the dot product of the facet area and momentum vectors by the summation of the absolute value of the dot product of the facet area and momentum vectors (surface mass flux):

For a particular position the mass weighted average can be calculated as

$$T_{a,d} = \frac{\int \rho \bar{u}.d\bar{A}}{\int \rho \bar{u}.d\bar{A}} \tag{5.1}$$

Temperature contour of along the length direction can be shown below

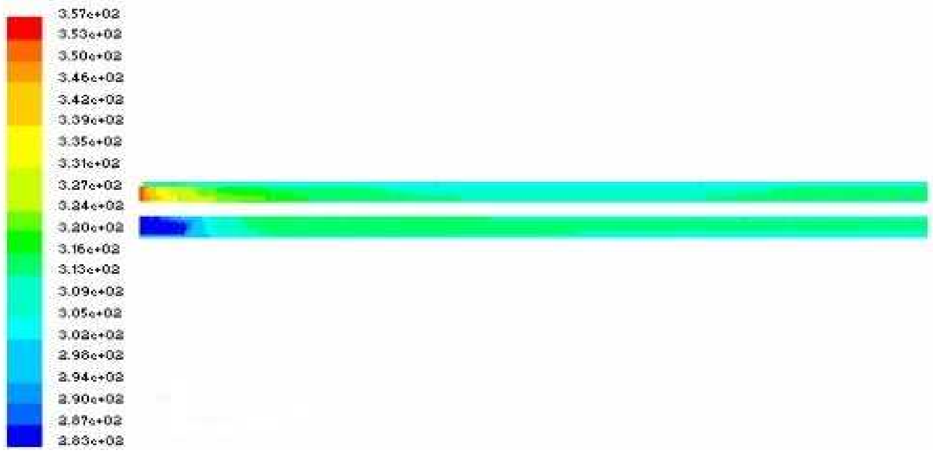

Figure 5.6: Temperature contour for maldistribution with backflow in one channel

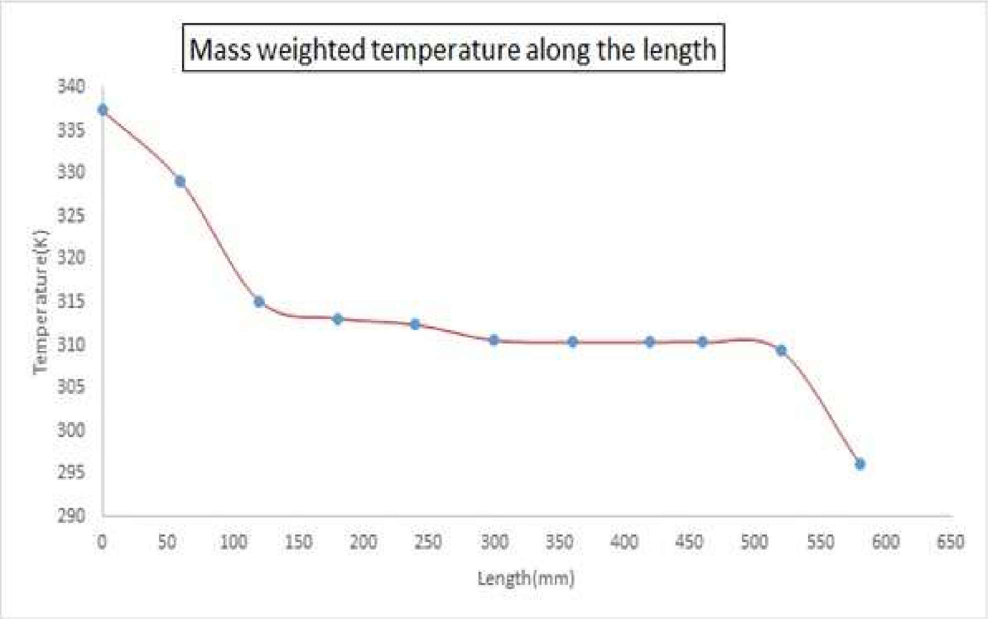

Figure 5.7: Temperature vs length of the heat exchanger

Hot fluid is entering through the channel in one channel the floe is positive and another channel the flow is backward direction.As their is backflow in one channel their is temperature jump at the inlet of channel as shown in 5.7 ,the reason for the temperature drop is that the backward flow of the fluid adiabatically mixed at the inlet bonnet of the heat exchanger.The backward flow is maldistributted.

5.2.2 Case 2

Calculation of Mach number has been done by using the computational domain where a tracer is injected for 0.1 sec at inlet ,after that the tracer injection stopped .The movement of trace can be observed for different time,all the simulation has run for unsteady state with taking the fluid as ideal one.The mixture used here also ideal.

Figure 5.8: Tracer after 0.1 sec

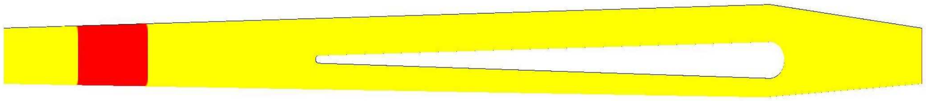

Figure 5.9: 0.2 sec

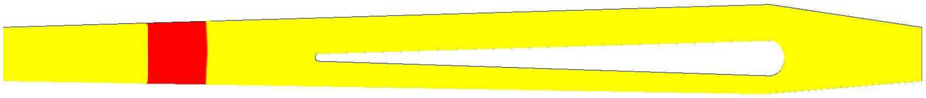

Figure 5.10: 0.3 Sec

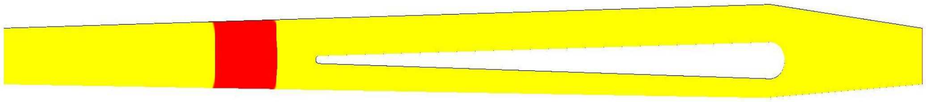

Figure 5.11: 0.4 Sec

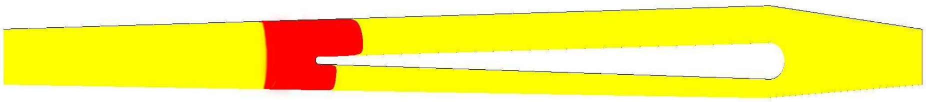

Figure 5.12: 0.5 Sec

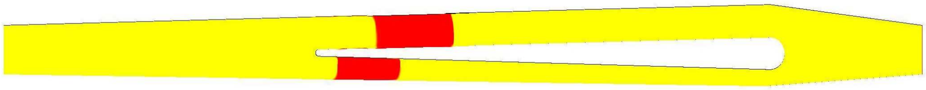

Figure 5.13: 0.6 Sec

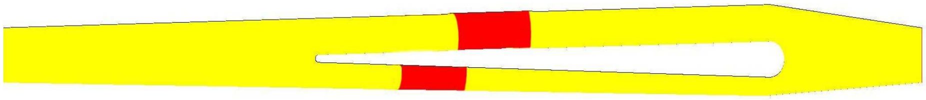

Figure 5.14: 0.7 Sec

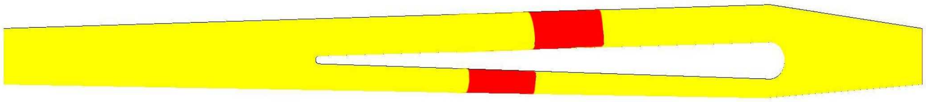

Figure 5.15: 0.8 Sec

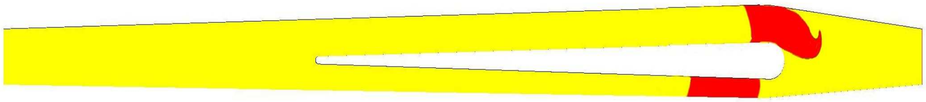

Figure 5.16: 1 Sec

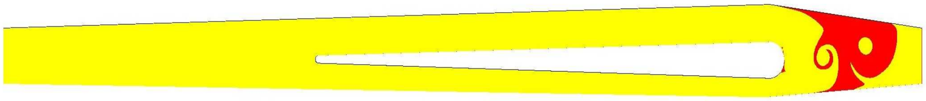

Figure 5.17: 1.2 Sec

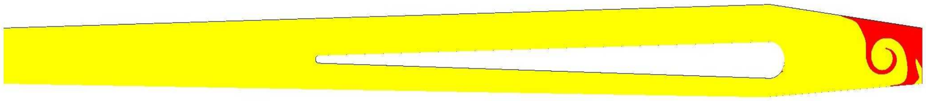

Figure 5.18: 1.4 Sec

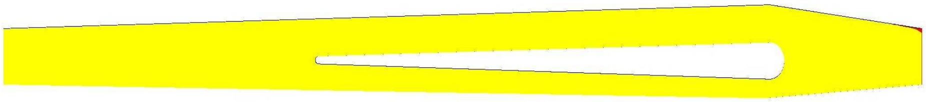

Figure 5.19: 1.6 Sec

At inlet tracer Mass fraction with respect to time for a 0.1 Sec is taken ,the signal at inlet is of square pulse signal which travels through the channel with two different velocity and at the outlet mass fraction of the tracer is plotted with respect to time . The graph at Inlet and Outlet found as

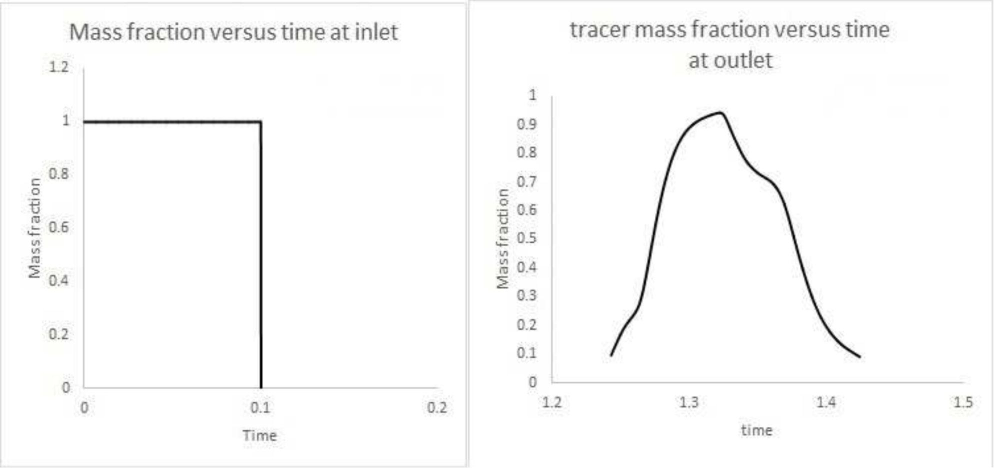

Figure 5.20: Tracer Mass fraction at the Inlet and Outlet

After injecting tracer for 0.1 Sec the injection stops and it flows through the inlet header after 0.5 sec tracer start dividing and moves with two different velocity ,the flow of the tracer is ideal after time 0.9 Sec the tracer in high velocity comes out through the pipe while in the less velocity channel tracer comes out after sometime.At the outlet header tracer from both the pipe mixed with each other and comes out at the outlet.Inside channel Plug flow is assumed i.e. all the pipe cross section have same velocity. Mass fraction is plotted against the time at Inlet and Outlet.At inlet plot between the mass fraction and time Fig. 5.20 observed to be a square signal but at outlet it is no longer square because the tracer is divided in two channel and taking different time to pass the domain ,at the header plot between tracer and time is more dispersed. Calculation of σ^2 for both inlet and outlet curve has been carried out by using equations 3.21 *to* 3.24 and it is found σ^2 for inlet is 0.00083 and for outlet is 0.0416.The values is well obeyed with the analytical analysis given by Roetzel and Ranong [2] for hyperbolic model on heat exchange which has variance value 0.00083 for inlet and 0.042500 at outlet.

5.2.3 Case 3

Constant heat flux at the base of channel is applied all four channel have different velocity and the simulation is run for the steady state.Water is used as material.The temperature contour inside the fluid element obtained can be seen as

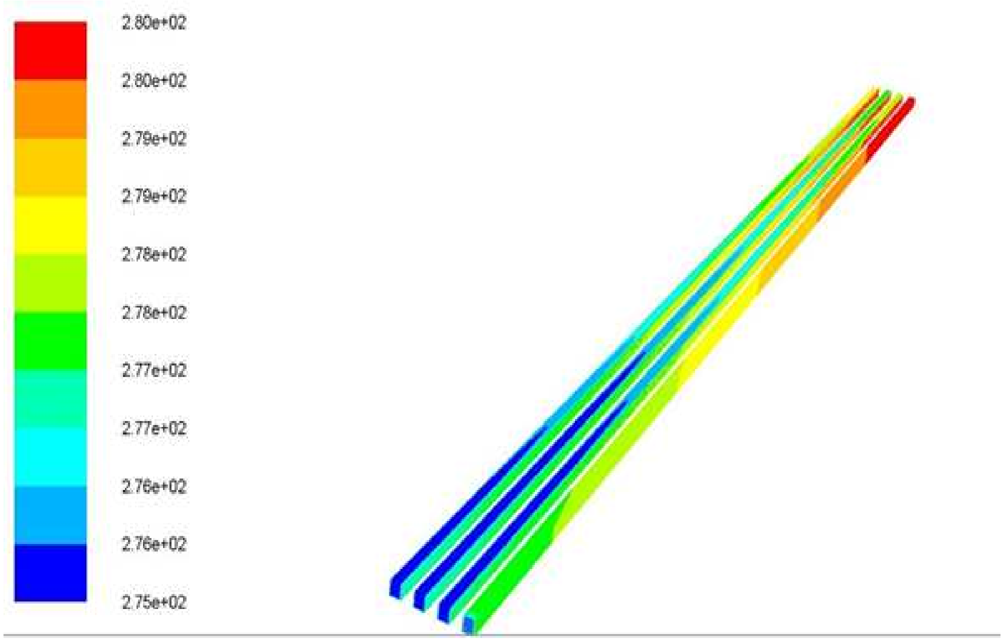

Figure 5.21: Temperature Contour with different velocity in each channel

From the contour it observed that due to Maldistribution effect some of the fluid are moving with higher mass flow rate and some are in less mass flow rate as a result the heat is taken by the fast moving fluid earlier and some are taking after some time which reduces the heat transfer efficiency.

Chapter 6

Conclusion

From the above analysis conclusion derived were

1. With maldistribution considering the effect of backflow in one channel their is a sharp drop in temperature at the inlet of the heat exchanger as backflow fluid is adiabatically mixing with the forward flow fluid.This can severely effect the effectiveness of heat exchanger.
2. With the help of tracer analysis it is possible to find the Mach number which is an indication of Maldistribution in heat exchanger.For two channels the results of tracer experiment exactly match with the result predicted in hyperbolic model.
3. Simple case of Mini-channel were taken with constant heat flux at the base with maldistribution effect at the channel and temperature contour is being observed.

Scope for future Work

1. More research has to be done on the tracer experiment for real case .Further it can be extended to more than two channels.
2. Experimental determination of the model parameter can be done.
3. Tracer can be applied to mini channel heat sink that were used for the cooling of electronics devices.

Bibliography

[1] Kandikar S.G. Microchannels and minichannels - history, terminology, classification and current research needs. In *ICMM2003-1000 presented at First International Conference on Microchannels and Minichannels, April 24-25, Rochester, New York, USA*.

[2] W. Roetzel and Ch. Na Ranong. Axial dispersion models for heat exchangers. *Int. J. Heat Technol.*, 18:7–17, 2000.

[3] Taylor SG. Dispersion of soluble matter in solvent owing slowly through a tube. *Proc. Royal Soc. London A*, 219:186–203, 1953.

[4] Danckwerts P.V. *Continuous ow systems distribution of residence times*, volume 2. 1953.

[5] Sahoo RK and Roetzel W. Hyperbolic axial dispersion model for heat exchangers. *Heat and Technology*, 18:1261–1270, 2000.

[6] P. Florent Lalot, S. and Bergles Langc, S.K. Flow maldistribution in heat exchangers. *Applied Thermal Engineering*, 111:847–863, 1999.

[7] Bengt Sunden Das S.K Prabhakara Rao, Bobbili. An experimental investigation of the port flow maldistribution in small and large plate package heat exchangers. *Applied Thermal Engineering*, 26:1919–1926, 2006.

[8] Anindya Roy and S.K. Das. An analytical solution for a cyclic regenerator in the warm-up period in presence of an axially dispersive wave. *Int. J. Therm. Sci.*, pages 21 –29, 2001.

[9] Spang B. Luo X. Roetzel, W. and S.K. Dash. Propagation of the third sound wave in fluid : hypothesis and theoretical foundation. *International Journal of Heat and Mass Transfer*, 41:2769 –2780, 1998.

[10] Zhe Zhang and YanZhong Li. Cfd simulation on inlet configuration of plate-fin heat exchangers. *Cryogenics*, 43:673–678, 2003.

[11] Wilfried Roetzel and Frank Balzereit. Axial dispersion in shell-and-tube heat exchangers. *Int. J. Therm. Sci.*, 39.

[12] A.C. Mueller. Effects of some types of maldistribution on the performance of heat exchangers. *Heat Transfer Engineering*, 8(2):75–86, 1987.

[13] Anil Kumar Dwivedi and Sarit Kumar Das. Dynamics of plate heat exchangers subject to flow variations. *International Journal of Heat and Mass Transfer Volume*, 50(13–14):2733–2743, 2007.

[14] Falk L. Corriou J. P. Commenge, J. M. and M. Matlosz. Optimial design for flow uniformity in microchannel reactors. *AIChE J.*, 2:345 –358, 2002.

[15] R. Chein and J Chen. Numerical study of the inlet/outlet arrangement effect on microchannel heat sink performance. *Int. J. Therm. Sci.,*, 48:1627–1638, 2009.

[16] Manikanda Kumaran R. Sornakumar T. Kumaraguruparan, G. and T Sundararajan. A numerical and experimental investigation of flow maldistribution in a microchannel heat sink. *Int. J. Commun. Heat Transfer*, 38:1349 –1353, 2011.

[17] Gavriilidis A. Amador, C. and Angeli. Flow distribution in different microreactor scale-out geometries and the effect of manufacturing tolerances and channel blockage. *Chem. Eng. J.*, 101:379 –390, 2004.

[18] O LEVENSPIEL. *Chemical Reaction Engineering.* New York: Wiley, 1999.