Performance Analysis of Shell and Tube Heat Exchanger

A Project Report

Submitted by

M.Mohamed Abdul Rahim - 71205114023
S.Venkateshwaran - 71205114053
A.Vinoth - 71205114058

in partial fulfillment for the award of the degree
of

Bachelor of Engineering
in
Mechanical Engineering

DEPARTMENT OF MECHANICAL ENGINEERING
KUMARAGURU COLLEGE OF TECHNOLOGY
COIMBATORE – 641 006

ANNA UNIVERSITY :: CHENNAI 600 025

APRIL – 2009

ANNA UNIVERSITY :: CHENNAI 600 025

BONAFIDE CERTIFICATE

Certified that this project report entitled **"Performance Analysis of Shell and Tube Heat Exchanger"** is the bonafide work of

Mr. M.Mohamed Abdul Rahim	-	Register No 71205114023
Mr. S.Venkateshwaran	-	Register No 71205114053
Mr. A.Vinoth	-	Register No 71205114058

Who carried out the project work under my supervision.

24/4/2009

Signature of the Head of the Department

(Dr.T.Kannan)

HEAD OF THE DEPARTMENT

Signature of the supervisor

(Mr.K.G.Maheswaran)

SUPERVISOR

27/4/2009

Internal Examiner

27/4/09

External Examiner

DEPARTMENT OF MECHANICAL ENGINEERING

KUMARAGURU COLLEGE OF TECHNOLOGY

COIMBATORE 641 006

ABSTRACT

Today manufacturing sector has entered into the path of reducing the size, shape, weight etc. But there comes the problem of heat dissipation from the machines. Heat that rejected during the operation is the losses of energy. Due to this the performance of the machines will reduce and brings the production a hault. Thus the optimum life time of the machines get decreases leading to a great loss in investment and decrease in reputation. For that we have to remove the heat from the machines by an effective and most economical method.

Our project "Performance analysis of heat pipe for shell and tube heat exchanger" aims at fabricating a heat exchanger with tubes made up of copper and shell made up of steel. The need for maximization of heat removal requires better technology than the present one. The convection mode of heat transfer will be effective than the conduction mode of heat transfer. Heat exchanger works on the principle of convection. Heat from the hot water passing through the tubes is transferred to cold water running through shell. Hence this could be more preferable method for heat removal between two liquids at different temperatures.Experiment is carried out varying the flow rate of cold water and using air as cold medium. Characteristics curves are drawn for the obtained calculation. From the analysis the various parameters and their behavior with respect to another is noted. The obtained correlation is helpful in analyzing the performance of the shell and tube heat exchanger. The numerical analysis were performed using the finite volume CFD code FLUENT.

ACKNOWLEDGEMENT

We thank the Almighty for giving us the strength and good health from which we were able to complete the project successfully.

We are blessed to have our supervisor and mentor **Mr.K.G.Maheswaran.,** without whose guidance and help we would not have been able to complete the project successfully with such precision.

We are blessed to have **Dr.T.Kannan.,** Dean and Head of the Mechanical Engineering Department who has been instrumental and encouraging by letting us use the lab facilities.

We acknowledge the support given to us by our Vice Principal **Dr.K.Annadurai.,** for providing all the amenities to do this project.

We are thankful to our Lab assistant **Mr.S.Manikavasam** and **Mr.R.Vincent** who shared their technical expertise with us for our project.

We thank all our beloved friends and well wishers for bearing with us all through the demanding times and helping us complete our project.

CONTENTS

Details	Page no.

LIST OF TABLES

LIST OF FIGURES

NOMENCLATURE

Symbol	**Definition**
STHE	Shell and tube heat exchanger
ε	Effectiveness
T_i	Inlet temperature of hot fluid (°C)
T_o	Outlet temperature of hot fluid (°C)
t_i	Inlet temperature of cold fluid (°C)
t_o	Outlet temperature of cold fluid (°C)
Q	Heat transfer (w)
Re	Reynolds No.
Pr	Prandtl No.
Nu	Nusselt No.
St	Stanton No.
NTU	No. of heat transfer units of an exchanger
lpm	Litres per minute

CHAPTER 1

INTRODUCTION

INTRODUCTION

1.1 CURRENT TREND

The huge demand in energy of this revolutionary world calls for various new sources of energy as the existing fossils are depleting. Hence various ways for new sources are been analyzed day to day. But today manufacturing sector has entered into the path of reducing the size, shape, weight etc. But there comes the problem of heat dissipation from the machines. Heat that rejected during the operation is the losses of energy. Due to this the performance of the machines will reduce and brings the production a hault. Thus the optimum life time of the machines get decreases leading to a great loss in investment and decrease in reputation. For that we have to remove the heat from the machines by an effective and most economical method. The energy conservation is a time need for an industry; the energy conservation produces the following benefits

1. Improvement in final product quality.
2. Reduction in operational and maintenance costs.
3. Reduction in overall size of the equipment.
4. Savings by recovered energy.
5. Reduction in energy losses.

1.2 PROJECT IMPORTANCE

To avoid this problem a heat exchanger is designed for cooling efficiently and also economically. We know that the heat exchanger is defined as the device used for transfer between two fluids that are at different temperatures. Thereby the heat exchanger decreases the temperature of the fluid so that the viscosity of the fluids remains unaltered. The transfer between two fluids is carried out either by direct contact, indirect contact or by separation value. Here we designed and analyzed the shell and tube heat exchanger to rectify this problem.

1.3 SCOPE OF THE PROJECT

We arrived at a solution of heat exchanger to cool the waste heat more efficiently than the available conventional type heat exchanger. The tubes are made up of copper. The fluid for convection is used as water. Though there are a number of tubes available for cooling purpose, our aim is to recover the heat from waste heat sources. Hence the convection mode of heat transfer could be better than the conduction mode. Cheaper metals like aluminium, stainless steel and others could be used instead of copper. Tubes are available for cooling applications. We can vary the heat transfer rate by varying the working fluids, so that they can be used for various operating temperatures. Similarly the length and cross section of the heat exchanger can be varied for increasing the heat recovery rate.

1.4 OUTLINE OF THE PROJECT REPORT

This project report is organized into following ten chapters which describes the purpose of the project.

Chapter 1: Introduction
Chapter 2: Line of attack
Chapter 3: Literature survey
Chapter 4: Overview about heat exchangers
Chapter 5: Shell and tube heat exchangers
Chapter 6: Design factors
Chapter 7: Pro-e models
Chapter 8: Fabrication
Chapter 9: Calculation
Chapter10: Computational fluid dynamics
Chapter11: Results and Discussion
Chapter12: Cost analysis
Chapter13: Conclusion
Chapter14: Reference

Chapter 1: Introduction presents the introduction of this project work with its importance, scope of the project work and an outline of the project report.

Chapter 2: Problem chosen states the problem chosen for this project work. It describes the nature of problem giving the details of present situation of the problem and the technique to be adopted to solve the problem.

Chapter 3: Literature survey reveals the relevant work done earlier related to the problem identified and the approach adopted to solve the problem. It gives the description of literature reviewed from various research papers published in international and national journals, proceedings of various conferences, etc. and books.

Chapter 4: Overview about heat exchangers describes about the basics of heat exchangers and its classifications along with its advantages and applications. It also explains about the working principle and the considerations that to be adopted while manufacturing the heat exchangers.

Chapter 5: Shell and tube heat exchangers describes about the basics of shell and heat exchangers and its classifications along with its advantages and applications. It also explains about the working principle and the considerations that to be adopted while manufacturing the heat exchangers.

Chapter 6: Design factors deals about the various factors involved in designing a heat exchanger and its formulas.

Chapter 7: Pro-e models deals about the various forms of designing the heat exchanger and gives 3d view.

Chapter 8: Fabrication describes the procedure adopted for creating a working model. It describes in detail about the fabricating process and equipments used.

Chapter 9: Calculation describes the experimental procedure adopted in this project work.

Chapter 10: Computational fluid dynamics is concerned with obtaining numerical solution to fluid flow problems by using computers. The advent of high-speed and large-memory computers has enabled CFD to obtain solutions to many flow problems including those that are compressible or incompressible, laminar or turbulent, chemically reacting or non-reacting.

Chapter 11: Results and Discussion analyses the result obtained from the experiment. It gives the comparison of the actual and theoretical results.

Chapter 12: Cost analysis provides cost information for production of one such component.

Chapter 13: Conclusion concludes with providing a brief summary about the work done in this project work. It also provides with a derived model from the experiments conducted along with the optimal parameters needed for improving the performance.

Chapter 14: Reference gives picture about books, journals refered for designing, fabrication and analysis.

CHAPTER 2

LINE OF ATTACK

LINE OF ATTACK

According to the law of conservation of energy "energy can neither be created nor be destroyed, but it can be transform to one form to another". Hence it is necessary to conserve the energy. In the present days the energy consumption is not proportional to that of the energy generation; hence energy conservation plays a vital role in the present days.

Apart from the energy conservation, the important factor that needs concentration is that of the pollution. Due to the increase of pollution in the industrial sector there are two main by-factors that are being emitted to the surrounding environment, and one among them is that of the heat. In the present days most of heat energy from the industries is being dissipated to the surroundings.

The method of extracting heat from the process in which useless heat is rejected from one system to another system is natural one for proper functioning. Heat removal is a saving process in which cost of a industrial unit or any other system that uses this technique is reduced. This removed heat helps used the system work properly. The removal of the heat energy can be done by means of a heat exchanger. Here in this project an attempt has been made to study the optimum performance of a shell and tube heat exchanger by means of selecting the appropriate parameters. And finally the optimum parameter is been selected accordingly by means of a suitable optimizing technique.

CHAPTER 3

LITERATURE SURVEY

LITERATURE SURVEY

The following papers are studied as a reference in order to have a basic idea as well as a deep overview of shell and tube heat exchangers. We have gone through the existing technologies and the problems which are in need of new solution and they motivated us to fabricate, test and analyze Shell and tube heat exchanger.

Numerical Analysis of Heat Transfer Characteristics for shell and tube Heat Exchanger-H.K. Choi, G.J. Yoo, S.S. Kim

Numerical analysis is performed to find flow and heat transfer characteristics for shell and tube heat exchanger. Location of inlet and outlet ports is one of the important design parameters of the coiled heat exchanger. 3-D numerical predictions are carried out for the coiled heat exchanger system with forced convection flow. For the analysis, three different connection types of inlet and outlet ports are selected with Reynolds number varying in the range of 2,000 and 200,000. From the prediction, pressure drop, flow rate and heat transfer coefficient are analyzed. Among three different connection types of inlet and outlet ports, middle location case is found to yield better characteristics than others.

Estimating number of shells and determining the log mean temperature difference correction factor of shell and tube heat exchangers -S. K. Bhatti, Ch. M. Krishna, Ch. Vundr

This problem aims at developing a practical and computational tool for LMTD Correction Factor 'F' and approximate number of shells in the process of designing a shell and tube heat exchanger. In order to analyze the performance of heat exchanger the available approaches are LMTD and effectiveness ε .Since the expressions for Log Mean Temperature Difference (LMTD) Correction Factor 'F' are difficult to evaluate, the traditional analysis methods rely on the heat exchanger charts.

Estimation of the performance of shell and tube heat exchanger systems when uncertainties exist-Al-Zakri

The estimation of the performance of systems of shell and tube heat exchangers when more than one heat exchanger is involved is very important in the evaluation of the total process system. To study the effect of uncertainties in the stream variables and heat exchanger parameters a computer program was developed for this purpose. The program uses the Monte Carlo technique of introducing random variables into the system model. The method was applied to a demonstration problem, a crude preheat train, and the temperature are discussed. The use of the method to predict the overdesign factor for each heat exchanger is presented along with a comparison with other methods.

Transient response of a parallel flow shell-and-tube heat exchanger and its control - S. K. Das, T. K. Dan

The shell and tube heat exchangers constitute the single largest component of the heat exchanger family. Though a vast number of literature is available for their steady state analyses, the number of works on transient analysis are comparatively fewer. The present paper brings out the transient response of a single pass shell and tube heat exchanger neglecting wall heat capacitance. The model is constructed with a distributed parameter approach. Unlike a number of recent analyses the numerical technique is avoided and the path of double Laplace transform has been chosen instead.

Performance of shell and tube heat exchangers used in immersion cooling for waste heat recovery-Quadir, G.A.; Leong, C.B.; Krishnan, G.M.; Seetharamu

In this investigation, closed loop immersion cooling of an package with external condenser-a shell and tube heat exchanger being attached to its enclosure-is analysed. The dielectric vapour leaving the enclosure flows through the tubes of the heat exchanger. The vapour is then cooled by ambient air in free or forced convection environments. The finite element method is used in the analysis. The dielectric fluid used is FC-72. The effects of parameters like the tube length, ambient temperature, mass flow rate of the dielectric fluid, etc., on the performance of the heat exchanger have been studied. The results are also presented in the form of ε-NTU (heat transfer effectiveness-number of transfer units) curves.

Design Operation and Performance of Heat Exchangers -Nabil Al- Khirdaji

It describes about mechanisms, types of fluid flow, heat capacity rate and performance monitoring and cleaning strategies. This paper describes the performance analysis of a cross-flow type heat exchanger for use as a liquid desiccant absorber (dehumidifier) and indirect evaporative cooler. The proposed absorber can be described as a direct contact, cross-flow, heat and mass exchanger, with the flow passages separated from each other by thin plastic,plates. One air stream (primary air) is sprayed by liquid desiccant solution, while the other stream (secondary air) is evaporatively cooled by a water sprayA parametric study for the primary air stream at 33°C, 0.0171 kg/kg humidity ratio and secondary air stream at 27°C and 0.010 kg/kg humidity ratio using calcium chloride solution was performed in this study.

CHAPTER 4

OVERVIEW ABOUT HEAT EXCHANGERS

OVERVIEW ABOUT HEAT EXCHANGERS

4.1 Introduction

A heat exchanger is a device built for efficient heat transfer from one medium to another, whether the media are separated by a solid wall so that they never mix, or the media are in direct contact. They are widely used in space heating, refrigeration, air conditioning, power plants, chemical plants, petrochemical plants, petroleum refineries, and natural gas processing. One common example of a heat exchanger is the radiator in a car, in which a hot engine-cooling fluid, like *antifreeze*, transfers heat to air flowing through the radiator.

4.2 Flow arrangement

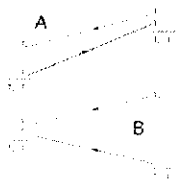


Fig 4.1 Countercurrent (A) and parallel (B) flows

Heat exchangers may be classified according to their flow arrangement. In *parallel-flow* heat exchangers, the two fluids enter the exchanger at the same end, and travel in parallel to one another to the other side. In *counter-flow* heat exchangers the fluids enter the exchanger from opposite ends. The counter current design is most efficient, in that it can transfer the most heat. In a *cross-flow* heat exchanger, the fluids travel roughly perpendicular to one another through the exchanger.

The driving temperature across the heat transfer surface varies with position, but an appropriate mean temperature can be defined. In most simple systems this is the log mean temperature difference (LMTD).

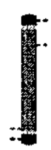


Fig. 4.2: Shell and tube heat exchanger, single pass (1-1 parallel flow)

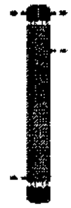


Fig. 4.3: Shell and tube heat exchanger, 2-pass tube side (1-2 crossflow)

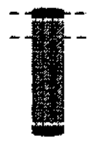


Fig. 4.4: Shell and tube heat exchanger, 2-pass shell side, 2-pass tube side (2-2 countercurrent)

4.3 Types of heat exchangers

4.3.1 Shell and tube heat exchanger

Shell and tube heat exchangers consist of a series of tubes. One set of these tubes contains the fluid that must be either heated or cooled. The second fluid runs over the tubes that are being heated or cooled so that it can either provide the heat or absorb the heat required. A set of tubes is called the tube bundle and can be made up of several types of tubes: plain, longitudinally finned, etc. Shell and Tube heat exchangers are typically used for high pressure applications (with pressures greater than 30 bar and temperatures greater than 260°C. This is because the shell and tube heat exchangers are robust due to their shape.

4.3.2 Plate heat exchanger

Another type of heat exchanger is the plate heat exchanger. One is composed of multiple, thin, slightly-separated plates that have very large surface areas and fluid flow passages for heat transfer. This stacked-plate arrangement can be more effective, in a given space, than the shell and tube heat exchanger. Advances in gasket and brazing technology have made the plate-type heat exchanger increasingly practical. In HVAC applications, large heat exchangers of this type are called *plate-and-frame*; when used in open loops, these heat exchangers are normally of the gasketed type to allow periodic disassembly, cleaning, and inspection.

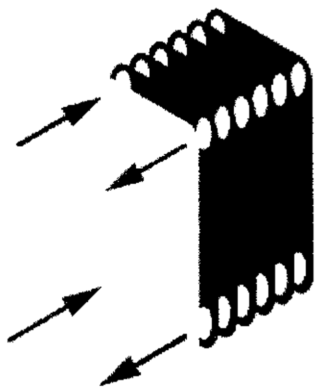


Fig. 4.5: Conceptual diagram of a plate and frame heat exchanger

Fig. 4.6: A single plate heat exchanger

4.3.3 Regenerative heat exchanger

A third type of heat exchanger is the regenerative heat exchanger. In this, the heat from a process is used to warm the fluids to be used in the process, and the same type of fluid is used either side of the heat exchanger (these heat exchangers can be either plate-and-frame or shell-and-tube construction). These exchangers are used only for gases and not for liquids. The major factor for this is the heat capacity of the heat transfer matrix. Also see: Countercurrent exchanger, Regenerator, Economizer

4.3.4 Adiabatic wheel heat exchanger

A fourth type of heat exchanger uses an intermediate fluid or solid store to hold heat, which is then moved to the other side of the heat exchanger to be released. Two examples of this are adiabatic wheels, which consist of a large wheel with fine threads rotating through the hot and cold fluids, and fluid heat exchangers. This type is used when it is acceptable for a small amount of mixing to occur between the two streams.

4.3.5 Plate Fin heat exchanger

This type heat exchanger uses "sandwiched" passages containing fins to increase the effectivity of the unit. The designs include crossflow and counterflow coupled with various fin configurations such as straight fins, offset fins and wavy fins.

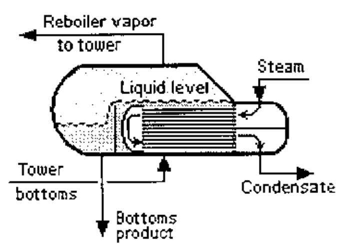
4.3.6 Fluid heat exchangers

This is a heat exchanger with a gas passing upwards through a shower of fluid (often water), and the fluid is then taken elsewhere before being cooled. This is commonly used for cooling gases whilst also removing certain impurities, thus solving two problems at once. It is widely used in espresso machines as an energy-saving method of cooling super-heated water to be used in the extraction of espresso.

4.3.7 Dynamic scraped surface heat exchanger

Another type of heat exchanger is called "dynamic heat exchanger" or "scraped-surface heat exchanger". This is mainly used for heating or cooling with high-viscosity products, crystallization processes, evaporation and high-fouling applications.

4.3.8 Phase-change heat exchangers



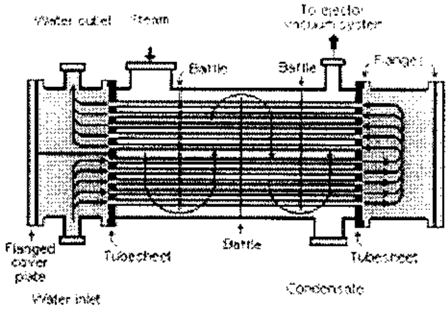


Fig. 4.8: Typical water-cooled surface condenser

In addition to heating up or cooling down fluids in just a single phase, heat exchangers can be used either to heat a liquid to evaporate (or boil) it or used as condensers to cool a vapor and condense it to a liquid. In chemical plants and refineries, reboilers used to heat incoming feed for distillation towers are often heat exchangers.

4.4 Multi-phase heat exchangers

4.5 HVAC air coils

One of the widest uses of heat exchangers is for air conditioning of buildings and vehicles. This class of heat exchangers is commonly called *air coils*, or just *coils* due to their often-serpentine internal tubing. Liquid-to-air, or air-to-liquid HVAC coils are typically of modified cross flow arrangement. In vehicles, heat coils are often called heater cores.

On the liquid side of these heat exchangers, the common fluids are water, a water-glycol solution, steam, or a refrigerant. For *heating coils*, hot water and steam are the most common, and this heated fluid is supplied by boilers, for example. For *cooling coils*, chilled water and refrigerant are most common. Chilled water is supplied from a chiller that is potentially located very far away, but refrigerant must come from a nearby condensing unit. When a refrigerant is used, the cooling coil is the evaporator in the vapor-compression refrigeration cycle. HVAC coils that use this direct-expansion of refrigerants are commonly called *DX coils*.

4.6 Spiral heat exchangers

A spiral heat exchanger (SHE), may refer to a helical (coiled) tube configuration, more generally, the term refers to a pair of flat surfaces that are coiled to form the two channels in a counter-flow arrangement. Each of the two channels has one long curved path. A pair of fluid ports are connected tangentially to the outer arms of the spiral, and axial ports are common, but optional.

4.6.1 Self cleaning

SHEs are often used in the heating of fluids which contain solids and thus have a tendency to foul the inside of the heat exchanger. The low pressure drop gives the SHE its ability to handle fouling more easily. The SHE uses a "self cleaning" mechanism, whereby fouled surfaces cause a localized increase in fluid velocity, thus increasing the drag (or fluid friction) on the fouled surface, thus helping to dislodge the blockage and keep the heat exchanger clean. "The internal walls that make up the heat transfer surface are often rather thick, which makes the SHE very robust, and able to last a long time in demanding environments." They are also easily cleaned, opening out like an oven where any build up of foulant can be removed by pressure washing.

4.6.2 Applications

The SHE is ideal for applications such as pasteurization, digester heating, heat recovery, pre-heating (see: recuperator), and effluent cooling. For sludge treatment, SHEs are generally smaller than other types of heat exchangers.

4.7 Selection

Due to the many variables involved, selecting optimal heat exchangers is challenging. Hand calculations are possible, but many iterations are typically needed. As such, heat exchangers are most often selected via computer programs, either by system designers, who are typically engineers, or by equipment vendors.

In order to select an appropriate heat exchanger, the system designers (or equipment vendors) would firstly consider the design limitations for each heat exchanger type.

Although cost is often the first criterion evaluated, there other several other important selection criteria which include:

- High/ Low pressure limits
- Thermal Performance
- Temperature ranges
- Product Mix (liquid/liquid, particulates or high-solids liquid)
- Pressure Drops across the exchanger
- Fluid flow capacity
- Cleanability, maintenance and repair
- Materials required for construction
- Ability and ease of future expansion

4.8 Monitoring and maintenance

Integrity inspection of plate and tubular heat exchanger can be tested in situ by the conductivity or helium gas methods. These methods confirm the integrity of the plates or tubes to prevent any cross contamination and the condition of the gaskets.

Condition monitoring of heat exchanger tubes may be conducted through Nondestructive methods such as eddy current testing.

The mechanics of water flow and deposits are often simulated by computational fluid dynamics or CFD. Fouling is a serious problem in some heat exchangers. River water is often used as cooling water, which results in biological debris entering the heat exchanger and building layers, decreasing the heat transfer coefficient. Another common problem is scale, which is made up of deposited layers of chemicals such as calcium carbonate or magnesium carbonate.

4.8.1 Fouling

Fouling occurs when a fluid goes through the heat exchanger, and the impurities in the fluid precipitate onto the surface of the tubes. Precipitation of these impurities can be caused by:

- Frequent use of the Heat Exchanger

- Not cleaning the Heat Exchanger regularly
- Reducing the velocity of the fluids moving through the heat exchanger
- Over-sizing of the heat exchanger

Effects of fouling are more abundant in the cold tubes of the heat exchanger, than in the hot tubes. This is because impurities are less likely to be dissolved in a cold fluid. This is because solubility increases as temperature increases.

4.8.2 Maintenance

Plate heat exchangers need to be dissembled and cleaned periodically. Tubular heat exchangers can be cleaned by such methods as acid cleaning, sandblasting, high-pressure water jet, bullet cleaning, or drill rods.

In large-scale cooling water systems for heat exchangers, water treatment such as purification, addition of chemicals, and testing, is used to minimize fouling of the heat exchange equipment. Other water treatment is also used in steam systems for power plants, etc. to minimize fouling and corrosion of the heat exchange and other equipment.

A variety of companies have started using water borne oscillations technology to prevent biofouling. Without the use of chemicals, this type of technology has helped in providing a low-pressure drop in heat exchangers.

4.9 Heat exchangers in nature

4.9.1 Counter current heat exchangers

Heat exchangers occur naturally in the circulation system of fish and whales. Arteries to the skin carrying warm blood are intertwined with veins from the skin carrying cold blood, causing the warm arterial blood to exchange heat with the cold venous blood. This reduces the overall heat loss in cold waters. Heat exchangers are also present in the tongue of baleen whales as large volumes of water flow through their mouths. Wading birds use a similar system to limit heat losses from their body through their legs into the water.

CHAPTER 5

SHELL AND TUBE HEAT EXCHANGERS

SHELL AND TUBE HEAT EXCHANGER

5.1 Introduction

A shell and tube heat exchanger is a class of heat exchanger designs. It is the most common type of heat exchanger in oil refineries and other large chemical processes, and is suited for higher-pressure applications. As its name implies, this type of heat exchanger consists of a shell (a large pressure vessel) with a bundle of tubes inside it. One fluid runs through the tubes, and another fluid flows over the tubes (through the shell) to transfer heat between the two fluids. The set of tubes is called a tube bundle, and may be composed by several types of tubes: plain, longitudinally finned, etc.

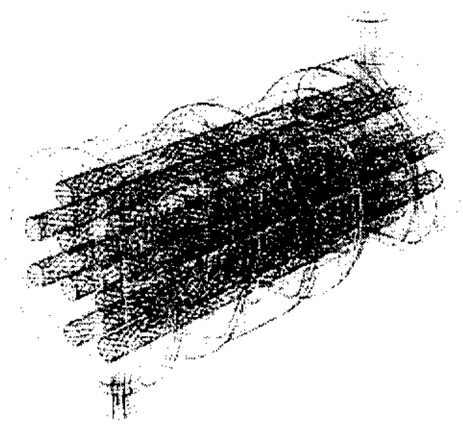


Fig. 5.1: Fluid flow simulation for a shell and tube exchanger

The shell inlet is at the top rear and outlet at in the foreground at the bottom

5.2 Principle

Two fluids, of different starting temperatures, flow through the heat exchanger. One flows through the tubes (the tube side) and the other flows outside the tubes but inside the shell (the shell side). Heat is transferred from one fluid to the other through the tube walls, either from tube side to shell side or vice versa. The fluids can be either liquids or gases on either the shell or the tube side. In order to transfer heat efficiently, a large heat transfer area should be used, leading to the use of many tubes. In this way, waste heat can be put to use. This is an efficient way to conserve energy.

5.3 Types of Shell and tube heat exchanger

There can be many variations on the shell and tube design. Typically, the ends of each tube are connected to plenums (sometimes called water boxes) through holes in tube sheets. The tubes may be straight or bent in the shape of a U, called U-tubes.

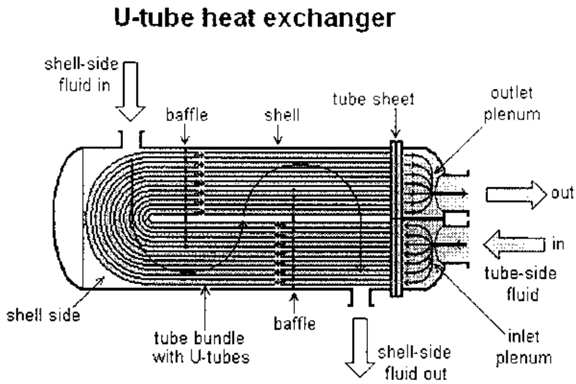


Fig. 5.2: U- tube heat exchanger

In nuclear power plants called pressurized water reactors, large heat exchangers called steam generators are two-phase, shell-and-tube heat exchangers which typically have U-tubes. They are used to boil water recycled from a surface condenser into steam to drive the turbine to produce power. Most shell-and-tube heat exchangers are either 1, 2, or 4 pass designs on the tube side. This refers to the number of times the fluid in the tubes passes through the fluid in the shell. In a single pass heat exchanger, the fluid goes in one end of each tube and out the other.

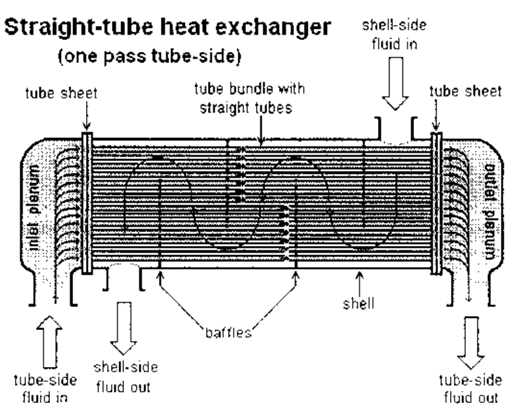


Fig. 5.3: straight- tube heat exchanger

5.4 Parts of Shell and tube heat exchanger

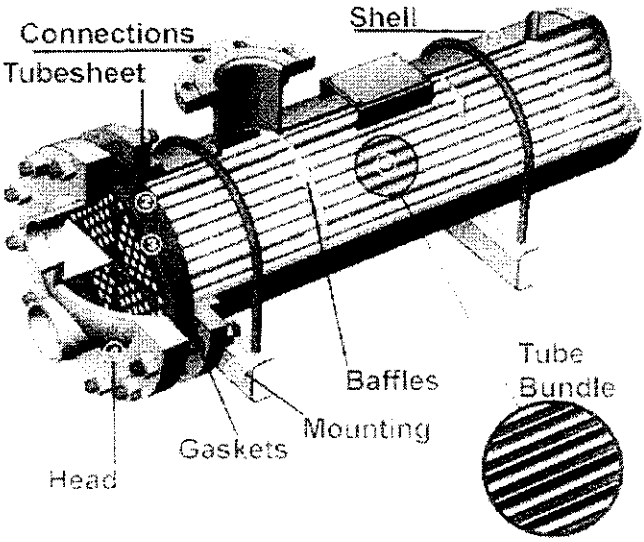


Fig. 5.4: parts of shell and tube heat exchanger in detail

Shell and tube heat exchangers have been around for over 150 years. Their thermal technology and manufacturing methods are well defined and applied by the modern manufacturer. Tube surfaces range from standard to exotic metals with plain or enhanced surface characteristics. They can help provide the least costly mechanical design for the flows, liquids and temperatures involved. The parts of shell and tube heat exchanger are listed below

- Shell
- Tube
- Baffle
- Connections
- Tube sheet
- Gasket
- Mounting
- Head

Detailed description of various parts are given below

5.4.1 Shell

Shell is the outermost covering of heat exchanger. It is made up of mild steel which accumulates the cold fluid and acts as boundary for fluid flow. The heat exchanger consists of a welded shell protected with high quality paint for corrosion resistance. Parameters for consideration are thickness, type of material, weight, corrosive resistant.

5.4.2 Tube

copper steel tubes which allow for strong, durable performance over a wide range of applications. Unique tube bundle layout (chevron corrugated pattern) minimizes deposit buildup at the edges and optimises media flow for high velocity flow turbulence.

5.4.2.1 Enhanced surfaces

Since there are so many different types of heat exchanger enhancements, it is highly unlikely that a commercial simulator could support them all. Furthermore, some propriety data from the manufacturers of the heat transfer enhancement might never be released. However, that does not mean that process and project engineers cannot perform some of the preliminary calculations for new technologies. The following provides background information on many different types of heat exchanger enhancements. Heat exchanger enhancement must always satisfy the primary goal of providing a cost advantage relative to the use of a conventional heat exchanger. Other factors that should be addressed include fouling potential, reliability and safety.

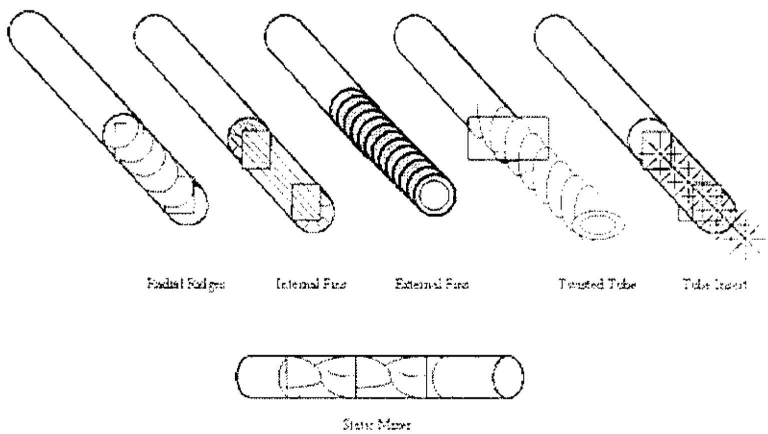


Figure 1. Examples of tubes with heat transfer enhancement.

Fig. 5.5: different types of tube

5.4.2.2 Finning

Tubes can be finned on both the interior and exterior. This is probably the oldest form of heat transfer enhancement. Finning is usually desirable when the fluid has a relatively low heat transfer film coefficient as does a gas. The fin not only increases the film coefficient with added turbulence but also increases the heat transfer surface area. This added performance results in higher pressure drop. However, as with any additional surface area, the fin area must be adjusted by efficiency. This fin efficiency leads to an optimum fin height with respect to heat transfer. Most of the heat transfer and film coefficients for finned tubes are available in the open literature and supported in most commercial heat exchanger rating packages. Recent papers also describe predicting finned tube performance. Data for the performance of low finned tubes as compared to generalized correlations are also available in the literature.

5.4.2.3 Tube Inserts

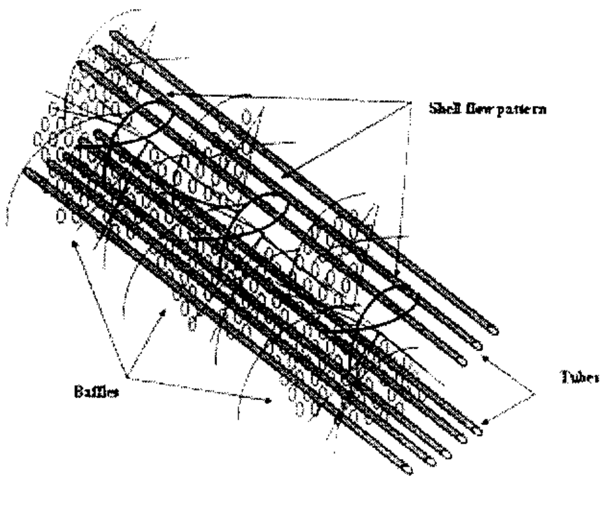
Inserts, tabulators, or static mixers are inserted into the tube to promote turbulence. These devices are most effective with high viscosity fluids in a laminar flow regime. Increases in the heat transfer film coefficients can be as high as five times. Inserts are used most often with liquid heat transfer and to promote boiling. Inserts are not usually effective for condensing in the tube and almost always increase pressure drop. Because of thecomplex relationships between the geometry of the insert and the resulting increase in heat transfer and pressure drop, there are no general correlations to predict enhancements. However, through the modification of the number of passes, a resulting heat transfer coefficient gain can be achieved at lower pressure drop in some situations

5.4.2.4 Tube Deformation

Many vendors have developed proprietary surface configures by deforming the tubes. The resulting deformation appears corrugated, twisted, or spirally fluted. Marto *et al* compares the performance of 11 different commercially available tubes for single tube performance. The surface condenses steam on the outside and heats water on the inside. The author reports a 400 % increase in the inside heat transfer film coefficient; however, pressure drops were 20 times higher relative to the unaltered tube at the same maximum inside diameter. Recently, Shilling describes some of the benefits of a new twisted tube technology including the fact that tube vibration can be minimized. Furthermore the author describes how baffles may be eliminated completely. Similar to the tube inserts, these twisted tubes promote turbulence and enhance boiling. Unfortunately, no quantitative results are provided to show the increase in film coefficients for both the shell and tube fluids.

5.4.3 Baffles

Baffles are designed to direct the shell side fluid across the tube bundle as efficiently as possible. Forcing the fluid across the tube bundle ultimately results in a pressure loss. The most common type of baffle is the single segmental baffle which changes the direction of the shell side fluid to achieve cross flow. Deficiencies of the segmented baffle include the potential for dead spots in the exchanger and excessive tube vibration. Baffle enhancements have attempted to alleviate the problems associated with leakage and dead areas in the conventional segmental baffles. The most notable improvement has resulted in a helical baffle as shown in Figure 2. Van der Ploeg and Master17 describes this baffle is most effective for high viscosity fluids and provide several refinery applications.



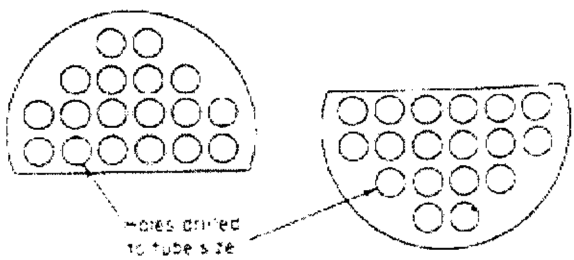


Fig. 5.6: Segmental baffles

5.4.3.1 Combined Enhancement

Several reports have discussed the use of combined enhancement including both the effects of passive and active methods. The combination of passive methods are somewhat limited, with the most common being both internal and external finned tubes. Other combinations may be nearly impossible because of the manufacturing techniques used in modifying the tube geometry. One recent article is of particular interest describing the use of both helical baffles and tube inserts. This exchanger was used in a crude preheat train and provides some quantitative comparisons for both the tube and shell side film coefficients along with some qualitative pressure drop information.

5.4.4 Connections

connections helps for the flow of fluid easily from inside to outside and vice versa. It come in standardized sizes for easy assembly and feature additional thread. It provides surface protection for clean installation.

5.4.5 Tube sheet

U-bend tubes expanded into a tube sheet which allow for tube expansions and contractions due to thermal fluctuations. It forms a great support for withholding the tubes inside shell at correct arrangement.

5.4.6 Gasket

gaskets that are made of high quality compressed fibres which lends to reusability. It provides additional support for heat exchanger.

5.4.7 Mounting

saddle attaches which make for quick and easy mounting

5.4.8 Head

a standard cast-iron or steel head for heavy duty services (also available as a spare part).

CHAPTER 6

DESIGN FACTORS

DESIGIN FACTORS

6.1 Mass flow rate

Mass flow rate is the mass of substance which passes through a given surface per unit time. Its unit is mass divided by time, so kilogram per second in SI units, and slug per second or pound per second in US customary units. It is usually represented by the symbol$\dot{m}$.

Mass flow rate can be calculated from the density of the substance, the cross sectional area through which the substance is flowing, and its velocity relative to the area of interest.

$$\dot{m} = \rho \cdot v \cdot A$$

where:

$\dot{m}$ = mass flow rate

ρ = density

v = velocity

A = flow area

This is equivalent to multiplying the volume flow rate by the density.

$$\dot{m} = \rho \cdot Q$$

where:

ρ = density

Q = volume flow rate

The symbol for mass flow rate is Newton's notation for a derivative:

$$\dot{m} = \frac{dm}{dt}$$

6.2 Log mean temperature difference

The log mean temperature difference (LMTD) is used to determine the temperature driving force for heat transfer in flow systems (most notably in heat exchangers). The LMTD is a logarithmic average of the temperature difference between the hot and cold streams at each end of the exchanger. The use of the LMTD arises straightforwardly from the analysis of a heat exchanger with constant flow rate and fluid thermal properties.

For Countercurrent flow (i.e. where the hot stream, liquid or gas, goes from say left to right, and the cold stream, again liquid or gas goes from right to left), is given by the following equation:

$$LMTD = \frac{(T_1 - t_2) - (T_2 - t_1)}{\ln\left(\frac{T_1 - t_2}{T_2 - t_1}\right)}$$

And for Parallel flow (i.e. where the hot stream,liquid or gas, goes from say left to right, and so does the cold stream), is given by the following equation:

$$LMTD = \frac{(T_1 - t_1) - (T_2 - t_2)}{\ln\left(\frac{T_1 - t_1}{T_2 - t_2}\right)}$$

T_1 = Hot Stream Inlet Temp.

T_2 = Hot Stream Outlet Temp.

t_1 = Cold Stream Inlet Temp.

t_2 = Cold Stream Outlet Temp.

It makes no difference which temperature differential is 1 or 2 as long as the nomenclature is consistent. The larger the LMTD, the more heat is transferred.Yet a third type of unit is a cross-flow exchanger, in which one system (usually the heat sink) has the same nominal temperature at all points on the heat transfer surface. This follows similar mathematics, in its dependence on the LMTD, except that a correction factor F often needs to be included in the heat transfer relationship.

There are times when the four temperatures used to calculate the LMTD are not available, and the NTU method may then be preferable.

6.3 Effectiveness

Effectiveness means the capability of producing an effect..

It is defined as the ratio of actual heat transfer to maximum possible heat transfer in the heat exchanger. It is denoted by ε.

$$\varepsilon = \frac{\text{Actual heat transfer}}{\text{Maximum possible heat transfer}}$$

6.3 NTU method

The Number of Transfer Units (NTU) Method is used to calculate the rate of heat transfer in heat exchangers (especially counter current exchangers) when there is insufficient information to calculate the Log-Mean Temperature Difference (LMTD). In heat exchanger analysis the fluid inlet and outlet temperatures are specified or can be determined by simple mass balance the LMTD method can be used, but when these information are not available The NTU or The Effectiveness method is used.

A quantity

$$q_{max} = C_{min}(T_{h,i} - T_{c,i})$$

is then found,where $q_{max}$is the maximum heat fluids.

. NTU is dimensionless parameter and a measure of heat transfer size of exchanger.

$$\text{NTU} = \frac{\text{UA}}{\text{C}_{\text{min}}}$$

Where,

U – Overall heat transfer coefficient in W/m^2K

A – Heat transfer area in m^2

C_{min} – minimum value among the product of mass flow rate and specific heat capacity of fluid.

6.5 Heat transfer coefficient

The heat transfer coefficient, in thermodynamics and in mechanical and chemical engineering, is used in calculating the heat transfer, typically by convection or phase change between a fluid and a solid:

$$h = \frac{\Delta Q}{A \cdot \Delta T \cdot \Delta t}$$

where

ΔQ = heat input or heat lost, J
h = heat transfer coefficient, W/(m^{2}K)
A = heat transfer surface area, m^2
ΔT = difference in temperature between the solid surface and surrounding fluid area, K
Δt = time period, s

From the above equation, the heat transfer coefficient is the proportionality coefficient between the heat flux, $Q/(A\Delta t)$, and the thermodynamic driving force for the flow of heat (i.e., the temperature difference, ΔT).

The heat transfer coefficient has SI units in watts per meter squared-kelvin (W/(m^{2}K)).

Heat transfer coefficient is the inverse of thermal insulance.

There are numerous methods for calculating the heat transfer coefficient in different heat transfer modes, different fluids, flow regimes, and under different thermohydraulic conditions. Often it can be estimated by dividing the thermal conductivity of the convection fluid by a length scale. The heat transfer coefficient is often calculated from the Nusselt number (a dimensionless number).

6.5.1 Dittus–Boelter correlation (forced convection)

A common and particularly simple correlation useful for many applications is the Dittus–Boelter heat transfer correlation for fluids in turbulent flow. This correlation is applicable when forced convection is the only mode of heat transfer; i.e., there is no boiling, condensation, significant radiation, etc. The accuracy of this correlation is anticipated to be +/-15%.

For a liquid flowing in a straight circular pipe with a Reynolds number between 10 000 and 120 000 (in the turbulent pipe flow range), when the liquid's Prandtl number is between 0.7 and 120, for a location far from the pipe entrance (more than 10 pipe diameters; more than 50 diameters according to many authors) or other flow disturbances, and when the pipe surface is hydraulically smooth, the heat transfer coefficient between the bulk of the fluid and the pipe surface can be expressed as:

$$h = \frac{k_w}{D_H} Nu$$

6.5.2 Thom correlation

There exist simple fluid-specific correlations for heat transfer coefficient in boiling. The Thom correlation is for flow boiling of water (subcooled or saturated at pressures up to about 20 MPa) under conditions where the nucleate boiling contribution predominates over forced convection. This correlation is useful for rough estimation of expected temperature difference given the heat flux:

$$\Delta T_{sat} = 22.5 \cdot q^{0.5} \exp(-P/8.7)$$

where:

ΔT_{sat} is the wall temperature elevation above the saturation temperature, K
q is the heat flux, MW/m^2
P is the pressure of water, MPa

Note that this empirical correlation is specific to the units given.

6.5.3 Heat transfer coefficient of pipe wall

The resistance to the flow of heat by the material of pipe wall can be expressed as a "heat transfer coefficient of the pipe wall". However, one needs to select if the heat flux is based on the pipe inner or the outer diameter.

Selecting to base the heat flux on the pipe inner diameter, and assuming that the pipe wall thickness is small in comparison with the pipe inner diameter, then the heat transfer coefficient for the pipe wall can be calculated as if the wall were not curved:

$$h_{wall} = \frac{k}{x}$$

where k is the effective thermal conductivity of the wall material and x is the wall thickness.

If the above assumption does not hold, then the wall heat transfer coefficient can be calculated using the following expression:

$$h_{wall} = \frac{2k}{d_i \ln(d_o/d_i)}$$

where d_i and d_o are the inner and outer diameters of the pipe, respectively.

The thermal conductivity of the tube material usually depends on temperature; the mean thermal conductivity is often used.

6.5.4 Combining heat transfer coefficients

For two or more heat transfer processes acting in parallel, heat transfer coefficients simply add:

$$h = h_1 + h_2 + \ldots$$

For two or more heat transfer processes connected in series, heat transfer coefficients add inversely:

$$\frac{1}{h} = \frac{1}{h_1} + \frac{1}{h_2} + \dots$$

For example, consider a pipe with a fluid flowing inside. The rate of heat transfer between the bulk of the fluid inside the pipe and the pipe external surface is:

$$Q = \left(\frac{1}{\frac{1}{h} + \frac{t}{k}} \right) \cdot A \cdot \Delta T$$

where

Q = heat transfer rate (W)

h = heat transfer coefficient (W/(m^2·K))

t = wall thickness (m)

k = wall thermal conductivity (W/m·K)

A = area (m^2)

6.6 Fouling

Fouling refers to the accumulation of unwanted material on solid surfaces, most often in an aquatic environment. The fouling material can consist of either living organisms (biofouling) or a non-living substance (inorganic or organic). Fouling is usually distinguished from other surface-growth phenomena in that it occurs on a surface of a component, system or plant performing a defined and useful function, and that the fouling process impedes or interferes with this function.

6.6.1 FOULING FACTORS

However, it is a well-known fact that the surface of a heat exchanger does not remain clear after it has been in use for some time. The surfaces become fouled with scaling or deposits which are formed due to impurities in the fluid, chemical reaction between the fluid and the wall material, rust formation etc. The effect of these deposits is felt in terms of greatly increased surface resistance affecting the value of U. This effect is taken care of by introducing an additional thermal resistance called the fouling resistance R_f.

Table 6.1. FOULING FACTORS

Fluid	R_f (m^2k/w)
Sea water and treated boiler feed water below 50°c	0.0001
Sea water and treated boiler feed water above 50°c	0.0002
River water below 50°c	0.0002 – 0.0001
Fuel oil	0.0009
Quenching oil	0.0007
Alcohol Vapours	0.00009
Steam, non-oil bearing	0.00009
Industrial air	0.0004
Refrigerating liquid	0.0002

6.7 Overall heat transfer coefficient

The overall heat transfer coefficient U is a measure of the overall ability of a series of conductive and convective barriers to transfer heat. It is commonly applied to the calculation of heat transfer in heat exchangers, but can be applied equally well to other problems.

For the case of a heat exchanger, U can be used to determine the total heat transfer between the two streams in the heat exchanger by the following relationship:

$$Q = UA\Delta T_{LM}$$

where

Q = heat transfer rate (W)
U = overall heat transfer coefficient (W/(m²·K))
A = heat transfer surface area (m^2)
ΔT_{LM} = log mean temperature difference (K)

The overall heat transfer coefficient takes into account the individual heat transfer coefficients of each stream and the resistance of the pipe material. It can be calculated as the reciprocal of the sum of a series of thermal resistances (but more complex relationships exist, for example when heat transfer takes place by different routes in parallel):

$$\frac{1}{UA} = \Sigma\frac{1}{hA} + \Sigma R$$

where

R = Resistance(s) to heat flow in pipe wall (K/W)

Other parameters are as above.

The heat transfer coefficient is the heat transferred per unit area per Kelvin. Thus area is included in the equation as it represents the area over which the transfer of heat takes place. The areas for each flow will be different as they represent the contact area for each fluid side.

Table 6.2. OVERALL HEAT TRANSFER COEFFICIENT

Physical situation	U (W/m^2k)
Brick exterior wall, plaster interior uninsulated	2.50
Frame exterior wall, plaster interior uninsulated	1.40
Rock wool insulation	0.40
Plate – Glass window	6.20
Double plate – Glass window	2.30
Alcohol condenser (Water in tubes)	250 – 700
Steam condenser (Water in tubes)	1100 – 5600
Ammonia condenser (Water in tubes)	800 – 1400
Water to water heat exchanger	850 – 1700

Water to oil heat exchanger	110 – 350
Finned tube heat exchanger	25 – 50

6.8 Nusselt number

In heat transfer at a boundary (surface) within a fluid, the Nusselt number is the ratio of convective to conductive heat transfer across (normal to) the boundary. Named after Wilhelm Nusselt, it is a dimensionless number. The conductive component is measured under the same conditions as the heat convection but with a (hypothetically) stagnant (or motionless) fluid.

A Nusselt number close to unity, namely convection and conduction of similar magnitude, is characteristic of "slug flow" or laminar flow. A larger Nusselt number corresponds to more active convection, with turbulent flow typically in the 100-1000 range.

The convection and conduction heat flows are parallel to each other and to the surface normal of the boundary surface, and are all perpendicular to the mean fluid flow in the simple case.

$$Nu_L = \frac{hL}{k_f} = \frac{\text{Convective heat transfer}}{\text{Conductive heat transfer}}$$

where:

- L = characteristic length
- k_f = thermal conductivity of the fluid
- h = convective heat transfer coefficient

Typically the average Nusselt number is expressed as a function of the Rayleigh number and the Prandtl number, written as: Nu = f(Ra, Pr). Empirical correlations for a wide variety of geometries are available that express the Nusselt number in the aforementioned form.

The mass transfer analog of the Nusselt number is the Sherwood number.

6.9 Reynolds number

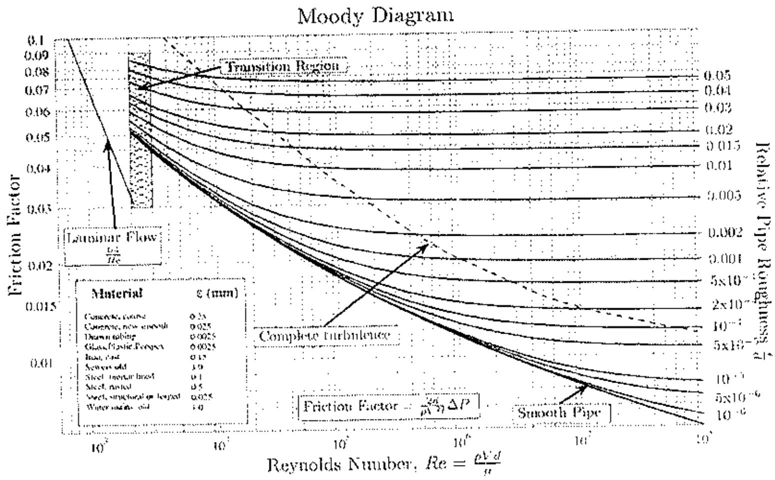


Fig. 6.1: moody diagram

Reynolds number can be defined for a number of different situations where a fluid is in relative motion to a surface. These definitions generally include the fluid properties of density and viscosity, plus a velocity and a characteristic length or characteristic dimension. This dimension is a matter of convention - for example a radius or diameter are equally valid for spheres or circles, but one is chosen by convention. For flow in a pipe or a sphere moving in a fluid the diameter is generally used today. For flow in a pipe or tube, the Reynolds number is generally defined as:

$$\mathrm{Re} = \frac{\rho \mathrm{V} D}{\mu} = \frac{\mathrm{V} D}{\nu} = \frac{\mathrm{Q} D}{\nu A}$$

where:

- V is the mean fluid velocity in (SI units: m/s)
- D is the diameter (m)

CHAPTER 7

PRO-E MODELS

PRO E MODELS

7.1 SHELL AND TUBE HEAT EXCHANGER

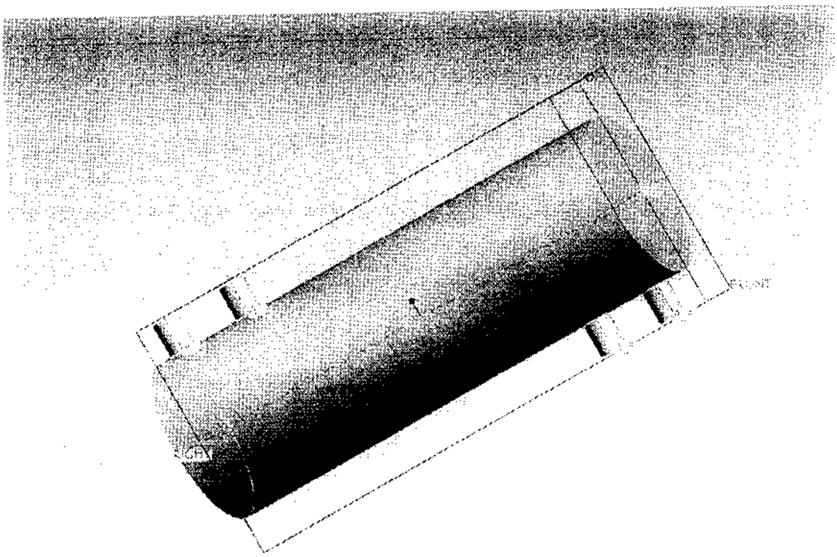


FIG: 7.1. MODEL 1

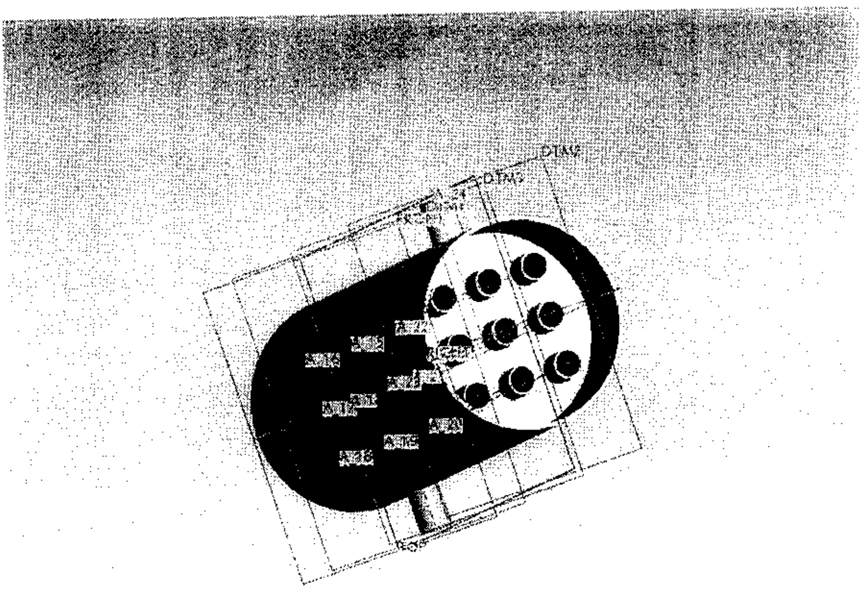


FIG: 7.2. MODEL 2

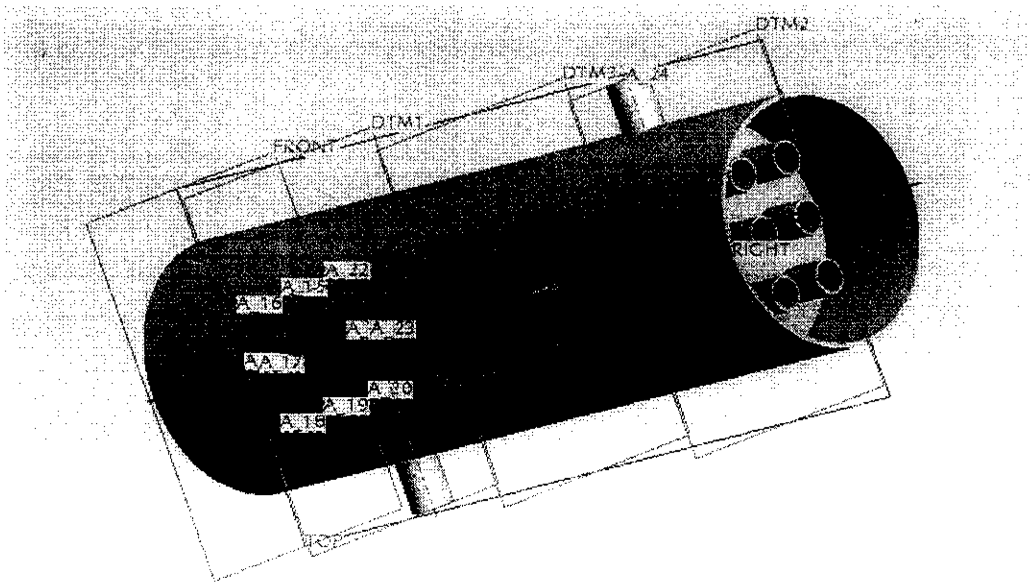


FIG: 7.3. MODEL 3

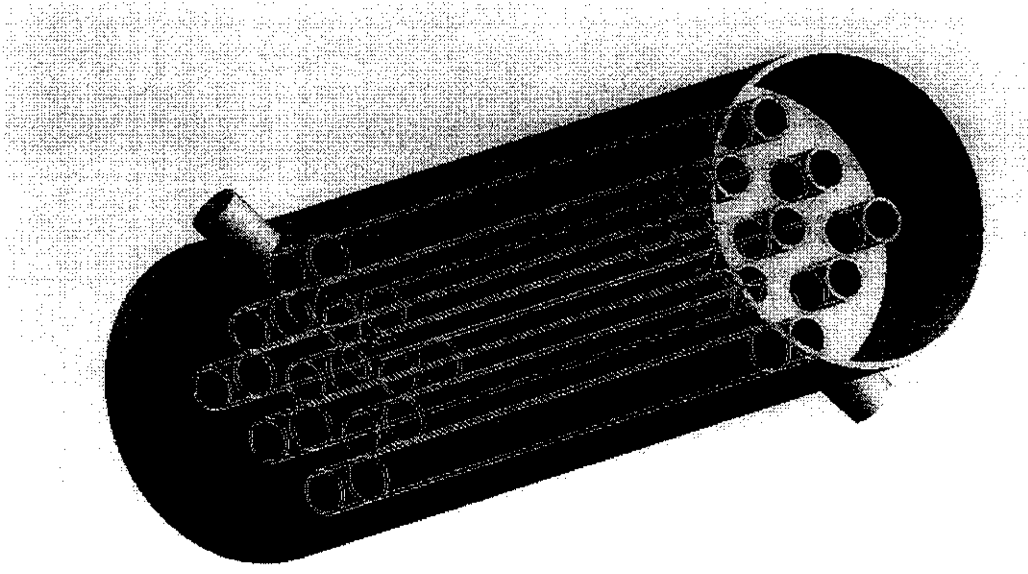


FIG: 7.4. MODEL 4

CHAPTER 8

FABRICATION

Metal fabrication

Metal fabrication is a value added process that involves the construction of machines and structures from various raw materials. A fab shop will bid on a job, usually based on the engineering drawings, and if awarded the contract will build the product. The fabricator may employ or contract out steel detailers to prepare shop drawings, if not provided by the customer, which the fabricating shop will use for manufacturing. Manufacturing engineers will program CNC machines as needed.

The fabrication of shell and tube heat exchanger can be done by the following procedure:

- Material for various parts are selected

8.1 Materials selection

8.1.1 Tube- copper

It is important to note that in contrast to the oxidation of iron by wet air that the layer formed by the reaction of air with copper has a protective effect against further corrosion

8.1.2 Shell – mild steel

Low carbon steel contains approximately 0.05–0.15% carbon and mild steel contains 0.16–0.29% carbon, therefore it is neither brittle nor ductile. Mild steel has a relatively low tensile strength, but it is cheap and malleable; surface hardness can be increased through carburizing.

8.1.3 Pipe fittings – PVC tubes

- Mild steel is rolled into the form of hollow cylinder

8.2 Rolling (metalworking) of mild steel

Rolling is a fabricating process in which the metal, plastic, paper, glass, etc. is passed through a pair (or pairs) of rolls. There are two types of rolling process, flat and profile rolling. In flat rolling the final shape of the product is either classed as sheet (typically thickness less than 3 mm, also called "strip") or plate (typically thickness more than 3 mm). In profile rolling the final product may be a round rod or other shaped bar, such as a structural section (beam, channel, joist etc).

- After rolling, mild steel is rolled into the form of hollow cylinder and the ends are arc welded

8.3 ARC welding

These processes use a welding power supply to create and maintain an electric arc between an electrode and the base material to melt metals at the welding point. They can use either direct (DC) or alternating (AC) current, and consumable or non-consumable electrodes. The welding region is sometimes protected by some type of inert or semi-inert gas, known as a shielding gas, and filler material is sometimes used as well.

- After arc welding, copper tubes are placed inside the shell by gas welding it with perforated mild steel plate. Again perforated plates are welded to inner surface of shell.

8.4 Gas welding

The most common gas welding process is oxyfuel welding, also known as oxyacetylene welding. It is one of the oldest and most versatile welding processes, but in recent years it has become less popular in industrial applications. It is still widely used for welding pipes and tubes, as well as repair work. It is also frequently well-suited, and favoured, for fabricating some types of metal-based artwork. Oxyfuel equipment is versatile, lending itself not only to some sorts of iron or steel welding but also to brazing, braze-welding, metal heating (for bending and forming), and also oxyfuel cutting.

- From outlet and to inlet pipes are fitted.

8.5 Pipe fitting

Pipe fitting is the occupation of installing or repairing piping or tubing systems that convey liquid, gas, and occasionally solid materials. This work involves selecting and preparing pipe or tubing, joining it together by various means, and the location and repair of leaks.

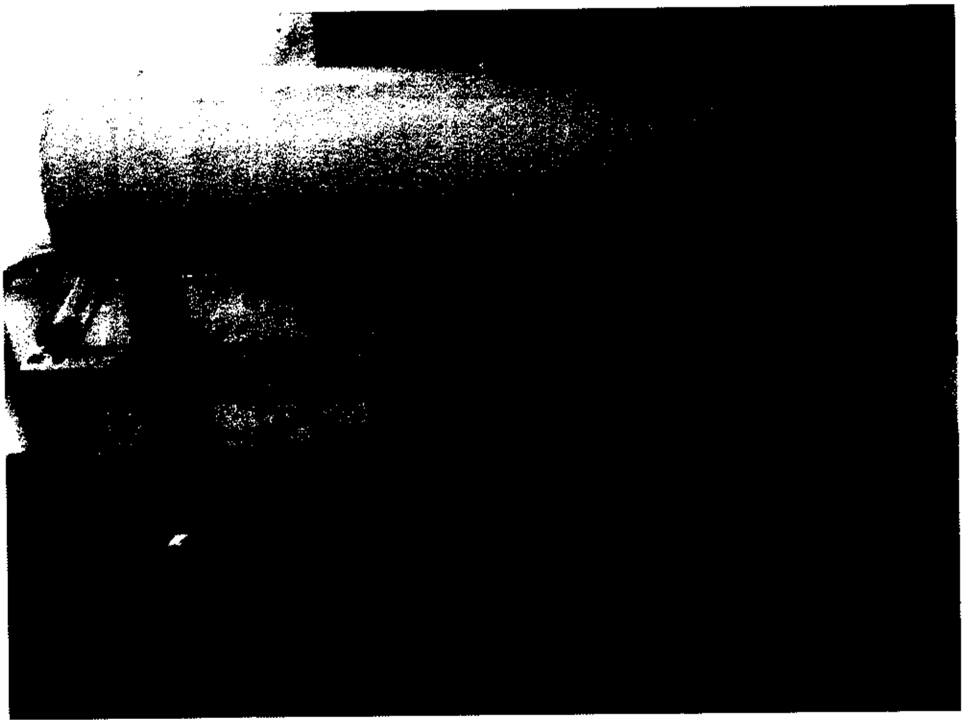


Fig 8.1: front view of heat exchanger setup

Fig 8.2: side view of heat exchanger setup

Fig 8.3: top view of heat exchanger setup

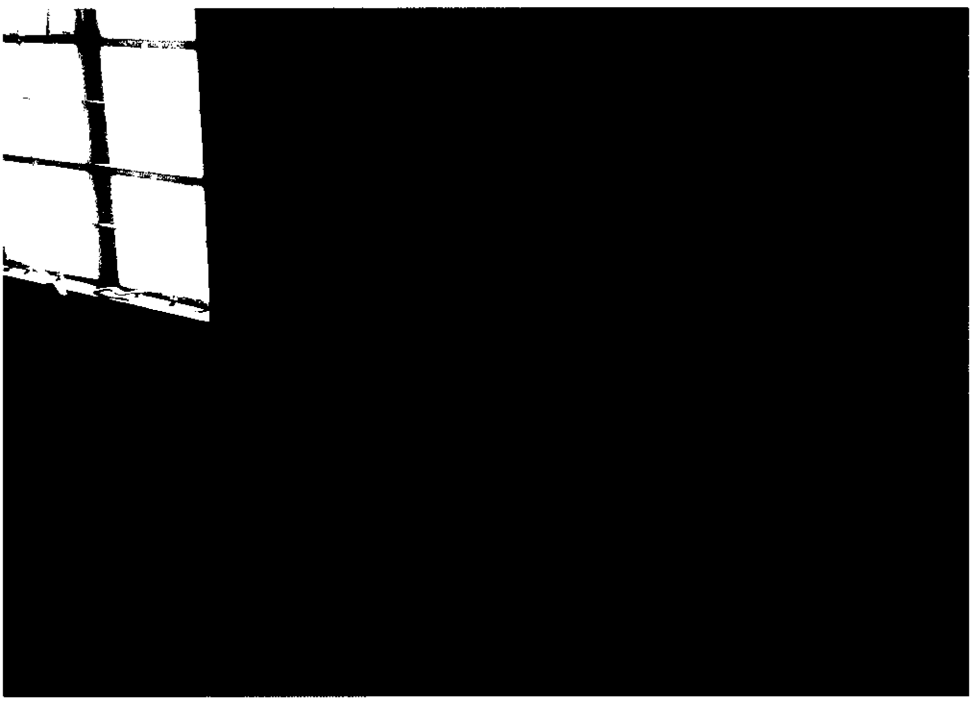


Fig 8.4: overall view of heat exchanger heater blower

CHAPTER 9

CALCULATION

Step 1: **selection of inner and outer pipe diameter**

With the help of HMT data book the suitable shell, tube outer and inner diameter is selected

Step 2: **selection of material**

Shell ----- mild steel

Tube----- copper

Step 3: calculation of heat transfer rate

$$\boxed{\begin{aligned} q_h &= \dot{m}_h c_{p,h}(T_{h,i} - T_{h,o}) \\ q_c &= \dot{m}_c c_{p,c}(T_{c,o} - T_{c,i}) \end{aligned}}$$

where

$\dot{m}_h$ - mass flow rate of hot fluid

c_p - specific heat of fluid

$T_{h,i}$ - Inlet temperature of hot fluid (°C)

$T_{h,o}$ - Outlet temperature of hot fluid (°C)

$\dot{m}_c$ - mass flow rate of cold fluid

$T_{c,o}$ - Inlet temperature of cold fluid (°C)

$T_{c,i}$ - Outlet temperature of cold fluid (°C)

Step 3 : **calculation of logarithmic mean temperature difference**

$$LMTD = \frac{(T_1 - t_2) - (T_2 - t_1)}{\ln\left(\frac{T_1 - t_2}{T_2 - t_1}\right)}$$

T_1 = Hot Stream Inlet Temp.

T_2 = Hot Stream Outlet Temp.

t_1 = Cold Stream Inlet Temp.

t_2 = Cold Stream Outlet Temp.

Step 4: **calculation of heat transfer coefficient**

Shell side & tube side

$$\mathrm{Re} = \frac{\rho V D}{\mu} = \frac{V D}{\nu} = \frac{Q D}{\nu A}$$

where:

- V is the mean fluid velocity in (SI units: m/s)

- D is the diameter (m)
- μ is the dynamic viscosity of the fluid (Pa·s or N·s/m^2)
- ν is the kinematic viscosity ($\nu = \mu / \rho$) (m^2/s)
- ρ is the density of the fluid (kg/m^3)
- Q is the volumetric flow rate (m^3/s)
- A is the pipe cross-sectional area (m^2)

If Reynolds number is lesser than 2300, flow is laminar.

If Reynolds number is greater than 2300, flow is turbulent.

For laminar,

$$Nu = 0.023(Re)^{0.8}(Pr)^{0.3}$$

For turbulent,

$$Nu = 0.10(Gr*Pr)^{0.333}$$

Where

pr- prandtl number,

Gr- grashoff number

$$Nu = (h_i * d_i)/k$$

Where

h_i- heat transfer coefficient (shell)

$$Nu = (h_0 * d_{i0})/k$$

Where

h_0- heat transfer coefficient (tube)

Step7: **calculation of overall transfer coefficient**

$$\frac{1}{UA} = \frac{1}{h_i A_i} + R_{cond} + \frac{1}{h_o A_o}$$

$$R_{cond} = \frac{\ln(D_o / D_i)}{2\pi k L}$$

Step 8: **calculation of heat transfer rate**

$$Q = UA\Delta T_{LM}$$

where

Q = heat transfer rate (W)

U = overall heat transfer coefficient (W/(m^2·K))

A = heat transfer surface area (m^2)

ΔT_{LM} = log mean temperature difference (K)

From this formula, heat transfer surface area is calculated. Hence we calculated the number of tubes

Step 8: **calculation of effectiveness**

$$\varepsilon = \frac{\text{Actual heat transfer}}{\text{Maximum possible heat transfer}}$$

Table 9.1 variation of parameters for constant flow rate(hot)- 2 lpm and constant hot inlet-45°c

Volumetric flow rate (lpm)		Cold water temperature (°C)		hot water temperature (°C)		log Temp Diff ΔTm (°C)	Heat Transfer rate q (kw)	Effectlve ness (%)	Overall Heat Transfer Coeff U (w/m^2k)
Cold	hot	inlet	outlet	inlet	outlet				
2	2	29	36	45	40.5	8.06	1.18	42.12	410.60
2.5	2	29	35	45	40	7.45	1.19	42.25	418.81
3	2	29	34.5	45	38	7.22	1.21	43.78	419.68
3.5	2	29	34.5	45	36	6.11	1.24	46.25	421.33
4	2	29	33	45	34	5.41	1.51	48.75	422.18
4.5	2	29	32.5	45	34	6.12	1.51	52.89	424.19
5	2	29	32.5	45	33.5	6.12	1.58	51.88	425.86
5.5	2	29	31.5	45	33	6.12	1.65	55.10	426.27
6	2	29	31	45	32.5	5.41	1.72	56.13	426.48
6.5	2	29	30	45	32	6.73	1.79	56.28	427.30

Table 9.2 variation of parameters for constant flow rate(hot)- 3 lpm and constant hot inlet-70°c

Volumetric flow rate (lpm) Cold hot		Cold water temperature (°C) inlet outlet		hot water temperature (°C) inlet outlet		log Temp Diff ΔTm (°C)	Heat Transfer rate q (kw)	EffectIve ness (%)	Overall Heat Transfer Coeff U (w/m^2k)
2	3	29	34	70	68	37.39	.825	28.59	145.29
2.5	3	29	33.5	70	67	37.12	1.022	30.11	150.25
3	3	29	33	70	66	36.88	1.153	31.09	154.29
3.5	3	29	32.5	70	64	36.04	1.274	33.59	157.16
4	3	29	32	70	62	35.21	1.329	35.01	160.23
4.5	3	29	32	70	60	34.08	1.411	36.25	162.95
5	3	29	31.5	70	57.5	32.93	1.566	37.51	165.24
5.5	3	29	31	70	56	32.34	1.605	38.26	169.22
6	3	29	31	70	54	31.14	1.792	39.89	174.59
6.5	3	29	30	70	53	31.14	2.092	41.46	177.35

Table 9.3 variation of parameters for constant flow rate(hot)- 3 lpm and constant hot inlet-45°c

Volumetric flow rate (lpm) Cold hot		Cold water temperature (°C) inlet outlet		hot water temperature (°C) inlet outlet		log Temp Diff ΔTm (°C)	Heat Transfer rate q (kw)	EffectIve ness (%)	Overall Heat Transfer Coeff U (w/m^2k)
2	3	29	34	45	43	12.17	.552	46.29	184.67
2.5	3	29	33.5	45	42	11.86	.623	49.21	191.26
3	3	29	33	45	41	11.54	.690	52.11	201.35
3.5	3	29	32.5	45	40	11.22	.759	54.29	210.45
4	3	29	32	45	39.5	11.22	.853	56.85	227.56
4.5	3	29	32	45	38	10.20	.966	59.45	249.15
5	3	29	31.5	45	37.5	10.09	1.035	62.56	271.59
5.5	3	29	31	45	35.5	9.07	1.311	65.12	284.16
6	3	29	31	45	35	8.67	1.380	67.71	305.26
6.5	3	29	30	45	34	8.66	1.519	68.75	336.48

Table 9.4 variation of parameters for constant flow rate(hot)- 2 lpm and constant hot inlet-70°c

Volumetric flow rate (lpm) Cold hot		Cold water temperature (°C) inlet outlet		hot water temperature (°C) inlet outlet		log Temp Diff ΔTm (°C)	Heat Transfer rate q (kw)	Effectlve ness (%)	Overall Heat Transfer Coeff U (w/m^2k)
2	2	29	35	70	66	22.68	.745	52.47	235.99
2.5	2	29	34.5	70	65	22.48	.811	55.68	249.88
3	2	29	34.5	70	63	21.65	.966	57.42	270.16
3.5	2	29	34	70	60	20.60	1.380	59.86	286.39
4	2	29	33.5	70	59	20.38	1.519	61.23	301.77
4.5	2	29	32.5	70	57	19.95	1.795	63.55	315.17
5	2	29	32.5	70	55	19.07	2.071	66.02	322.45
5.5	2	29	31.5	70	53	18.62	2.347	68.22	349.25
6	2	29	31	70	52	18.39	2.485	69.27	361.78
6.5	2	29	30	70	50	17.93	2.761	71.23	378.54

Table 9.5 variation of parameters when air is used as cold medium

Volumetric flow rate (lpm) Cold hot		Cold air temperature (°C) inlet outlet		hot water temperature (°C) inlet outlet		log Temp Diff ΔTm (°C)	Heat Transfer rate q (w)	Effectlve ness (%)	Overall Heat Transfer Coeff U (w/m^2k)
0.25	2	32	45	53	44	6.113	966.67	30.43	772.987
0.5	2	33	42	56	48	6.569	1104.76	34.78	789.78
0.75	2	33.5	45	59	48	6.790	1242.86	41.86	893.45
1	2	36	46	60	49	9.308	1242.86	42.85	1054.25
1.25	2	39.5	48	61	52	10.098	1237.34	43.13	1167.54
1.5	2	40	51	62	52	10.404	1380.96	45.45	1212.56
1.75	2	42	50	65	58	10.513	1398.42	45.83	1222.09
2	2	47	56	68	56	12.651	1519.05	57.14	1313.83
2.25	2	53	64	71	60	12.720	1582.98	61.11	1419.56
2.5	2	54	62	73	61	14.671	1657.15	63.16	1497.45

CHAPTER 10

COMPUTATIONAL FLUID DYNAMICS

COMPUTATIONAL FLUID DYNAMICS

10.1 INTRODUCTION

Computational Fluid Dynamics or simply CFD is concerned with obtaining numerical solution to fluid flow problems by using computers. The advent of high-speed and large-memory computers has enabled CFD to obtain solutions to many flow problems including those that are compressible or incompressible, laminar or turbulent, chemically reacting or non-reacting.

The equations governing the fluid flow problem are the continuity (conservation of mass), the Navier-Stokes (conservation of momentum), and the energy equations. These equations form a system of coupled non-linear partial differential equations (PDE's). Because of the non-linear terms in these PDE's, analytical methods can yield very few solutions. In general, closed form analytical solutions are possible only if these PDE's can be made linear, either because non-linear terms naturally drop out (e.g., fully developed flows in ducts and flows that are in viscose and irrotational everywhere) or because nonlinear terms are small compared to other terms so that they can be neglected (e.g., creeping flows, small amplitude sloshing of liquid etc.). If the non-linearity in the governing PDE's cannot be neglected, which is the situation for most engineering flows, then numerical methods are needed to obtain solutions.

CFD is the art of replacing the deferential equation governing the Fluid Flow, with a set of algebraic equations (the process is called Discretisation), 1 that in turn can be solved with the aid of a digital computer to get an approximate solution. The well-known Discretisation methods used in CFD are Finite Deference Method (FDM), Finite Volume Method (FVM), Finite Element Method (FEM), and Boundary Element Method (BEM).

Commercial CFD codes such as FLUENT, PHOENIX, CFX, CFD++ and Star-CD have also developed to a point that the user need not be an expert on CFD in order to successfully put it to use. The end-user must however have a good knowledge of fluid dynamics for a successful simulation with credible results to be obtained. Therefore, used correctly, CFD codes can reduce time and cost of experiments in product development or process improvement.

The rest of the chapter discusses the various areas of CFD in the following order:

- Governing equations
- Grid generation techniques
- Boundary conditions that define a CFD problem
- Solution algorithms and convergence criteria
- Basic background on the CFD solver used (FLUENT)

10.2 GOVERNING EQUATIONS

The governing equations of fluid behavior are given in equations. These equations are given for compressible flow, but can be easily simplified for incompressible flow. In the Eulerian system, the particle derivative is described as follows

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + \left(\overline{V}.\nabla\right) \qquad \text{----- (4.1)}$$

Where

$$\left(\overline{V}.\nabla\right) = d\overline{V} = \frac{\partial u}{\partial x} + \frac{\partial V}{\partial y} + \frac{\partial w}{\partial Z} \qquad \text{----- (4.2)}$$

This particle derivative will be used in the sections to follow to present the Navier-Stokes equations in conservative form.

10.2.1 Conservation of Mass

The equation for conservation of mass in conservative form is given as

$$\frac{\partial \rho}{\partial t} + \nabla .\left(\rho \overline{V}\right) \qquad \text{----- (4.3)}$$

Where ρ is the density and $\overline{V}$ is the vector velocity of the fluid.

10.2.2 Conservation of Momentum

The equations for conservation of momentum in the three Cartesian directions are presented.

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial\left(\rho u^2\right)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial Z}$$
$$= \frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left(\lambda \nabla \overline{V} + 2\mu.\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right] + \frac{\partial}{\partial z}\left[\mu\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right)\right] + \rho g_x$$

----- (4.4)

$$\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho uv)}{\partial x}+\frac{\partial\left(\rho v^2\right)}{\partial y}+\frac{\partial(\rho vw)}{\partial Z}$$

$$=-\frac{\partial p}{\partial y}+\frac{\partial}{\partial y}\left(\lambda\nabla\overline{V}+2\mu.\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial x}\left[\mu\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)\right]+\frac{\partial}{\partial z}\left[\mu\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right)\right]+\rho g_y$$

----- (4.5)

$$\frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho uw)}{\partial x}+\frac{\partial(\rho vw)}{\partial y}+\frac{\partial\left(\rho w^2\right)}{\partial Z}$$

$$=-\frac{\partial p}{\partial z}+\frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)\right]+\frac{\partial}{\partial y}\left[\mu\left(\frac{\partial w}{\partial x}+\frac{\partial v}{\partial z}\right)\right]+\frac{\partial}{\partial z}\left[\lambda\nabla.\overline{V}+2\mu\frac{\partial w}{\partial z}\right]+\rho g_z$$

----- (4.6)

These equations can be rewritten as a single vector equation using indicial notation:

$$\frac{D\rho\overline{V}}{Dt}=\rho\overline{g}-\Delta p+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial v}{\partial x_j}+\frac{\partial v_j}{\partial x_i}\right)+\delta_{ij}\lambda d\overline{V}\right] \quad \text{----- (4.7)}$$

10.2.3 The Energy Equation

The energy equation, which in essence is the first law of thermodynamics, is given in its most economic form as follows:

$$\rho\frac{D}{Dt}\left[e+\frac{p}{\rho}\right]-\frac{Dp}{Dt}+d\left(K\nabla T\right)+\tau_{ij}\frac{\partial u_i}{\partial x_j} \quad \text{----- (4.8)}$$

Where the viscous stresses are given by the stress tensor:

$$\tau_{ij}=\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) \quad \text{----- (4.9)}$$

10.3 PROGRAM STRUCTURE

Figure 4.1 shows the organizational structure of these components.

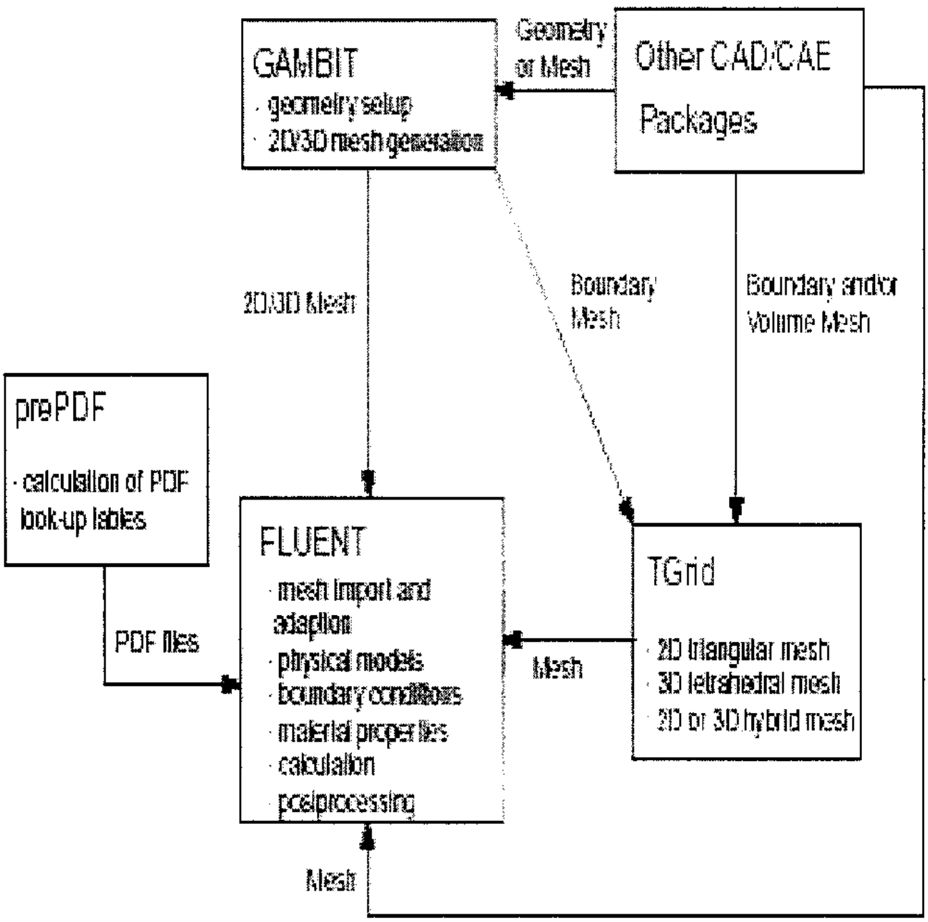


FIGURE 10.1 BASIC PROGRAM STRUCTURE OF CFD

FLUENT package includes the following products:

FLUENT	-	the solver.
PrePDF	-	the preprocessor for modeling non-premixed combustion in FLUENT.
GAMBIT	-	the preprocessor for geometry modeling and mesh generation.
Tgrid	-	an additional preprocessor that can generate volume meshes from existing Boundary meshes.

Translators - (translators) for import of surface and volume meshes from CAD/CAE packages such as ANSYS, CGNS, I-DEAS, NASTRAN, PATRAN, and others.

GAMBIT is used to create geometry and grid. See the GAMBIT documentation for details. Also use TGrid to generate a triangular, tetrahedral, or hybrid volume Mesh from an existing boundary mesh (created by GAMBIT or a third-party CAD/CAE Package). See the TGrid User's Guide for details. It is also possible to create grids for FLUENT using ANSYS (Swanson Analysis Systems, Inc.), CGNS (CFD general notation System), or I-DEAS (SDRC); or MSC/ARIES, MSC/PATRAN, or MSC/NASTRAN (all from MacNeal-Schwendler Corporation). Interfaces to other CAD/CAE packages may be made Available in the future, based on customer requirements, but most CAD/CAE packages can export grids in one of the above formats.

Once a grid has been read into FLUENT, all remaining operations are performed within the solver. These include setting boundary conditions, defining fluid properties, executing the solution, refining the grid, and viewing and post processing the results. Note that preBFC and GeoMesh are the names of Fluent preprocessors that were used before the introduction of GAMBIT. You may see some references to preBFC and GeoMesh in this manual, for those users who are still using grids created by these programs.

10.3.1 Grid generation and GAMBIT

The grid generation process or meshing involves dividing the flow domain into smaller control volumes over which the discretised Navier-Stokes equations are solved. Grid (mesh) types can be classified into two categories namely; structured and unstructured grids. The mentioned types of grids find use in different applications and used in the meshing process in this study.

Structured grids consist of grid lines with a characteristic of not crossing or overlapping. The position of any grid point is uniquely identified by a set of two (2-D) or three (3-D) dimensional indices, e.g., (*i, j, k*). Unstructured grids make no assumption about any structure in the grid definition and usually consist of triangular (tetrahedral (tet) in 3D) elements.

A numerically generated structured grid or mesh, is understood here to be the organized set of points formed by the intersections of the lines of a boundary conforming

10.3.2 Boundary Conditions

In mathematics, any solution to a set of partial differential equations (PDE's) requires a set of boundary conditions for closure and the solution of the governing equations is no exception.

CFD simulations largely depend on the boundary conditions specified: hence correct boundary specification improves convergence to a correct solution. Incorrect boundary and initial conditions, however, can give convergence although not to a correct solution. There are wide variety of boundary types available in FLUENT, but only those used in this study will be given.

Flow inlet and exit boundaries

Pressure inlet: Used to define the total pressure and other scalar quantities at flow inlets.

Pressure outlet: Used to define the static pressure at flow outlets (and also other scalar variables, in case of backflow). The use of a pressure outlet boundary condition instead of an outflow condition often results in a better rate of convergence when backflow occurs during iteration. Note that when backflow occurs, this boundary acts like a pressure inlet boundary.

Wall and symmetry:

Wall: Used to define a solid-fluid interface where viscous flow is considered, thus applying a no-slip condition. The boundary condition on a surface assumes no relative velocity between the surface and the gas immediately at the surface.

Symmetry: Used to define surfaces at which normal velocity and normal gradients of all other variables are zero. This boundary type is essential where the geometry is symmetrical in nature, and only half the domain is specified.

10.3.3 Solution Algorithms

The governing equations of fluid flow are particularly difficult to solve because of their non-linear nature. Much work has been done in numerical methods to solve for these types of equations. Some proven and popular methods worthy of note are SIMPLE, SIMPLE-C, SIMPLER, QUICK and PISO. These methods are appreciated because of their robustness when applied to a variety of problems. In this study, steady state and transient flows are solved using the SIMPLE and PISO algorithms respectively.

10.4 BASIC ELEMENTS OF CFD

CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. All CFD codes contain three main elements.

- Pre-processor
- Solver
- Post-processor

10.4.1 Pre-processor

Pre-processing consists of the input of a flow problem to a CFD program by means of an operator-friendly interface and the subsequent transformation of this input into a form suitable for use by the solver.

- Definition of the geometry of the region of interest in the computational domain.
- Grid generation is the sub-division of the domain into a number of smaller, non-overlapping sub-domains-a grids of cells.
- Selection of the physical and chemical phenomena that need to be modeled.
- Definition of fluid properties.
- Specification of appropriate boundary conditions at cells, which coincide with or touch the domain boundary.

10.4.2 Solver

The numerical methods that form the basis of the solver perform the following steps:

- Approximation of the unknown flow variables by means of simple functions.
- Discretisation by substitution of the approximations into the governing flow equations and subsequent mathematical manipulations.
- Solution of the algebraic equations.

10.4.3 Post-processor

CFD packages are now equipped with versatile data visualization tools.

These include

- Domain geometry & Grid display.
- Vector plot.
- Line and shaded contour plot.
- 2D & 3D surface plots.
- Particle tracking.
- View manipulation (Translation, Rotational, Scaling).
- Color postscript output.

10.4.4 CFD Simulation

The process of performing single CFD simulation is split into components.

Pre-Processor → Solver → Post-Processor

Setting up the simulation	: Pre-processor
Solving for the Flow Field	: Solver
Visualizing the Results	: Post-processor

10.5 PROBLEM SOLVING STEPS

Once you have determined the important features of the problem you want to solve, you will follow the basic procedural steps shown below.

1. Creating the model geometry and grid.
2. Starting the appropriate solver for 2D or 3D modeling.
3. Importing the grid.
4. Checking the grid.
5. Selecting the solver formulation.
6. Choosing the basic equations to be solved: laminar or turbulent (or in viscid), chemical Species or reaction, heat transfer models, etc. Identify additional models needed: Fans, heat exchangers, porous media, etc.
7. Specifying material properties.
8. Specifying the boundary conditions.
9. Adjusting the solution control parameters.
10. Initializing the flow field.
11. Calculating a solution.
12. Examining the results.
13. Saving the results.

CHAPTER 11

RESULTS & DISCUSSION

RESULTS & DISCUSSION

11.1 Effect of flow rate of the cold fluid:

Increase in the flow rate of cold fluid results in increase in the overall heat transfer coefficient as can be seen from tables. This is because increase in the flow rate increases the Reynolds number, which in turn increases the Stanton number and thereby the film heat transfer coefficient. The increase in film heat transfer coefficient will increase the overall heat transfer coefficient. This will also cause a decrease in the tube outlet temperature, as can be observed form tables. This is because increase in the volumetric flow rate increases the mass flow rate in a much faster rate than over all heat transfer coefficient or the heat energy transferred

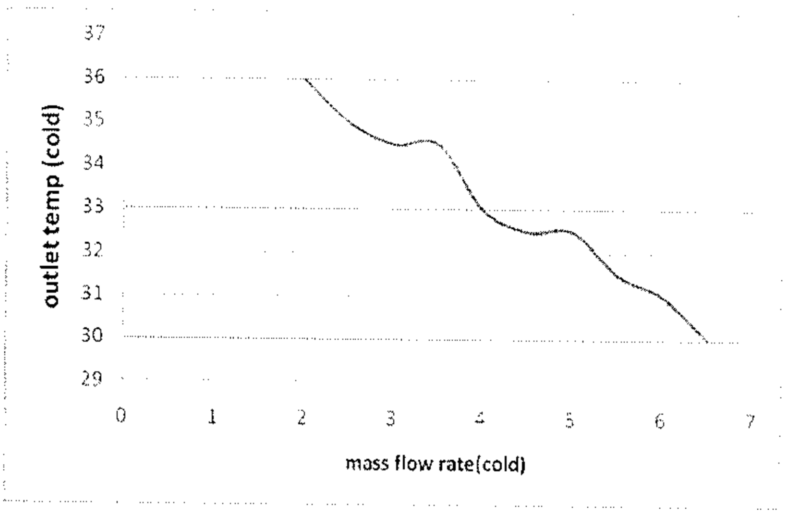


Fig 11.1 effect of flow rate(cold) on outlet temperature for constant flow rate(hot)- 2 lpm and constant hot inlet-45°c

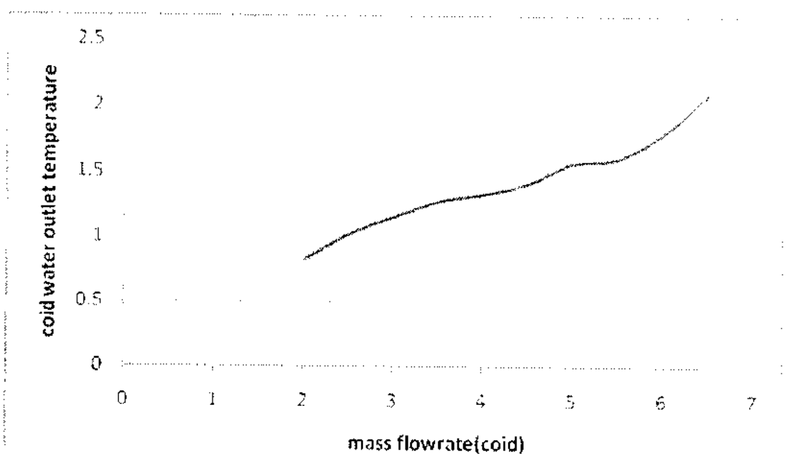


Fig 11.2 effect of flow rate(cold) on outlet temperature for constant flow rate(hot)- 3 lpm and constant hot inlet-70°c

Fig 11.3 effect of flow rate(cold) on outlet temperature for constant flow rate(hot)- 3 lpm and constant hot inlet-45°c

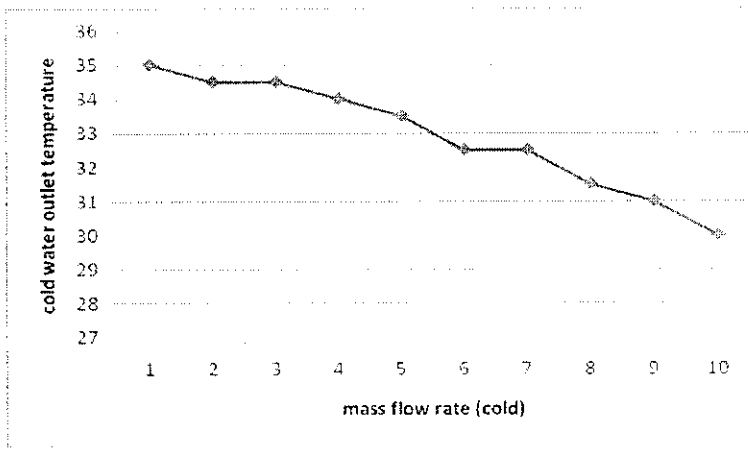


Fig 11.4 effect of flow rate(cold) on outlet temperature for constant flow rate(hot)- 2 lpm and constant hot inlet-70°c

Fig. 11.1,2,3,4 illustrates the variation of the outlet water temperature with water mass flow rate for different curvature ratios. It is found that when the inlet cold water and average tube wall temperatures are kept constant, the outlet water temperature decreases with increasing cold water mass flow rate. This is because the heat transfer rate increases as water mass flow rate increases. But, the increasing of the heat transfer is less than that of cold water mass flow rate. Therefore, the outlet water temperature tends to decrease as cold water mass flow rate increases. For given water mass flow rate, the outlet water temperature at lower curvature ratio is higher than those at higher ones. This is because the tube length for lower curvature ratio is higher than that higher ones. Considering the clearly seen that the predicted outlet water temperature is slightly higher than the measured one.

11.2 Overall heat transfer coefficient for Shell and Tube Heat Exchanger:

As the volumetric flow rate of the tube side fluid is increased from 2 to 7 lpm, the overall heat transfer coefficient increased from 426.167 to 450.15 W/(m^2 K). For the same volumetric flow rates, the simulated values varies from 421.805 to 448.605 W/m^2K respectively, i.e., almost same as experimental values.

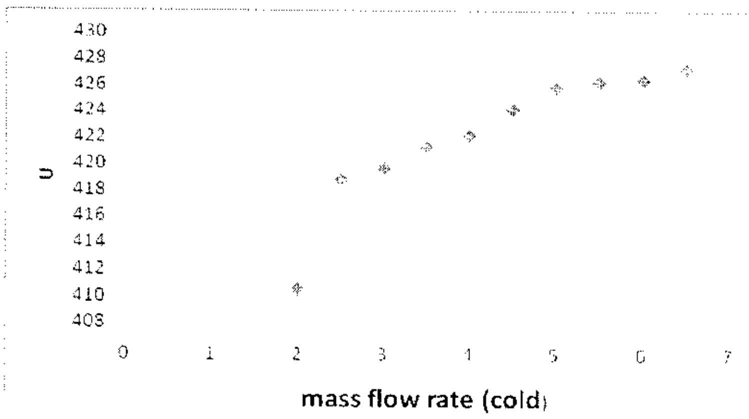


Fig 11.5 effect of flow rate(cold) on overall heat transfer coefficient for constant flow rate(hot)- 2 lpm and constant hot inlet-45°c

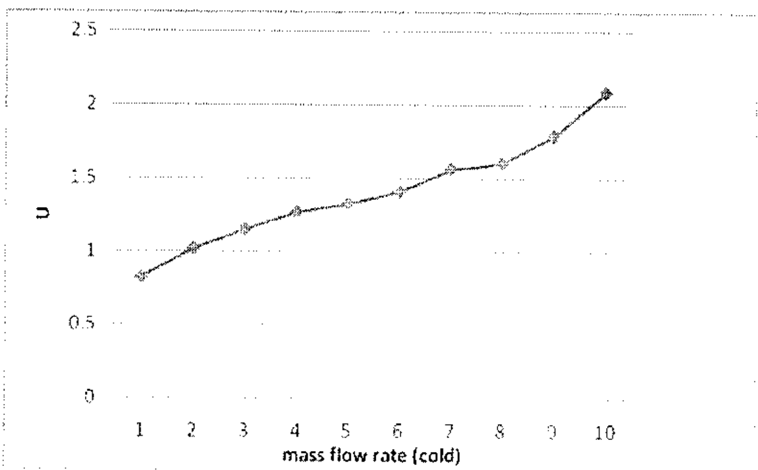


Fig 11.6 effect of flow rate(cold) on overall heat transfer coefficient for constant flow rate(hot)- 3 lpm and constant hot inlet-70°c

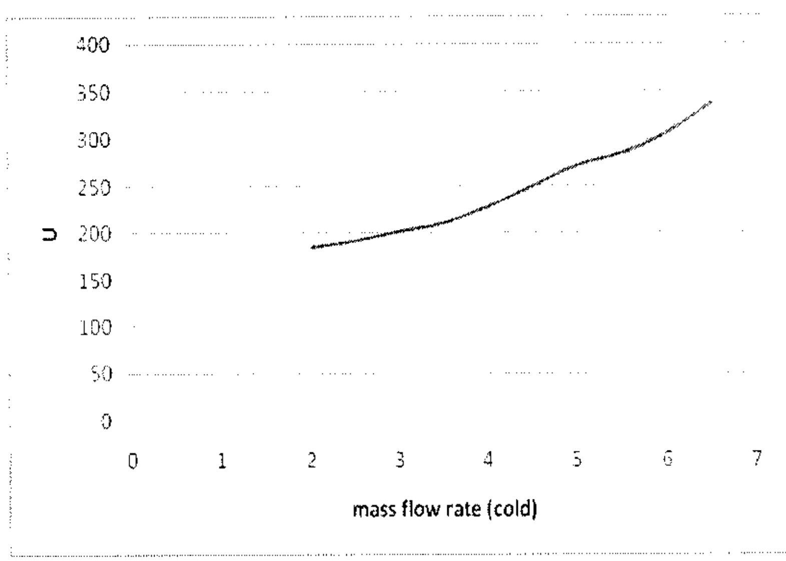


Fig 11.7 effect of flow rate(cold) on overall heat transfer coefficient for constant flow rate(hot)- 3 lpm and constant hot inlet-45°c

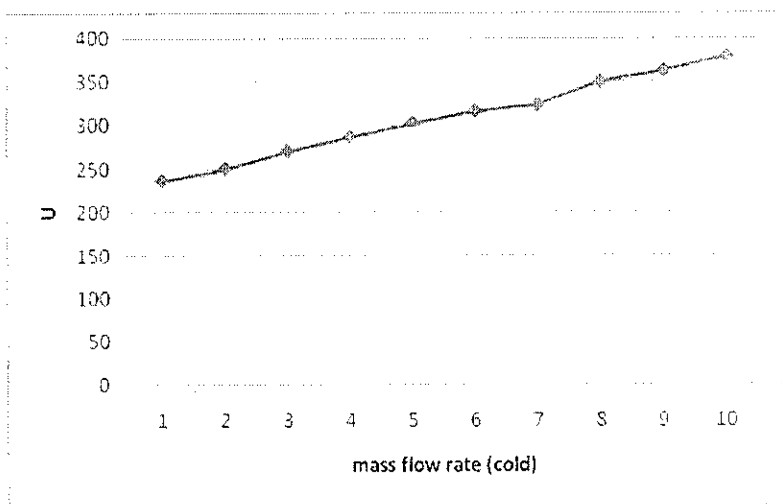


Fig 11.8 effect of flow rate(cold) on overall heat transfer coefficient for constant flow rate(hot)- 2 lpm and constant hot inlet-70°c

11.3 Shell outlet temperature for Shell and Tube Heat Exchanger:

For the flow rate increments from 2 to 7 lpm, the outlet temperature of the shell side fluid

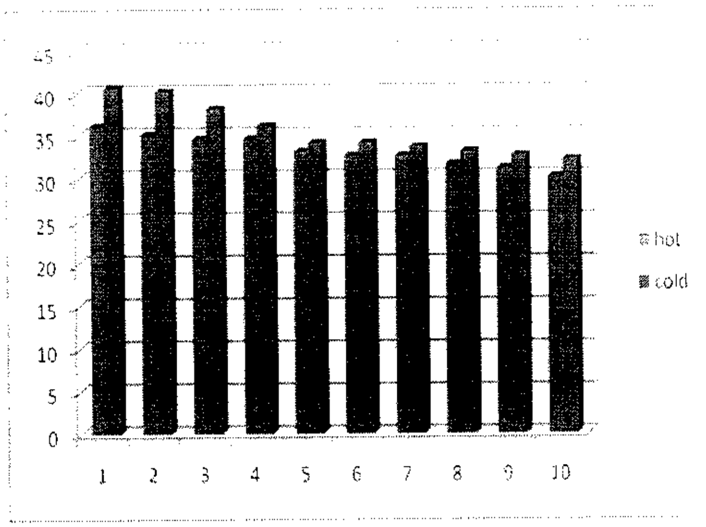


Fig 11.9 variation of outlet temperature for constant flow rate(hot)- 2 lpm and constant hot inlet-45°c

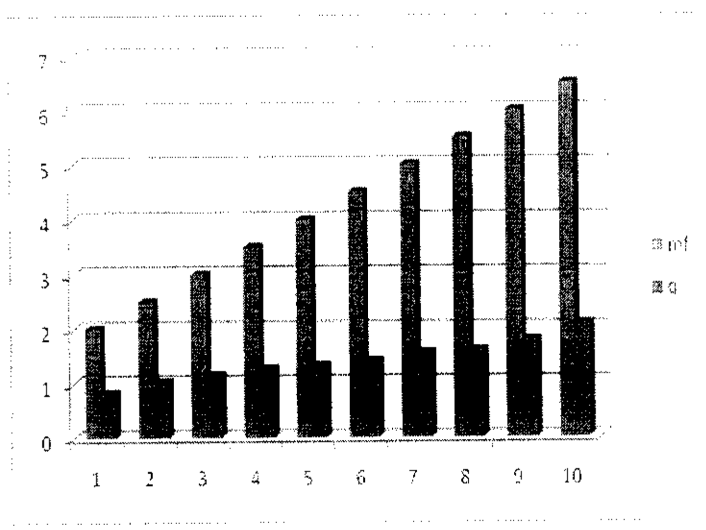


Fig 11.10 variation of outlet temperature for constant flow rate(hot)- 3 lpm and constant hot inlet-70°c

11.4 Tube outlet temperature for Shell and Tube Heat Exchanger:

For the flow rate increments from 2 lpm to 7 lpm, the outlet temperature of tube side fluid varied from 35 to 41.5°C, whereas the simulated values were 38 to 40°C respectively.

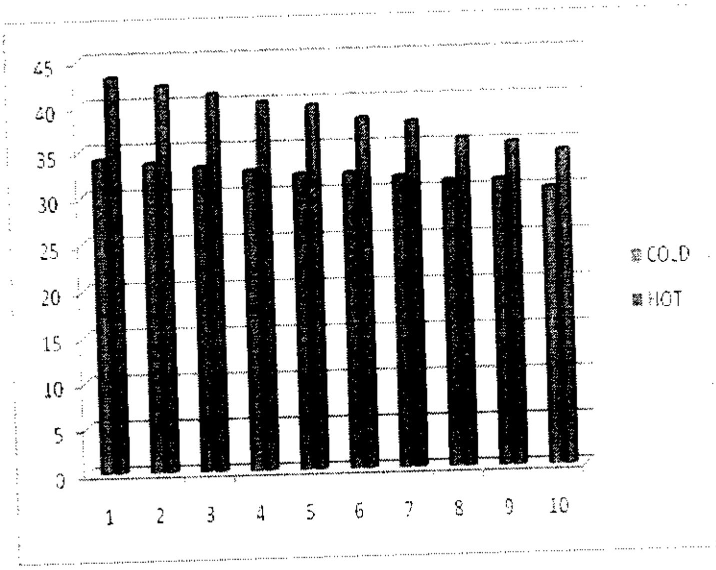


Fig 11.11 variation of outlet temperature for constant flow rate(hot)- 3 lpm and constant hot inlet-45°c

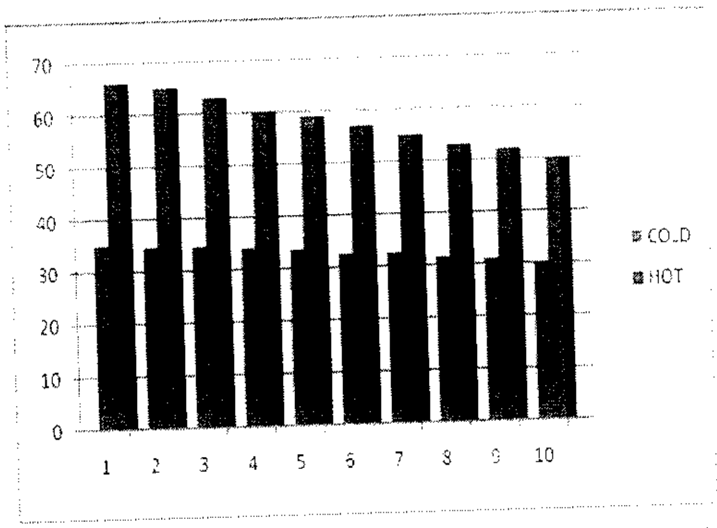


Fig 11.12 variation of outlet temperature for constant flow rate(hot)- 2 lpm and constant hot inlet-70°c

11.5 Effect of heat transfer rate of the cold fluid:

Figure show the variation of the heat transfer rate with cold water mass flow rate. As expected, the heat transfer rate is directly proportional to the water mass flow rate. In addition, it can be noted that the curvature ratios have significant effect on the heat transfer rate. The same explanation described above as for Figs. 6 and 7 can be given. However, this effect can be clearly seen at higher water mass flow rate.

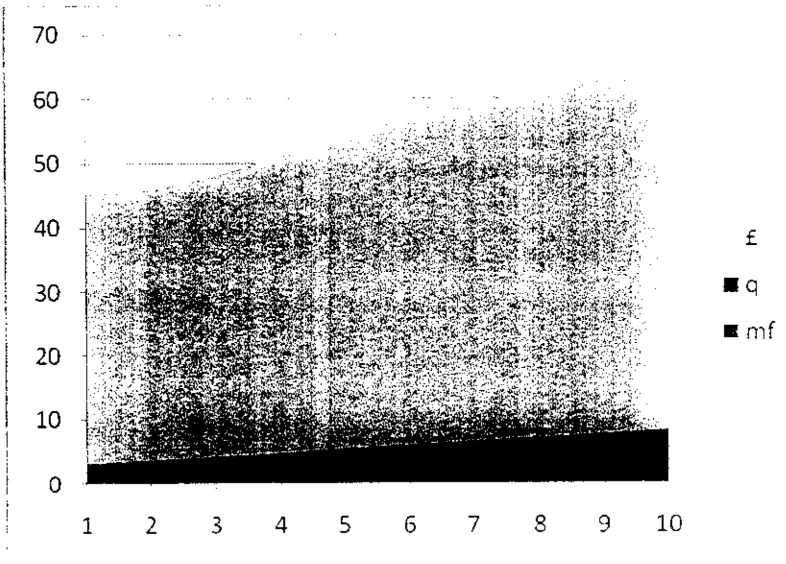


Fig 11.13 variation of heat transfer rate for constant flow rate(hot)- 2 lpm and constant hot inlet-45°c

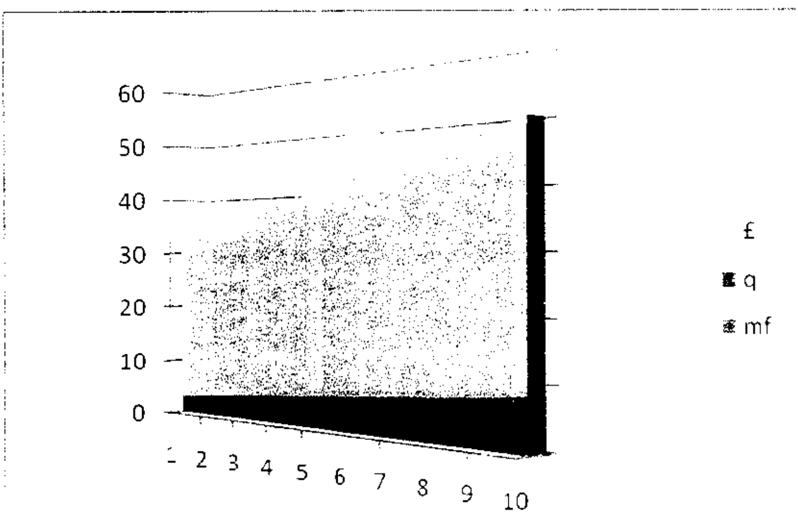


Fig 11.14 variation of heat transfer rate for constant flow rate(hot)- 3 lpm and constant hot inlet-70°c

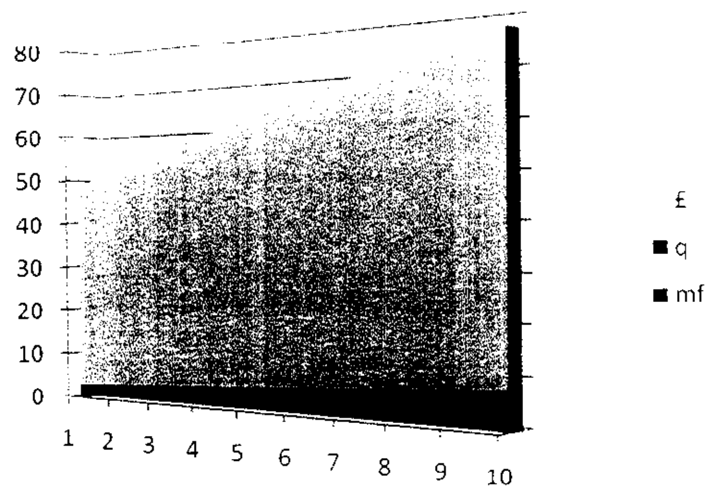


Fig 11.15 variation of heat transfer rate for constant flow rate(hot)- 3 lpm and constant hot inlet-45°c

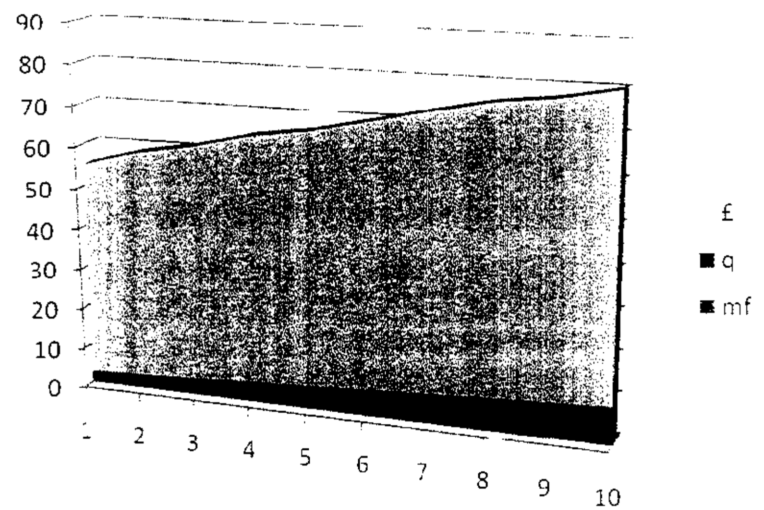


Fig 11.16 variation of heat transfer rate for constant flow rate(hot)- 2 lpm and constant hot inlet-70°c

air is cold medium

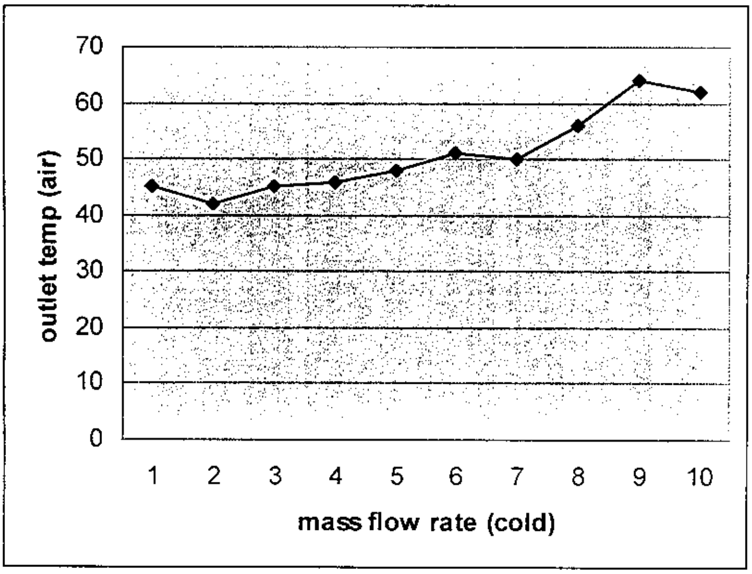


Fig 11.17 effect of flow rate(cold) on outlet temperature when air is cold medium

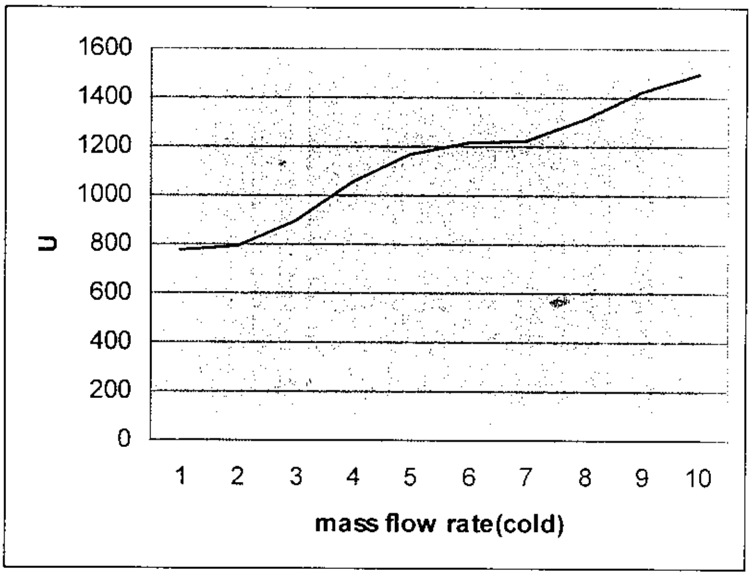


Fig 11.18 effect of flow rate(cold) on overall heat transfer coefficient when air is cold medium

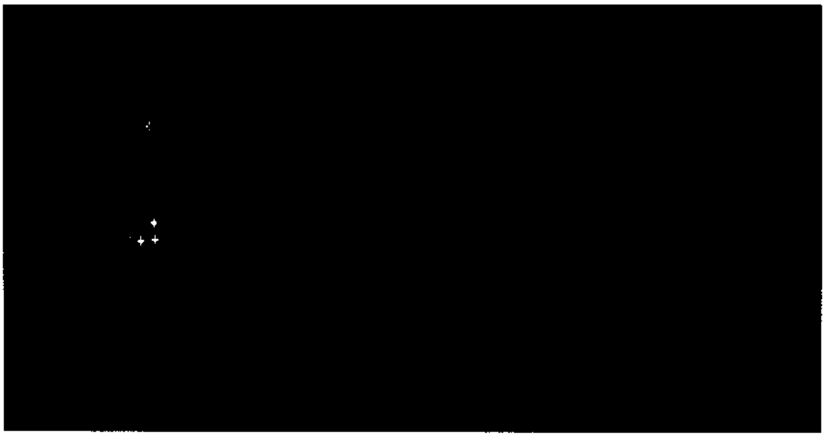


Fig 11.21: GAMBIT MODEL

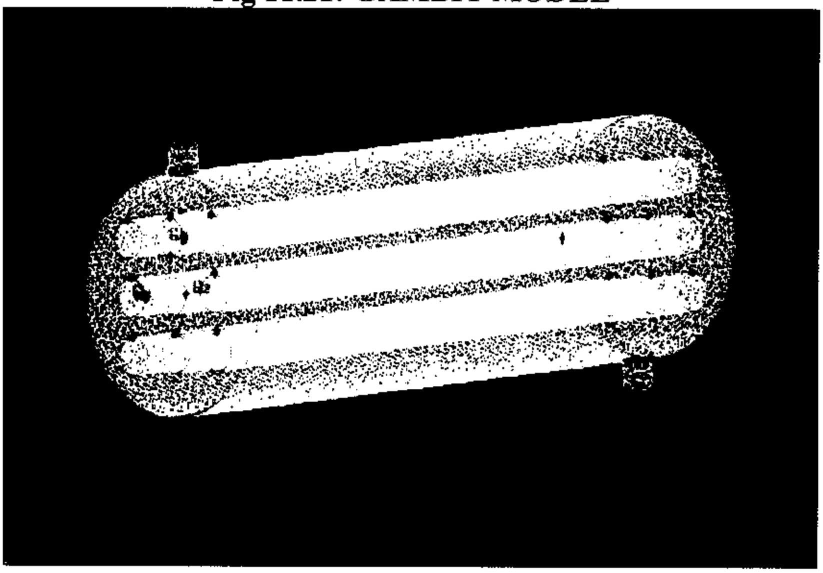
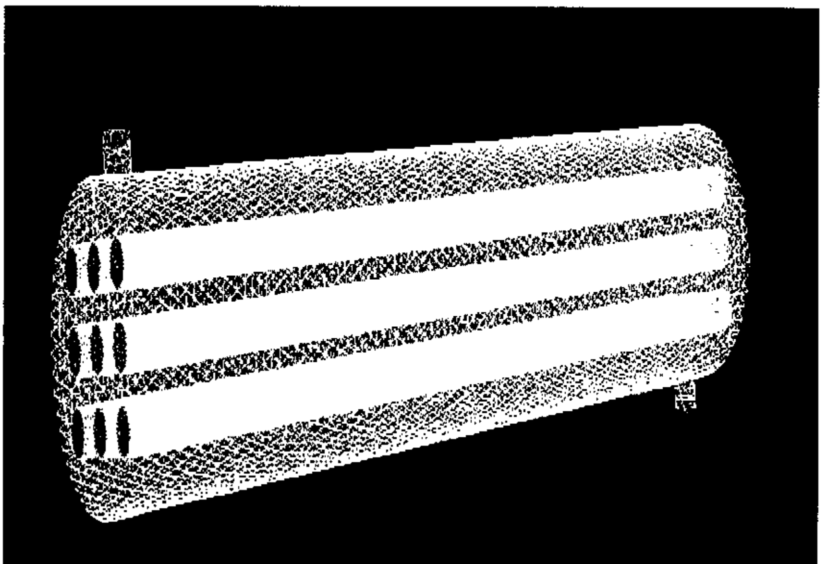


Fig 11.22: GAMBIT MODEL MESH



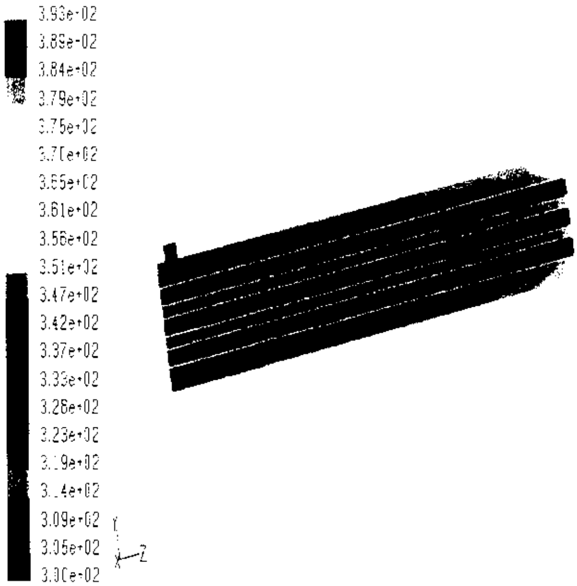


Fig 11.24: TEMPERATURE DISTRIBUTION

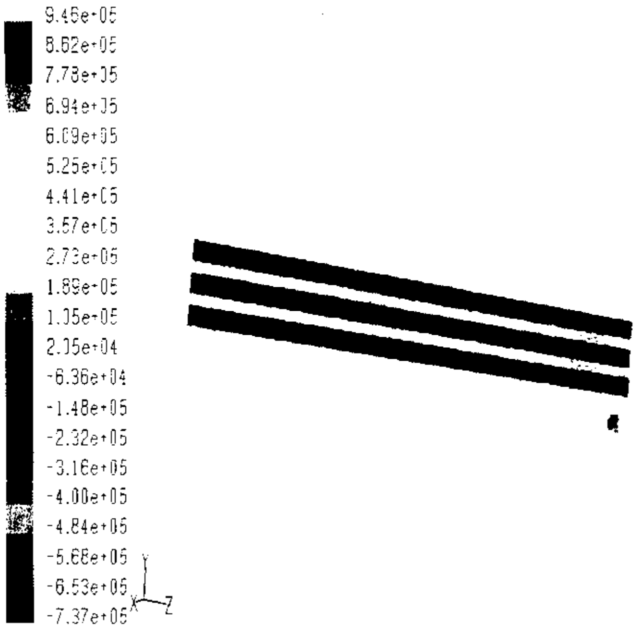


Fig 11.25: VELOCITY DISTRIBUTION

CHAPTER 12

COST ANALYSIS

COST ANALYSIS

Calculation of Material Cost:

For Shell:

Volume, V_1 $= (\pi/4) * D^2 * L$

$= (\pi/4) * (498.04^2 - 484.38^2) * 850$

$= 8.958 * 10^{-3}\ m^3$

For Side Cover:

Volume, V_2 $= (\pi/4) * d^2 * L$

$= (\pi/4) * 498.04^2 * 6.83$

$= 1.3305 * 10^{-3}\ m^3$

Since there two side covers, then volume

$= 1.3305 * 10^{-3}\ m^3 * 2$

V_2 $= 2.661 * 10^{-3}\ m^3$

Since there two side covers, then volume

$= 1.3305 * 10^{-3}\ m^3 * 2$

V_2 $= 2.661 * 10^{-3}\ m^3$

For Dome:

Volume, V_3 $= (2/3) * \pi (r_1^3 - r_2^3)$

$= 1.526 * 10^{-5}\ m^3$

Since there two dome, then volume

$= 1.526 * 10^{-5} * 2$

V_3 $= 3.053 * 10^{-5}\ m^3$

For Inlet & Outlet pipe:

Volume, V_4 $= (\pi/4) * d^2 * L$

$= (\pi/4) * (48.26^2 - 34.6^2) * 500$

$= 4.445 * 10^{-4}\ m^3$

Total volume $= V_1 + V_2 , V_3 , V_4$

$= 0.01206\ m^3$

Weight of steel = volume * density

Density of steel = 7833 Kg/ m^3

Weight of steel = 94.46 Kg

Cost of steel = Rs.170/ Kg

Total cost = Rs. 1606

For Tubes:

Volume of the Tube $= (\pi/4) * D^2 * L$

$= (\pi/4) * (38.1^2 - 30.4^2) * 850$

$= 3.521 * 10^{-4}\ m^3$

For Tubes

Volume $= 0.0172\ m^3$

Weight of the copper = 0.0172 * 8954

= 154 Kg

Cost of copper = Rs.380/Kg

Cost of tube material = Rs. 5852

Calculation of Welding cost:

For shell

Length of weld $= 2 * \pi * 249.02$

= 1564.639 mm

For 4 sides = 4 * 1564.639

= 6258.55 mm

For inlet & outlet pipes

Length of the weld $= 2 * \pi * 48.26$

= 151.61 mm

For four pipes = 4 * 151.61

= 606.45 mm

Total length of the weld = 6865 mm

Cost of the weld = Rs.4/ inch

Cost for welding shell = Rs. 1081

For tubes

Length of weld $= \pi * 38.1$

$= 119.69$ mm

For 49 tubes on both sides $= 98 * 119.69$

$= 461.79$ inch

Cost for welding = Rs. 5/ inch

Cost for welding tubes = Rs. 308.98

Calculation of machining cost:

For drilling = Rs. 3/drill

Total cost for drilling = Rs. 3 * 9

= Rs. 27

Calculation of Total cost:

Cost of steel = Rs. 6,658.50

Cost of copper = Rs. 58.52

Cost for welding = Rs.1389.98

Cost for drilling = Rs. 27

Total cost = Rs. 8,862.48

= Rs. 8,863 (rounded off)

CHAPTER 13

CONCLUSION

CONCLUSION

Through this project we have designed, fabricated and analyzed a Shell and Tube heat exchanger which are used by the industries for the process of heat removal . Detailed flow analysis on shell and tube profiles has ensured optimum variation in temperature, flow, velocity, pressure distribution.

Experiments were conducted on a 1-1 Shell and Tube heat exchanger with different cold side flow rates and different temperature of cold fluid. The effect of these parameters on the shell outlet temperature, tube outlet temperature and overall heat transfer coefficients were studied. It was found that cold fluid outlet temperature decreases and the overall heat transfer coefficient increases with increase in flow rate of cold fluid. Also the outlet temperature of cold fluid decreases and overall heat transfer coefficient increases with increase in temperature of water. The overall effectiveness of heat exchanger was found to increase with decrease in temperature of water. It was found that the Shell and Tube Heat Exchanger is the most effective compared with the other heat exchanger. A GAMBIT model of this system is developed, simulated using CFD - FLUENT and compared with the experimental values. Finally experimental values are compared with stimulated values.

SCOPE FOR FUTURE IMPROVEMENT

1. Analyze the cost of fabrication of our design
2. Find the payback period for the same to take back the investment

To improve the heat transfer rate,

1. Design of improved baffles, which changes the flow direction of the refrigerant.
2. Try out with some other materials instead of mild steel and mild steel for better thermal conductivity.
3. Design of multi pass shell and tube heat exchanger for the same application.

Design of tubes with pyramid like projections on the surface of the tubes.

CHAPTER 14

REFERENCE