

TWO PHASE FLOW CFD ANALYSIS OF REFRIGERANTS IN A
CONDENSER PIPE FOR PREDICTION OF
PRESSURE DROP AND
PUMPING POWER

by

ANIKET RAMCHANDRA KALAMBE

Presented to the Faculty of the Graduate School of
The University of Texas at Arlington in Partial Fulfillment
Of the Requirements
For the Degree of

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

THE UNIVERSITY OF TEXAS AT ARLINGTON

December 2015

Copyright © by Aniket Ramchandra Kalambe 2015
All Rights Reserved



ACKNOWLEDGEMENTS

I would like to take this opportunity to thank my supervising professor Dr. Dereje Agonafer for his constant encouragement, support and guidance during the course of my research and studies at this University. The invaluable advice and support provided by him was the major driving force, which enabled me to complete my thesis.

I would like to thank Dr. Haji-Sheikh and Dr. Veerendra Mulay and for taking time to serve on my thesis committee.

I am obliged to Ms. Sally Thompson and Ms. Debi Barton for helping me out in all educational matters. They have been very kind and supportive whenever I needed their help.

I would like to thank all my friends in the EMNSPC team and in the University for helping me throughout my time here at this University. Finally, I would like to thank my parents for their support, both emotionally and financially, without which I would not have been able to complete my degree.

November 23, 2015

ABSTRACT

TWO PHASE FLOW CFD ANALYSIS OF REFRIGERANTS IN A
CONDENSER PIPE FOR PREDICTION OF
PRESSURE DROP AND
PUMPING POWER

Aniket Ramchandra Kalambe, MS

The University of Texas at Arlington, 2015

Supervising Professor: Dereje Agonafer

With the ever growing need of achieving optimum cooling at chip level at the expense of low input pumping power and keeping in check the Global Warming Potential (GWP) & Ozone Depletion Potential (ODP) has given rise to “two-phase on chip cooling” with nature friendly refrigerants. Currently, “air-cooling” and “single-phase water on-chip cooling” using copper micro-channel coolers are being used but “two-phase on chip cooling” with refrigerants have a very large scale & long lasting advantages over the prior. However, due to the perceived complexity of modelling two-phase flow, this solution is not yet well understood. Modelling of two phase flow, particularly liquid – vapor under diabatic conditions inside a horizontal tube using CFD analysis is difficult with the available two phase models in FLUENT due to continuously changing flow

patterns. This study is an attempt at modelling a two-phase flow for various refrigerants for the proper prediction of pressure drop and pumping power. In the present analysis, CFD analysis of two phase flow of refrigerants inside a horizontal condenser tube of inner diameter, 0.0085m and 1.2m length is carried out using homogeneous model under adiabatic conditions. The refrigerants considered here are R134a, R407C and the newly developed Du-Pont/Honeywell R1234yf. The analysis is performed at saturation temperature to evaluate the local frictional pressure drop. Using Homogeneous model, average properties are obtained for each of the refrigerants that is considered as single phase pseudo fluid. The so obtained pressure drop data is compared with experimental data and the separated flow models available in literature.

Table Of Contents

ACKNOWLEDGEMENTS	iii
ABSTRACT	iv
List of Illustrations	ix
List of Tables	xi
Chapter 1 INTRODUCTION.....	1
1.1. Two-Phase On Chip Cooling	4
1.1.1. An Introduction.....	4
1.1.2. Two-Phase Refrigerant Cooling Vs Single-Phase Water Cooling.....	5
1.1.3. Alternative Refrigerants.....	7
1.2. Refrigerant Vapor Compression Cycle.....	9
1.2.1. Theory.....	9
1.2.2. Importance of Two Phase Pressure Drop Measurement.....	12
1.3. Multi-Phase Flows	13
1.3.1. Theory.....	13
1.3.2. Examples and Applications.....	13
1.3.3. Characteristics of Two-Phase Flow	15
1.3.4. Multi-Phase Flow Regimes.....	16

Chapter 2 LITERATURE REVIEW	18
2.1. General	18
2.2. Separated Flow Model	20
2.3. Analytical Correlations	22
2.3.1. Lockhart-Martinelli Correlation.....	22
2.3.2. Gronnerud Correlation	23
2.3.3. Muller-Steinhagen & Heck Correlation.....	23
2.4. Mathematical Formulation.....	24
Chapter 3 CFD ANALYSIS	26
3.1. Introduction to CFD Analysis (FLUENT-ANSYS)	26
3.2. Governing Equations	27
3.3. Turbulence Modeling (Standard k-epsilon model).....	30
3.3.1. K-Epsilon Turbulence Model	31
Chapter 4 DESIGN & ANALYSIS METHODOLOGY	33
4.1. Geometry Specifications.....	33
4.2. Meshing.....	35
4.3. CFD Analysis.....	39
Chapter 5 SCENARIOS CONSIDERED & RESULTS.....	45
5.1. Set 1	46
5.1.1. Scenario 1	46
5.1.2. Scenario 2.....	47

5.1.3. Scenario 3.....	48
5.1.4. Scenario 4.....	49
5.2. Set 2	50
5.2.1. Scenario 1.....	50
5.2.2. Scenario 2.....	51
Chapter 6 CONCLUSION & FUTURE WORK	52
6.1. Conclusion	52
6.2. Future Work	52
References.....	53
Biographical Information.....	55

List of Illustrations

Figure 1-1: Heat Flux trend of commercial processors (1995-2014)	2
Figure 1-2: Moore’s Law predicted trend of transistors (1971-2011)	3
Figure 1-3: Chemical formula/structure of R1234yf	8
Figure 1-4: Typical Refrigerant Vapor Compression Cycle.....	10
Figure 1-5: Temperature-Entropy diagram for Vapor Compression Cycle.....	11
Figure 1-6: Various Multi-Phase Flow Regimes	16
Figure 1-7: Flow Regimes at high Void Fractions of air-water mixture (“Ewing et al [1999]”).....	17
Figure 3-1: Representation of a 3D grid	30
Figure 4-1: ANSYS Space-Claim Model	34
Figure 4-2: ANSYS Meshing Model	34
Figure 4-3: Mesh Sensitivity Analysis.....	35
Figure 4-4: Meshed Inlet/Outlet.....	37
Figure 4-5: Meshed Wall	37
Figure 4-6: Orthogonal Quality of Meshed Model.....	38
Figure 4-7: Skew-ness Quality of Meshed Model	39
Figure 4-8: Kinematic Viscosity Model Comparison, $G = 176 \text{ kg/m}^2\text{-s}$	42
Figure 4-9: Kinematic Viscosity Model Comparison, $G=528 \text{ kg/m}^2\text{-s}$	42
Figure 5-1: Pressure drop comparison, R134a, $T_s = 40\text{c}$, $G = 600\text{kg/m}^2\text{-s}$	46
Figure 5-2: Pressure drop comparison, R134a, $T_s = 40\text{c}$, $G = 400\text{kg/m}^2\text{-s}$	47

Figure 5-3: Pressure drop comparison, R407C, Ts =40c, G =600kg/m ² -s	48
Figure 5-4: Pressure drop comparison, R407C, Ts =40c, G =400kg/m ² -s	49
Figure 5-5: Pressure Drop comparison, R1234yf vs R134a, Ts = 25 degrees, G = 600kg/m ² -s	50
Figure 5-6: Pressure Drop comparison, R1234yf vs R134a, Ts = 25 degrees, G = 400kg/m ² -s	51

List of Tables

Table 4-1: Mesh Sensitivity Analysis	36
Table 4-2: Averaged Properties of R134a at 40 degree Celsius	41

Chapter 1

INTRODUCTION

Data Centers have gained utmost importance in today's modern age of technology as they are the facilities housing computer systems and other components, like data storage units, telecommunication systems, etc. which store all our "online data". These data center equipments require a certain controlled environment for its optimum functionality. This requires an efficient cooling system as well, since the huge amount of IT equipment within the data center produces a large amount of heat. So, the entire data center is an expensive facility that needs to be maintained and protected from any sort of damage or factors that would cause malfunctioning.

Advances in modern data center cooling technologies are facing challenges such as the need to mitigate highly & large concentrated heat fluxes because of the ever growing performance of new generation processors. Also reduction of energy consumption is strongly required to keep in check the global warming potential. In US, the data center operation energy consumption was more than 100 billion kWh in 2011, which represented an annual cost of approximately \$7.4 billion [1]. Cooling datacenters costs up to 45% [2] of the total consumption using current air cooling technologies. In US, it is about 45 billion kWh usage with an annual cost of \$3.3 billion for cooling. A major drawback with the current data center cooling technologies is that all the used energy is converted into heat and is rejected into

the atmosphere. Thus, using this waste heat can decrease the overall operating cost and also the carbon footprint of the datacenter. Data centers now-a-days make use of air cooling technologies but limits of air cooling are being reached due to the recent microprocessor performance increase (Figure: 1-1). Thus in order to take advantage of increasing computing power while having a smaller carbon footprint, alternative solutions to air cooling are needed.

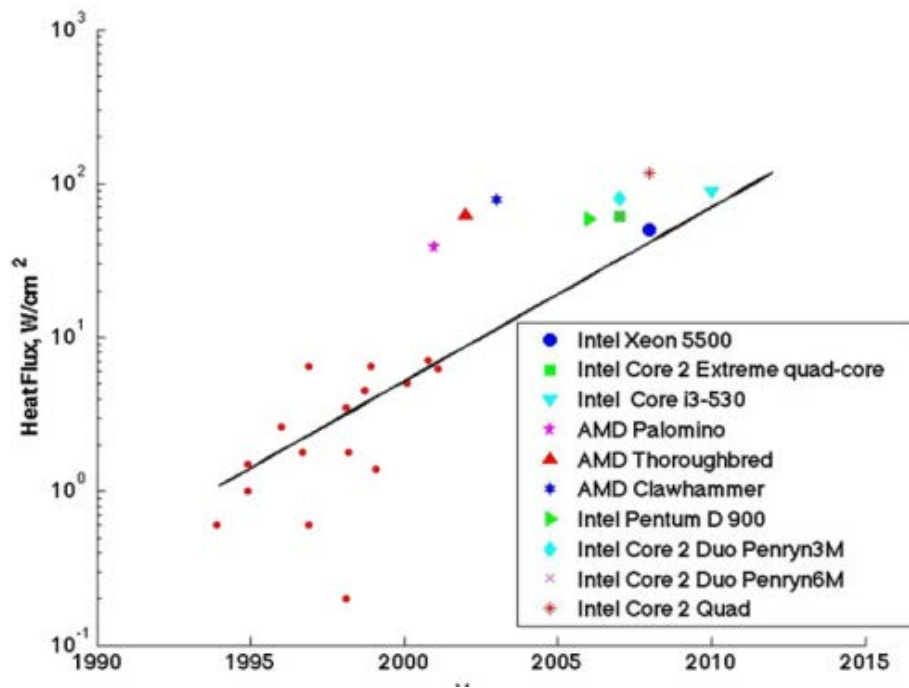


Figure 1-1: Heat Flux trend of commercial processors (1995-2014)

Microprocessor Transistor Counts 1971-2011 & Moore's Law

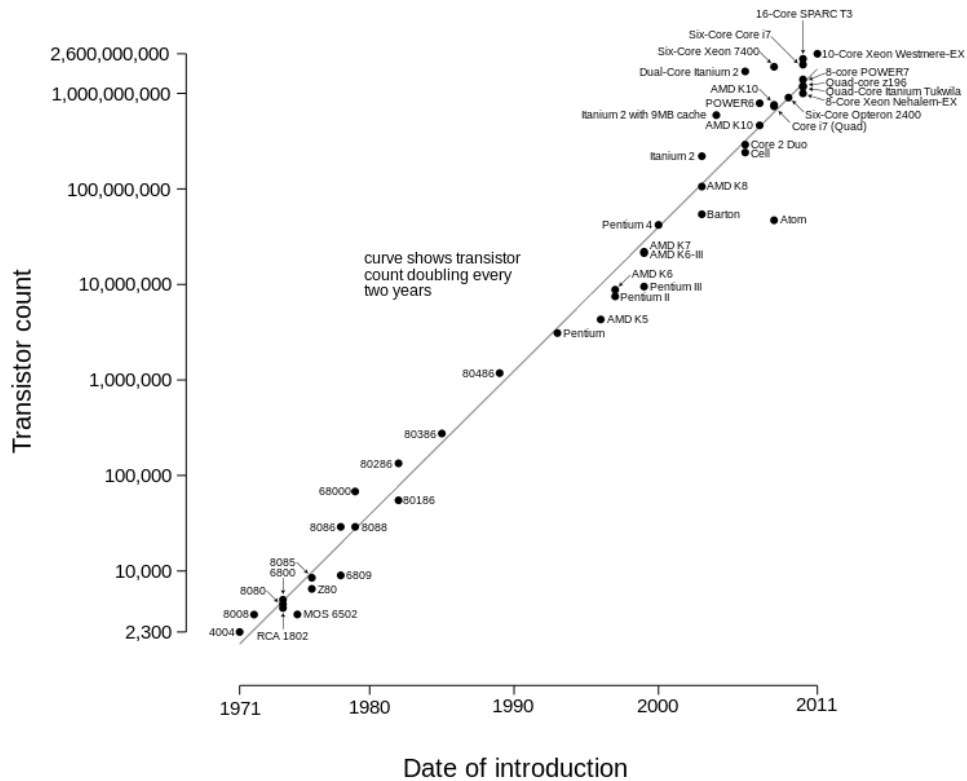


Figure 1-2: Moore's Law predicted trend of transistors (1971-2011)

https://en.wikipedia.org/wiki/Moore%27s_law#/media/File:Transistor_Count_and_Moore%27s_Law_-_2011.svg

Gordon Moore was the co-founder of INTEL. Moore's law is an observation that the numbers of transistors will double every two years. Figure 1-2 justifies this trend by depicting the increase in the number of transistors over the years. This increase basically means the increase in the number of heat generating devices. This all culminates to developing new novel data center cooling technologies.

1.1. Two-Phase On Chip Cooling

1.1.1. An Introduction

On-chip cooling with primary coolant as a liquid or refrigerant is brought right up to the main processor as a replacement to air is one solution attracting a lot of interest in the current data center cooling industry. The main advantage of two-phase (refrigerant) micro-channel flow is that the latent heat of the fluid, which is much more effective in removing heat than the sensible heat of a single-phase (water) fluid. The latent heat also makes sure of the uniformity in chip temperatures. It's been shown that heat fluxes as high as 300 W/cm^2 [4] can be achieved by using a phase changing two-phase refrigerant while maintaining temperatures of chips below their critical value. Heat flux values of 1000 W/cm^2 have been cooled using refrigerants [5]. Also it was shown that heat flux values of 180 W/cm^2 can be removed with a saturation temperature of 60 C , while maintaining the temperature of chip below 85 C [6]. The various advantages of a refrigerant are that it is a dielectric fluid with a long successful history in industrial applications; also it is inert to most engineering materials and readily available & inexpensive. The newly developed 4th generation refrigerants have a negligible impact on the environment. The most important advantage of using two phase on-chip micro-channel cooling for datacenters is that the heat gained from the chip cooling can be reused somewhere else. This is because the heat removal process is local to the chip and the refrigerant's heat carrying capacity is higher

than that of air, eventually decreasing the environmental losses. This has the potential to reduce datacenter energy costs and also its carbon footprint & its environmental impact.

1.1.2. Two-Phase Refrigerant Cooling Vs Single-Phase Water Cooling

Compatibility with Electronics: The combination of water and electronics is risky while refrigerants being dielectric fluids will keep the electronics safe even in the case of leakage. The earlier generation super computers CRAY-2 & CRAY T90 used pool boiling as the main heat transfer mechanism by submerging their electronics in refrigerants. Also, IBM has been using two-phase cold plates for cooling their Z-series mainframes.

Material Compatibility: In the refrigeration and air-conditioning industry, refrigerants have a long and successful history with well-known compatibility with various materials without being corrosive. Refrigerants are mostly coupled with copper and aluminum, with some refrigeration systems known for running over 30 years. On other hand the use of water is not long-running, as it attacks metallic components unless treated.

Organic/Fouling: Water tends to degrade the heat transfer performance of cold plate because it mobilizes the growth of organic matter and tends to foul its containment by blocking the micro-channels and rendering the micro-channel cooler unreliable. Refrigerants do not cause any organic growth or fouling.

Erosion: When shear stress is exerted by the fluid flowing on the wall of a channel it leads to the occurrence of erosion. Guidelines suggest that water velocities should be kept below 0.5-1.2 m/s for water flowing inside copper pipes, depending on the temperature of water. Erosion can degrade the performance of fins in a micro-channel cooler by thinning of the pipe. Also erosion leads to contamination of the fluid with particles, which need to be filtered out, before they have an adverse effect on the fluid pumping pump. Erosion is a main concern while designing because for proper cooling of electronics water needs to flow at high mass flow rates. Refrigerants require low flow rates and thus not causing any erosion.

Harsh Climates: The freezing point of water is 0 C which creates a major problem in the winter on most places on earth to deploy water cooled systems. As refrigerants have a very low freezing temperatures ($< -100^{\circ}\text{C}$) they are ideal for harsh winter conditions.

Heat Dissipation: Another advantage of using two-phase refrigerants is because of their ability of heat dissipation to higher temperatures using the cooling cycle of vapor compression. Thus, maintaining the chip at optimum operating temperature and thus increasing the life cycle of electronic components in harsh operating conditions.

1.1.3. Alternative Refrigerants

With the advancement in cooling technologies utmost importance while designing new one's is given towards its impact on the environment. The two main factors generally being taken under consideration are ozone depleting potential (ODP) and global warming potential (GWP) and become the deciding criteria's for the development of new refrigerants apart from CFC refrigerants because of their contribution towards ozone layer depletion and global warming. The main advantages of hydrocarbon refrigerants are that they are environmentally friendly, non-toxic and non-ozone depleting replacement over chlorofluorocarbons (CFCs), hydro-chlorofluorocarbons (HCFCs) and hydro-fluorocarbons (HFCs). Looking from a purely chemical perspective, a hydrocarbon (HC) is a naturally occurring simplest organic compound, consisting entirely of hydrogen and carbon. It is mostly found in crude oil, where the decomposition of organic matter provides lot of hydrogen and carbon. At present HFC's are the major type of refrigerants to replace CFCs and HCFCs in refrigeration and air-conditioning industry. GWP of HFCs is lower than CFCs but it is much higher than HCs which is why they are considered the major in place of HFCs [7].

Natural refrigerants occur in nature's biological and chemical cycles without human intervention. Natural refrigerants include a range of organic and inorganic compounds. These materials include ammonia, carbon dioxide and natural hydrocarbons. Due to the advantages of natural refrigerants they have been

significantly used in the recent years in applications mostly served by fluorocarbons.

Du-Pont/Honeywell Inc. co-developed a new refrigerant R1234yf which is considered to be a potential substitute of R134a. This fluid has a “Global Warning Potential” of only 6 against 1410 of R134a, i.e., it is considered as an immediate/future replacement for R134a. Both HFC134a and HFC1234yf are dielectric fluids and thus compatible with electronics.

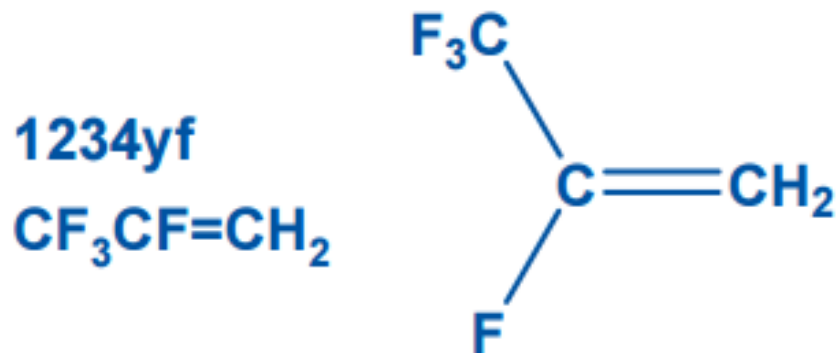


Figure 1-3: Chemical formula/structure of R1234yf

1.2. Refrigerant Vapor Compression Cycle

1.2.1. Theory

A two phase on-chip micro-channel basically follows a regular vapor compression refrigeration cycle. The micro-channel plays the role of evaporator in the cycle. The liquid-vapor mixture which will be at a low pressure coming from the pump/metering device will enter the micro-channel evaporator coil. Now, through the micro-channel evaporator coil, the refrigerant will pass. While doing this, it will absorb the heat flux given out by the micro-processor chips. Now upon this heat absorption, the refrigerant will undergo phase change. Now as the vapor undergoes phase change, due to the latent heat of evaporation of the refrigerant more heat will be absorbed. In a single phase, only the sensible heat takes place the heat flux absorption process. Thus as the latent heat of absorption is very much higher than the sensible heat, the refrigerant will absorb a lot more heat flux given out by the micro-processor chips than that due sensible heat of a single phase. The liquid will get evaporated into vapor phase and it will exit the evaporator. The vapor will be at low pressure while leaving the evaporator. The low pressure vapor will go through a compressor. At the compressor stage, the low pressure vapor will be converted to high pressure vapor. Now this high pressure vapor will undergo the condensation process at the condenser. At the condenser, this high pressure vapor gets condensed to high pressure liquid. This takes place because of the cooling effect from some outside ambient air discharge.

The high pressure liquid then passes through a metering device/pump where it is again converted to a low pressure liquid/vapor mixture. This liquid/vapor mixture again enters the evaporator/micro-channel and thus completing the vapor compression cycle for a refrigerant.

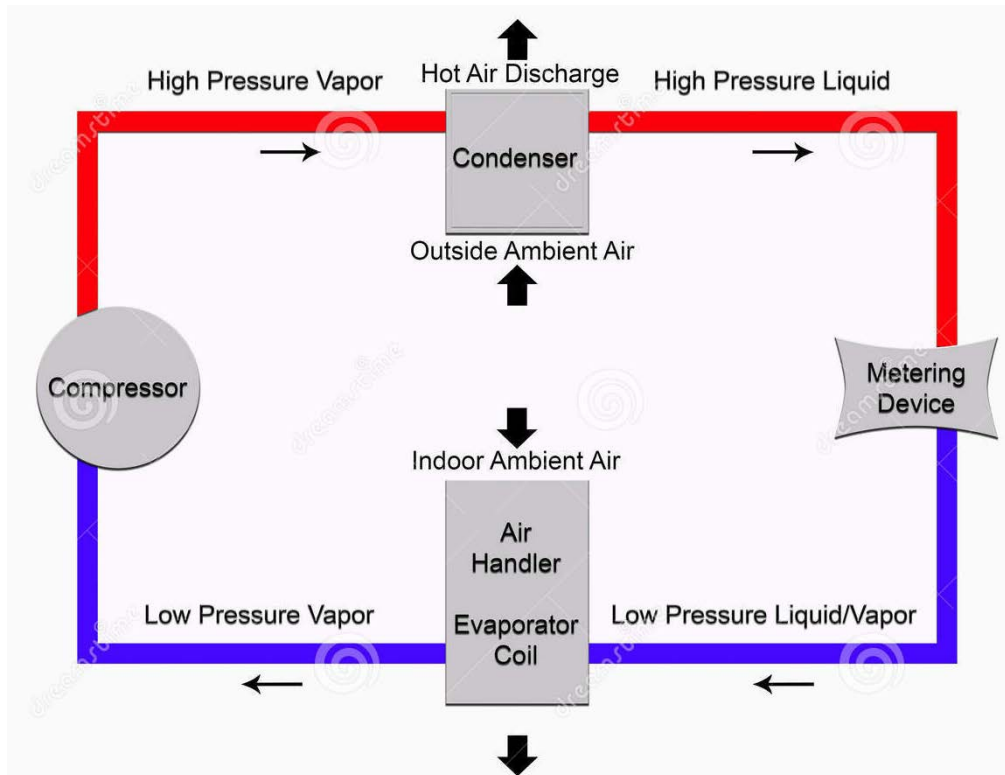


Figure 1-4: Typical Refrigerant Vapor Compression Cycle

(<http://thumbs.dreamstime.com/z/basic-refrigeration-cycle-26303864.jpg>)

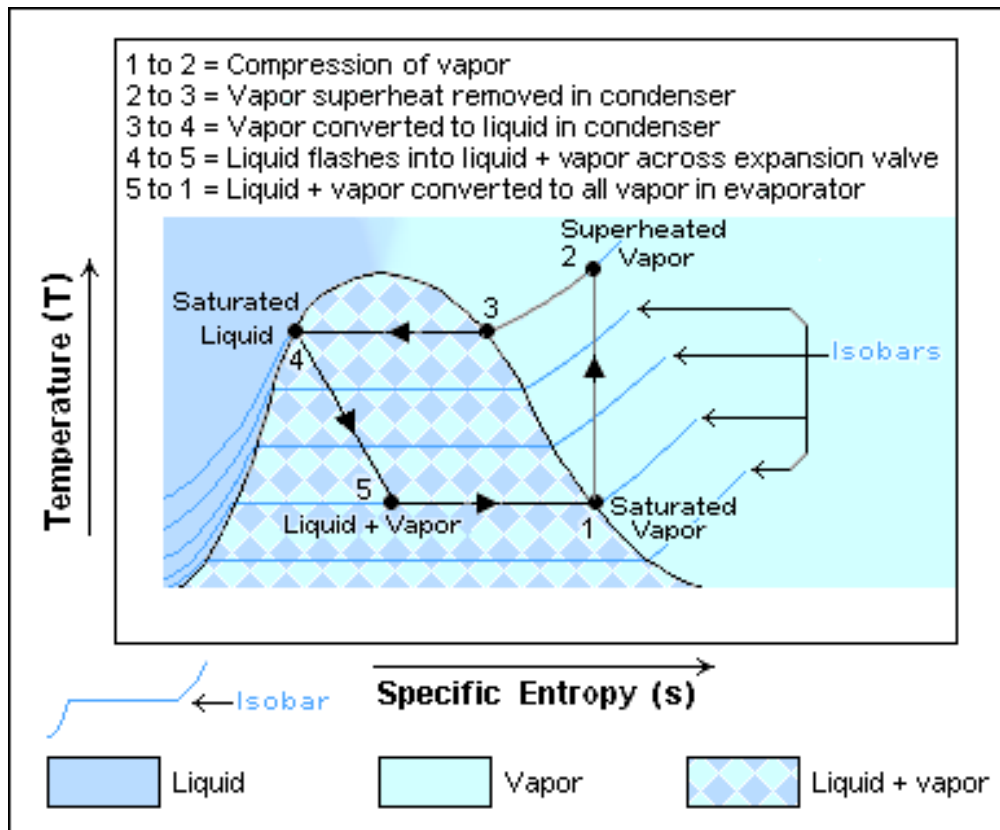


Figure 1-5: Temperature-Entropy diagram for Vapor Compression Cycle

(https://en.wikipedia.org/wiki/Heat_pump_and_refrigeration_cycle#/media/File:RefrigerationTS.png)

Figure 1-4 and 1-5 properly explain the theory and thermodynamics of the refrigerant vapor compression cycle.

1.2.2. Importance of Two Phase Pressure Drop Measurement

From the theory of the refrigerant vapor compression cycle, the importance of properly designing the condenser and evaporator is exemplified. Now for the proper design of the condenser/evaporator, the most important parameter to be analyzed is the pressure drop taking across the length of the condenser/evaporator. Pressure losses occur in two-phase flow systems due to friction, acceleration and gravitational effects. If a fixed flow is required, then the pressure drop determines the power input of the pumping system. If the available pressure drop is fixed, the relationship between velocity and pressure drop needs to be invoked in order to predict the flow rate. If a condenser/evaporator is inaccurately designed with an under prediction two-phase pressure drop, then the efficiency of the system will hamper from the more than expected fall in saturation temperature and pressure through the condenser/evaporator. Now if the pressure drop is over predicted then less number of tubes of longer length could have been used to get a small unit. Thus an accurate prediction of two-phase pressure drops is an important aspect in the first law and second law optimization of these systems. Pressure drop in two-phase flow is also a major design variable because it governs the pumping power required to transport two-phase fluids and also the recirculation rate in natural circulation systems. Thus due to all the above reasons the proper prediction of pressure drop is of utmost importance while designing a condenser/evaporator.

1.3. Multi-Phase Flows

1.3.1. Theory

In fluid mechanics, two-phase flow occurs in a system containing gas and liquid with a meniscus separating the two phases. Two-phase flow is a particular example of multiphase flow. Two-phase flow can occur in various forms. For example, there are transient flows with a transition from pure liquid to a vapor flow as a result of external heating, separated flows and dispersed two-phase flows where one phase is present in the form of particles, droplets, or bubbles in a continuous carrier phase (i.e. gas or liquid) [8]

1.3.2. Examples and Applications

Historically, probably the most commonly studied cases of two-phase flow are in large-scale power systems. Coal and gas-fired power stations used very large boilers to produce steam for use in turbines. In such cases, pressurized water is passed through heated pipes and it changes to steam as it moves through the pipe. The design of boilers requires a detailed understanding of two-phase flow heat-transfer and pressure drop behavior, which is significantly different from the single-phase case. Even more critically, nuclear reactors use water to remove heat from the reactor core using two-phase flow. A great deal of study has been performed on the nature of two-phase flow in such cases, so that engineers can

design against possible failures in pipework, loss of pressure, and so on (a loss-of-coolant accident (LOCA)). [8]

Another case where two-phase flow can occur is in pump cavitation. Here a pump is operating close to the vapor pressure of the fluid being pumped. If pressure drops further, which can happen locally near the vanes for the pump, for example, then a phase change can occur and gas will be present in the pump. Similar effects can also occur on marine propellers; wherever it occurs, it is a serious problem for designers. When the vapor bubble collapses, it can produce very large pressure spikes, which over time will cause damage on the propeller or turbine. [8]

The above two-phase flow cases are for a single fluid occurring by itself as two different phases, such as steam and water. The term 'two-phase flow' is also applied to mixtures of different fluids having different phases, such as air and water, or oil and natural gas. Sometimes even *three*-phase flow is considered, such as in oil and gas pipelines where there might be a significant fraction of solids. [8]

Other interesting areas where two-phase flow is studied includes in climate systems such as clouds, and in groundwater flow, in which the movement of water and air through the soil is studied. Other examples of two-phase flow include bubbles, rain, waves on the sea, foam, fountains, mousse, cryogenics and oil slicks. [8]

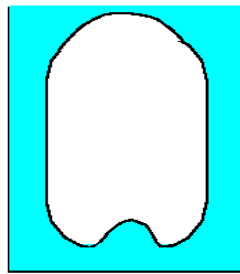
1.3.3. Characteristics of Two-Phase Flow

Several features make two-phase flow an interesting and challenging branch of fluid mechanics: [8]

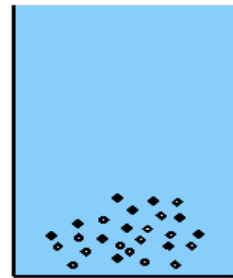
- Surface tension makes all dynamical problems nonlinear.
- In the case of air and water at standard temperature and pressure, the density of the two phases differs by a factor of about 1000. Similar differences are typical of water liquid/water vapor densities.
- The sound speed changes dramatically for materials undergoing phase change, and can be orders of magnitude different. This introduces compressible effects into the problem.
- The phase changes are not instantaneous, and the liquid vapor system will not necessarily be in phase equilibrium.
- The change of phase means flow-induced pressure drops can cause further phase-change (e.g. water can evaporate through a valve) increasing the relative volume of the gaseous, compressible medium and increasing exit velocities, unlike single-phase incompressible flow where closing a valve would decrease exit velocities.

1.3.4. Multi-Phase Flow Regimes

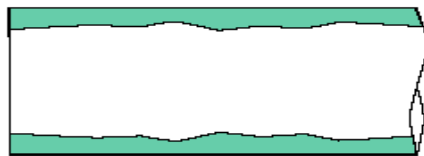
- Bubbly flow: discrete gaseous bubbles in a continuous liquid.
- Droplet flow: discrete fluid droplets in a continuous gas.
- Particle-laden flow: discrete solid particles in a continuous fluid.
- Slug flow: large bubbles in a continuous liquid.
- Annular flow: continuous liquid along walls, gas in core.
- Stratified and free-surface flow: immiscible fluids separated by a clearly-defined interface.



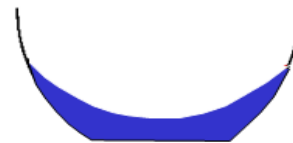
Slug flow



Bubbly flow
Droplet flow particle-laden
flow



Annular flow



free-surface flow

Figure 1-6: Various Multi-Phase Flow Regimes

The void fraction ϵ is an important factor used in characterization of two phase flows. It is most important physical value for determining various other parameters like, two phase viscosity and density and for finding out the relative average velocity of the two phases. Also it of fundamental importance in models for predicting heat transfer, pressure drop and transmissions in flow patterns. The multi-phase flow regimes depending on high void fractions were observed by *Ewing et al [1999]* and are shown in Figure: 1-7

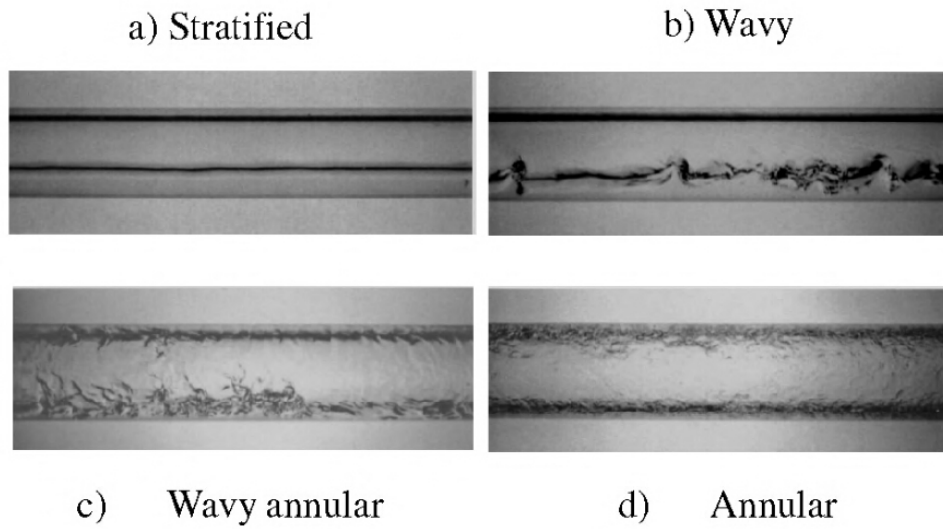


Figure 1-7: Flow Regimes at high Void Fractions of air-water mixture (“Ewing et al [1999]”)

Chapter 2

LITERATURE REVIEW

2.1. General

[9] In internal condensation there is simultaneous flow of liquid and vapor inside the condenser pipe. And thus the two phase resulted is much more physically complicated than a simple single phase flow. The single phase flow is only affected by pressure, inertia and viscous forces. On the other hand, the two phase flows are also affected by the wetting physiognomies of the liquid on the tube wall, the interfacial tension forces and the exchange of momentum between the liquid and vapor phases in the flow. And due to these effects the two phase flow pattern morphology varies according to the orientations and geometries of the tubes and channels under consideration. Horizontal tube condensation is guided by the both gravity and interfacial shear stresses and this combination changes with the change in geometry and flow pattern. Thus analysis of general in nature if two phase flow is complicated in nature.

A general one dimensional flow of one component two phase flows especially for stratified flow regime is developed where there is a common interface and both the phases are in contact with the channel. The momentum equation and energy equations resulted are solved for separated flow model and the special case of separated flow model i.e. the homogeneous separated flow model.

The separated flow model was initially used for prediction of two phase frictional pressure drop for an isothermal flow in horizontal pipe by Lockhart and Martinelli in 1944 [10] by using two-phase multiplier. Later to cover the acceleration component the Martinelli-Nelson correlation was developed for forced circulation boiling and condensation pressure drop measurement. Further on, Thom [10], Baroczy [10] and Chisholm [10] developed the method for calculating the two phase friction multiplier.

Recently, Muller-Steinhagen and Tribbe [11] combined and presented a massive 35 two phase pressure drop correlations results from predictive methods for air-water and several refrigerants. It was discovered that the Muller-Steinhagen and Heck [12] correlations were giving the most consistent and reliable results. In the Engineering Data book 3, Wolverine Tube Inc. [13] the Gronnerud and the Muller-Steinhagen and Heck correlations were described as the best. The Friedel correlation came third in comparison to the other correlations. The recommendations made in the book are as follows:

- $\left(\mu_l/\mu_g\right) < 1000 \ \& \ G < 2000 \frac{kg}{m^2s}$, *Friedel Correlation be used*
- $\left(\mu_l/\mu_g\right) > 1000 \ \& \ G > 100 \frac{kg}{m^2s}$, *Chisholm correlation be used*
- $\left(\mu_l/\mu_g\right) > 1000 \ \& \ G < 100 \frac{kg}{m^2s}$, *Lockhart – Martinelli be used*

Kattan et al. [14] differentiated the data according to the flow regimes by making use of flow pattern map. They found that the flow regimes are not working in sync with the predictive methods and thus not able to say the whole story if the flow regimes changed. Moreno, Quiben and Thome [15, 16] work comprises of an expansive study of experimental analysis and then by using the new pattern flow map by Wojtan et al. [17] constructed a new flow pattern model for pressure drop predictions.

2.2. Separated Flow Model

The equations of a separated flow model do not rely on any specific flow pattern. Also the velocities of the phases are considered to be constant.

Condensation (In-Tube) pressure drop is obtained from the momentum equation [9] depending upon the separated flow model. The each phase velocities are assumed to be constant at any zone occupied by the phase. Now using the momentum equation for two phase flow, the horizontal tube condensation equation for pressure drop can be calculated as:

$$\begin{aligned}
 -\left(\frac{dp}{dz}\right) &= -\left(\frac{dp}{dz}\right)_f - \left(\frac{dp}{dz}\right)_a - \left(\frac{dp}{dz}\right)_z \\
 -\left(\frac{dp}{dz}\right)_z &= g[\epsilon\rho_g + (1 - \epsilon)\rho_l] \\
 -\left(\frac{dp}{dz}\right)_a &= G^2 d \left\{ \left(\frac{x^2}{\rho_g \epsilon}\right) + \left(\frac{1 - x^2}{\rho_l (1 - \epsilon)}\right) \right\} / dz
 \end{aligned}$$

Here, $-\left(\frac{dp}{dz}\right)_z$ is gravity pressure drop component, $-\left(\frac{dp}{dz}\right)_a$ is acceleration pressure drop gradient and $-\left(\frac{dp}{dz}\right)_f$ is frictional pressure drop gradient.

Now for horizontal tubes the gravity pressure drop component is not applicable as it is only applicable to long vertical tubes. The acceleration pressure drop gradient develops only if there is pressure increase at the exit than the inlet. In the case of flow which is condensing in nature the outlet flow has less kinetic energy than that of the inlet. Thus the acceleration component of pressure gradient is neglected. These exclusions provide for certain level of design conservation.

Also, for calculation of gravity and acceleration pressure drop the void fraction data is of requirement. Now because of high vapor density on the condenser due to very high pressure, if the vapor velocity is compared at any given vapor quality and mass flux, it will be always less than that of on the evaporator. Thus a stratified flow regime will develop in the condenser bringing the flow closer to one another. The results evaluated from any common void fraction model in this stratified region would be inaccurate. Thus the only consideration of frictional pressure drop is further more justified.

[18] The condensation heat transfer coefficients experimental values and the pressure drop experimental values obtained in such a way in recent literature are quasi local, obtained when fewer variations in quality occur in the testing tube. The experimental process aids in knowing the condensation phenomenon. The process takes place in such a way that the part of condensation takes place in the

testing section. The data is obtained for average quality of liquid/vapor properties in the testing section. This data from the experiment is used in the present study for comparison with the CFD results and data from the various correlations for refrigerants R134a and R407C. On the basis of this evaluation the CFD process is further used for refrigerant R1234yf to predict the pressure drop and pumping power in comparison with R134a. The experimental analysis is conducted in horizontal condenser tube of ID = 8.5mm and length of tube = 1.2mm.

2.3. Analytical Correlations

2.3.1. Lockhart-Martinelli Correlation

This method is the original method which predicted the two-phase frictional pressure drop based on the two-phase multiplier for both the vapor and liquid phases,

$$\Delta P_{frict} = \phi_{Ltt}^2 \Delta p_L$$

$$\Delta P_{frict} = \phi_{Ltt}^2 \Delta p_G$$

Equation,

$$\Delta P_L = 4f_L \left(\frac{L}{d_i} \right) \dot{m}_{total}^2 \left(\frac{1}{2\rho_L} \right) \text{ Is used for } \Delta P_L \text{ with } (1-x)^2$$

multiplied with the mass velocity term and Δp_G Is obtained as,

$$\Delta P_G = 4f_G \left(\frac{L}{d_i} \right) \dot{m}_{total}^2 x^2 \left(\frac{1}{2\rho_G} \right)$$

Now, f_L and f_G are obtained from, $f = \frac{0.079}{Re^{0.25}}$, The respective two phase multipliers are given as,

$$\phi_{Ltt}^2 = 1 + \frac{C}{X_{tt}} + \frac{1}{X_{tt}^2}, \text{ for } Re_L > 4000$$

$$\phi_{Gtt}^2 = 1 + CX_{tt} + X_{tt}^2, \text{ for } Re_L < 4000$$

Where, X_{tt} is the Martinelli parameter for both the phases given as,

$$X = \left(\frac{1-x}{x} \right)^{0.9} \left(\frac{\rho_G}{\rho_L} \right)^{0.5} \left(\frac{\mu_L}{\mu_G} \right)^{0.1}$$

2.3.2. Gronnerud Correlation

$$\Delta P_{frict} = \phi_{gd} \Delta P_L$$

And the two phase multiplier is given as,

$$\phi_{gd} = 1 + \left(\frac{dp}{dz} \right)_{Fr} \left[\frac{\left(\frac{\rho_L}{\rho_G} \right)}{\left(\frac{\mu_L}{\mu_G} \right)} - 1 \right]$$

Frictional pressure drop gradient is given as,

$$\left(\frac{dp}{dz} \right)_{Fr} = f_{Fr} [x + 4(x^{1.8} - x^{10} f_{Fr}^{0.5})]$$

2.3.3. Muller-Steinhagen & Heck Correlation

$$\left(\frac{dp}{dz} \right)_{frict} = G(1-x)^{\frac{1}{3}} + Bx^3$$

Where the factor G,

$$G = A + 2(B - A)x$$

Where A and B are frictional gradients for the liquid.

2.4. Mathematical Formulation

[9] The separated flow model consists of a special case where the vapor and liquid velocities are not only considered to be constant but also equal. This model is known as the homogeneous separated flow model. This model converts the two phase flow into a single phase pseudo fluid flow. The properties are taken to be averaged form both the phases. The basic equations for horizontal tube condensation from the steady homogeneous flow model in the reduced form are given as:

$$\textit{Continuity Equation: } \dot{m} = \bar{\rho}\bar{u}A$$

$$\textit{Momentum Equation: } -Adp - d\bar{F} - A\bar{\rho}gdz = \dot{m}d\bar{u}$$

Here, the total walls shear force $d\bar{F}$ in terms of wall shear stress, τ_w acting over the inside area of the tube can be expressed as,

$$d\bar{F} = \tau_w(Pdz)$$

The two phase friction factor for the averaged properties is given by the Blasius equation as,

$$f_{TP} = 0.079 \left[\frac{Gd}{\bar{\mu}} \right]^{-0.25}$$

The averaged properties taken for the pseudo single phase fluid from the liquid and vapor properties are given by Collier [9].

The averaged fluid density given as:

$$\frac{1}{\bar{\rho}} = \left[\frac{x}{\rho_g} + \left(\frac{1-x}{\rho_l} \right) \right]$$

Then for mean two phase viscosity, $\bar{\mu}$ based on limiting conditions, at $x=0$, $\mu_l = \bar{\mu}$ and at $x=1$, $\mu_g = \bar{\mu}$ are:

$$\frac{1}{\bar{\mu}} = \left[\frac{x}{\mu_g} + \left(\frac{1-x}{\mu_l} \right) \right] \text{ (McAdams)}$$

$$\bar{\mu} = x\mu_g + (1-x)\mu_l \text{ (Cicchitti)}$$

$$\bar{\mu} = \bar{\rho} \left[\frac{x\mu_g}{\rho_g} + \frac{(1-x)\mu_l}{\rho_l} \right] \text{ (Duker)}$$

Now the frictional pressure drop equation which is the Fanning equation is given from the above averaged properties for a horizontal tube of internal diameter d as:

$$\Delta P = \frac{2f_{TP}G^2L}{\bar{\rho}d}$$

Chapter 3

COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS

3.1. Introduction to CFD Analysis (FLUENT-ANSYS)

CFD is a branch of Fluid Dynamics which deals with the analysis of problems involving fluid flow and heat transfer. It uses numerical methods and algorithms to solve and analyze problems. Computational fluid dynamics is applied to simulate and analyze the behavior of fluids in various systems. The major advantage of numerical methods is that, the problem is discretized based on certain parameters and solved. A mathematical model is generated, which represents to actual physical system and then it can be solved and analyzed. CFD is concerned with the numerical simulation of fluid flow, heat transfer and related processes such as radiation. The objective of CFD is to provide the engineer with a computer-based predictive tool that enables the analysis of the air-flow processes occurring within and around different equipment, with the aim of improving and optimizing the design of new or existing equipment.

In this case the study the CFD solver FLUENT by ANSYS is used. This study involves the two phase flow analysis of refrigerants like R134a, R407C and R1234yf entering a horizontal condenser tube at saturation temperature. The CFD analysis is performed to find out the pressure drop across the length of the horizontal condenser tube. This pressure drop prediction is important in order to

predict the pumping power required by the compressor to pump the high pressure vapor onto the horizontal condenser tube. Adiabatic conditions are considered on the wall of the horizontal condenser tube for simplicity in calculations. The two phase flow is also converted into single phase pseudo flow by averaging out the liquid and vapor properties at that particular saturation temperature. The pressure drop results from the CFD analysis are then compared with the correlations mentioned in the literature review section and experimental results [18], to access who predicted the result nearer to the experimental results.

3.2. Governing Equations

The numerical solution for most problems are obtained by solving a series of three differential equations, collectively referred to as the Navier-Stokes' Equations. These differential equations are the conservation of mass, conservation of momentum and conservation of energy.

But in this particular case, adiabatic conditions are considered on the wall of the condenser and the effect of flow is analyzed. Hence, only conservation of mass and conservation of momentum equations are solved.

In general form,

The conservation of mass is given by:

$$\frac{\partial(\rho)}{\partial t} + \nabla \cdot (\rho u) = 0$$

The conservation of momentum is given by:

$$\frac{\partial(\rho u)}{\partial t} + (\rho u \cdot \nabla)u = \nabla \cdot (\mu \nabla u) - \nabla p + \rho f$$

The solution domain is the region or space within which these differential equations are solved. The solutions are obtained by imposing certain boundary conditions for this solution domain. The boundary conditions for most problems include ambient temperature, pressure, wind conditions and other environmental conditions. Also, if there is heat transfer involved then, type of heat transfer, such as conduction, convection or even radiation are considered. The conditions at the domain wall are also specified, whether they are open, closed or symmetrical in nature. The fluid properties like density, viscosity, diffusivity and specific heat need to be specified.

The governing equations for many problems are solved using numerical techniques like Finite Element Method, Finite Volume Method and Finite Difference Method. In FEM, the elements are varied and approximated by a function, in FVM the equations are integrated around a mesh element whose volumes are considered and in FDM the differential terms are discretized for each element.

In the CFD technique used in FLUENT, the conservation equations are discretized by sub-division of the domain of integration into a set of non-overlapping, continuous finite volumes referred to as 'grid cells', 'control cells' or quite simply as 'cells'. The governing equations are solved by considering the

volume of the grid cells and the variables to be calculated are situated at the center of these grid cells.

The finite volume method is more advantageous than other computational methods as it the governing equations are conserved even on coarse grids and it also does not limit cell shape. A set of algebraic equations are used for discretizing the results, each of which relates the value of a variable in a cell to its value in the nearest-neighbor cells.

For example let T denote the temperature, this can be calculated using the algebraic equation:

$$T = \frac{C_0 T_0 + C_1 T_1 + C_2 T_2 + \dots + C_n T_n + S}{C_0 + C_1 + C_2 + \dots + C_n}$$

Where T_0 represents temperature value in the initial cell, T_1, T_2, \dots, T_n are values in the neighboring cells; $C_0, C_1, C_2, \dots, C_n$ are the coefficients that link the in-cell value to each of its neighbor-cell values. S denotes the terms that represent the influences of the boundary conditions.

These algebraic equations are solved for the variables like T, u, v, w and p . This means that if there are 'n' cells in the solution domain, a total of '5n' equations are solved.

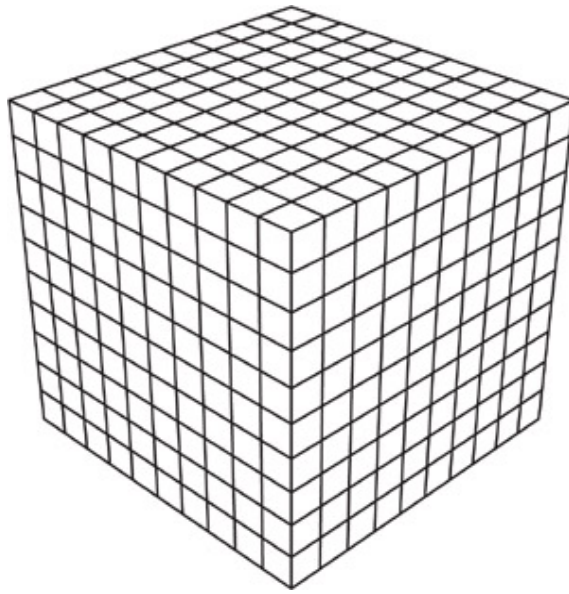


Figure 3-1: Representation of a 3D grid

3.3. Turbulence Modeling (Standard k-epsilon model)

[19] A flow is said to be turbulent when the fluid undergoes irregular fluctuations or mixing. The velocity of the fluid at a point is continuously undergoing changes in both magnitude and direction, as opposed to laminar flow wherein the fluid moves in smooth paths or layers. Usually fluid with large Reynolds number are considered to be turbulent, while fluids with low Reynolds number are considered laminar. ANSYS FLUENT uses various methods for turbulence modelling. In this case, the standard k-epsilon method was selected for better accuracy at the cost of less computational time.

3.3.1. K-Epsilon Turbulence Model

Two-equation turbulence models allow the determination of both, a turbulent length and time scale by solving two separate transport equations. The standard k-epsilon model in ANSYS Fluent falls within this class of models and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism.

The standard k-epsilon model is a model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ϵ). The model transport equation for k is derived from the exact equation, while the model transport equation for ϵ was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart.

In the derivation of the k - ϵ model, the assumption is that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard k - ϵ model is therefore valid only for fully turbulent flows.

The transport equations for the standard k-epsilon model are given as,

For turbulent kinetic energy k ,

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k + P_b - \rho \epsilon - Y_M + S_k$$

For dissipation ε ,

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (P_k + C_{3\varepsilon} P_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon$$

In these equations, P_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, P_b is the generation of turbulence kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{3\varepsilon}$ are constants. σ_k and σ_ε are the turbulent Prandtl numbers for k and ε respectively. S_k and S_ε are user-defined source terms.

The turbulent (or eddy) viscosity, μ_t , is computed by combining k and ε as follows;

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

Where C_μ is a constant.

The model constants $C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$, σ_k and σ_ε have the following default values:

$$C_{1\varepsilon} = 1.44, \quad C_{2\varepsilon} = 1.92, \quad C_\mu = 0.09, \quad \sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3$$

These default values have been determined from experiments for fundamental turbulent flows including frequently encountered shear flows like boundary layers, mixing layers and jets as well as for decaying isotropic grid turbulence. They have been found to work fairly well for a wide range of wall-bounded and free shear flows.

Chapter 4

DESIGN & ANALYSIS METHODOLOGY

The Computational Fluid Dynamics model is of a horizontal condenser pipe. CFD modelling was done by considering adiabatic conditions on the condenser pipe wall. The model was created in ANSYS Space-Claim. Meshed in ANSYS Meshing with Mesh Sensitivity Analysis performed. The CFD analysis was done in ANSYS FLUENT by considering a homogeneous separated flow model.

4.1. Geometry Specifications

The horizontal condenser pipe is of inner diameter 8.5mm and of length 1.2m. These geometry details were taken from [18], as the CFD analysis was performed to simulate the results of pressure drop in a horizontal condenser pipe obtained from the experiment performed. ANSYS Space-Claim was used to create the geometry (Figure: 4-1). The Space-Claim model was then imported into ANSYS Meshing with properly marked normally selected surfaces, NS-Inlet and NS-Outlet (Figure: 4-2).

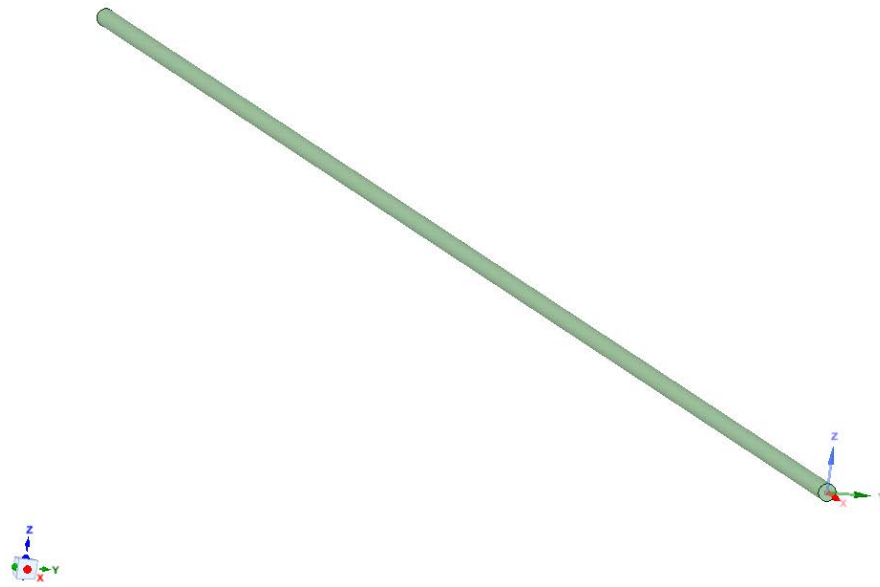


Figure 4-1: ANSYS Space-Claim Model

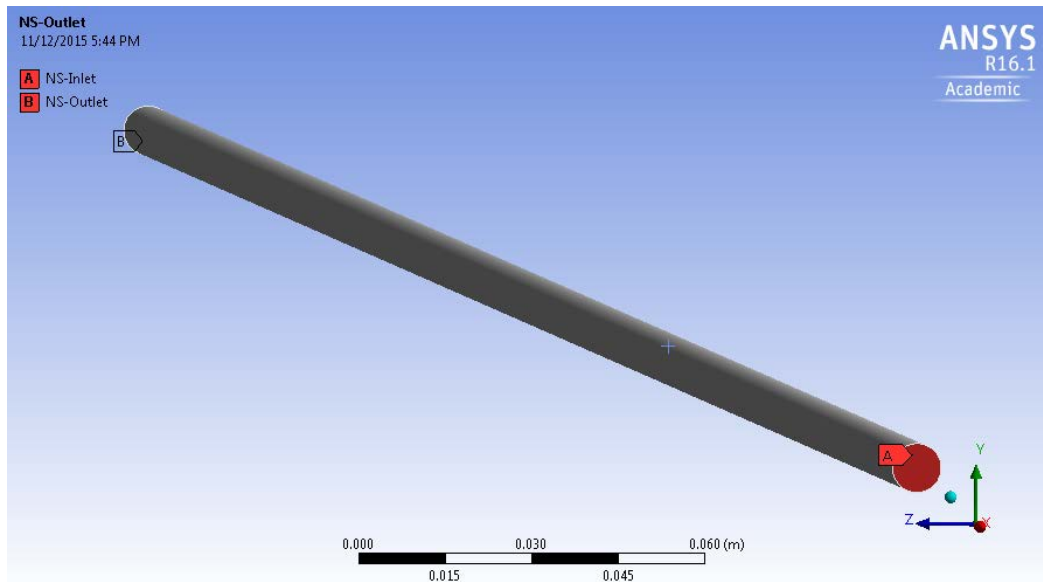


Figure 4-2: ANSYS Meshing Model

4.2. Meshing

The imported model from ANSYS Space-Claim was meshed in ANSYS Meshing with Mesh Sensitivity Analysis performed. The variation of pressure drop was measured at a particular monitor point. The obtained data of pressure drop vs grid size was plotted onto a graph (Figure 4-3) to narrow down on the number of elements to be selected for the optimum grid size. The obtained data from graph was then tabulated (Table 4-1).

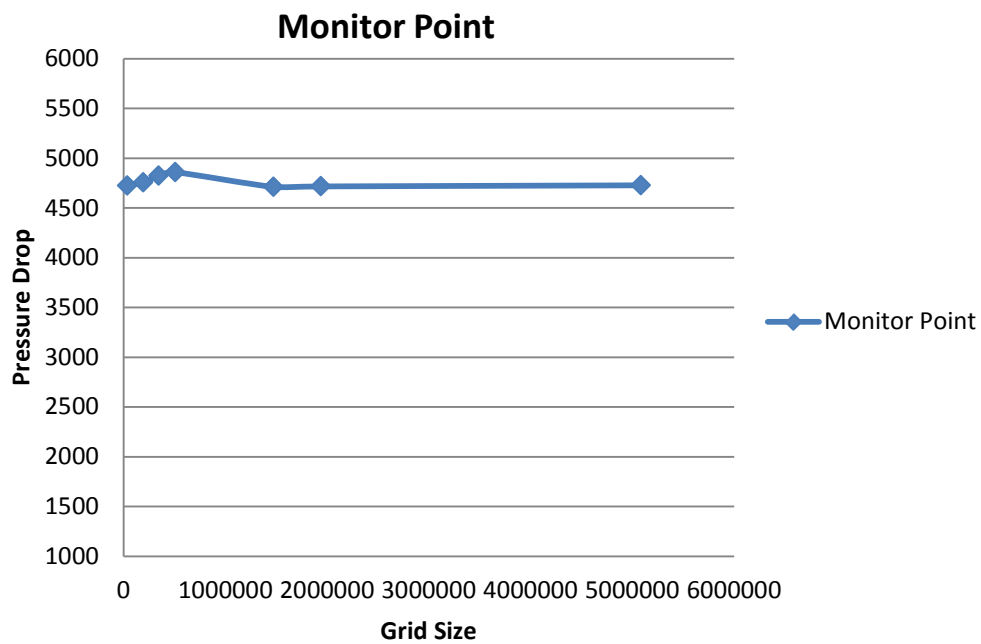


Figure 4-3: Mesh Sensitivity Analysis

Table 4-1: Mesh Sensitivity Analysis

Grid Size	Monitor Point
36859	4725
191812	4758
342573	4825
507276	4859
1474458	4712
1939084	4717
5084000	4728

The grid size was varied from 36859 elements to 5084000 elements. The pressure drop variation was noted down for a particular monitor point. Not much variation was observed across the variation of the grid size. The optimum grid size was selected to be 1474458. And the CFD analysis was done at this grid size. The Meshed model on the Inlet and Outlet and the wall is shown in Figures: 4-4 and 4-5 respectively.

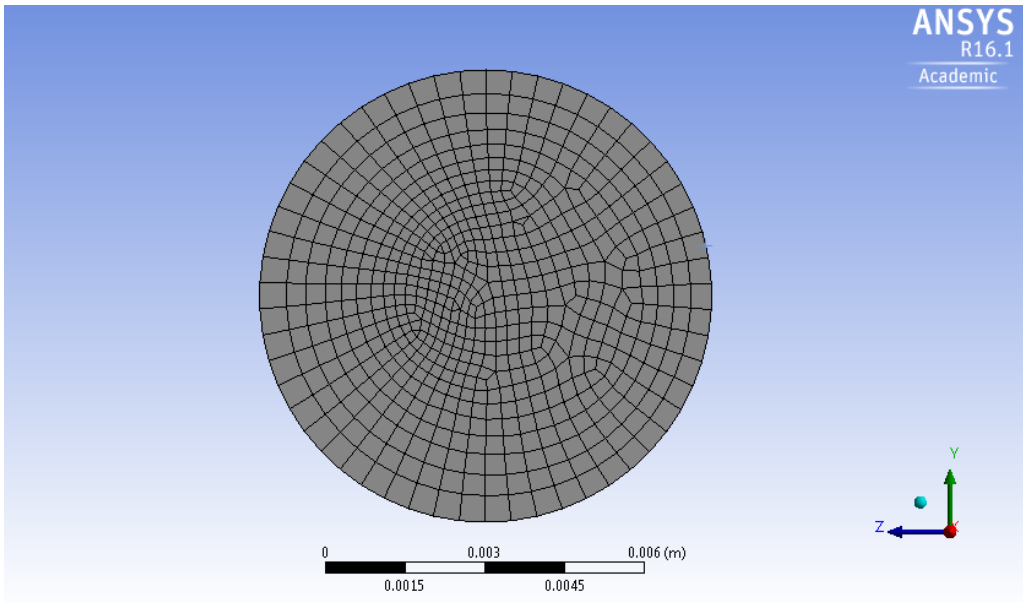


Figure 4-4: Meshed Inlet/Outlet

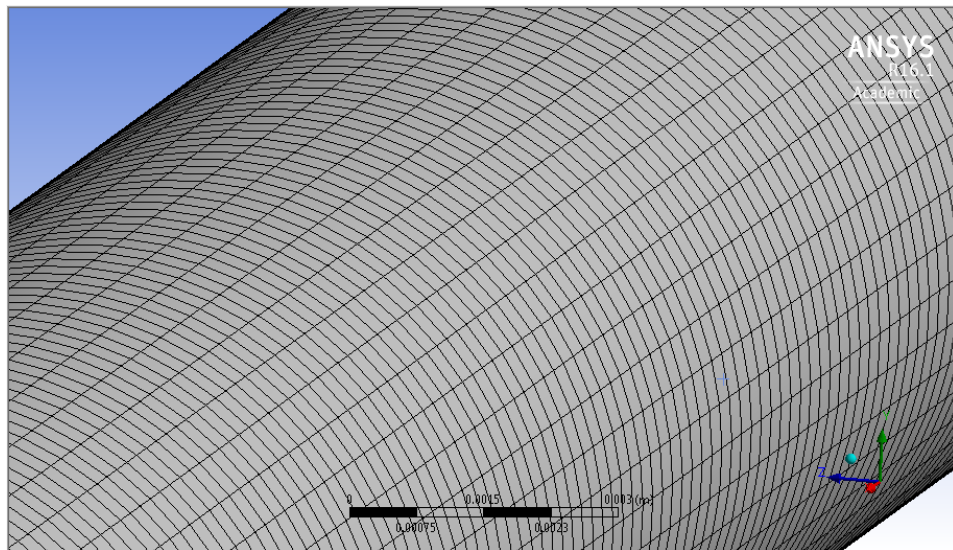


Figure 4-5: Meshed Wall

The Orthogonal Quality of the meshed model selected according to the optimum grid size is:

Statistics	
<input type="checkbox"/> Nodes	1534652
<input type="checkbox"/> Elements	1474458
Mesh Metric	Orthogonal Quality
<input type="checkbox"/> Min	0.73946
<input type="checkbox"/> Max	0.9997
<input type="checkbox"/> Average	0.98577
<input type="checkbox"/> Standard Deviation	2.7428e-002

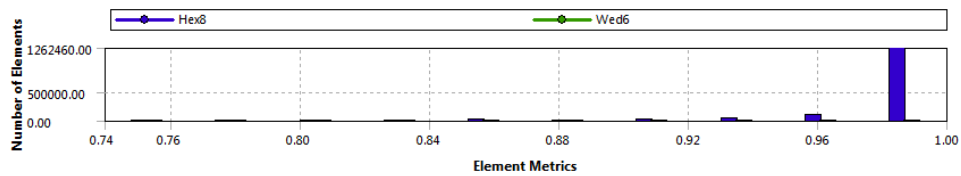


Figure 4-6: Orthogonal Quality of Meshed Model

The orthogonal quality of the meshed model selected according to the optimum grid size is towards 1.00, thus solidifying the mesh quality as optimum.

The skew-ness quality of the meshed model selected according to the optimum grid size is:

Statistics	
<input type="checkbox"/> Nodes	1534652
<input type="checkbox"/> Elements	1474458
Mesh Metric	Skewness
<input type="checkbox"/> Min	1.5025e-002
<input type="checkbox"/> Max	0.45578
<input type="checkbox"/> Average	0.10554
<input type="checkbox"/> Standard Deviation	8.4414e-002

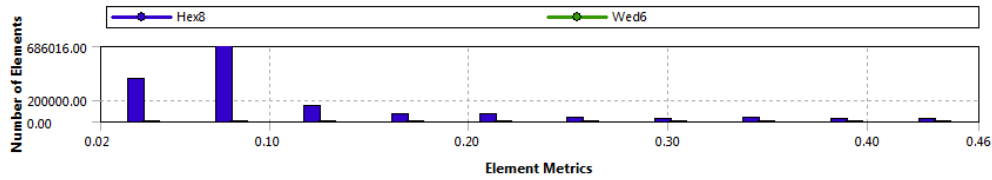


Figure 4-7: Skew-ness Quality of Meshed Model

The Skew-ness Quality of the meshed model selected according to the optimum grid size is very much less than 0.80, thus solidifying the mesh quality as optimum.

4.3. CFD Analysis

The Homogenous Separated Flow model is selected for performing the CFD Analysis. The homogeneous separated flow model is explained in the literature review. The homogenous separated flow model takes into account the averaged flow properties. In this study, the condenser pipe wall is taken at adiabatic conditions. So the properties to be averaged are density and kinematic viscosity. The averaged density is given by a single equation. But to find out the averaged

kinematic viscosity there are three linear kinematic viscosity models. The analytical analysis for finding the pressure drop using the homogenous separated flow model was undertaken to select between the *Cicchitti*, *McAdams*, *Duker* kinematic viscosity model. The kinematic viscosity model with the highest pressure is to be selected.

The liquid and vapor properties of R134a at the saturation temperature of 40 degree Celsius were taken into consideration for performing the analysis.

Liquid & vapor properties of Refrigerant R134a at saturation temperature of 40°C [20]:

$$\rho_l = 1146.7 \text{ kg/m}^3$$

$$\rho_g = 50.085 \text{ kg/m}^3$$

$$\mu_l = 1.42 * 10^{-6} \text{ Pa} - \text{s}$$

$$\mu_g = 2.57 * 10^{-6} \text{ Pa} - \text{s}$$

These averaged properties are then substituted into the average density formula and the three averaged kinematic viscosity models. These values were obtained at various vapor qualities (x). The obtained values are shown in table 4.3.1.

The various averaged properties model equations are shown below:

$$\frac{1}{\bar{\rho}} = \left[\frac{x}{\rho_g} + \left(\frac{1-x}{\rho_l} \right) \right] \text{ (Averaged Density Model)}$$

$$\frac{1}{\bar{\mu}} = \left[\frac{x}{\mu_g} + \left(\frac{1-x}{\mu_l} \right) \right] \text{ (McAdams Averaged Kinematic Viscosity Model)}$$

$$\bar{\mu} = x\mu_g + (1-x)\mu_l \text{ (Cicchitti Averaged Kinematic Viscosity Model)}$$

$$\bar{\mu} = \bar{\rho} \left[\frac{x\mu_g}{\rho_g} + \frac{(1-x)\mu_l}{\rho_l} \right] \text{ (Duker Averaged Kinematic Viscosity Model)}$$

Table 4-2: Averaged Properties of R134a at 40 degree Celsius

X	$\bar{\rho}$	$\bar{\mu}_C * 10^{-6}$	$\bar{\mu}_{MA} * 10^{-6}$	$\bar{\mu}_D * 10^{-6}$
0.1	358.44	148.67	74.24	55.08
0.2	212.41	133.55	48.00	34.95
0.3	150.92	118.42	35.47	26.47
0.4	117.04	103.3	28.12	21.80
0.5	95.58	88.17	23.30	18.84
0.6	80.78	73.04	19.88	16.80
0.7	69.94	57.92	17.35	15.31
0.8	61.67	42.79	15.38	14.17
0.9	55.14	27.68	13.81	13.27

Substituting the averaged properties for R134a at 40°C saturation temperature

(T_s) at flow-rates (G) of 176 & 528 kg/m²s into pressure drop (Fanning) equation

as mentioned in the literature review chapter & plotting the various linear kinematic viscosity models.

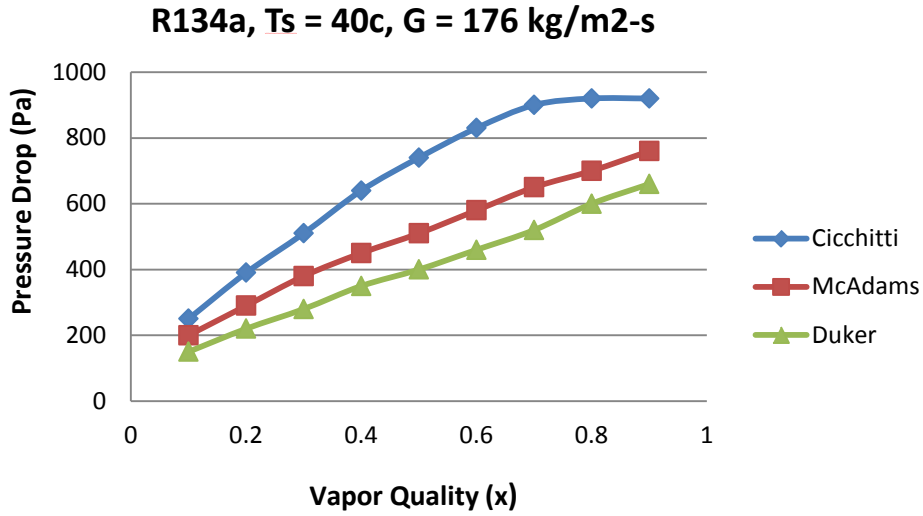


Figure 4-8: Kinematic Viscosity Model Comparison, $G = 176 \text{ kg/m}^2\text{-s}$

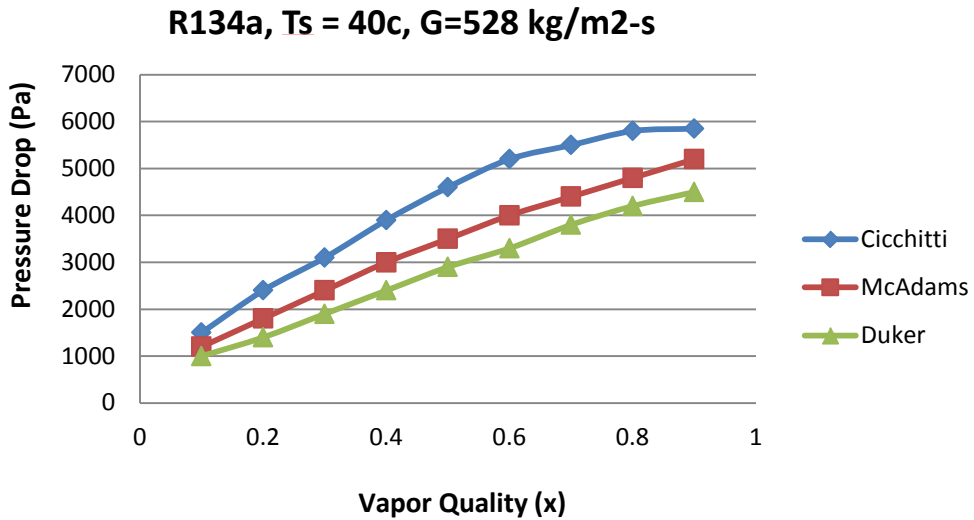


Figure 4-9: Kinematic Viscosity Model Comparison, $G=528 \text{ kg/m}^2\text{-s}$

Form the above mentioned pressure drop results, the Cicchitti model of kinematic viscosity gives higher pressure drop at any vapor quality. So, the Cicchitti model is of kinematic viscosity is selected for CFD analysis.

CFD Modelling procedure:

The CFD analysis was carried out on ANSYS FLUENT. Three refrigerants R134a, R407C and R1234yf were selected for the analysis. The geometry was selected as per the experimental analysis [18]. The steady state pressure based solver with standard K-epsilon model was selected because of low speed incompressible turbulent flow. The enhanced wall function while taking into account the pressure gradient effects was also selected. The averaged material properties were taken from the analytical homogenous separated flow model analysis. The operating pressure was set as the saturation pressure at the saturation temperature considered. The mass flow rate was varied from 0.01 kg/sec to 0.06 kg/sec depending upon the limitations provided from literature to avoid corrosion and other effects. The SIMPLE algorithm for pressure-velocity coupling given by Dr. Suhas Patankar is used. The analysis was set up in the following way:

- Working Fluid: Refrigerant R134a, R407c, R1234yf
- Dimensions: ID of tube = 8.5mm, Length of tube = 1.2m.

- Meshing: Mesh sensitivity analysis performed and optimum mesh size is selected.
- Selection of Solver: 3D segregated solver with implicit formulation is selected. The solver conditions are set for steady flow with enhanced wall treatment. Adiabatic conditions are considered at the wall.
- Selection of Viscous Model: K-epsilon model is considered as the Reynolds number for entire range of flow-rates for averaged properties exceeds 2300.
- Material Properties: Averaged liquid & vapor properties of R134a at a particular saturation temperature are calculated. *Cicchitti* model of kinematic viscosity is used.
- Setting Operating Conditions: The operating pressure is set as per the saturation pressure at considered saturation temperature.
- Applying Boundary Conditions: *Mass Flow Inlet*: 0.01 kg/s to 0.06 kg/s are considered. *Outlet Condition*: Outflow condition is given. *Wall*: No slip condition.
- Solution Controls: *Pressure-Velocity Coupling*: SIMPLE algorithm. For *Gradient discretization*, Least-Square Cell Based scheme is selected. For *pressure discretization*, Standard scheme is selected. Second Order Upwind method is selected for *discretization of momentum, turbulence K.E. & turbulence dissipation rate*.
- Convergence: 10e-03 convergence criteria are considered.

Chapter 5

SCENARIOS CONSIDERED & RESULTS

The CFD results were carried out in two different sets.

- The scenarios considered for the first set of CFD results is first carrying out the analysis with R134a and R407C at saturation temperature of 40 degree Celsius at two different flow-rates of 400 and 600 kg/m²-sec basically to compare the CFD results with the analytical results of 3 correlations (*Gronnerud, Lockhart-Martinelli and Muller-Steinhagen & Heck*) [18] and finally with the experimental results [18]. In order to find whether the CFD procedure better predicts the experimental pressure drop results than the analytical correlations. Thus, in order to verify the validity of the two-phase CFD procedure set up.
- Upon verification of the two-phase CFD method from the results of set 1, the set 2 analysis is carried out where the refrigerant under consideration is the newly developed Du-Pont/Honeywell R1234yf, which is considered to be better environmentally than R134a and thus seen as a potential one to replace R134a. Along with the environmental aspect the commercial aspect of pumping power required is verified by performing a CFD analysis and comparing the pressure drop results with that of R134a at the same conditions. Pumping power can be easily calculated from pressure drop by multiplying it the flow-rate and dividing by the pump efficiency.

5.1. Set 1

5.1.1. Scenario 1

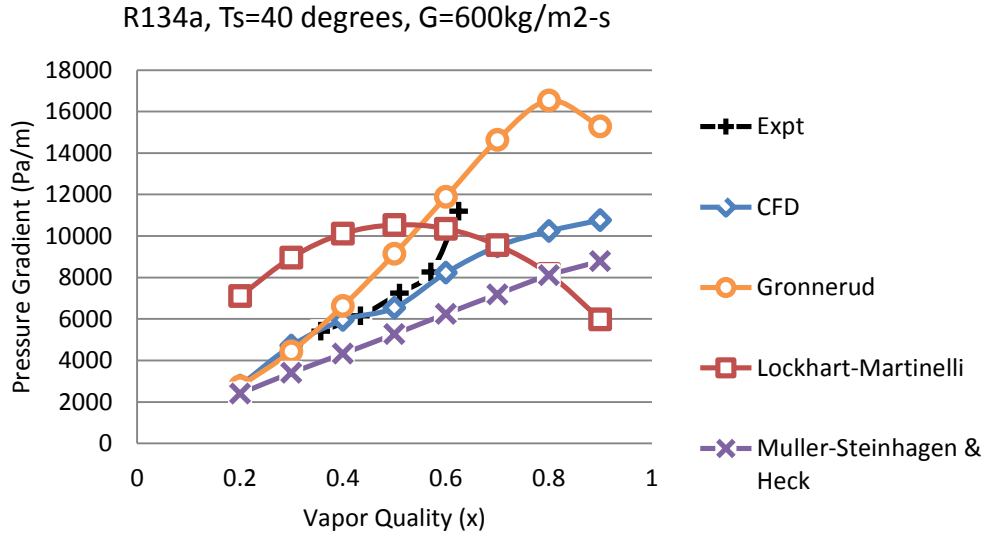


Figure 5-1: Pressure drop comparison, R134a, $T_s = 40^\circ\text{C}$, $G = 600\text{kg/m}^2\text{-s}$

The graph shows a plot of pressure gradient vs vapor quality for the experimental results [18], the CFD results and the correlations result (*Gronnerud*, *Lockhart-Martinelli* and *Muller-Steinhagen & Heck*) [18]. The graphical representation shows that the experimental results line is being best emulated by the CFD results line than any of the correlations result line. Amongst the correlational results, Gronnerud correlation predictions are best after CFD. Lockhart-Martinelli over predicts all the results while Muller-Steinhagen & Heck under predicts all the results.

5.1.2. Scenario 2

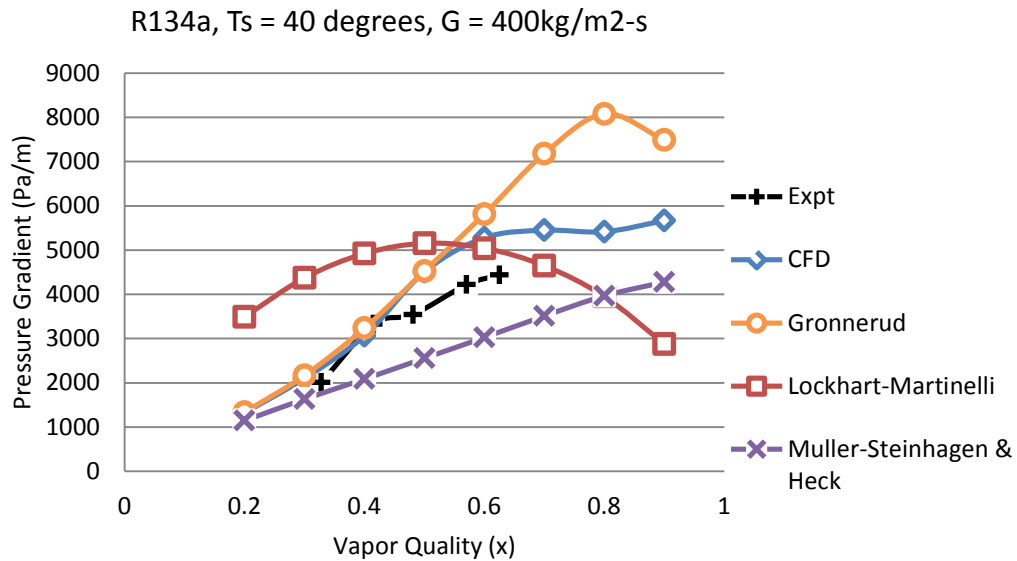


Figure 5-2: Pressure drop comparison, R134a, $T_s = 40^\circ\text{C}$, $G = 400\text{kg/m}^2\text{-s}$

The graph shows a plot of pressure gradient vs vapor quality for the experimental results [18], the CFD results and the correlations result (*Gronnerud*, *Lockhart-Martinelli* and *Muller-Steinhagen & Heck*) [18]. The graphical representation shows that the experimental results line is being best emulated by the CFD results line than any of the correlations result line. Amongst the correlational results, Gronnerud correlation predictions are best after CFD. Lockhart-Martinelli over predicts all the results while Muller-Steinhagen & Heck under predicts all the results.

5.1.3. Scenario 3

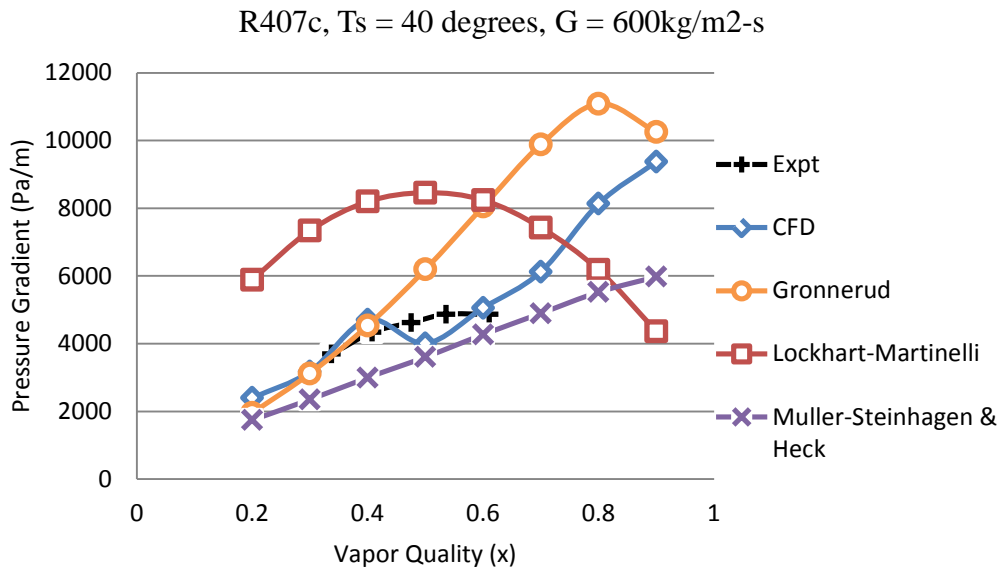


Figure 5-3: Pressure drop comparison, R407C, $T_s = 40^\circ\text{C}$, $G = 600\text{kg/m}^2\text{-s}$

The graph shows a plot of pressure gradient vs vapor quality for the experimental results [18], the CFD results and the correlations result (*Gronnerud*, *Lockhart-Martinelli* and *Muller-Steinhagen & Heck*) [18]. The graphical representation shows that the experimental results line is being best emulated by the CFD results line than any of the correlations result line. Amongst the correlational results, Gronnerud correlation predictions are best after CFD. Lockhart-Martinelli over predicts all the results while Muller-Steinhagen & Heck under predicts all the results.

5.1.4. Scenario 4

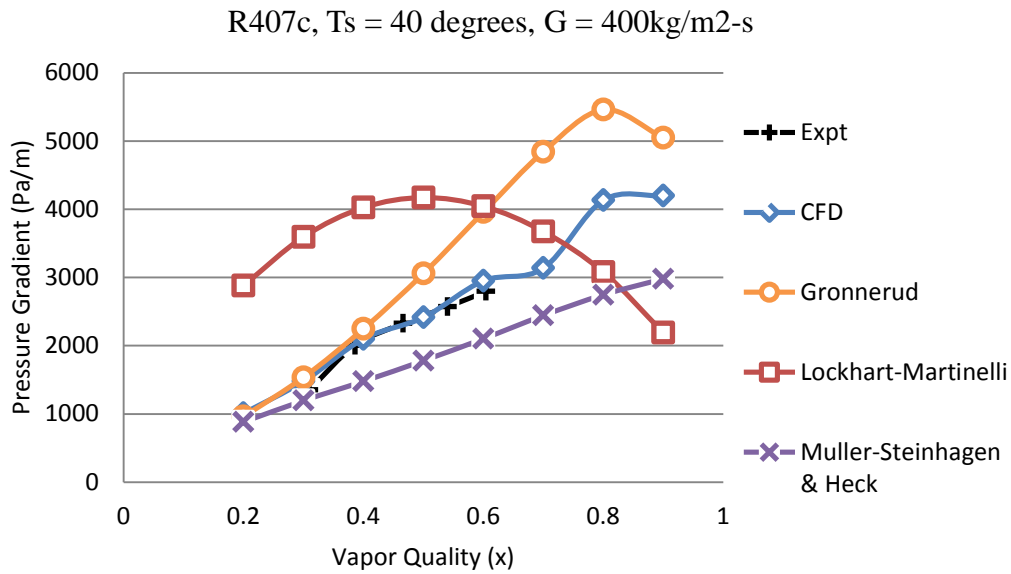


Figure 5-4: Pressure drop comparison, R407C, $T_s = 40\text{c}$, $G = 400\text{kg/m}^2\text{-s}$

The graph shows a plot of pressure gradient vs vapor quality for the experimental results [18], the CFD results and the correlations result (*Gronnerud*, *Lockhart-Martinelli* and *Muller-Steinhagen & Heck*) [18]. The graphical representation shows that the experimental results line is being best emulated by the CFD results line than any of the correlations result line. Amongst the correlational results, Gronnerud correlation predictions are best after CFD. Lockhart-Martinelli over predicts all the results while Muller-Steinhagen & Heck under predicts all the results. Thus from all the above set 1 results the conclusion can be drawn is that the CFD results are better predicting the experimental results than any of the analytical correlations.

In general deviation terms, the CFD results give about 5% – 10% deviation from the experimental results, while the analytical correlational results give about 30% - 40% deviation from the experimental results. Thus the validity of the two phase CFD analysis method is verified.

Now this set up of two phase flow analysis is applied to set 2 part of the CFD analysis. The set 2 results are carried out at a saturation temperature of 25 degree Celsius for both R1234yf and R134a and pressure drop results are compared.

5.2. Set 2

5.2.1. Scenario 1

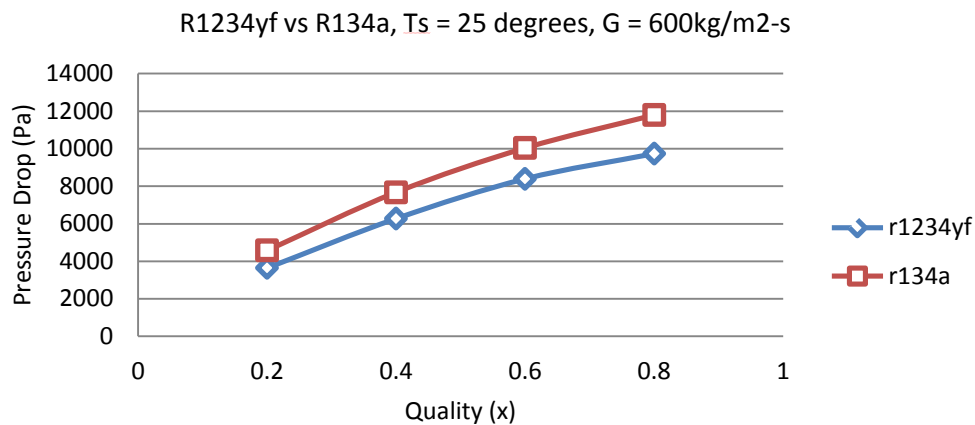


Figure 5-5: Pressure Drop comparison, R1234yf vs R134a, Ts = 25 degrees, G = 600kg/m²-s

5.2.2. Scenario 2

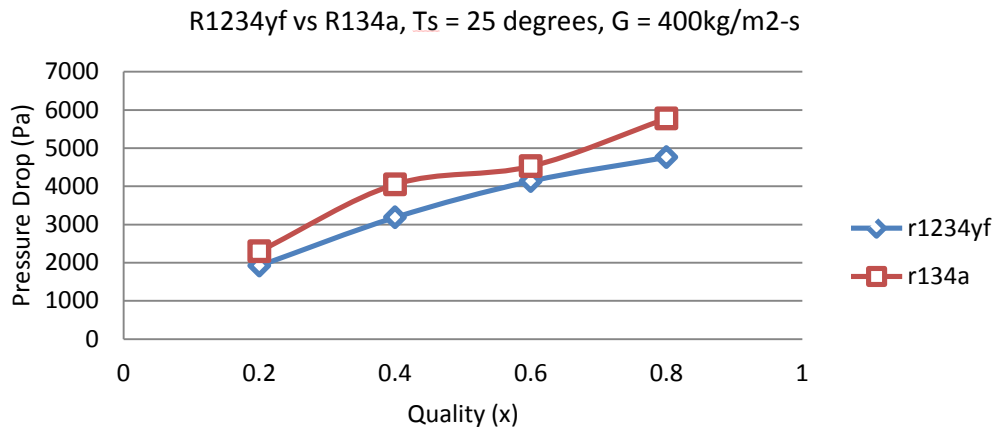


Figure 5-6: Pressure Drop comparison, R1234yf vs R134a, Ts = 25 degrees, G = 400kg/m2-s

From the above scenarios of pressure drop results, the conclusion can be drawn as that at any flow rate and at any required vapor quality, the pressure drop of R1234yf is less than that of R134a. Now on considering the following equation,

$$Pumping\ Power = \frac{Flow\ Rate * Pressure\ Drop}{Efficiency}$$

It is evident that if the pressure drop is less at any particular flow rate at a particular efficiency of pump, then the pumping power will be less for that refrigerant. Thus the pumping power required by the compressor for R1234yf will be less than that of R134a at any flow rate and efficiency. This proves the commercial advantage of R1234yf over R134a along with the environmental.

Chapter 6

CONCLUSION AND FUTURE WORK

6.1. Conclusion

- ✓ The two-phase on chip cooling method was proposed.
- ✓ A CFD model for 2 Phase flow of R134a and R407c was established.
- ✓ Pressure Drop results from CFD were compared with the that of the experiment and analytic correlations.
- ✓ CFD results better predicted the pressure drop.
- ✓ R1234yf was proposed as an alternative for R134a in the refrigeration cycle.
- ✓ CFD analysis was carried out to compare the pumping power for R1234yf and R134a.

6.2. Future Work

- ✓ Experimental work can be done on the available cold plates with using two-phase on chip cooling with refrigerants as the major cooling medium.
- ✓ Natural refrigerants such as R600a (Iso-Butane), R1270 (Propylene), R1150 (ethene/ethylene), R170 (ethane) can also be used.

References

- [1] EPA, Report to Congress on Server and Data Center Energy Efficiency Public Law pp. 109e431, U.S. Environmental Protection Agency, 2007.
- [2] J.G. Koomey, Estimating Total Power Consumption by Servers in the U.S. and the World February 15,, Analytics Press, Oakland, CA, 2007.
- [3] R.R. Schmidt, B.D. Notohardjono, High-end server low-temperature cooling, IBM Journal of Research and Development 46 (2002) 739e751.
- [4] Park, J.E., 2008. "Critical heat flux in multi-microchannel copper elements with low pressure refrigerants", PhD Thesis, École Polytechnique Fédérale de Lausanne, Switzerland.
- [5] J. Lee, I. Mudawar, Low-Temperature two-phase microchannel cooling for high-Heat-Flux thermal Management of Defense electronics, IEEE Transactions on Components and Packaging Technologies 32 (2) (2009) 453e465.
- [6] Y. Madhour, J.A. Olivier, E. Costa-Patry, S. Paredes, B. Michel, J.R. Thome, Flow Boiling of R134a in a Multi-Microchannel Heat Sink with Hotspot Heaters for Energy-Efficient Microelectronic CPU Cooling Applications, IEEE Transactions on Components, Packaging and Manufacturing Technology 1 (6) (2011) 873e883.
- [7] Hammad, M.A., Alsaad, M.A., The use of Hydrocarbon Mixture as Refrigerants in Domestic Refrigerators, Applied Thermal Engineering, 19 (1999) pp. 1181-1189.
- [8] https://en.wikipedia.org/wiki/Two-phase_flow
- [9] P. Bhramara, V. D. Rao, K. V. Sharma, T. K. K. Reddy, CFD Analysis of Two Phase in a Horizontal Pipe – Prediction of Pressure Drop.
- [10] John G. Collier, Convective Boiling and Condensation. Mc Graw Hill, 1972. ch.2.
- [11] C. Tribbe, H. Muller-Steinhagen, An evaluation of the performance of phenomenological models for predicting pressure gradient during gas- liquid flow in horizontal pipelines, Int. J. Multiphase Flow. Vol.26, 2000, pp. 1019-1036.

- [12] H. Muller-Steinhagen, K. Heck, A simple friction pressure drop correlation for two-phase flow in pipes, Chem. Eng. Process, vol. 20 , 1986, pp. 297-308.
- [13] Engineering Data Book III, Wolverine Tube Inc., Revised in 2006. ch.13.
- [14] M.B. Ould-Didi, N. Kattan, J.R. Thome, Prediction of two phase pressure gradients of refrigerants in horizontal tubes, Int. J. Refrigeration. Vol. 25, 2002, pp.935-947.
- [15] J. Moreno Quibe'n, J.R. Thome, Flow pattern based two-phase frictional pressure drop model for horizontal tubes, Part I: Diabatic and adiabatic experimental study, Int. J. Heat Fluid Flow, vol.28, Issue. 5,2007, pp. 1049-1059.
- [16] J. Moreno Quibe'n, J.R. Thome, Flow pattern based two-phase Frictional pressure drop model for horizontal tubes, part II: New phenomenological model, Int. J. Heat Fluid Flow, vol.28, Issue.5, 2007, pp. 1060-1072.
- [17] L. Wojtan, T. Ursenbacher, J.R. Thome, Investigation of flow boiling in horizontal tubes: part I e a new diabatic two phase flow pattern map, Int. J. Heat Mass Transfer. Vol. 48, 2005, pp. 2955-2969.
- [18] P. Bhramara, K. Reddy, K. V. Sharma, Experimental and CFD analysis of two phase flow of refrigerants inside a horizontal tube for the evaluation of pressure drops.
- [19] ANSYS FLUENT Help Manual, Release 16.1.
- [20] REFPROP mini

Biographical Information

Aniket Ramchandra Kalambe was born in Mumbai, India. He received his Bachelor's degree in Mechanical Engineering from Mumbai University, India in 2012. He completed his Master of Science degree in Mechanical Engineering at the University of Texas at Arlington in May 2015.

His primary research area is Computational Fluid Dynamics. He worked as Research and Development Intern at Peerless Mfg. Co. during his Master's education. He has worked on the two phase flow CFD analysis of refrigerants in condenser pipe for predicting pressure drop and pumping power as his thesis topic.

He joined the EMNSPC research team under Dr. Dereje Agonafer in December 2013.